Research Progress in Mechanical and Manufacturing Engineering Vol. 2 No. 2 (2021) 399-405

RPMME

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/rpmme
e-ISSN : 2773-4765

Investigation on the Air Filter with Industrial Ducting System

Muhammad Aiman Nasri[1], **Mohamad Farid Sies**[1,*]

[1]Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia, Parit Raja, 86400, Batu Pahat, Malaysia

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2021.02.02.045
Received 02 Aug. 2021; Accepted 27 Nov. 2021; Available online 25 December 2021

Abstract: Air filter is one of the element in Local Exhaust Ventilation (LEV) system that have been widely used in industry to encounter air pollution. The objective of this study is to investigate the velocity distribution and pressure drop in air filter by using computational fluid dynamic (CFD) software. A simulation study was carried out to obtain the velocity and pressure distribution in air filter. This study was performed with three types of filters that is fibrous filter, pleated filter and High Efficiency Particulate Air (HEPA) filter. The analysis of velocity and pressure behavior in air filter is from the inlet to the outlet of the system. The CFD software used in this study is Ansys Fluent Fluid Flow. It can be observed that the pressure drop and the performance of tested filter is influenced by the permeability of the filter. From the result obtained, HEPA filter is the most preferable because it has the least value of pressure drop and the velocity distribution on the system is more balance. Furthermore, high permeability filter is more efficient compared to low permeability filter when it operates in the system.

Keywords: Air Filter , CFD, Velocity Distribution, Pressure Drop

1. Introduction

The rapid growth of the industry now presents significant problems for the human living environment. Based on the observation and studies, one of the most severe problems is indoor air pollution which can easily cause diseases and cancers in humans. One of the most effective ways to encounter this problem is by using the Local Exhaust Ventilation (LEV) system. Local Exhaust Ventilation (LEV) is not a modern form of air ventilation system. The LEV system has been widely used in industrial application which aims to control the transmission of contaminant and extract contaminant locally [1]. The LEV is operating on the concept of capturing contaminants before they are dispersed in the workplace environment.

Air filter is one of the elements in the LEV system that is designed to remove contaminants that are carried in the contaminated air from hood which cannot be discharged into the community environment or to recover materials that have a salvage value [2]. Air filters are usually consist of a sturdy frame, filled with some kind of filter media, and sealed to avoid leaks between the frame and

**Corresponding author: farids@uthm.edu.my*

media. There are several types of air filters that are commonly used in industry such as fiberglass, pleated, carbon and High Efficiency Particulate Air (HEPA) filter and the ones we used in this simulation are fibrous, pleated and HEPA filter.

The two most important parameters to measure air filter performance are pressure drop and capture efficiency. The main factors affecting the filtration performance include filter material characteristics, aerosol characteristics and operating conditions. Experimental models for predicting the pressure drop have been proposed by earlier researchers such as Bortolassi, Miguel and Theron [3-5]. In summary, there are few studies on the performance of filters under unsteady flow, and the existing studies are all experimental tests. Therefore, the current studies only focus on simulation to study the effects of different types of air filter to velocity distribution and pressure drop in the air filter.

2. Models and Methods

2.1 Computational model

Geometry model for fluid flow analysis can be design in various software such as ANSYS Design Modeler, ANSYS Space Claim Direct Modeler or import the appropriate geometry. If a geometry model designed from another software such as Solidworks 2019, the geometry must be converted to IGES or STEP file first before import it into the ANSYS Fluent software [6].

Typically, the geometry is created or modified in some other types of CAD software and saved in the format that compatible with the simulation software. The design of a real geometry consists of intricate details such as solids, screw, mounting, joints, sealing and nuts. The geometry needs to be as simple as possible [7]. Tiny details which do not significantly influence the flow should be removed from the flow path. The Fluent Fluid Flow is one of the systems in ANSYS Workbench that can be set up and solve a three-dimensional (3D) turbulent fluid flow. In this simulation, we generated air filter model from Solidworks 2019 and import it into ANSYS Workbench. The geometry model consists of an inlet, an outlet and a filter media. The input, output and the wall are determined from the generated model where the input was assigned velocity inlet type, and the output was assigned pressure outlet type.

2.2 Meshing

After the geometry model have been designed or created, the geometry must generate a computational mesh throughout the flow volume. For this project, meshing process will be conducted by using ANSYS Meshing application. Meshing is an integral part of the computer aided engineering simulation process. This process is allow the software to run calculation based on the domain. The mesh affects the accuracy, convergence and speed of the solution [8].

The maximum velocity analysis in the subsystem was performed to evaluate the results of each mesh and determine the best mesh size for the flow. The results obtained become more accurate as of the number of cells increases while the computational time also increases. However, given that the mesh size is finer and the number of cells is increased, it is done when the results obtained do not impact the mesh size slightly [9].

2.3 Boundary Condition

Boundary conditions and parameters are one of the vital steps in CFD calculation because it involves identifying the location of the boundaries and providing information at the boundaries. When applying the boundary conditions, make sure that they are not overly defined or under-defined [10].

Table 1 tabulated the data required before running the calculation. In this study, the inlet velocity is constant to 2.7 m/s with 5% of turbulence intensity and the turbulent viscosity ratio is 10.

For the outlet, the pressure is 0 Pa with 5% backflow turbulent intensity and backflow turbulent viscosity ratio is 10.

This research has 3 types of filters that is fibrous filter, pleated filter (cellulose fiber) and High Efficiency Particulate Air (HEPA) filter (glass borosilicate). These 3 types of filters has a different permeability value that is $4.05 \times 10^{-11}\ m^2$, $3.8 \times 10^{-11}\ m^2$ and $7.28 \times 10^{-9}\ m^2$respectively. The viscous resistance value of a filter is determine by using inverse of absolute permeability value of a filter. The permeability value can be found in the past experimental study [3–5].

Table 1: Boundary condition and parameters

Model		**Viscous**	**k- epsilon (standard)**
Materials		Fluid	Air
		Solid	Aluminum
Filter media properties	Test 1	Permeability (m^2)	4.05×10^{-11}
		Viscous resistance (m^{-2})	24691358020
		Porosity	0.90
	Test 2	Permeability (m^2)	3.8×10^{-11}
		Viscous resistance (m^{-2})	26315789470
		Porosity	0.88
	Test 3	Permeability (m^2)	7.28×10^{-9}
		Viscous resistance (m^{-2})	840336134500
		Porosity	0.92
Inlet		Type of Boundary	Velocity - inlet
		Velocity Magnitude (m/s)	2.7
		Turbulent Intensity (%)	5
		Turbulent Viscosity Ratio	10
Outlet		Type of Boundary	Pressure - outlet
		Gauge Pressure (Pascal)	0
		Backflow Turbulent Intensity (%)	5
		Backflow Turbulent Viscosity Ratio	10
Wall		Wall motion	Stationary
		Shear condition	No slip
		Wall roughness	Standard
		Roughness Height (m)	0
		Roughness Constant	0.5
Methods		Pressure-Velocity Coupling	SIMPLE
Solution Initialization		Initialization Methods	Standard
		Compute from	Inlet

2.4 Grid Independent Test

The grid Independent Test (GIT) is performed to establish an optimal grid. The method that is used in this test is to reduce the cell size to obtain more nodes and elements numbers. As more nodes and smaller cell size used, the calculation should become more accurate. Therefore, an optimal grid design for the CFD model is required for the accurate results of CFD.

3. Results and Discussion

3.1 Velocity vector for all tests

The velocity in all test is explained in the form of the velocity vector. In this simulation, a fibrous filter is used in test 1 while test 2 is pleated filter and test 3 uses HEPA filter. Figure 1 illustrate the graph of velocity against location for test 1, test 2 and test 3. Based on Figure 1, the highest velocity occurs at the outlet of the system for all tests and the lowest velocity occur at the filter media. The air flow decrease in velocity before passing through the air filter media. However, air velocity gradually increase after passing through the filter and at the outlet is the highest velocity value in the system.

For test 1, test 2, and test 3, the maximum velocity that occurs in the whole system is 4.28 m/s, 4.28 m/s and 4.11 m/s and the average velocity is 1.71 m/s, 1.71 m/s and 1.70 m/s respectively. Test 1 and test 2 has approximately same maximum velocity value due to the different of permeability value between them is small. However, test 3 has the lowest maximum velocity because its permeability value is the highest among three of them which cause the difficulties for air to flow through it. Test 1 has the highest suction velocity at the outlet of the system which is 4.07 m/s while test 2 is 4.06 m/s and test 3 is 3.91 m/s because it needed to maintain the inlet velocity at 2.7 m/s.

Table 2 illustrate the velocity result obtained at ten locations for test 1, test 2 and test 3. From the velocity result obtained, HEPA filter is the most efficient air filter media compared to fibrous filter and pleated filter. This is because the velocity vector and the result that has been taken at ten location show the air flow in test 3 is smooth. Test 1 and test 2 show that there is fluctuation in velocity vector and the high velocity region that form in the flow causing the system to be inconsistent.

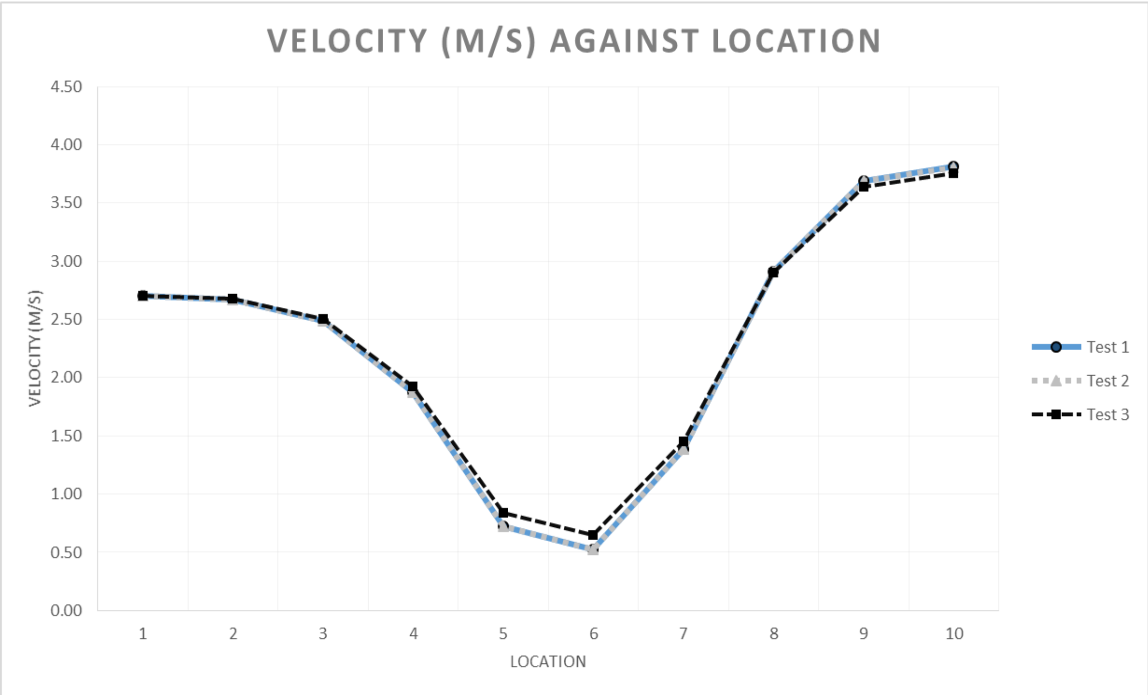

Figure 1: Velocity against location for test 1, test 2 and test 3

Table 2: Velocity result obtained from 10 location on the system for test 1, test 2 and test 3

Location	Velocity m/s		
	Test 1	Test 2	Test 3
1	2.70	2.70	2.70
2	2.67	2.67	2.68
3	2.49	2.49	2.51
4	1.87	1.87	1.93
5	0.72	0.72	0.84
6	0.53	0.52	0.65
7	1.39	1.39	1.45
8	2.91	2.91	2.90
9	3.69	3.69	3.64
10	3.81	3.81	3.75

3.2 Pressure vector for all test

The determination of pressure is important to show the pressure loss occur in the system. Pressure is one of the parameters that can be obtained from the simulation study. Figure 2 illustrate the graph of pressure distribution for test 1, test 2, and test 3. The highest pressure occur at the inlet of the system with a magnitude of 5546.22 Pa for test 1 while test 2 is 5903.58 Pa. For test 3, the highest pressure occur at the filter media with a magnitude of 42.30 Pa. Higher pressure region is at the body inlet which before filter while lower pressure region is at the body outlet.. The differences between the inlet and outlet pressure of the filter during the passage of the air stream is common phenomenon which is called pressure drop. Location 10 is at the outlet of the system that is kept constant at 0 Pa to get the actual pressure distribution in the system. At location 1 which is at the inlet shows that test 3 has lowest pressure, 41.94 Pa compared to test 1 and test 2 that is 5545.89 Pa and 5903.22 Pa respectively.

At location 2, test 2 has the highest pressure that is 5903.31 Pa and compared to test 1 and test 3 that is 5545.97 Pa and 42.02 Pa respectively. There is slight increase in pressure from location 2 to location 3 for test 1, test 2 and test 3. At the location 5 and 6 which is at the air filter shows that a dramatically decreasing in pressure for test 1, test 2 and test 3 from 5549.91 Pa, 5907.25 Pa and 45.87 Pa to 8.08 Pa, 8.07 Pa and 7.57 Pa respectively. The pressure differences between two locations is known as pressure drop. From the result at location 5 and 6, it can be concluded that test 3 has the lowest pressure drop compared to test 1 and test 2. From location 7 to location 10, the pressure for test 1, test 2 and test 3 is decreasing from 7.19 Pa, 7.18 Pa and 6.66 Pa to 0 Pa respectively.

Table 3 presents the pressure results at ten locations for test 1, test 2 and test 3. From the pressure result obtained, HEPA filter is the most efficient filter compared to fibrous filter and pleated filter. This is because the result that has been taken at the ten location show that test 3 requires lower pressure to maintain the proper airflow in the system. Test 1 and Test 2 show that higher pressure needed for the system to maintain the airflow which cause the fan use more energy for the system. There is also no significant loss in pressure in test 3 compared to test 1 and test 2.

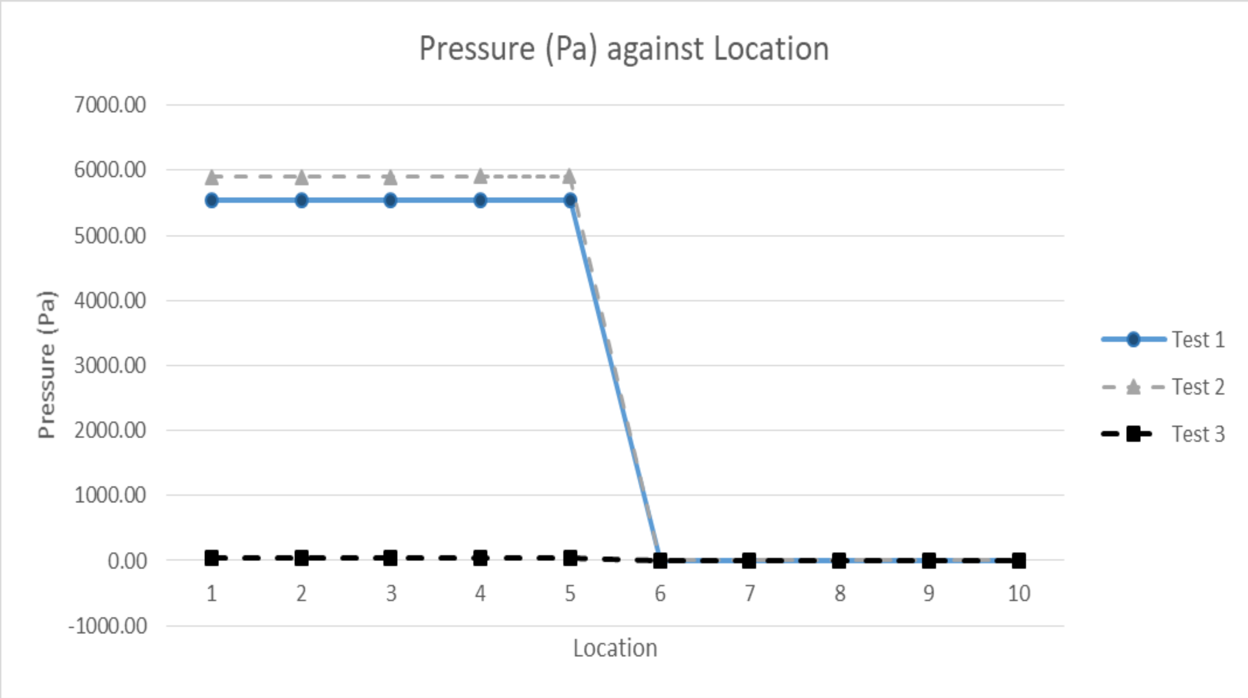

Figure 2: Pressure against location for test 1, test 2 and test 3

Table 3: Pressure result obtained from 10 location on the system for test 1, test 2 and test 3

Location	Pressure (Pa)		
	Test 1	Test 2	Test 3
1	5545.89	5903.22	41.94
2	5545.97	5903.31	42.02
3	5546.53	5903.87	42.55
4	5548.16	5905.50	44.10
5	5549.91	5907.25	45.87
6	8.08	8.07	7.57
7	7.19	7.18	6.66
8	3.45	3.44	3.15
9	0.47	0.47	0.42
10	0.00	0.00	0.00

4. Conclusion

In conclusion, these simulation consist of three different types of air filters with the same geometry of model. Three types of filter that used in this simulation is fibrous filter, pleated filter and HEPA filter. The model for this simulation consist of an inlet, an outlet and a filter where the inlet and outlet have the same diameter and surface area. The velocity and pressure distribution is recorded from the inlet to the outlet of the system. All the test are using the same calculation method and boundary condition except the filter properties.

Based on the data acquired in this simulation, test 3 is the most preferable because the velocity distribution in the system is more balance. In addition, the pressure distribution in test 3 is more even and less pressure drop occur because it did not require high pressure to maintain the air flow in the system at the desired inlet velocity.

To summarize, test 3 is using High Efficiency Particulate Air (HEPA) filter which makes it the best filter and has the best performance in this study. The pressure drop is less and the velocity of air flow is evenly distributed. Hence, HEPA filter is suitable for the best filtration performance.

Acknowledgement

The authors would also like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for its support.

References

[1] A. Q. Ahmed, S. Gao, and A. K. Kareem, “Energy saving and indoor thermal comfort evaluation using a novel local exhaust ventilation system for office rooms,” *Appl. Therm. Eng.*, vol. 110, pp. 821–834, 2017.

[2] Department of Occupational Safety and Health (DOSH) Malaysia, *on Occupational Safety and Health for Design, Inspection, Testing and Examination of Local Exhaust Ventilation System*. 2014.

[3] A. C. C. Bortolassi, V. G. Guerra, and M. L. Aguiar, “Characterization and evaluate the efficiency of different filter media in removing nanoparticles,” *Sep. Purif. Technol.*, vol. 175, pp. 79–86, 2017.

[4] A. F. Miguel, “Effect of air humidity on the evolution of permeability and performance of a fibrous filter during loading with hygroscopic and non-hygroscopic particles,” *J. Aerosol Sci.*, vol. 34, no. 6, pp. 783–799, 2003.

[5] F. Théron, A. Joubert, and L. Le Coq, “Numerical and experimental investigations of the influence of the pleat geometry on the pressure drop and velocity field of a pleated fibrous filter,” *Sep. Purif. Technol.*, vol. 182, pp. 69–77, 2017.

[6] ANSYS Fluent Tutorial Guide 18, “ANSYS Fluent Tutorial Guide 18,” *ANSYS Fluent Tutor. Guid. 18*, vol. 15317, no. April, pp. 724–746, 2018.

[7] CFD SUPPORT S.R.O., “TCFD Manual, version 18.06,” 2018.

[8] ANSYS Inc., “ANSYS Meshing,” p. 4, 2007.

[9] I. Aniekan Essienubong and E. Kelly Orhorhoro, “Pressure Losses Analysis in Air Duct Flow Using Computational Fluid Dynamics (CFD),” vol. 3, no. 2, pp. 55–70, 2016.

[10] ANSYS inc, “Chapter 1 : Introduction to Using ANSYS Fluent in ANSYS Workbench : Fluid Flow and Heat Transfer in a Mixing Elbow,” pp. 1–72, 2009.