CLARIFYING LAB DESIGN

After much research, the National Institutes of Health can now provide a methodology for optimization of laboratory hoods

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

Division of Engineering, National Institutes of Health, USA

ABASTRACT

A research program was undertaken by the National Institutes of Health (NIH) to investigate ventilation performance of different laboratory configurations and their effect on the laboratory hood. The intention is to provide a basis for guidelines aimed at maximizing laboratory hood containment. This research showed that laboratory air movement is strongly coupled with the performance of the laboratory hood in terms of containment, and that when all other parameters remain constant, a direct link between changing one parameter (such as distance of the supply air diffuser from the hood face) and the hood performance can be demonstrated. Previous work has been almost entirely based on an empirical approach, but this by its nature has limited the scope of variations in design due to the prohibitive cost of modifying real installations and accuracy of the measurement devices. As a result, design guidance has been extremely limited, attempting to identify gross simplifications to ensure good hood containment. The growing awareness of health and safety issues (and thus the need to limit exposure to many substances) makes this research essential in order to provide an understanding of the way in which complex interactions of room air flows can affect hood containment performance.

INTRODUCTION

While millions of dollars are spent for building and renovating laboratories, and detailed design changes are made to laboratory hoods to improve performance, current standards do not address different laboratory operating conditions and configurations, making only limited reference to the requirements for controlled laboratory air movement. The ventilation requirements for laboratories provide a unique demand on HVAC design. In particular, there is a need to control the migration of airborne contaminants and maximize the laboratory hood containment. Further, it is necessary to provide sufficient ventilation air to accommodate fresh air requirements for staff and to maintain an acceptable thermal environment. It is well accepted that room geometry, mechanical HVAC equipment, diffuser placement, and operational procedures within a laboratory all play a role in the containment performance of laboratory hoods and safety cabinets. Literature on the subject of laboratory hood containment is sparse. Recommendations from the literature do not deal with the complexities of laboratory layout and equipment placement that exist at most of research facilities. Some empirical information derived in a generic laboratory is available on the effect of airflow velocities near the hood face on laboratory hood containment. The sophisticated research conducted in laboratories such as at the NIH necessitates the use of a great deal of equipment within the laboratory. This large amount of equipment degrades the ability of the hoods to perform satisfactorily and also tends to limit the design effectiveness of the supply diffusers, thus aggravating the problem. Current laboratory hood guidelines and standards are designed to ensure containment of any contaminant release that may occur in the hood. In practice, these cannot address the fact that the performance is inextricably linked with the air movement in the laboratory around the sash opening. This paper concentrates on data that describe the complex interaction of the room airflow in relation to the hood. The results can be used to optimize the ventilation and contaminant removal effectiveness.

METHODS

Air flow and heat transfer within a fluid are governed by the principles of conservation of mass, momentum, and thermal energy. In order to predict the airflow and temperature, as well as the distribution of contaminants at any given point in space, Computational Fluid Dynamics (CFD) techniques are used to represent the fundamental laws of physics describing fluid flow and heat transfer. The science of computational fluid dynamics is made up of many different disciplines from the fields of aeronautics, mathematics, and computer science. A scientist or engineer working in the CFD field is likely to be concerned with topics such as stability analysis, graphic design, and aerodynamic optimization. CFD may be structured into two parts, generating or creating a solution, and analyzing or visualizing the solution. Often the two parts overlap, and a solution is analyzed while it is in the process of being generated in order to ensure no mistakes have been made.

The governing equations of Computational Fluid Dynamics represent the conservation of mass, momentum, and energy for a fluid continuum. The conservation of mass states that mass cannot be created or destroyed, and the conservation of energy is similar. The conservation of momentum is simply Newton's Law of Motion (force = mass x acceleration) that is cast in a form suitable for fluid dynamics. Because the governing equations are the three conservation laws, they are also referred to as the conservation law equations. The governing equations receive their name because they determine the motion of a fluid particle under certain boundary conditions. The governing equations remain the same, but the boundary conditions will change for each problem 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. The Navier-Stokes Equations are a set of partial differentia equations that represent the equations of motion governing a fluid continuum. The set contains equations for , mass conservation, three components of momentum conservation energy conservation, and etc. In addition, certain properties of the fluid being modeled, such as the equation of state, must be specified. The equations themselves can be classified as non-linear, and coupled. Non-linear, for practical purposes, means that solutions to the equations cannot be added together to get solutions to a different problem (i.e., solutions cannot be superimposed). Coupled means that each equation in the set of five depends upon the others so that they must all be solved simultaneously. If the fluid can be treated as incompressible and non-buoyant, then the conservation of energy equation can be de-coupled from the others and a set of only four equations must be solved simultaneously, with the energy equation being solved separately, if required. For this research FLOVENT[1] CFD software from Flomerics Ltd., UK was used.

This research took over 4,800 Computer hours (CRAY equivalent) and over 4000 man-hours. During the course of this research several progress meetings were held and the findings up to date were discussed. The meeting was attended by some 50 key personnel covering the full spectrum of laboratory design and operation, including architects, engineers, industrial hygienists, scientists, and experimentalists from a multitude of organizations such as: The National Cancer Institute, The National Institute of Environmental Health, Sciences, The Center for Disease Control and Prevention, Massachusetts Institute of Technology, Harvard University, Pennsylvania State University, Johns Hopkins University, Duke University, University of North Carolina at Chapel Hill, The American Society of Heating, Refrigerating and Air- conditioning Engineers technical committee for Laboratory Systems, The American Institute of Architects, The Building Services Research and Information Association, England and many representatives from the private sector. The research shows the sensitivity of hood containment performance to small changes in design and configuration. 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. The Address is http://des.od.nih.gov/farhad/cover.htm.

RESULTS

The configuration parameters varied in the study include lab size, hood position, nominal hood face velocity, supply diffuser type, supply diffuser layout, room ventilation rate, make-up air, supply air temperature, and the presence or absence of a scientist in front of the hood. Three parameters were selected as the primary fume-hood containment indicators. They are:

Sash leakage factor: This measures the leakage through the sash opening as the fraction of the contamination released inside the hood that leaks back into the laboratory against the flow. This was found to correlate well with the level of turbulence in the air.

Box leakage factor: This measures the leakage that makes its way further out into the body of the laboratory in terms of the escape of contaminants from an imaginary 12-in box placed just outside of the sash opening. This was found to be a result not only of the turbulence that induces leakage through the sash opening, but also because of air currents immediately outside the sash opening.

Box/sash leakage proportion: This provides box performance independently of sash performance, and can also be defined as the proportion of the contaminants reaching the box that leak out through the box.

---

[1] Mention of company names or products does not constitute endorsement by the National Institutes of Health

Specific Recommendations

The following is some of the finding of this research. The 520 page report illustrates the data shown using scatter diagrams of sash leakage factor vs. box / sash leakage proportion and flow diagrams. This data shows 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 through the hood, thus preventing the room filling with particles. These data largely represent new knowledge.

Hood position

Six single-fume hood laboratory configurations were studied with the hood at the center of the long side wall. These studies were then repeated with the hood at the corner of the lab against the long wall and at the center of the short wall.

The results indicate that the corner position performs substantially better in terms of both sash and box leakage for different supply air diffusers and different laboratory sizes. The only exception is when the hood at the corner is adjacent to the transfer grille. In this case, the jet from the transfer grille passing just above and in front of the sash opening falls down into the working zone of the hood as it mixes with the cool supply air from the diffuser. It thus falls down in front of the open sash to the detriment of hood containment in terms of box/sash leakage proportion.

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

Bulkhead effect

The effect of extending the fume hood cabinet to the ceiling with a bulkhead was studied in terms of six single-hood lab configurations, each with either a bulkhead or absent. Containment was found to vary with lab configuration.

When thin high-velocity jets from a diffuser meet above and in front of the sash opening, the bulkhead forces the jets down directly in front of the open sash, increasing both sash and box leakage. With larger diffusers, the effect is different, in that only box leakage is affected. In both cases, the resulting jet impacts the air entering the hood, thereby reducing containment.

This does not hold true if the jets are thicker and lower in velocity, since they then feed the flow into the hood rather than disrupting it. In these instances, leakage is reduced by the presence of bulkhead. The latter effect is likely with a bulk-flow diffuser such as a Total Air Diffusers. However, care should be taken to avoid having the jet roll past the open sash. This can easily occur with a down-flow diffuser, causing a loss of box containment wherever the bulkhead acts to enhance the circulation.

Recommendation: A bulkhead can reduce leakage when:

- using a diffuser layout that gently feeds low-velocity air to the hood
- avoiding a diffuser layout that generates high-velocity thin jets across the face of the hood from above
- avoiding the use of a down-flow diffusers that cause roll in front of the hood.

Effect of diffuser blanking

When a diffuser is close to a laboratory hood or a side wall, it is common to blank the jet in the direction of the hood or wall in an effort to protect containment. Doing this can be dangerous, if the paths of the other jets create a circulation around the laboratory that disrupts hood performance.

Recommendation: Avoid blanking when the increased-velocity jets have a path back to the hood.

Effect of jet thickness

While square diffusers of the same size are often assumed to perform similarly, jet thickness and angle can substantially affect the flow patterns generated, and thus affect hood containment. Three simulations were used to investigate the effect of changing jet thickness (and velocity to maintainvolume flow rate). A typical design having 12-in. square diffusers laid out on the quarters was used as a reference. When the jet thickness was doubled, the velocity was halved, and vice versa.

A thick jet does not have enough momentum, so it dumps into the working zone of the hood. This creates an asymmetric flow into the hood from below, causing leakage from the sash opening. Further jet thickness reductions increase penetration of the thin higher-velocity jets, which cause a down-flow in front of the sash opening. Although the thin jet reduces the sash leakage factor continuously, this is at the risk of increasing the box/sash leakage proportion.

Effect of diffuser position

The variation in hood containment performance was tested as a 24-in. square diffuser was moved laterally from one side of the hood, to near the edge of the hood, to in front of the hood. The overall result in the laboratory was found to be dominated by leakage at the box. The greatest leakage into the bulk of the laboratory occurs when the diffuser is in line with the center of the hood. Although the sash leakage is reduced by approximately 5% by placing the diffuser in front of the hood, the box/sash leakage proportion is increased by around 70%, resulting in much larger spread around the lab.

Recommendation: Place a square diffuser on the centerline and in front of the hood to minimize the sash leakage and thus minimize exposure for the scientist. This however does not minimize the leakage into the laboratory.

With the diffuser in front of the hood, the leakage was reduced by the presence of a bulkhead. Although moving the diffuser further away did reduce the sash leakage factor, the improvement was less than 3%, a figure that is constrained by the width of the laboratory. Further, the box/sash leakage proportion increased by almost 12%.

Moving the diffuser closer to the hood reduces the time the jet has to thicken, preventing the thick jet from forming down the front of the hood and allowing more air to escape sideways from the bulkhead at a high level. This results in substantially reduced leakage into the lab—the sash leakage factor is reduced by 14% compared with the middle position, and the box/sash leakage drops by 39%.

Where there is insufficient distance to move the diffuser well away from the hood, position the diffuser in line with the center of the hood, close to the bulkhead, so that the jet cannot fully develop.

hoods on same wall

In general, simulations involving two hoods show that it is extremely difficult to achieve as good containment as with just one hood when the ventilation rate is dominated by the hood flow rate.

Performance for two hoods on the same wall were simulated for separations from 2 ft to 8 ft and for a low transfer grille flow rate of 66 cfm.

The leakage from the hoods increases especially when they are close together, however leakage is different for each hood in the pair because of their different position in the room. When the two hoods are only separated by 2 ft, the highest sash leakage factor and highest box/sash leakage proportion are almost double compared with the leakage for a single hood . The highest sash leakage factor is reduced to about 65% above that for a single hood when hood separation is increased to 4 ft or 6 ft. Box/sash leakage proportion for these larger separation is only about 40% above that of a single hood. Leakage is further reduced at 8 ft separation possibly because on hood is now near a corner.

hoods on opposite walls

The containment is generally better than for hoods on the same wall except, however, where two hoods are opposite or separated by just 2 ft.

hoods on perpendicular walls

Hoods on perpendicular walls perform best, and can achieve a box/sash leakage proportion that is lower (more than half) than for the single hood, with sash leakage less than 20% higher.

Recommendation: Separate hoods by more than 4 ft. Placing two hoods on perpendicular walls is likely to produce a better performance than placing them on opposite walls. In turn, either of these configurations can achieve lower leakage than hoods on the same walls. Maximize the distance between the two hoods and the transfer grille.