Suction Duct Optimization for Fume Extraction System

Jeevan Singh Rawat[1], Rucha Date[2], Chandra Shekhar Singh[3]

[1, 2] Mechanical Engineering Graduate, Guru Gobind Singh Indraprastha University, Dwarka, Delhi, India
[3] HOD, Department of Mechanical Engineering, JEMTEC, Guru Gobind Singh Indraprastha University, Delhi, India

***

Abstract - *The industry is full of processes which are fume-generating. The vapor, dust, or smoke generated by a substance as a result of burning, explosion or any other related event is termed as fume. It is irritating, harmful and sometimes poisonous. Fumes-producing industrial operations can represent a substantial health and safety risk to workers. Extracting fumes from the work station for safety and efficient operation has always been a key engineering issue. This paper focuses on improving the design of suction duct for optimal fume extraction from hot operations such as degreasing and phosphating without increasing the blower capacity. ANSYS Fluent is used to evaluate the performance of the proposed design. The results are then compared to the industry standard suction duct design.*

***Key Words*: Extraction System, CFD, ANSYS, Suction Duct, CAD, Fluid Flow, Solidworks, Keyshot**

1. INTRODUCTION

Fume extraction system comprises of a blower, several inlets as per requirement, and an enclosed path. The end is generally connected to a filtration system. The centrifugal blower creates a negative pressure inside the fume extractor system which extracts out the dust and harmful particles from the inlets and sends it to a contained filtration system. Fume extraction system is a necessity in several industries like mechanical industries where fine metalwork or lasers are used, electrical industries where soldering is done and chemical industries which generate harmful or toxic fumes.

The non-uniform extraction at successive inlet points is the main issue with the fume extraction system of tanks. This results in a larger mass flow rate for extraction from the tank closest to the centrifugal blower. With each tank, the mass flow rate decreases noticeably, with the farthest tank having the lowest mass flow rate of extraction. It's crucial to achieve sufficient pressure consistently via all the inlets in order to maximize the extraction process. The typical solution to this issue is to use a blower with a bigger capacity, although doing so considerably increases both cost and consumption. The primary goal of this study is to suggest design adjustments to the extraction system that will result in uniform mass flow rate through all the inlets using the centrifugal blower of the same capacity.

1.1 Phosphating

The chemical process of phosphating involves reacting an aqueous phosphate solution with a metallic surface. This produces a metal phosphate conversion layer that is hardly soluble. The material is first washed with acid for this purpose, and after that, the phosphate layer is created. The application of phosphate coating serves as a pretreatment step before coating or painting, enhancing corrosion protection and enhancing the frictional characteristics of sliding components. Phosphate chemical conversion (PCC) coatings have been investigated for improving surface protection of magnesium alloys in aerospace, automobile, electronics, sports goods, and biomedical applications [1]. Zinc and manganese phosphate coatings are offered by metal coating. These standard processes in industry produce hazardous gaseous effluents. The mists and vapors produced by heated tanks must be removed from the area, since they are dangerous for the workers. The use of extraction and ventilation systems is a necessity in such set-ups to make the working environment free of hazardous gases and fumes efficiently.

1.2 Overview of the simulation

Design of an extraction system depends on many factors, which include, efficiency, cost and scale. There are several advantages of one design and certain drawbacks as well. In this journal, we have compared the conventional extraction system design (case 1) to a proposed design (case 2). The results of the analysis done are then used to conclude the various advantages of the proposed design over the conventional one.

Observing the flow and calculating the pressure at both the inlet and outlet side of the extraction system enables to identify the efficiency of both the systems. The mass flow rate at the inlets is taken as the parameter to define degree of extraction. Setting up an experiment for the same can be a time and resource consuming task, hence analysis comes into play. Two design models of fume extraction system are analyzed and compared using Ansys Fluent in this journal. Both the models are analyzed under the effect of a blower of same capacity.

1.3 Centrifugal Blower

A centrifugal blower, also known as a centrifugal fan, is a motor or pump that circulates and moves air. The air is drawn into the blower and then forced out at a 90° angle. The impeller and the motor are the two major parts of a centrifugal blower. It has high efficiency and produces less noise. The centrifugal blower has relatively small airflow with high static pressure. It is a space saving blower hence the ideal choice for fume extraction systems.

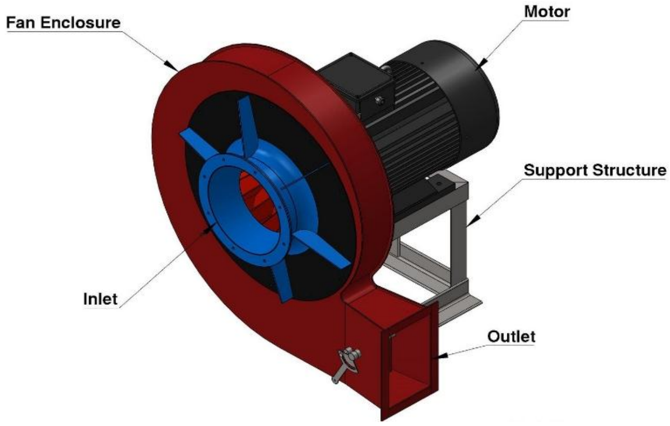

Fig -1: Centrifugal Blower

2. DESIGN

The geometry of a fume extractor is the most important part for determining the fluid flow uniformity. In the present study, numerical analysis of the fluid flow characteristics and design optimization of inlet duct geometry for improving flow uniformity are to be done, hence precise design of the models is a must. Designs of both case 1 and case 2 are done in DS Solidworks 2020 and rendered in Keyshot 9. The overall dimensions of both the cases are equalized for comparison purpose. The dimensions are with reference to the extraction system used in industries at present. For analysis purpose, the design is made as a solid body and not as a sheet metal design as in real life.

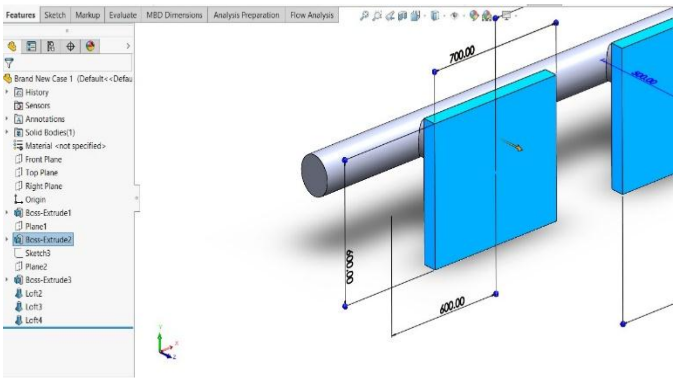

Fig -2: CAD in making (Solidworks)

3. ANALYSIS SETUP

Ansys Fluent module of Workbench is utilized for the analysis. Meshing is optimized for precision in both the designs and a very fine mesh is generated, with 65247 nodes and 321839 elements in the 2nd case. All inlets are kept at atmospheric pressure whereas the outlet is provided with a mass flow rate of 0.4Kg/s, to generate the suction effect.

The extraction system is examined using the K-epsilon model. It is a two-equation model, which implies that two additional transport equations are included to depict the turbulent characteristics of the flow. As a result, a two equation model may take historical influences like convection and turbulent energy diffusion into consideration.

Turbulent kinetic energy, abbreviated k, is the first transferred variable. The turbulent dissipation, or epsilon, it is the variable that determines the scale of turbulence, whereas the first variable, 'k', determines the energy in the turbulence.

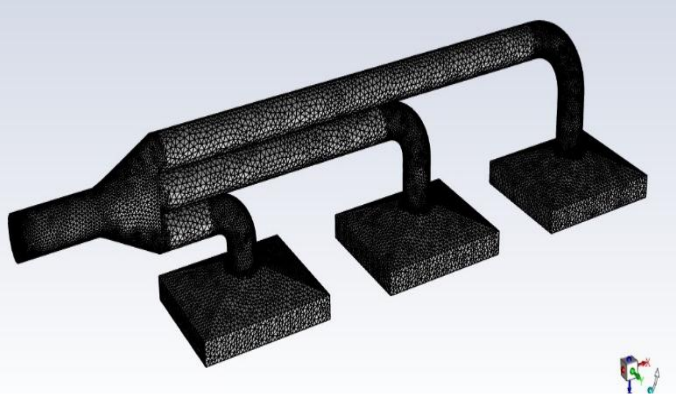

Fig -3: Meshing: Case 2 (ANSYS Mechanical)

4. CASE 1: CONVENTIONAL SUCTION DUCT DESIGN

The conventional design of the suction duct (fig 4) of the fume extraction system is one of the most common design used in small scale industry due to the design simplicity and affordability. It includes one big main pipe with goes throughout the length of the line and multiple inlet ducts are connected to it. A blower is connected to one end of the main pipe. This type of fume extraction system can still be found in many small-scale industries. It is very primitive but is easy and cheap to manufacture using sheet metal. There are several drawbacks of this design discussed below.

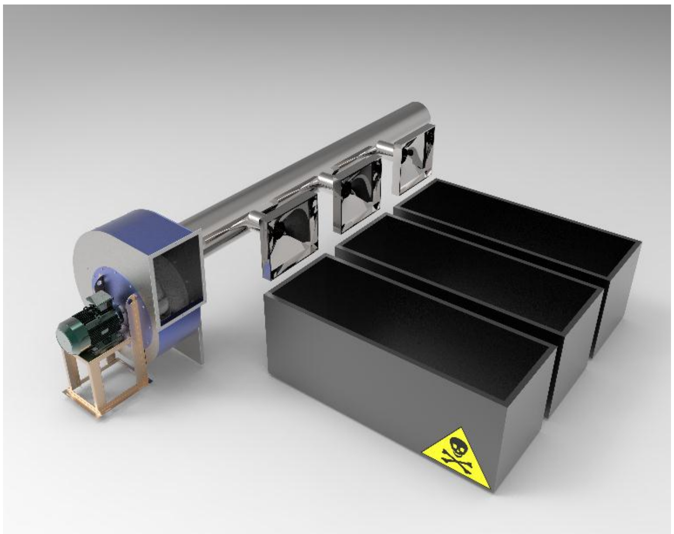

Fig -4: Conventional Suction Duct Design (Case1)

4.1 Drawbacks of Case 1

Simulation of the conventional suction duct design is done under the fluent module of ANSYS Workbench 2019 R2. The results of the same are displayed in fig 3 and fig 4. The velocity contour (fig-5) clearly shows the reduction in velocity as the number of inlets and distance from the blower end is increased. The pressure contour (fig-6) displays the reduction in pressure as the number of inlets and distance from the blower end is increased. Another important observation is that the pressure and velocity are hindered at the junction of inlets to the main pipe. Thus there is a sudden change in the flow due to a lack of continuity in the design. All this leads to a lack of suction pressure at the inlet end as we move further from the blower.

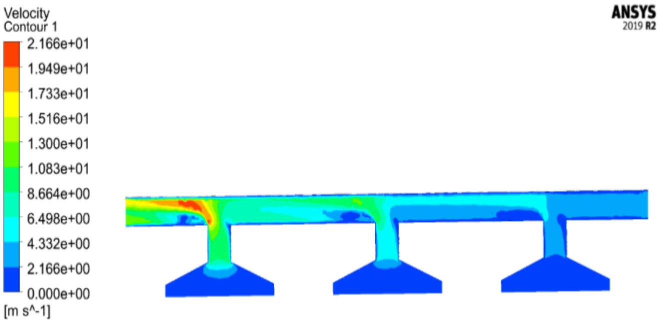

Fig -5: Velocity Contour (Case 1)

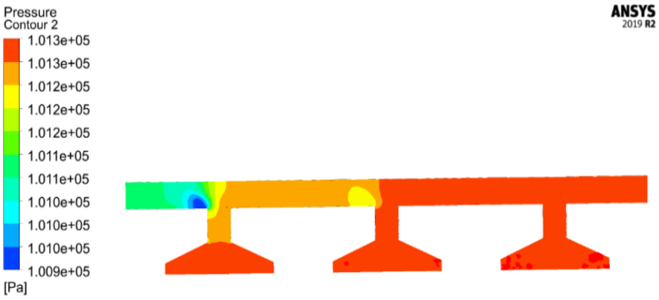

Fig -6: Pressure contour (Case 2)

5. CASE 2: PROPOSED SUCTION DUCT DESIGN

The proposed design of the suction duct (fig 7) of the fume extraction system is a new and improved concept that overcomes the drawbacks of the conventional suction duct design. The main features of this design include

1. Individual paths for each inlet to reduce the traffic which increases the fluid flow.
2. Junctions are provided with smooth curved path which allows the flow to be continuous and does not get hindered by a sudden change in path.

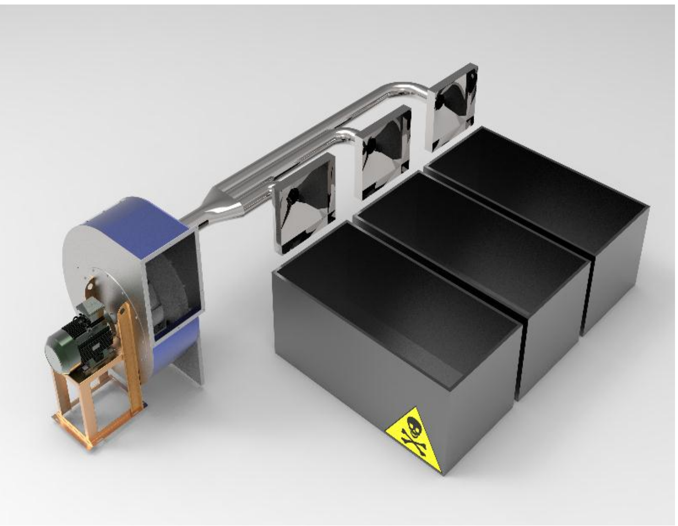

Fig -7: Proposed Suction Duct Design (Case 2)

5.1 Simulation results of Case 2

The simulation results confirm the high efficiency and improved flow in the extraction system. As shown in the pressure contour (fig-8), uniformity in the pressure can be seen unlike case 1. The velocity contour (fig-9) verifies that the velocity of the fluid has noticeably increased at the far ends as well and does not get hindered over the junctions.

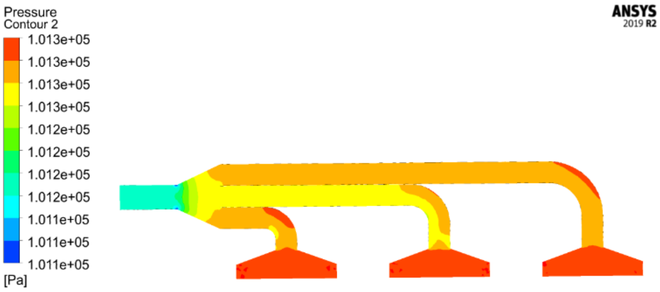

Fig -8: Pressure contour (Case 2)

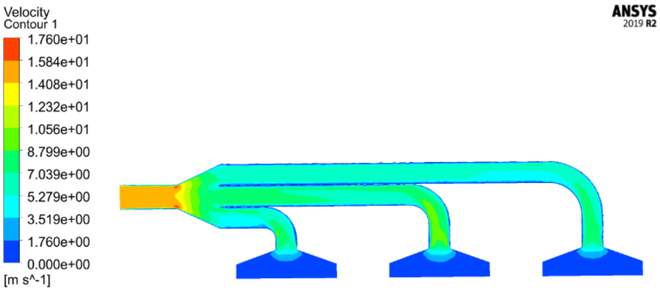

Fig -9: Velocity contour (Case 2)

6. FUTURE SCOPE (Case 3)

Optimization of the conventional design can further be extended with the application of guiding strips inside the extraction system. The guiding strips (fig: 10) further separates the traffic inside of the vent preventing cluster formation. The number of strips to be applied are calculated in reference to the cross section of the vent as well as the blower formation. Too many strips can have an adverse effect as they will reduce the area and will act as barriers to the flow, hence it is crucial to decide the number of strips as well as the dimensions of the strips precisely.

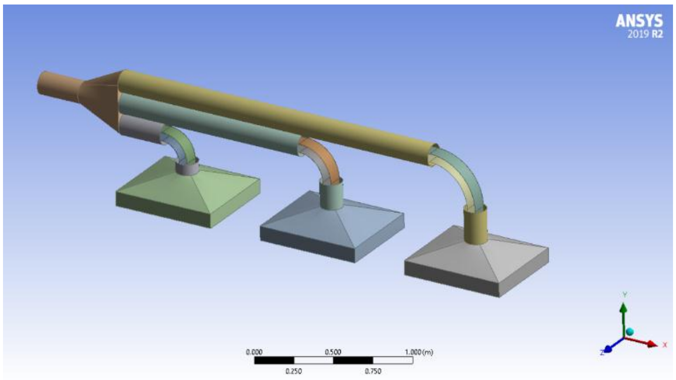

Fig -10: Case 3 (Guiding Strips) -Future Scope

7. CONCLUSION

Mass flow rate at every inlet was noted from the results and a graph was plotted for both the case (Suction ducts are represented on X axis and mass flow rate (Kg/s) on the Y axis). With respect to the observations made (refer to fig: 11), we can conclude that the proposed design of the suction duct has increased the uniformity in the mass flow rate at the suction inlets throughout the extraction system. Pressure build-up and fluid flow is also observed to be more uniform even at the junctions. Thus, the proposed design must be considered against the conventional one when the efficiency of the extraction system is a must.

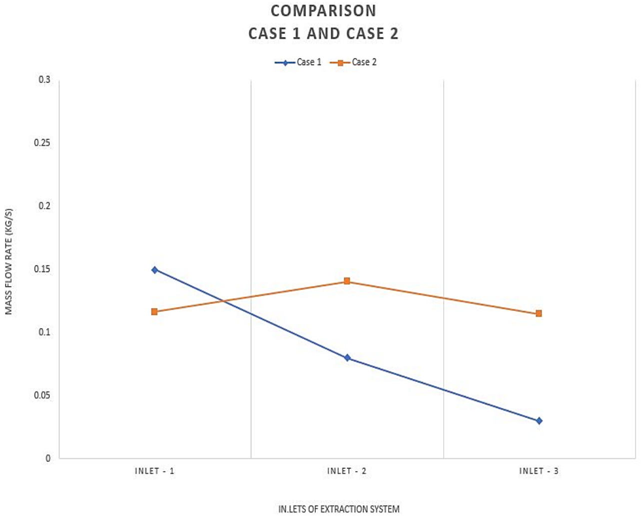

Fig -11: Comparison on the basis of '*mass flow rate'(y)* at suction inlets(x)

REFERENCES

[1] M. A. Hafeez, A. Farooq, A. Zang, A. Saleem & K. M. Deen "Phosphate chemical conversion coatings for magnesium alloys: a review"

[2] Anderson, D. A., Tannehill, J. L, and Pletcher, R.H., "Computational fluid mechanics and Heat Transfer"

[3] John David Anderson, "Computational Fluid Dynamics: The Basics with Applications", McGraw Hill, New York