Modeling the Flow of Gas and Particulates Within a Respirator

Kathryn M. Butler

NIST, BFRL (Building and Fire Research Laboratory)
100 Bureau Drive Mail Stop 8665
Gaithersburg, MD 20899-8665
Email: kathryn.butler@nist.gov

A computational model of the flow inside the facepiece of a respirator has several potential benefits. The respiratory uptake of toxic gases and fine particulates in the presence of a leak can be predicted. A model can supplement experimental studies of fit testing and respirator performance under a variety of work rates. It can also determine the best locations to place sensors for representative measurement of conditions within the facepiece. This paper presents the progress on a computational fluid dynamics (CFD) study of the flow and pressure fields within a respirator.

The complex geometry within the facepiece is obtained by converting three-dimensional scans of actual heads and masks into data sets for entry into the CFD software. For comparison with experimental results, laboratory headforms are prepared. CAD files of facepieces are used when available from the manufacturer.
Computational fluid dynamics (CFD) numerically simulates fluid flow by solving the equations of motion. The solutions can be obtained for variables that are not easily measured and in detail that is impossible to achieve in experiments, especially in three-dimensional space. Visualization tools that display the computational results as they vary with time and space enhance our understanding of the flow phenomena.

The flow of gases and of particles with mass in a respirator under normal operating conditions will be presented as a base case. An investigation of the effects of leaks and breathing patterns from resting to high work rates will follow.