RESEARCH ARTICLE OPEN ACCESS

An Investigation on Shape Optimization of Pressurized Inlet Diffuser in Steam Vent Silencer By Using Computational Fluid Dynamics

Rajgopal[1], Kiran K[2], Yeshwantraya Ashtagi[3]
[1]Asst. Professor, Dept. of Mechanical Engineering, Basavakalyana Engineering College, Basavakalyana-27.
[2] Asst. Professor, Dept. of Mechanical Engineering, East Point College Of Engineering & Technology, Bangalore-49,
[3]Asst. Professor, Dept. of Thermal Power Engineering, Dept. of PG Studies, VTU RO, Gulbarga-05.

ABSTRACT
Recently, research on new techniques for noise reduction of steam vent silencer has been addressed. However, research work on shape optimization of steam vent silencer along with work on pressure drop has been sorely neglected. Therefore, a numerical simulation has been investigated on the pressure drop using steam vent silencer is introduced in this research work. This emphasizes the variation of pressure drop with the influence of holes with various geometries like circular holes, square holes and square extrusion in pressurized inlet diffuser. The sensitivity of pressure drop is less in square extrusion, compared to all other geometry. So it can conclude that shape optimization of pressurized inlet diffuser in steam vent silencer is obtained with the application of square extrusion.
Key Words: Steam vent silencer, Shape optimization, Pressure drop, Numerical simulation

I. INTRODUCTION:

A vent silencer or a blow off silencer is a device used to reduce unwanted noise created by gas or steam flow in a pipeline discharging directly into the atmosphere. The resulted noise can be generated due to the high velocity flow through the valve and turbulence created around any obstacle in the line that suddenly restricts or changes the direction of flow such as valve or an orifice. Vent silencers find wide applications in steam vents, safety relief valve outlets, high pressure vents, system blow downs and purge outlets. Vent and blow down noise is a function of upstream pressure and temperature, the valve size and type, type of gas being vented, plus the effect of downstream piping. Each vent silencer is designed to attenuate the noise level to the required sound pressure level criteria at a given distance from the silencer. For this reason that vent and blow down silencers are seldom a catalogue selection.

In any steam blow off system, the primary reason for noise to occur is at the open stack exit. This blow off silencer is installed either within the stack or at the stack outlet to intercept this noise before it escapes into the environment. These are two fundamental noise reduction principles used in passive silencer design. Dissipative components (using sound absorbing material) provide balanced noise reduction over a broad frequency range. This reactive components, using resonant reflections within tuned chambers and passages, provide peak noise reduction in a more concentrated frequency band. The SV series vent silencers combines both dissipative and reactive technology in a highly efficient design.

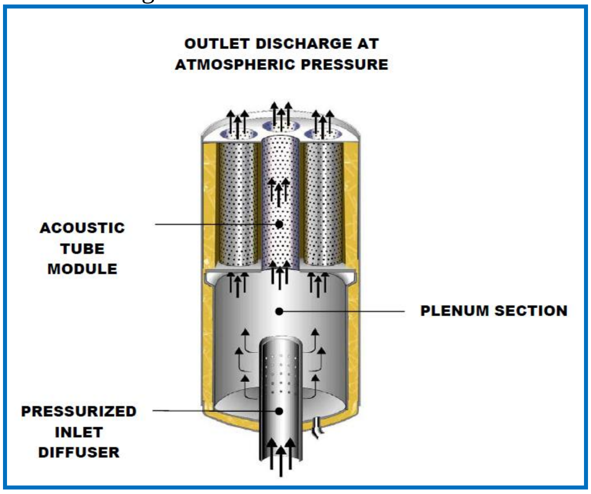

Fig -1: Absorptive type of silencer

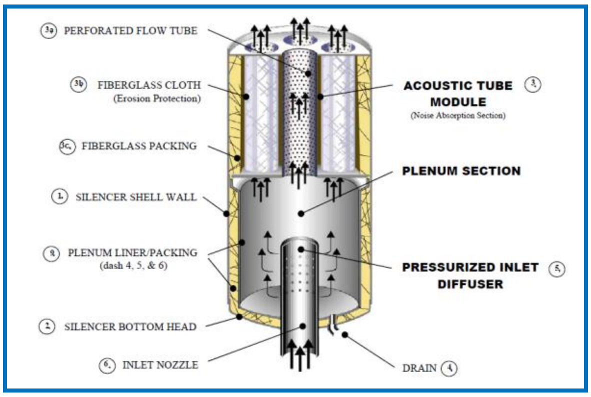

Fig -2: Typical Absorptive silencer with parts

1.1. PRESSURE DROP:

Pressure drop of exhaust system includes losses due to piping, silencer, and termination. High backpressure can cause a decrease in engine efficiency or increase in fuel consumption, overheating, and may result in a complete shutdown of the system potentially causing significant damage. Hence it is necessary that the pressure drop in the silencer is as less as possible.

According to the Energy Equation for a fluid, the total energy can be summarized as elevation energy, velocity energy and pressure energy. The Energy Equation can then be expressed as:

$$p_1 + \rho\, v_1^2 / 2 + \rho\, g\, h_1 = p_2 + \rho\, v_2^2 / 2 + \rho\, g\, h_2 + p_{loss}$$

where,

p = pressure in fluid (Pa (N/m^2))

p_{loss} = pressure loss (Pa (N/m^2))

ρ = density of the fluid (kg/m^3)

v = flow velocity (m/s)

g = acceleration due to gravity (m/s^2)

h = elevation (m)

For horizontal steady state flow, $v_1 = v_2$ and $h_1 = h_2$.

$p_{loss} = p_1 - p_2$

The pressure loss is divided into

- **Major loss** due to friction and
- **Minor loss** due to change of velocity in bends and valves.

The pressure loss in pipes and tubes depends on the flow velocity, pipe or duct length, pipe or duct diameter, friction factor based on the roughness of the pipe or duct, and whether the flow is turbulent or laminar - the Reynolds Number of the flow.

1.2. PRESSURE DROP:

The Mechanical performance criterion specifies the material properties of the exhaust system to ensure that it is durable and requires little maintenance when incorporated into service. Material selection is especially important in cases involving high temperature or corrosive gases. Traditional carbon steels will typically be sufficient for the majority of applications. Aluminized steel is available from many silencer manufacturers and is often preferred for general applications. Aluminized steel is slightly more heat resistant than carbon steel and offers an increased resilience to corrosion and is often selected as an economical alternative to specifying a stainless steel system.

1.3. STRUCTURAL PERFORMANCE:

The Structural performance criterion can specify the geometric restrictions and/or maximum allowable volume/weight of the silencer that can substantially influence the silencer design process. Secondary loading outside of the weight of the silencer can also affect the design and cost of the exhaust system. A standard silencer is not traditionally designed to absorb substantial loads due seismic activity, wind or thermal growth of adjacent piping. Silencers that are specifically incorporated as an element of an exhaust "stack" should be designed to accommodate the loads that will be absorbed due to potentially high wind loads as well as seismic activity.

II. LITERATURE SURVEY:

George Feng, Vadim Akishin, [1] presented a paper which deals with aerodynamic aspects for designing industrial acoustic silencers, in this they analyzed the flow field in a silencer for pressure and velocity distribution and the comparison of pressure drop from different methods is discussed and the impact of the silencer to the flow system is considered and also the consideration for flow characteristics in the design of a silencer is discussed.

Sanjay S. Gosavi, Vinayak M. Juge, Mayur M. Nadgouda [2] presented a paper on "Optimization of Suction Muffler Using Taguchi's DOE method" which evaluates the effect of variables like length of the insertion tube, cross sectional areas of tubes, volume of expansion chamber, speed of sound in the refrigerant media etc. on functional parameters such as transmission loss characteristics and pressure drop across the muffler. The significance of the above variables in terms of the resulting parameters is decided by using Taguchi's DOE methodology. Optimization for acoustic characteristics is done using commercially available tool "SYSNOISE". The pressure drop characteristics of the muffler are analyzed using "FLUENT". The optimized muffler is validated experimentally using "Bench Test" in the laboratory. A fair agreement between the theoretical and sound test laboratory results is observed.

V. Koteswara Rao, Mahesh S. Murthy, S. Arumuga Raja & Dattu Kumar [3] in their paper presented a case study wherein CFD (Computational Fluid Dynamics) technique was used to study the suction gas flow through the suction muffler for improving the efficiency and reduce the noise level of a hermetic refrigerator compressor. In this paper two factors were considered for volumetric efficiency improvement one is suction gas superheating and the other is restriction of passage for gas flow. This is a very complex phenomenon, which requires a great deal of time and money to study and understand. To understand the refrigerant behaviour through suction muffler, CFD was used in analyzing these complex phenomena to overcome the time constraint otherwise involved.

C.J. Deschampsa, F.C. Possamaib and E.L.L. Pereira [4] presented a paper on numerical simulation of pulsating flow in Suction mufflers in which Head losses of a suction muffler are evaluated with the model taking into account the pulsating flow condition in a dynamic simulation of the whole compressor. The results are compared with those given by an analytical acoustic model and with experimental data so as to assess the model capability. The acoustic model uses an equation to estimate the friction losses but requires empirical loss coefficients that vary according to the compressor working conditions. On the other hand, the computational fluid dynamics model estimates directly the friction losses from its transport equations and therefore requires no such adjustments. Another benefit provided by the fluid dynamics model is the evaluation of gas temperature throughout the muffler, which is considered to be constant in the acoustic model.

Middelberg, Barber, T.J., Leong, S. S., Byrne, K.P and Leonardi, E. [5] published a paper in the proceedings of acoustic 2004 entitled "Computational Fluid Dynamics Analysis Of The Acoustic Performance Of Various Simple Expansion Chamber Mufflers" in which Different configurations of simple expansion chamber mufflers, including extended inlet/outlet pipes and baffles, have been modelled numerically using Computational Fluid Dynamics (CFD) in order to determine their acoustic response. The CFD results are compared with published experimental results. The CFD model consists of an axisymmetric grid with a single period sinusoid of suitable amplitude and duration imposed at the inlet boundary. The time history of the acoustic pressure and particle velocity is recorded at two points, one point in the inlet pipe and one point in the outlet pipe. These time histories are Fourier Transformed and the transmission loss of the muffler is calculated. Calculated results show excellent agreement with the published data. The mean flow performance has also been considered. The mean flow model of the muffler uses the same geometry, but has a finer mesh and has a suitable inlet velocity applied at the inlet boundary and the pressure drop across the muffler is found.

Dr. S. Rajadurai Suresh Natarajan and N.Manikandan [6] published a paper titled "Muffler Pre-Processing Methodology and Comparative Study" in which they summarize the Pre-processing methodology of Exhaust system muffler using pre- processing tool Hyper Mesh for calculating transmission loss in the muffler. Also pre-processing methodology from CFD tool was explained. The advantages of using Hyper Mesh in reducing the pre-processing time are explained in this paper. The transmission loss for particular frequency range i.e. 50 -1000 Hz from experimental test setup i.e. Impedance tube are correlated with simulation results from 3D simulation tool for the developed FE model from Hyper Mesh and CFD tool are compared and explained in detail in this study.

Shital Shah, Sai Sankaranarayana K, Kalyankumar S Hatti, Prof. D. G.Thombare [7] presented a paper titled "A Practical Approach towards Muffler Design, Development and Prototype Validation" which deals with a practical approach to design, develop and test muffler particularly reactive muffler for exhaust system, which will give advantages over the conventional method with shorten product development cycle time and validation. This paper also emphasis on how modern CAE tools could be leveraged for optimizing the overall system design balancing conflicting requirements like Noise & Back pressure.

Lee Ming Wong G. Gary Wang [8] presented a paper titled "Development of an automatic design and optimization system for industrial silencers" in which they used CAD system supported by Design of Experiment (DOE) based optimization methods to integrate product life-cycle considerations into the design It is expected that the developed system can help the development of similar systems for other products. Through the development of this system, some further research issues are identified.

Ar-15 Reflex Sound Suppressor Modelling and Testing by **Christopher Gingrich Daniel Emery Ricky Lynch [9]** The objective of this project is to model a military-style sound suppressor using computational fluid dynamics (CFD) and ANSYS simulations to optimize the design. Suppressor development, since its invention, has generally been through trial and error along with a common understanding of fluid flow. The goal is to use the computer model, once correctly calibrated, to optimize the baffle arrangement and which reduces resources such as materials and labour normally associated with design. The projectile being tested is a standard 5.56mm cartridge currently used by United States military and fellow NATO military forces. The CFD computer model will be simulated using a FLUENT 2-D, axisymmetric geometry and mesh. ANSYS structural models will model loads on suppressor parts caused by high-pressure combustion gases. A physical suppressor, manufactured with 7075 and 6061 aluminium, will be attached to an AR-15 rifle for testing.

Investigation and optimization of the acoustic performance of exhaust systems [10] by **Sara Elsaadany** presented a thesis in which a number of problems relevant for the design of modern exhaust systems for vehicles are addressed. First the modelling of perforated mufflers is investigated. Fifteen different configurations were modelled and compared to measurements using 1D model. The limitations of using 1D model due to 3D or non-plane wave effects are investigated. It is found that for all the cases investigated the 1D model is valid at least up to half the plane wave region. But with flow present, i.e., as in the real application the 3D effects are much less important and then normally a 1D model works well.

III. METHODOLOGY:

3.1. PROBLEM DEFINITION:

The purpose of the work is to investigate the flow pattern of steam in the vent silencer and hence, optimize the design of the vent silencer for better flow process and less noise output.

CFD analysis is carried out on the available design of the vent silencer and it is found that the flow in the silencer is non-uniform due to which the efficiency of the vent silencer to absorb the noise is reduced.

3.2. Objective:

The flow in the silencer is mainly through the diffuser which is designed with multiple circular holes rectangular holes and rectangular extrusions through which the flow is passing from the diffuser to the plenum section. From literature survey it is found that the number of holes and the diameter of holes play a major role on the flow characteristics.

The objective of the project is to design and analyze a steam vent silencer in such a way that the flow of steam into the silencer is uniform and finely distributed in order to get higher Noise Transmission loss from the silencer. The Pressure drop in the silencer is to be maintained as low as possible, since pressure drop causes energy loss. The length and diameter of holes on the diffuser are the two factors considered in this project.

3.3. Grid Independence study:

A grid independence study on unstructured meshes for the silencer was carried. The meshing was carried out by giving different size of the mesh element like 6mm, 5mm, 4mm, 3mm and 2mm and running the simulations for the silencer with different mesh numbers, comparing the results and getting a grid independent mesh size. It was found that the solution no longer changes for a size of the mesh element of 3 mm which gives a total number of elements as 20,00,000 and therefore this was kept constant for simulations.

IV. RESULTS AND DISCUSSION:

Analysis set up of all cases is carried out in Ansys Workbench 14.5. Result analysis is done using ANSYS Post processor. K-epsilon turbulence model is used for Analysis. Velocity and pressure plots are plotted for all the cases of study.

Case No.1: Model having circular holes on inlet diffuser:

The silencer with inlet diffuser having 24 numbers of circular holes of 30mm diameter is considered in this case. This design is modelled in the CATIA V5 software, meshed in ANSYS Mesher and analysed using the Ansys Workbench 14.5. The results of velocity and pressure distribution are shown in the Figure 4.1 and 4.2 below:

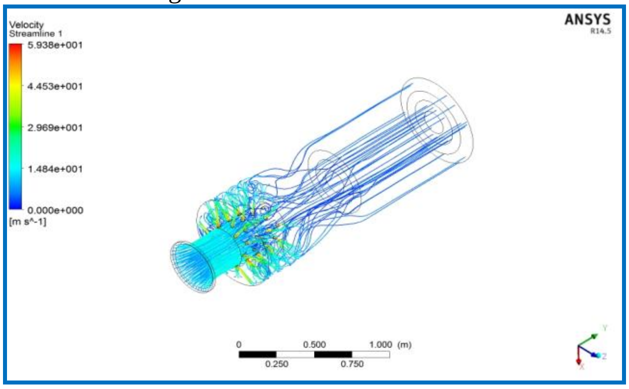

Figure 4.1: Velocity Streamlines For The Model Having Circular Holes On Inlet Diffuser.

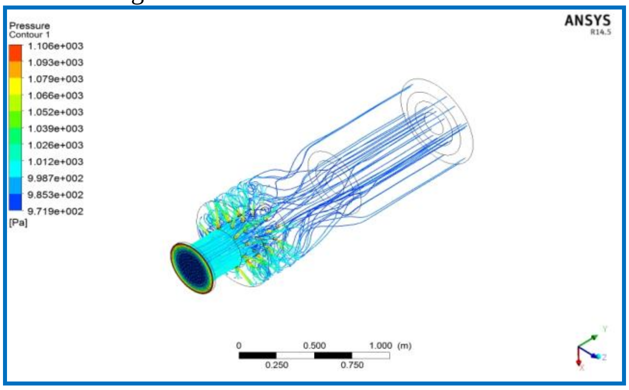

Figure 4.2: Total Pressure Distribution At The Inlet And Outlet For The Model Having Circular Holes On Inlet Diffuser.

Figure 4.1 shows the velocity streamline of the silencer. Having circular holes on the inlet diffuser. Here the colour code shows the velocity variation at different sections of the silencer.

Mass flow rate at which the fluid enters is 0.53 kg/s. Figure 4.2 shows the pressure contours for circular holes on inlet diffuser. In the figure colour code indicates the variation of pressure from inlet to the outlet of the silencer. Pressure varies uniformly at the inlet of the diffuser as the flow progress towards the outlet of silencer pressure drops. From Figure 4.2 we can see that the average pressure at the inlet is 1106 Pa and at the outlet it is 972 Pa showing a pressure drop of 134 Pascal.

Case no 2: Model having rectangular holes on inlet diffuser:

In this case a design of the vent silencer with 24 number of rectangular holes of 30*40mm dimension is considered whose results are shown in the figure 4.3 & 4.4 below.

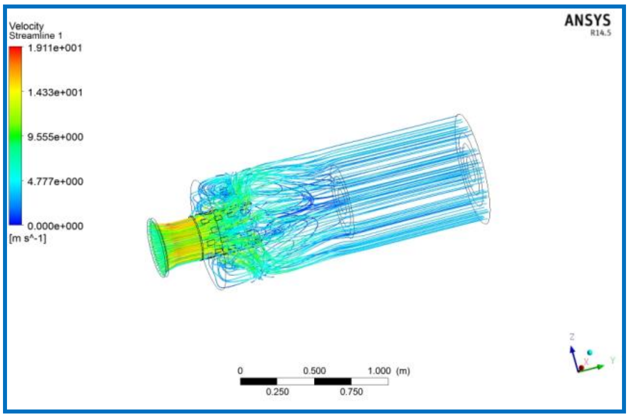

Figure 4.3: Velocity Streamline for the Silencer with 24 Number of Rectangular Holes of 30*40mm Dimension.

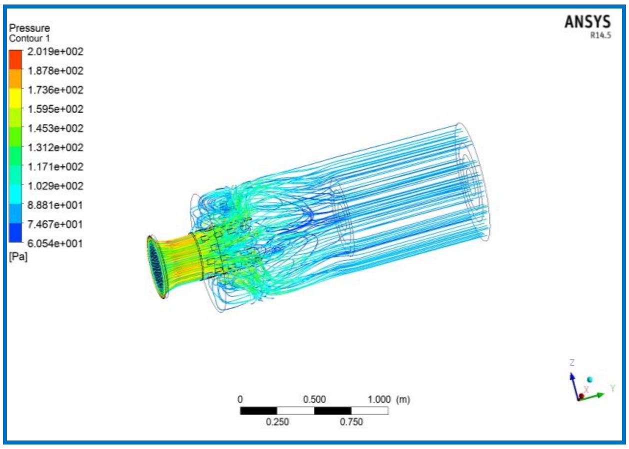

Figure 4.4: Total Pressure Distribution At The Inlet And Outlet For The Silencer With 24 Number Of Rectangular Holes Of 30*40mm Dimension.

Figure 4.3 and 4.4 shows the velocity streamline and pressure contour of the silencer. The colour code indicates the velocity streamline and pressure variation from inlet to the outlet in the vent silencer. Compared to the previous case pressure drop is less in this case. Mass flow rate at which the fluid enters is 0.53 kg/s. From Figure 4.4 it can be observed that the average pressure at the inlet is 202Pa and at the outlet it is 80.5 Pa showing a pressure drop of 121.5 Pascal.

Case No 3: Model having rectangular holes with extrusions on inlet diffuser:

In this case a design of the vent silencer with 24 number of rectangular extrusions with 30*40 mm dimensions and height of 20mm is considered whose results are shown in the figure 4.5 & 4.6 below.

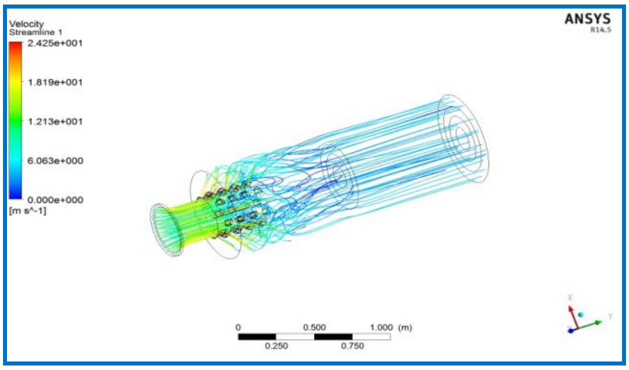

Figure 4.5: Velocity Streamline For Silencer With 24 Holes Of Rectangular Extrusions Of 30*40 Mm Dimension And 20mm Height.

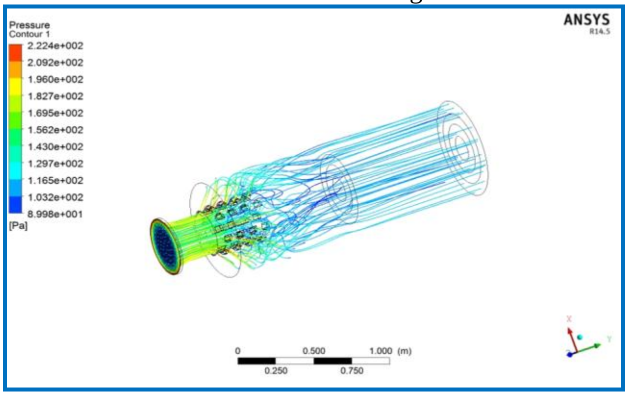

Figure 4.6: Total Pressure At The Inlet And Outlet For Silencer With 24 Holes Of Rectangular Extrusions Of 30*40 Mm Dimension And 20mm Height.

Figure 4.5 and 4.6 shows the velocity streamline and pressure contour of the silencer.
The colour code indicates the velocity streamline and pressure variation from inlet to the outlet in the vent silencer. Mass flow rate at which the fluid enters is 0.53 kg/s.

From Figure 4.6 we can see that the average pressure at the inlet is 223 Pa and at the outlet it is 90 Pa showing a pressure drop of 133 Pascal. Here in this case pressure drop is more than the previous case. This gives better results than the previous case.

Observations:

- Figure 4.1, 4.3 & 4.5 shows the velocity streamline for the cases considered for optimization.
- Figure 4.2 & 4.6 shows the maximum pressure drop and velocity streamline having circular holes and rectangular extrusions.
- Case number 2 gives lesser pressure drop than case 1& 3.
- Therefore, Case number 1&3 are considered as the optimum. This gives much more good results than case number 2.

From these observations we can infer that the design of the silencer with diffuser having 24 number of circular holes with 30mm diameter and holes of rectangular extrusions with 30*40mm dimension and 20mm height has given good results than case number 2. That is diffuser having 30*40mm rectangular holes and diffuser having 30mm diameter circular holes can be considered as best designs compared to the other designs of the silencers.

V. CONCLUSION AND RECOMMENDATIONS:

5.1. CONCLUSIONS:

The work demonstrate the possibilities of improving the performance of a Vent Silencer by changing the design of the Silencer Diffuser by differing the hole size and number of holes.

Following conclusions were drawn during this assessment.

- As the flow becomes uniform the pressure in the Silencer decreases which gave better performance of the silencer.
- The flow becomes uniform as there is increase in the size of the holes and number of holes.
- Three different cases of the silencer were tried with different size and number of holes on the diffuser.
- It is found that the design of the silencer with inlet diffuser having 24 numbers of rectangular extrusions of 30*40 mm dimensions with 20mm height and circular holes of 30mm diameter gave satisfactory results with the flow of the steam into the silencer taking place uniformly and giving maximum pressure drop.
- From these assessments, it can be concluded that 24 number of circular holes and 24 number of rectangular extrusions on inlet diffuser gives more pressure drop than the case 2. However, considering the manufacturing cost, 24 number of circular holes with 30mm diameter on inlet diffuser is less costlier than the others and also it gives maximum pressure drop.

5.2. RECOMMENDATIONS:

By extending this work one can optimize more features to see if further improvements can be achieved. For example, the length and diameter of the whole silencer can be changed to see if further improvements can be made. Furthermore in this work uniform distance between the holes is considered, simulations can be carried out by changing the orientation of the holes and making it random. Experimental tests are not conducted for this study, it is recommended to verify the results with experimental test to ensure the reliability of results.

REFERENCES

[1] George Feng., VadimAkishin., “Flow Consideration in Industrial Vent Silencer” *Journal of Mechanical Engineering Science, v. 17, n. 5, pp. 271-279, 1975.*

[2] Sanjay S. Gosavi, Vinayak M. Juge, Mayur M. Nadgouda., “Optimization of Suction Muffler Using Taguchi’s DOE method” *the Proceedings of the 2000 ASME Design Engineering Technical Conferences, Computers and Information in Engineering Conference*, DETC2000/DAC-14275, Baltimore, Maryland, September 10-13, 2000.

[3] G. Gundareddy., Srimanth Taith “CFD (Computational Fluid Dynamics) technique to study the suction gas flow through the suction muffler for improving the efficiency and reduce the noise level of a hermetic refrigerator compressor” *International Journal of Engineering and Research Technology volume 4 issue 2 2011 page no 20-28.*

[4] C.J. Deschampsa., F.C. Possamaib and E.L.L. Pereira., “Numerical simulation of pulsating flow in Suction mufflers” in *International Compressor Engineering Conference.*

[5] Middelberg, J.M., Barber, T.J., Leong, S. S., Byrne, K.P. and E. Leonardi., “Computational Fluid Dynamics Analysis of The Acoustic Performance of Various Simple Expansion Chamber Mufflers” published a paper in the *Proceedings of Acoustic 3-5 November 2004 Australia.*

[6] Dr. S. Rajadurai., Suresh. Natarajan and N.Manikandan., “Muffler Pre-Processing Methodology and Comparative Study” published in the *Simulation Driven Innovation HTC 2012.*

[7] Shital Shah Sai Sankaranarayana K, Kalyankumar S. Hatti Prof. D. G.Thombare., “A Practical Approach towards Muffler Design, Development and Prototype Validation” *Journal of Engineering Optimization, Vol. 33, No. 6, pp. 707-734, 2001*.

[8] Lee Ming Wong G. Gary Wang., “Development of an automatic design and optimization system

for industrial silencers" the proceedings of the 1997 Design Engineering Technical Conferences, Computers and Information in Engineering Conference, DETC97/DAC-3992, Sacramento, California, September 14-17, 1997.

[9] Christopher Gingrich., Daniel Emery. and Ricky Lynch,. "Ar-15 Reflex Sound Suppressor Modeling and Testing By Christopher Gingrich Daniel Emery Ricky Lynch"

[10] Sara Elsaadany "Investigation and Optimization of the Acoustic Performance of Exhaust Systems"