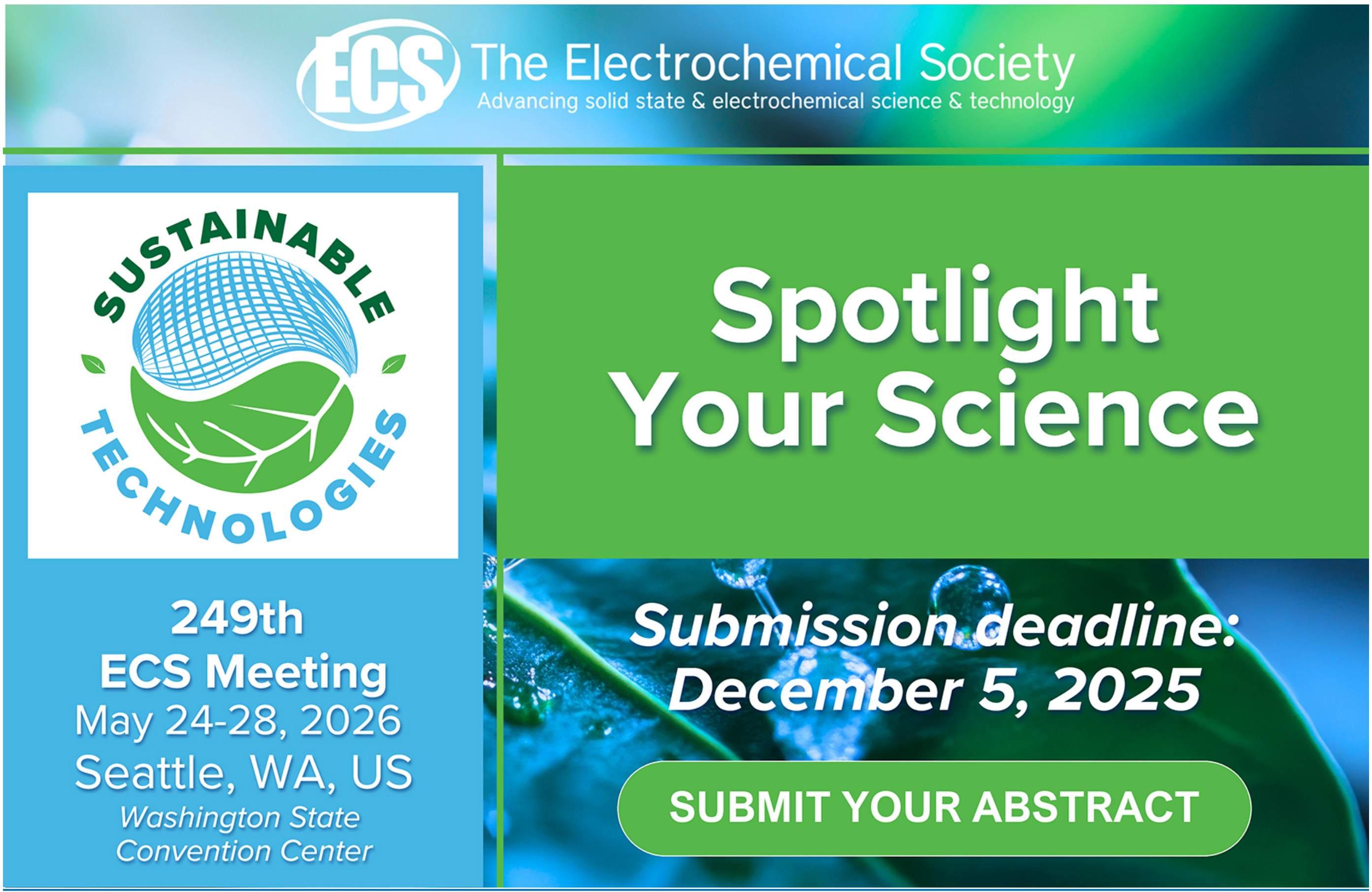
IOP Conference Series:
Earth and
Environmental Science

PURPOSE-LED
PUBLISHING™

Performance evaluation of a retrofitted multi-cyclone using computational fluid dynamic

To cite this article: M Dewika *et al* 2020 *IOP Conf. Ser.: Earth Environ. Sci.* **476** 012122



Performance evaluation of a retrofitted multi-cyclone using computational fluid dynamic

M Dewika[1], M Rashid[2] , C M Hasrizam[3] M P Khairunnisa[4] , J NorRuwaida[5], Y Y Sara[6]

[1]Centre of American Education, Sunway University, Bandar Sunway, 47500 Selangor, Malaysia.
[2,3,4,5]Air Resources Research Laboratory, Malaysia Japan International Institute of Technology, 54100 UTM Kuala Lumpur, Malaysia.
[6]School of Environmental Engineering, University Malaysia Perlis, 02600 Arau, Perlis, Malaysia

E-mail: dewikan@sunway.edu.my

Abstract. Multi-cyclone is widely used in industries as air pollution control device due to several advantages over other available separation units such as its low capital, operating, and maintenance cost and as well as its usability under a wide range of operational conditions. However, it is merely a pre-cleaner as it is inefficient in collecting fine particulate especially, particulate matter with size less than 10 µm (PM_{10}) and below. Hence a simple, cost-effective retrofit on a Conventional Multi-cyclone (CMC) with the motivation of increasing its overall performance on fine particulate emission control was carried out. The retrofit was performed by creating higher negative pressure inside the dust hopper of the CMC by extracting 10% and 24% from the total volumetric airflow rate of the unit with the means of an external Induced Draft Fan. The Computational Fluid Dynamics (CFD) with Reynold Stress Model (RSM) turbulence model was performed and validated using experimental data to gain a better understanding in pressure distribution, velocity profile and particulate movement between the CMC and the Retrofitted Multi-cyclone (RMC). The CFD results show deviation between 0% to 8% for pressure at inlet and outlet of cyclone compared to the experimental results. In addition, CFD results depict that the RMC has higher pressure at the inlet and lower pressure inside dust hopper of CMC, which cause the finer particle to be pulled in through suction outlet. Also, the emission of fine particulate is reduced in RMC by 9% to 16%. compared to the CMC. Moreover, the phenomena at the suction duct can be clearly explained with the usage of CFD. The finding suggests that a simple, cost-effective retrofit at the multi-cyclone has increased the overall performance in the fine particulate collection, and the understanding of the phenomena could be enhanced by the CFD.

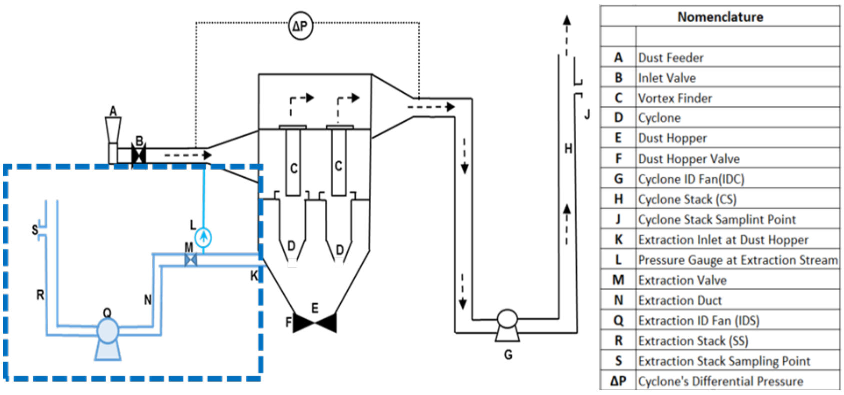
1. Introduction

Cyclone separators are commonly used as an air pollution control device to remove particulate matter due to its low operating and maintenance cost Multi-cyclone which comprises of miniature cyclone structured in parallel are commonly used in the industries compared to single cyclone. The cyclones employ a centrifugal force generated by spinning gas stream to separate the particulate from the flue gas. However, the multi-cyclone is efficient in collecting coarse particulate rather than the fine particulate. Hence, a Conventional Multi-cyclone (CMC) was retrofitted at its dust hopper, where a small portion of the gas stream was drawn out using an Induced Draft Fan. The experimental work of the Retrofitted Multi-cyclone (RMC) has successfully demonstrated an increase in collection efficiency, especially in collecting PM_{10}, $PM_{2.5}$ and PM_1. However, the phenomena occurring within the cyclone with the presence of the retrofit could not be explained experimentally. Therefore, due to these limitations, the Computational Fluid Dynamics (CFD) was employed to demonstrate the pressure, velocity and particulate trajectories profiles within the CMC and RMC units.

2. Methodology

2.1. *The rig and experiments*

Figure 1 shows the experimental setup of the studied pilot plant scale multi-cyclone system [1], [2]. The dotted box presents the retrofitted section with an additional suction stream attached to the dust hopper. The CMC system consists of four identical miniature cyclones in a single compartment equipped with a dust feeder and a pressure gauge to determine cyclone's differential pressure, dust hopper, cyclone ID fan (designated as ID Cyclone or IDC) and cyclone stack (CS). The airflow rate of the IDC is manipulated by a control panel attached to the pilot plant. The retrofitted line has an additional suction inlet pipeline with the suction valve connected to an ID Suction fan or IDS, controlled by a separate control panel. The experiment was run with and without suction flow created at the dust hopper via the suction stream by the IDS. The gas velocity and volumetric air flow rate were measured according to US EPA Method 2 - Determination of Stack Gas Velocity and Volumetric Gas Flow Rate (Type S pitot tube). The experimental work was performed at cyclone flowrate of 11 and 19 m^3/s with the suction flowrate varying from 0% to 20 % of the cyclone flowrate and fed with sample of Palm oil mill boiler fly ash with size fraction below 100 μm. The sampling of concentration of particulate by size segregation was done using the the GRIMM Portable Laser Aerosol Spectrometer dust monitor Model 1.108/1.109.



Nomenclature	
A	Dust Feeder
B	Inlet Valve
C	Vortex Finder
D	Cyclone
E	Dust Hopper
F	Dust Hopper Valve
G	Cyclone ID Fan(IDC)
H	Cyclone Stack (CS)
J	Cyclone Stack Samplint Point
K	Extraction Inlet at Dust Hopper
L	Pressure Gauge at Extraction Stream
M	Extraction Valve
N	Extraction Duct
Q	Extraction ID Fan (IDS)
R	Extraction Stack (SS)
S	Extraction Stack Sampling Point
ΔP	Cyclone's Differential Pressure

Figure 1. The pilot plant multi-cyclone system with the retrofitted stream in the dotted region.

2.2. *Computational fluid dynamics*

CFD model template in Ansys-Fluent was used to predict the flow pattern within the cyclone unit at points beyond reachable of experiments. The geometry of CMC and RMC was modelled by using SolidWorks and simulated via ANSYSFLUENT 15.0 in ANSYS Workbench tools. The simulation values were compared with the experimental values, which was obtained from dry runs at the multi-cyclone unit itself. Since the difference between the simulation and the experimental value was below 10%, thus the application simulation was accepted. Hence, the simulation was used to demonstrate the movement of particulate flow, velocity contour, and pressure contour inside the unit to which could not be done by experiments. The CFD RSM modelling and simulation was performed with two different values of effective inlet velocities represented by different volumetric airflow of 0.11 m^3/s and 0.19 m^3/s of the CMC. The k-ε RNG and RSM method was used to predict the turbulence dispersion and discrete phase model (DPM).

3. Results and discussion

Evidently, experimental results had proven that the collection efficiency of the RMC was much higher than the CMC by 9 to 16%. The emission concentration of PM_{10}, $PM_{2.5}$ and PM_1 reduced by 87%, 58% and 24% for a flowrate of 0.11m^3/s, respectively with the introduction of suction. In order to understand the cyclone's better, CFD simulation was deployed. The study was performed on the velocity contour, pressure contour, and particulate trajectory by size. The simulation results were compared with conventional cyclone, which is designated as 0% suction. Before accepting the CFD values, validation was performed to check on the accuracy of CFD results.

3.1. Validation of experimental results with CFD

Table 1 shows the comparison between the experimental measurement and CFD results for the RMC. It is observed that the static pressure difference at the outlet between experimental and CFD is less than 2% for all cyclone volumetric airflow rate despite the varied percentage of suction. While the static pressure difference between CFD and experimental value at the inlet was at 8% maximum deviation where the higher deviation is observed for the cyclone flow rate of 0.19 m^3/s. The difference between CFD and experimental is less than 10% indicate that the CFD model has achieved convergence with good accuracy. Therefore, the CFD results obtained was used for further data acquisition and analysis, which is a limitation to experimental work such as pressure contour, velocity contour and particulate trajectory by size.

Table 1. The experimental and CFD static pressure reading for the RMC.

Cyclone Flow rate (m^3/s)	Suction (%)	Inlet Static Pressure (N/m^2)			Outlet Static Pressure (N/m^2)		
		CFD	Exp.	Difference	CFD	Exp.	Difference
0.11	10%	238	225	6%	301	308	2%
	20%	253	235	7%	313	313	<1%
0.19	10%	648	598	8%	843	843	<1%
	20%	662	608	8%	853	853	<1%

3.2. Pressure drop profile

Figure 2 shows the pressure contour for CMC and RMC at the cyclone flow rate of 0.11 m^3/s and 0.19 m^3/s, where the pressure variation is given in the illustration. It was mentioned that the reverse flow structure in a cyclone is apparently originated by the pressure field inside the cyclone, and not influenced by the conical shape or the geometrical of the cyclone [3]. Since the suction creates more negative pressure in the hopper leading to the increase in pressure difference, therefore this may eventually cause more disruption in the inversion.

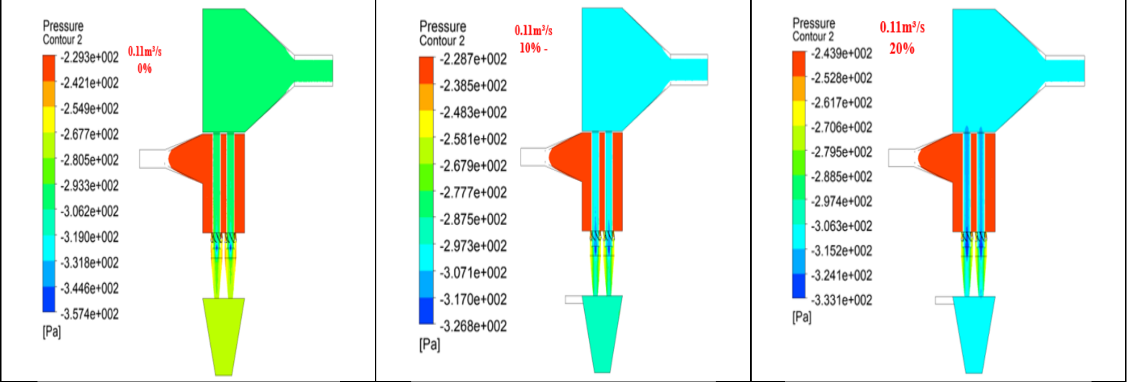
It is depicted that the inner vortex is – 293 N/m^2 and -771 N/m^2 for Figure 2a) and Figure 2d), respectively where it was observed that the negativity of pressure in all parts of the cyclone increases with the flow rate of the cyclone. This is due to the airflow increases the inlet velocity and eventually increases the pressure drop too.

Also, interestingly the negativity of pressure in the inner vortex of the cyclone further increases with the suction percentage. It was observed that the pressure in the inner vortex for cyclone flow rate 0.11 m^3/s of a) b) and c) is -293 N/m^2, -307 N/m^2 and -315 N/m^2 for suction 0%, 10% and 20%, respectively. Meanwhile, for cyclone flow rate 0.19 m^3/s , the pressure in the inner vortex is -771 N/m^2, -785 N/m^2 and -822 N/m^2 for suction 0%, 10% and 20%, respectively. The suction at the RMC made changes to the airflow, causing an increase in negativity in pressure within the cyclone at the fixed flow rate. It was stated that higher inlet velocities increase the pressure drop across the cyclone and give higher collection efficiencies for a given cyclone [4]. The authors claim was similar to this work as shown in Figure 1 where the presence of suction increased the pressure negativity at the dust hopper leaving a more negative pressure area. It is expected that the vortex carrying particulate would be pulled further into the dust hopper, indirectly creating more collision and eventually increasing its collection efficiency. This finding explains the increase in collection efficiency of particulate in the experimental work.

a) Q_c =0.11m^3/s; Suction 0% (CMC)

b) Q_c =0.11m^3/s; Suction 10% (RMC)

c) Q_c =0.11m^3/s; Suction 20% (RMC)



d) Q_c =0.19m^3/s; Suction 0% (CMC)

e) Q_c =0.19m^3/s; Suction 10% (RMC)

f) Q_c =0.19m^3/s; Suction 20% (RMC)

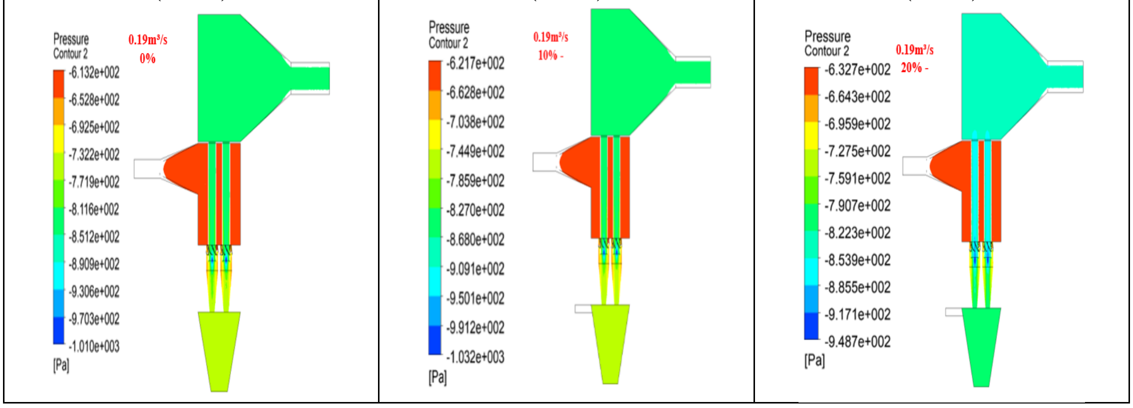


Figure 2. The pressure contour at cyclone flow rate of 0.11 and 0.19 m^3/s with suction 0, 10 and 20%.

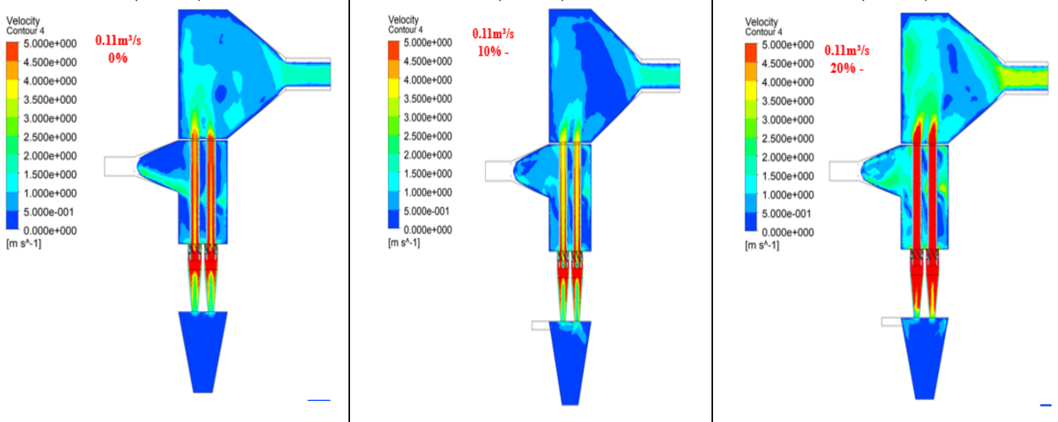
3.3. Velocity profile comparison

Figure 3 shows the velocity contour for the multi-cyclone without, and with suction at the cyclone flow rate of 0.11 m^3/s and 0.19 m^3/s, It is observed that the velocity of the inner vortex of flow rate of 0.11 m/s and 0.19 m/s is 4.5 m/s and 5 m/s, respectively. The increase in flow rate increases the velocity of air within the cyclone where the phenomena was similar to the performance of a new generation multichannel cyclone studies [5].

The comparison between Figure 3 a) and b) at a constant flow rate of 0.11 m/s, showed that the increase in suction to 10% had reduced the vortex velocity slightly. However, the vortex velocity increased with the increase of the suction stream at 20%. Similar reduction and increase in vortex velocity are depicted at a higher flow rate of 0.19 m^3/s in Figure 3 d), e) and f).

Figure 3 b) and c) for cyclone flow rate of 0.11 m^3/s and e) and f) for flow rate 0.19 m^3/s, demonstrated some air flow in the dust hopper. It is believed that the airflow is the extension from the vortex into the dust hopper where the intensity of the extension increases with the increase in suction percentage. Evidently, this finding indicates that the presence of suction stream has pulled the vortex further down into the dust hopper. The increase of negativity in pressure in the vortex with the presence of suction as Figure 2 could be the reason for this phenomenon. The urge of balancing the displaced air due to the pressure difference, pulls the vortex further into the dust hopper. This phenomenon also supports the increase in collection efficiency as obtained experimentally throughout the experiment.

a) Q_c =0.11m^3/s; Suction 0% (CMC) b) Q_c =0.11m^3/s; Suction 10% (RMC) c) Q_c =0.11m^3/s; Suction 20% (RMC)



d) Q_c =0.19m^3/s; Suction 0% (CMC) e) Q_c =0.19m^3/s; Suction 10% (RMC) f) Q_c =0.19m^3/s; Suction 20% (RMC)

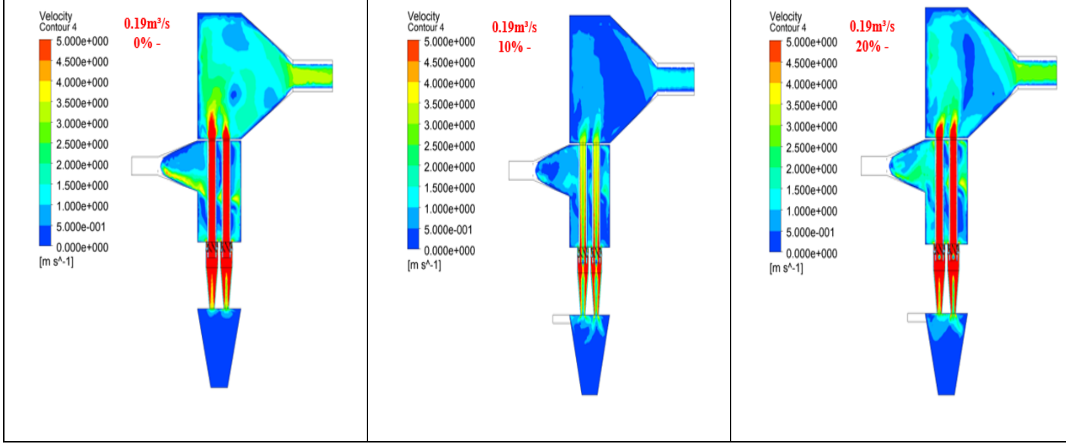


Figure 3 The velocity contour at cyclone flow rate of 0.11 and 0.19 m^3/s with suction 0, 10 and 20%.

3.4. Particulate trajectory

Figure 4 depicts the PM_1, $PM_{2.5}$ and PM_{10} trajectory in the multi-cyclone without and with suction at the cyclone flow rate of 0.11 m^3/s which showed the velocity of particulate by size. Figure 4 depicts that PM_1 at the inlet velocity is 4.86 m/s, 4.89 m/s and 5.23 m/s for suction 0%, 10% and 20%, respectively are observed despite that constant cyclone flow rate of the particulate velocity at the entry of the cyclone is affected by the suction stream. Similarly, the velocity for $PM_{2.5}$ was 5.17 m/s, 5.15 m/s and 5.29 m/s for suction 0%, 10% and 20%, respectively. Despite the slight drop in particulate velocity at 10% suction, however, as an overall, it shows an increase. PM_{10} demonstrates a similar trend where the velocity was 5.48 m/s, 5.52 m/s and 5.68 m/s for suction 0%, 10% and 20%, respectively.

Also, Figure 4 depicts particulate velocity in the vortex increases with the increase in suction percentage where the vortex velocity for PM_1 is 4.86, 5.58 m/s and 6.10 m/s for suction 0%, 10% and 20%, respectively. While for $PM_{2.5}$ the vortex velocity is 5.17 m/s, 5.15 m/s and 5.29 m/s for suction 0%, 10% and 20%, respectively. Expectedly, PM_{10} showed a similar trend with PM_1 and $PM_{2.5}$. The increase in particulate velocity intensified the collision between particulate despite the size which explains the increase in collection efficiency of the experimental work.

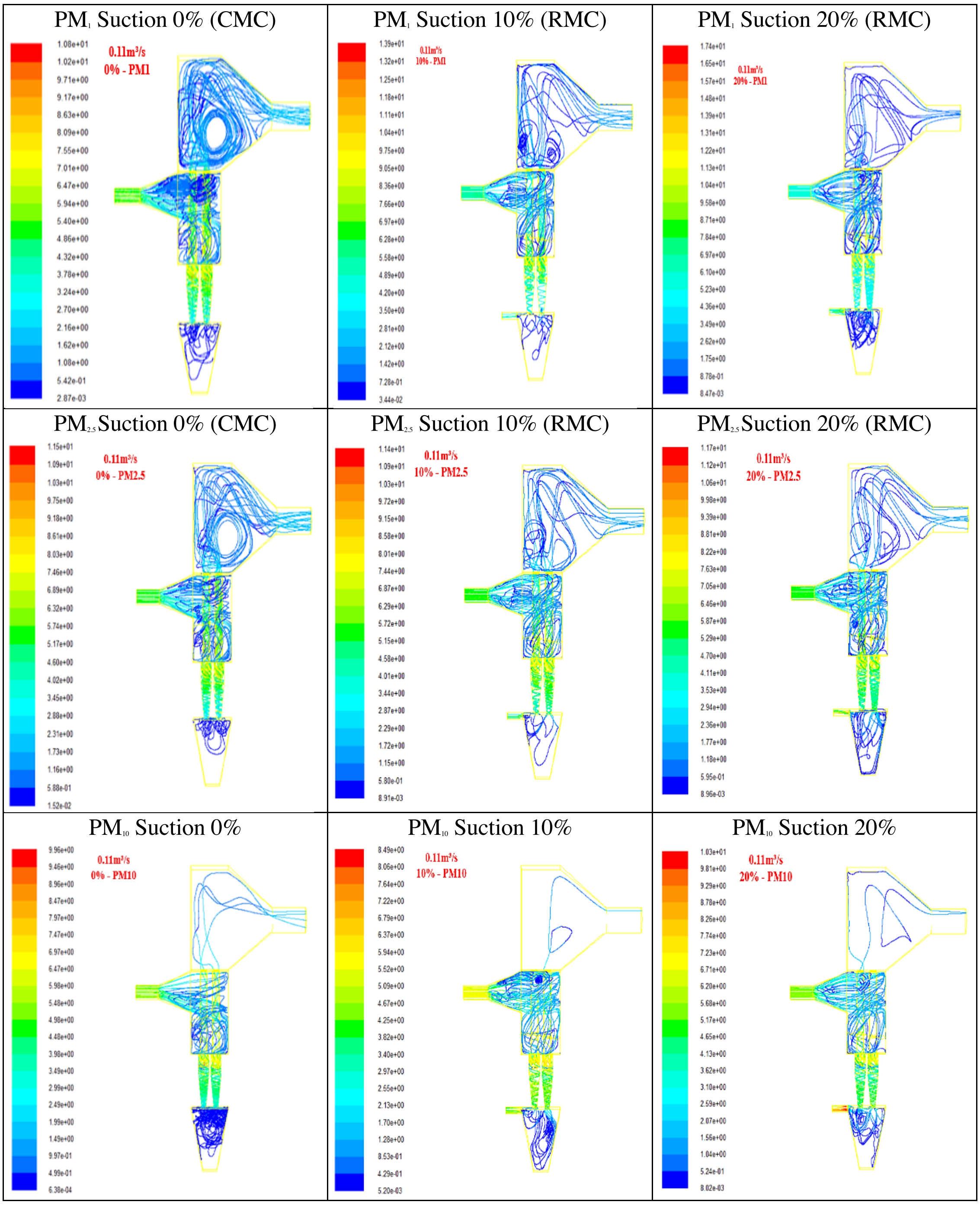


Figure 4. PM$_1$, PM$_{2.5}$ and PM$_{10}$ trajectory at constant flow rate of 0.11 m^3/s with suction 0%, 10% and 20%.

4. Conclusion

The experimental collection efficiency of the RMC is higher than the CMC by 9 % to 16%, and this occurrence was explained by the pressure profile, velocity profile and particulate trajectory with the usage of CFD. The pressure, airflow velocity and particulate velocity increased with cyclone flowrate and further enhanced with the presence of the suction at the RMC. The phenomena at the suction duct were clearly explained with the usage of CFD. The simple, cost-effective retrofit at the multi-cyclone has increased the overall performance in the fine particulate collection.

Acknowledgement
The first author was a doctoral student at the MJIIT, Universiti Teknologi Malaysia, during the execution of experimental work. She is now attached to the Sunway University. The authors would also like to express appreciation for the supporting grant of UTM Tier 2 Q.K130000.2643.15J50.

References

[1] Rashid M Huda N Norelyza H and Hasyimah N 2015 *Sains Malaysiana* vol **44** no 4 pp 565–569
[2] Norelyza H and Rashid M *J. Environ. Res. Dev.* 2013 vol **7** no 4 pp 1392–1398
[3] Cortés C and Gil A *Prog. Energy Combust. Sci.* 2007 vol **33** no 5 pp 409–452
[4] Griffiths W D and Boysan F 1996 *J. Aerosol Sci.*, vol **27** no 2 pp 281–304
[5] Baltrenas P Pranskevicius M and Venslovas A 2015 *Energy Procedia* vol **72** pp 188–195