A laminar fluid flow model for study of ventilation systems in micro-electromechanical systems (MEMS) clean room

Srinivasa Rao Vutla¹, Srinivasa Prakash Regalla ¹*, Kannan Ramaswamy ²
¹Department of Mechanical Engineering
²Department of Physics
Birla Institute of Technology & Science Pilani, Hyderabad Campus, Hyderabad-500078, Telangana, India
*Email: regalla@hyderabad.bits-pilani.ac.in, Phone: +91-9010202879

Abstract. This paper reports the simulation based analysis results of air flow in a MEMS clean room aimed at predicting the air borne distribution of 0.5 μm particles as per the ISO standard. These particulate contaminants, if not flushed by proper design of the ventilation system of the clean room can get deposited and accumulated on the MEMS devices under fabrication, contributing possibly to the affected functionality of the device, particularly high precision devices such as single cantilever type precision MEMS sensors and actuators. The results of computational fluid dynamics (CFD) based flow analysis done in COMSOL™ estimated for adverse conditions such as a desired ISO-5 cleanliness degrading to ISO-8 cleanliness due to poor maintenance have been presented. Based on the flow conditions, conclusions on the particle accumulation have been drawn.

Keywords: MEMS clean room, particle flow analysis, contamination, ventilation, bottom ventilation, side ventilation

1. Introduction

High precision MEMS devices depend on the freedom from contamination for delivering expected performance. Performance of precision multi-functional MEMS devices with moving parts is sensitive to cleanliness during fabrication in cleanroom. Deposition of contaminant particles in sensitive locations of the device can degrade its performance. Contamination of MEMS devices can happen during their fabrication in the clean room due to many causes, the most likely cause being the improper maintenance or power outage conditions resulting in deviations of temperature, humidity and particle count out of the ranges prescribed. Although clean rooms are generally well maintained with the help of several instruments and sensors, knowing the effect of adverse conditions of cleanliness on levels of contamination will help predict deviations from expected performance of the MEMS device [1]. The successful integration of micro-mechanical components is critically dependent on minimization of contaminant particle deposition at all stages of fabrication and packaging, which are typically done in a cleanroom. It is important therefore to clearly understand the particulate density and distribution in the cleanroom, temperature and humidity conditions for common arrangements of equipment and human operators.

ISO-5 (maximum 3520 particles of size 0.5 μm or larger per m³) level of cleanliness is generally used for MEMS component manufacturing in cleanrooms [2]. In applications involving structures with further smaller end of the size from micro to nano electrical systems, the decoupling of mass and stiffness and the influence of surface contamination become important [3]. The sensitivity and response time of the MEMS sensors are affected in the presence of contamination. MEMS
cleanrooms need innovatively designed with mini-environments, novel ventilation arrangements, air flow systems, air filters and air handling units and water chillers to maintain the necessary tolerances of particulate matter within the specified ISO-5 standard. The knowledge of optimizing these for biotechnology cleanrooms is abundantly available but the same for MEMS cleanrooms is not available. The ‘adverse conditions’ of cleanroom parameters deviating from standard and specified tolerances and their effect can be studied by experimentation and numerical simulation of particle and contaminant depositions on the component surfaces, non-uniform thermal distortions and condensation of moisture.

The sources of contaminant particles in a MEMS cleanroom include human operators, machine surfaces, internal surfaces of the walls, ventilation systems and fabrication processes themselves, as they may release liquid and gaseous ions. The release of these positive and negative air ions were found especially to be promoted by high-luminosity lighting, which is a requirement of the clean room, with the corona discharge of the lighting reacted with the surrounding air resulting in creation of ions [4]. Study of the effect of placing the FFUs at different locations in mini-environments and larger clean rooms on contamination enabled optimum design of placement of Fan Filter Units (FFU) placed in the ceiling plenum of the MEMS clean room [5]. These design studies included trials of placing additional FFUs and reducing air-flow rate of exhaust fans to contain high contamination. Micro CMM probe performance in measurement of coordinate dimensions of complex geometry parts was found affected contaminant particle depositions during the fabrication in a MEMS cleanroom [6]. Scanning electron microscopy (SEM) and energy-dispersive x-ray spectroscopy (EDX) were used to determine the shape and size of contaminant on the ruby tip of the CMM probe.

Mini-clean room arranged within a regularly sized ISO-5 cleanroom environments were found provide extra cleanliness and its design, operation and effectiveness were studied [7]. This work recommended building mini-environments within the main cleanroom. These mini-environments are meant to house micro-fabrication facilities to achieve higher levels of cleanliness up to ISO-5 while the larger enclosing cleanroom may still have been designed with ISO-6 cleanliness. Similar studies on effectiveness of different ventilations systems on cleanliness were done [8-16] for a variety of applications including, hospital operation theatre, food industry and residential building but no research work exists relevant to MEMS cleanrooms.

In the present work, the results of fluid flow patterns obtained in a MEMS cleanroom by conducting computational fluid dynamic analysis for various ventilation conditions are presented and discussed. The novelty of the present work lies in considering several different ventilation boundary conditions including bottom vents and the effort to derive certain insights into particle flow patterns through fluid flow patterns. Since actual perforated tiles have been simulated in this present computational model, a full three dimensional analysis proving cumbersome and highly time consuming, a two dimensional cross-sectional model at cross-section containing maximum details has been analyzed.

2. Numerical simulation of clean room ventilation

In the present work, computational fluid dynamic (CFD) model of the cleanroom is analyzed using finite element method. The cleanroom with typical arrangement of equipment and humans working in it is modeled as a two-dimensional control volume. Since the 0.5 μm particles have insignificant mass, it is assumed that the flow of these particles essentially governed by the fluid flow, resulting in the possibility of predicting the path and presence of particles in the domain.

The CFD simulation model presented in this paper is for the 536 sq. ft. MEMS clean room present in the authors’ Institute. In this section, first the details of the geometric setup of the clean
room, flow conditions along with the arrangement of equipment and the supporting furniture are explained. The velocity of air at the exit of the fan filter units (FFU) in the ceiling, the arrangement combinations of the FFUs in the ceiling and the level of contamination sources in the air in accordance with the ISO level of cleanliness have been simulated as per the regular practices and the same are also explained in the next section.

### 2.1. Cleanroom CFD simulation model setup

The objective is to predict the flow paths and locations of accumulation of the contamination particles in the ISO-5 clean room environment. Fig. 1 shows a two-dimensional approximation of a typical clean room (4.6m×2.4m) fabrication room. It consists of two sets of workbenches (1mx0.7m) where film deposition/etching takes place along with the associated personnel operating these processes.

A quantitative estimation of the contamination level and its effect on the performance of the device necessitates the understanding of the nature and magnitude of the contamination and its geometry of distribution over the device. Several different modeling approaches have been developed for predicting the flow characteristics of fluid-particle suspensions, ranging from discrete, particle-based methods to macroscopic, semi-empirical two-phase descriptions. Particle based methods, in which each particle’s path is computed and traced, are suitable when the number of solid particles suspended in the fluid is limited. On the other hand, when there are many particles, particle based methods become computationally very expensive and unwieldy. For such cases, instead, use of a macroscopic or averaged model is recommended. In such average models, the volume fractions of particle concentrations at time \( t=0 \) are specified and the tracking of the volume fractions of the phases is done for \( t>0 \). A further simplified model possible when the particles are of insignificant density is carry out the single phase flow of the fluid and the flow patterns established automatically represent the flow of particles with the fluid, and this is the model presented in this paper. The work was carried out to qualitatively predict the extent of particle deposition on the component using computational fluid dynamic analysis of the clean room.

The clean room roof has six fan filter units (FFU) and the floor has vents (seven lumped vents each of width 0.1m) for the air flow. In a typical ISO-5 clean room air flow occurs at a velocity of \( v=0.5 \text{ m s}^{-1} \) from the upper vents to the lower vents [7]. The contaminants also travel with the air flow and there is a possibility of these getting deposited on the desired cantilever beam being fabricated. In order to understand this process thoroughly we have first carried out a Computation Fluid Dynamic (CFD) analysis of the air flow along with the contaminants. The flow of the fluid is governed by the control volume based conservative laminar continuity and momentum equations [17].

The continuity is given by:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0
\]  

(1)

and the momentum equation (Navier-Stokes equations) are given by:

\[
\frac{\partial \mathbf{u}}{\partial t} + \Delta \left( \rho \mathbf{u} \mathbf{u} \right) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + \rho f_x
\]  

(2a)

\[
\frac{\partial \mathbf{v}}{\partial t} + \Delta \left( \rho \mathbf{v} \mathbf{v} \right) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} + \rho f_y
\]  

(2b)
\[
\rho \frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = - \nabla p + \frac{\partial \mathbf{\tau}}{\partial x} + \frac{\partial \mathbf{\tau}}{\partial y} + \frac{\partial \mathbf{\tau}}{\partial z} + \rho f_z
\]  

(2c)

which can be written in compact form in a single equation as:

\[
\rho \frac{\partial \mathbf{u}}{\partial t} + \rho (\mathbf{u} \cdot \nabla) \mathbf{u} = - \nabla p + \nabla \cdot \left[ - \rho \mathbf{I} + \mathbf{\tau} \right] + \mathbf{F}
\]  

(2d)

where \( \mathbf{u} = u \mathbf{i} + v \mathbf{j} + w \mathbf{k} \) is the velocity vector and \( \mathbf{F} = f_x \mathbf{i} + f_y \mathbf{j} + f_z \mathbf{k} \) is the body force vector, \( \rho \) is density, \( p \) is the hydrostatic pressure and \( \mathbf{\tau} \) is the total stress tensor. Since the entire operation has been assumed to take place at a constant temperature, the energy balance has been ignored in the present finite element analysis.

The strain-rate tensor components are given by

\[
\mathbf{S} = \frac{1}{2} \left( \nabla \mathbf{u} + (\nabla \mathbf{u})^T \right)
\]  

(3)

The Stoke’s fluid flow (Ma < 0.3) here is treated as compressible single phase laminar flow, which obeys the non-Newtonian constitutive law for shear stress in the fluid layers as given below.

\[
\mathbf{\tau} = 2 \mu \mathbf{S} - \frac{2}{3} \mu (\nabla \cdot \mathbf{u}) \mathbf{I}
\]  

(4)

The above mentioned CFD equations have been modelled and solved in COMSOL™ software.

2.2. Finite Element Solution to the Flow Problem

Figure 1 shows the domain and the geometric approximate model considered typical boundary conditions for all CFD analyses conducted in the present work, and the Fig-2 shows the finite element mesh wherein a free meshing triangular element has been used. The flow has been assumed to be laminar because all the obstructions to flow, namely tables and men were modeled to be smooth surfaced. The problem was solved in steady state situation.
Figure 1: Typical ISO-5 clean room set up used for simulations
In the next two following sections, the details of the CFD simulation for two different ventilation conditions have been presented. The first scenario is that of ventilation outlets provided exclusively through floor tiles. The second is that of bottom side outlets. The nature of inlets and fan filter unit configuration are the same for both the models.

2.3. Ventilation through side outlets

The technique of the side ventilation on the walls is an established practice in the clean room design for different applications including operation theaters of hospitals and biotechnology manufacturing plants. In this technique, there is typically no false flooring and the vents of selected height are provided at the bottom of the side walls. The air along with particles are drawn into these vents, wherefrom the double wall structure enables the recirculation of the air into the fan filter units and therefrom into the room again.

Figure-3 shows the domain and boundary conditions of the side vents model. The height of the side vents is a design parameter. The typically used value of 300 mm has been analyzed in the present work. In the model, all the floor tiles are solid and no tiles with vent holes are incorporated. The rest of the model remains as in Figure 1.
2.4. Ventilation outlets through the floor tiles

The second model ventilation in the present investigation consists of the false floor, which is raised from the floor of the ground at 300 mm. The recirculation is achieved by the ventilation through the holes of the floor tiles, which are commercially available Comey Hae-Kwang moulded aluminium panel with very high mechanical resistance, damp- and liquid-resistant, incombustible, non-magnetic (Fig-4).

Different combinations of solid and perforated panels allow different levels of ventilation achieved. The same scenario has been captured in the CFD simulation by appropriately defining the outlet boundary conditions at the floor, as shown in Fig-5.
3. Results and discussion

One sample of CFD analysis results are shown in Fig. 6 to facilitate explanation of the method of inference on the contaminant particle flow and accumulation in the cleanroom under a given set of inlet conditions provided by fan filter units and a given set of outlet ventilation conditions provided by side vents or floor-tile vents.

This figure shows the flow field when only floor-tile vents are provided. As can be seen in the figure, the steady-state flow field is highly streamlined at the entry level when the flow leaves the FFUs. The flow would have continued to be largely unaffected until it reached the floor-tiles under the non-usage conditions of the cleanroom indicated no presence of machines and users. However, when the cleanroom is under regular usage condition, the flow is obstructed by the machines and
people and hence it deviates from straight line streamlines. The deflection of the streamlines results, as can be seen, in several zones of flow reversal, vorticity and stagnation. These undesirable deviations serve as clues to the inference of zones of particle accumulation, stagnation and deposition on MEMS device being processed.

3.1. Contaminant particle distributions in side ventilation model

The flow field with side ventilations is shown in Fig-7. As can be seen, side ventilation prevents any substantial change in flow velocity from inlet (FFUs) to the vents. The flow velocity remains largely unchanged in its path except in the close vicinity of the side vents themselves. The superimposed arrow surface of flow field indicates that flow has not taken

![Flow field obtained by CFD analysis of cleanroom with side ventilation model](image)

The Fig-8 shows the vorticity plot for side vents model. It is evident from this plot little or negligible level of vorticity is created by the side vents, mainly because negligible shear for most part of the flow. The vorticity is the curl of the velocity vector is given in two-dimensional compressible flow by:

$$\vec{\omega} = \nabla \times \vec{v} = \left( \frac{\partial}{\partial y}, \frac{\partial}{\partial z}, \frac{\partial}{\partial x} \right) \times (v, v, 0)$$

(5)

In the side vent model the vorticity is limited to the small zones around corners of the tables.
Fig-8: Vorticity variation in the flow field of the cleanroom

Fig-9 shows the pressure plot, which can used to observe the stagnation points in the fully developed flow region near around the MEMS device on the top of the tables. It is evident from the figure that the pressures acquires high values around the corners of the tables and near around the head of the user. This can indicate possible accumulation of contaminant particles around the MEMS device at the centers of the tables.

Fig-9: Pressure contour plot for side vents model
3.2. Contaminant particle distributions in floor-tile ventilation model

The velocity field with 50% area of floor-tiles having vent holes is shown in Fig-10.

![Velocity field streamlines for 50% area floor-tile ventilation model](image1)

Fig-10: Velocity field streamlines for 50% area floor-tile ventilation model

The Fig-11 shows the velocity field with 75% area of floor tiles having vent holes.

![Velocity field distribution for 75% area floor-tile ventilation model](image2)

Fig-11: Velocity field distribution for 75% area floor-tile ventilation model
The Fig-12 shows the velocity field with 100% area of floor tiles having vent holes.

![Streamline: Velocity field](image1)

**Fig-12: Velocity distribution for 100% area floor-tile ventilation model**

It is clear from the Fig-10, Fig-11 and Fig-12 that the maximum velocity with 50% area floor-tile ventilation is about 60% more than that in the 100% area floor-tile ventilation. However, the flow is far more streamlined in 100% area floor-tile ventilation than in the 50% area floor tile ventilation with more uniform velocity in the entire cleanroom. The results are intermediate with 75% area floor-tile ventilation.

Fig-13 shows the vorticity distribution with 50% area floor-tile ventilation.

![Surface: Vorticity magnitude](image2)

**Fig-13 shows the vorticity distribution with 50% area floor-tile ventilation.**
Fig-13: Vorticity distribution for 50% area floor-tile ventilation model

Fig-14 shows the vorticity distribution with 100% area of floor-tiles.

![Vorticity Distribution](image)

Fig-14: Vorticity distribution for 100% area floor-tile ventilation model

It is evident from the Fig-13 and Fig-14 that the vorticity is significant in 50% area floor-tiles and it occurs around the heads of the users and on the corner zones of the tables. This indicates the possibility of contaminant particles in these zones to get recirculated and ultimately get deposited onto the device surfaces rather than smoothly flushed. The vorticity is insignificant in the 100% area floor-tiles model.

Fig-15 shows the pressure distribution in 50% area floor-tile ventilation model.

![Pressure Distribution](image)
**Fig-15:** Pressure distribution for 50% area floor-tile ventilation model. 

**Fig-16:** Pressure distribution plot in 100% area floor-tile ventilation model.

From Fig-15 and Fig-16, the high values of pressure around the table top zones in 50% area floor-tile ventilation model indicates high levels of flow stagnation resulting in possible accumulation of the contaminant particles on the MEMS devices. The stagnation zones of flow also extend to the sides of the tables in this model. Comparatively, the flow stagnation is far less in 100% area flow-tiles model, indicating better flushing of particles down into the ventilation holes.

### 4. Conclusions

Computational fluid dynamics based numerical simulation of the flow in a MEMS cleanroom has been conducted to predict the contamination particle flow, stagnation and vorticity distributions with a view to predicting the level contamination that the MEMS device being processed in the cleanroom may undergo. Flow velocity arrow distribution, pressure distribution and vorticity distribution have been determined for two major types of ventilation systems, namely, side vents model and floor-tile ventilation model. From the results obtained it may be concluded that the side vents model cannot assure uniform velocity distribution and is prone to some level of flow stagnation around the corners of the tables and users thus resulting in the possibility of contaminant particles getting deposited on the MEMS device. Similarly, 50% area floor-tile ventilation model also results in significant flow stagnation and vorticity around the table top zones and around user heads. This can mean significant levels contaminant particles getting accumulated around these areas. On the other hand 100% area floor-tiles result in negligible flow stagnation and vorticity in the cleanroom. Further studies incorporating turbulence and explicitly modeled initial particle concentration and their eventual stagnation and accumulation areas are necessary to get the in-depth knowledge and preventive design measures for cleanroom interior are necessary.

### References
[1] Noh KC, Kim HS and Oh MD: (2010) “Study on contamination control in a minienvironment inside clean room for yield enhancement based on particle concentration measurement and airflow CFD simulation”, Building and Environment, 45, pp825-831.

[2] Mrad F: (1999) “The characterization of a clean room assembly process”, IEEE Transactions on Industry Applications, 35, pp399-404.

[3] Sadeghian H, Yang CK, Gavan KB, Goosen JFL, Drift EWJMVD, Zant HSJVD, Bossche A, French PJ and Keulen FV: (2009) “Surface Some considerations of effects-induced errors in resonant cantilevers with the laser deflection method”, Journal of Micromechanics and Microengineering, 20, pp400-403.

[4] Whyte W: (2010) “Cleanroom Technology: Fundamentals of Design, Testing and Operation”, John Wiley, USA.

[5] Sakata S and Okada T: (1994) “Effect of hydrated cluster-ion formation in a clean room corona discharge neutralizer”, Journal of Aerosol Science, 25, pp879-893.

[6] Habeb RR and Kinnell PK: (2012) “Evaluation Tip Cleaning for a Micro CMM Touch Trigger Stylus Sensor”, Procedia Engineering, 47, pp306-309.

[7] Xu T: (2007) “Characterization of minienvironments in a clean room: Design characteristics and environmental performance”, Building and Environment, 42, pp2993-3000.

[8] Yang Lu, Jing Liu, Hiroshi Yoshino, Rie Takaki & Genta Kurihara (2015), Study on Distribution and Deposition of House Dust by Different Ventilation Systems in a Full-scale Residential Room, International Journal of Ventilation, 13:4, 417-426

[9] Bin Xia, Da-Wen Sun, Applications of computational fluid dynamics (CFD) in the food industry: a review, Computers and Electronics in Agriculture 34 (2002) 5–24.

[10] Yunlong Liu, Alfred Moser, Kazuyoshi Harimoto, Numerical Study of Airborne Particle Transport in an Operating Room, International Journal of Ventilation Volume 2 No 2.

[11] M. Cheng, G.R. Liua, K. Y. Lama W. J. Caib E L Leeb, Approaches for improving air flow uniformity in unidirectional flow Cleanrooms, Building and Environment 23 pp. 164-173.

[12] Chun Yong Khoo, Chung-Chieh Lee, Shih-Cheng Hu, An experimental study on the influences of air change rate and free area ratio of raised-floor on cleanroom particle concentrations, Building and Environment 48 (2012) 84-88.

[13] Ti Lin b, Yun-Chun Tung a, Shih-Cheng Hu b,*, Yen-Jhih Chen, Experimental study on airflow characteristics and temperature distribution in non-unidirectional cleanrooms for electronic industry, Building and Environment 46 (2011) 1235-1242

[14] Whyte, W., Hejab, M., Whyte, W.M., and Green, G. (2010) Experimental and CFD airflow studies of a cleanroom with special respect to air supply inlets. International Journal of Ventilation, 9 (3). pp. 197-209.

[15] Zhuang RN, Li X and Tu J: (2015) “An Energy Saving Ventilation Strategy for Short-Term Occupied Rooms based on the Time-Dependent Concentration of CO2”, International Journal of Ventilation, 14(1), pp39-52.

[16] Karki KC and Patankar SV: (2006) “Airflow distribution through perforated tiles in raised-floor data centers”, Building and Environment, 41, pp734–744.

[17] Anderson JD: (2012) “Computational Fluid Dynamics: The Basics and Applications”, McGraw-Hill, Inc., Singapore.