Section I

INTRODUCTION

SECTION I : TABLE of CONTENTS

1 INTRODUCTION

1.1 Objective of the Research

This document describes a research program undertaken by the National Institutes of Health (NIH), Office of Research Services (ORS), Division of Engineering Services (DES) 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 will help laboratory planners and mechanical system designers to minimize the adverse impact of the laboratory layout and ventilation system on hood containment performance. This work was corrected in collaboration with Flomerics Ltd. In the United Kingdom.

Previous research has shown 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.

1.2 Background

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 facilities such as those at the NIH. 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 research 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. However, this program of work will not address issues related to the design of the laboratory hood itself.

A wealth of experimental work has been undertaken to investigate laboratory hood performance. Indeed, it is standard practice to evaluate new designs not only by measuring uniformity of flow into the hood through the sash opening but by measuring leakage, often according to the American Society of Heating Ventilating and Air-Conditioning (ASHRAE) 110 standard, which specifies tracer gas source and measurement quantities and positions in order to identify the containment performance. The biggest problem in such an approach is that the tests are costly and they identify the problem only after the laboratory has been designed and built.

1.3 Project Summary

At project conception, NIH recognized that a very comprehensive study could be undertaken using Computational Fluid Dynamics (CFD) to predict air movement, heat transfer, and contaminant dispersion in the laboratory setting. CFD is an advanced three dimensional mathematical technique that can be used to compute the motion of air, water, or any other gas or liquid. If you were learning fluid dynamics as recently as, say 1960, you would be operating in the “two-approach world” or theory and experiment. However, the advent of the high-speed digital computer combined with the development of accurate numerical algorithms for solving physical problems on these computers has revolutionized the way we study and practice fluid dynamics today. It has introduced this fundamentally important approach of CFD. The CFD approach provides a tool with which you can carry out numerical experiments and in this way undertake the comprehensive study envisaged.

It was clear from the beginning that the disadvantage of this approach would be the vast quantities of data generated for each and every simulation, compounded by undertaking many simulations. This project presents these data in a simplified manner to allow the designer or specifier to use it to identify a satisfactory laboratory configuration, or to undertake measurements that will confirm current laboratory performance. To achieve this, a significant effort during the project centered on developing analysis techniques for identifying the relative performance of the hood for each configuration modeled, and then determining the links between the containment, or loss, and the design configuration.

To provide the database, the project has involved more than 250 laboratory configurations using CFD. For this project, the CFD program used was FLOVENT [1.1] by Flomerics that is specifically designed for analyzing ventilation for the built environment. The intent was to investigate how changes in laboratory design affect hood containment in order to provide a basis for new design guidelines. There are in fact an infinite number of combinations of laboratory and ventilation configuration and hence reaching a conclusion about best practice is almost impossible without a considerable study focused on the effect of the many parameters. The different configurations analyzed for this research are as follows, and Section III - “Modeling”, details the actual configurations simulated:

Laboratory size
Hood position
Nominal hood face velocity
Supply diffuser type
Supply diffuser layout
Room ventilation rate (ACH)
Make-up air
Supply air temperature
Presence or absence of a scientist in front of the hood

When calculating the flow in the built environment, one of the most important effects is that of turbulence. This is particularly so in the laboratory because the leakage at the hood face normally can be entirely accounted for by turbulent diffusion back against the fast inward moving convective flow. Those not familiar with this type of physics can easily recognize the effects through a simple everyday happening.

Consider the smoke rising from a cigarette (figure 1.1). Initially, the smoke rises in a well mannered flow as a narrow column due to its warmth and consequent positive buoyancy. It spreads very little horizontally, and the spreading it undergoes is due purely to molecular diffusion. After rising in this manner for a short distance, it bursts into a chaotic flow with the smoke spreading laterally as quickly as it rises. This lateral motion is dominated by turbulence, the sideways motion being caused by random fluctuations in air movements superimposed on the initial buoyant convective flow. This spread is known as turbulent diffusion, the accurate prediction of which determines the ability of a laboratory hood to contain a contaminant.

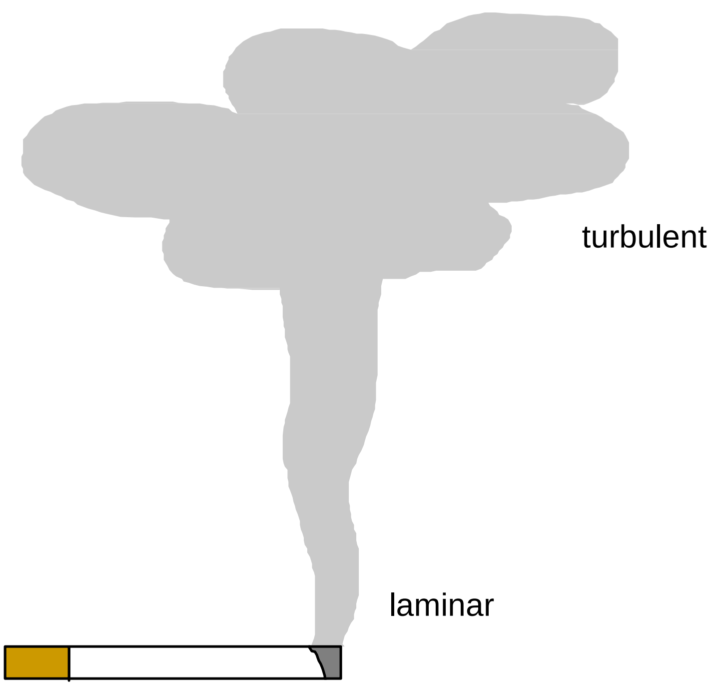

Figure 1.1 Schematic spread of smoke rising from a cigarette.

To determine the flow, heat transfer, and contaminant distribution, it is necessary to calculate eight different quantities on a grid of many locations in the laboratory. Calculating pressure, three velocities, temperature, two turbulence quantities, and concentration on a grid of around 80,000 to 100,000 calculation points or cells generates some 640,000 to 800,000 items of data describing the conditions. The equations for these data have to be solved iteratively until the convergence criteria is satisfied. This is an extremely computer intensive operation that requires the use of powerful state-of-the-art workstations.

Some 250 different configurations have been undertaken during this research, which has taken over 4,800 computer hours (CRAY equivalent) and 4000 man-hours to complete. It should be noted that this work was carried out using Systems International (S.I.) units, that is kilogram, meter, second, Pascal (1 Newton per square meter), watt (1 joule per second), and degrees Celsius. The conversions to Imperial units are approximate and are displayed for reference only.