THERMAL ANALYSIS OF MOTOR BIKE EXHAUST SILENCER FOR ENHANCING HEAT TRANSFER RATE BY ADDING DIMPLE AS ENHANCING FEATURES

[1]Dattatray Dilip Giripunje, [2]Vilas B. Shinde, [3]Swapnil S. Kulkarni
[1]ME-CAD/CAM & Robotics- Student, Datta Meghe COE, Airoli, Mumbai, India
[2]Director, Amity School of Engineering & Technology (ASET), India
[3]Assosciate, Advent Tool Tech PVT. LMD, Pune, India

ABSTRACT

The exhaust system of automobile are directly exposed to high temperatures as they form the passage for the hot gases which generate from combustion of fuel and released to the atmosphere. It consist of three parts such as exhaust manifold, catalytic converter and resonator out of those resonator having very short life span as there is lot of restriction provided to the flow of hot gases in order to reduce the noise level hence gases staying more time in this section as compare to other two part of exhaust system. Hence that area needs to be focused during design phase. The uniform heat distribution over the entire exhaust system is important for ensuing enhanced life of elements in the sub-system. The problem identified for this dissertation work is to assess the heat flow along the passage of hot gases and design the passage or passage surface such as to minimize the harmful effects of hot-spots over the length of the silencer, especially at the outer body of silencer. Silencer proposed in this work used for 180cc petrol engine which having configuration 17.02 @ 8500 (PS@RPM) and silencer designed such way to achieve configuration such as TL (transmission loss noise) 30 dB and allowable back pressures 10 in H2O.

KEYWORDS: *Exhaust system, Local heat transfer coefficient, Effective heat transfer area, Dimple pattern.*

NOMENCLATURE:

P= Pitch Between The Two Adjacent Dimple, D= Diameter Of Tube,H=Depth Of Tube, PD= Print Diameter Of Dimple, ST= Latral Distance Between Two Dimple Row, SL=Longitudinal Pitch Between Two Dimples, DX=Raidius Of Dimple Along X- Axies,
DY = Raidius of Dimple Along Y- Axies, PD/H= Profile Stand Ratio, PD/P= Print Diameter To Pitch Ratio.

I. INTRODUCTION

Internal combustion engines are typically equipped with an exhaust muffler to suppress the acoustic pulse generated by the combustion process. A high intensity pressure wave generated by combustion in the engine cylinder propagates along the exhaust pipe and radiates from the exhaust pipe termination. The pulse repeats at the firing frequency of the engine which is defined by f= (engine rpm x number of cylinders)/120 for a four stroke engine. The frequency content of exhaust noise is dominated by a pulse at the firing frequency; Silencer is also used in many other engines and generators. The size, shape and construction vary according to the type and size of the engine. The primary function of the silencer is to reduce engine noise level. The exhaust system of automobile or any IC engine consists of three components such as tail pipe, catalytic converter and resonator (i.e. silencer or muffler). So all the gaseous generated they have to pass through this complete exhaust system. But because of functional requirement exhaust gaseous staying longer time in resonator (i.e. silencer section) in order to reduce noise level which al so result thermal failure of resonator and shorter life span. Construction wise silencer classified in to two types first is reactive and second is dissipative or absorptive silencer. In general, sound waves propagating along a pipe can be attenuated using either a dissipative or a reactive muffler. There are many author worked in heat exchanger field and use different technique to enhance heat transfer rate. The some technique they used is passive and some are active one. In passive technique they change the design of actual component by adding some extra features in that [1] [2] [4] [5] [6] [7] [8] [44]. Those responsible to enhance heat transfer rate from the same surface. They using shallow spiral, dimple pattern, surface roughness, fins, and twisted tape or different type of inserts wise features. And in active techniques [10] [12] [13] [14] they try to make flow more turbulent by adding swirl generator, nozzle and multiphase flow. But out of those techniques only few are possible to adopt on the surface of silencer body but rest are only possible to adopt in the flow or inside of silencer. So to enhance heat transfer rate from the surface of silencer body we have to adopt such features which possible to built on the surface of it. Again some of them are possible and some are not. Those are listed in table 1.

Table 1 Comparison of different heat transfer technique

Features	Inboard (Inner side of silence)	Outboard (Outer side of silencer)	Comment as per Silencer application
1.Fins	NA	A	It possible but may harmful for rider. (it may cut driver or co passenger skin as it located near to their legs)
2.Dimple pattern	A	A	It is possible to employ in case of silencer.
3.Shallow extended spiral	A	A	It is possible to employ in case of silencer.
4.Surface roughness	A	A	It is possible to employ in case of silencer but become ineffective after deposition of mud on silencer body.

(A: applicable, NA: not applicable)

As mention in table some techniques are inboard type and some are outboard. The inboard technique are those which can located at inner side of silencer and outboard are those which can located at outer side. But inboard techniques are responsible to generate back pressure as it directly present in the flow field. So it may build back pressure on engine as it located inside of silencer. So from the table dimple pattern are the best enhancement features in case of silencer.

II. Methodology

1. Study of design of existing silencer :

The silencer used in our case is reactive type of silencer. Hence it consists of gas expansion chambers in to it and those are design on the basis of acoustic and back pressure (i.e. transmission loss noise 30 dB and allowable back pressures 10 in H2O) consideration. It is made of aluminized mild steel.

2. Geometry modeling of existing silencer :

The dimensions and fluid domain modeling of silencer are shown in figure 1 and 2 respectively.

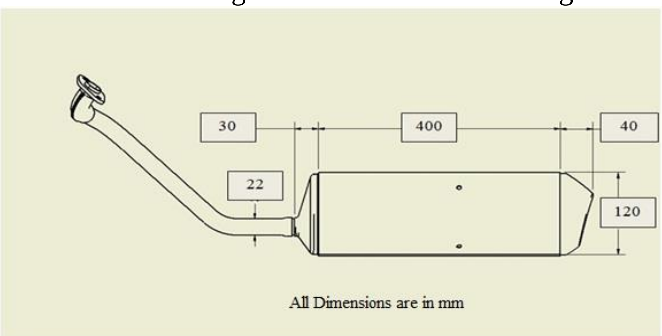

Figure1: Dimension of silencer

As shown in figure 1 bike silencers are mainly having three section inlet tail pipe, silencer body and its outlet. But only tail pipe section of silencer has tube in tube structure; as it is directly deals with high gas pressure and temperature which swept out from combustion chamber. The inner tube of tail pipe having maximum thickness as compare to outer tube so it can withstand at high temperature.

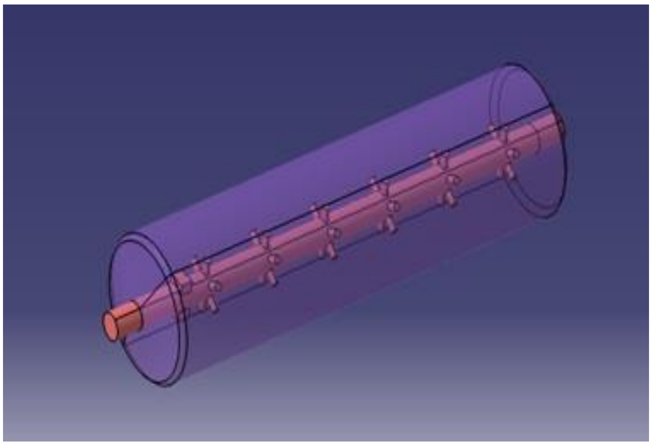

Figure 2 : Fluid domain model of silencer

But as we goes further silencer body not having tube in tube structure; instead of that it only made of 1.5 mm thick aluminized mild steel or CRCA or Stainless steel sheet. In which various type of obstacle present to the flow of gaseous. Because of such arrangement it has to handle high temperature for long time. Hence to identify the maximum temperature that can gain by silencer body from exhaust gaseous; we are only considering silencer body and outlet section without tail pipe.

3. CFD Analysis of existing silencer for “thermal (heat dissipation)” :

Geometry modeling of complete silencer is done on CATIA modeling software. After that it imported in to GAMBIT 2.2.30 software for preprocessing purpose (i.e. for meshing and fluid definition). Once all preprocessing is done it converted in to mesh file that file can importable in to Ansys Fluent for solving purpose. Analysis of existing silencer consist two parts. In first part we are considering complete silencer body with some heat transferring wall at atmospheric condition which is shown in figure 3. From that analysis we will get the maximum temperature that can be attained by silencer body. And in second part we are only considering body of silencer which is initially at maximum temperature getting from first part and subjected to same heat transfer atmospheric condition. The schematic figures of first and second parts of analysis are shown in figure 3 and 4 respectively.

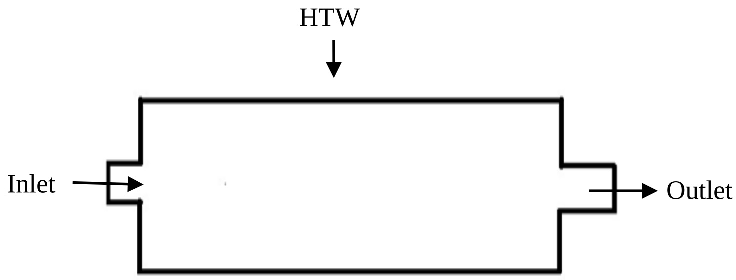

Figure 3: Schematic diagram of first part of analyses

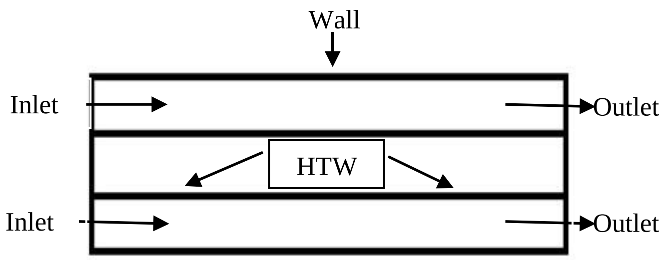

Figure 4: Schematic diagram of second part of analyses

Assumption for CFD analysis:
1) Flow is considered to be steady.
2) Air is considered as fluid for computations.
4) Flow considered as Turbulent (K-Ɛ model).
5) Inlet considered as velocity inlet.
6) Outlet considered as pressure outlet.

As per schematic diagram in figure 3 and 4 we carried out the analysis in two parts. In first part we considering complete silencer with the following boundary condition.

For inlet: - Velocity inlet

Inlet flow velocity (m/s)	70
Inlet temperature (K)	853
Turbulence intensity (%)	10
Hydraulic diameter (m)	0.12

For outlet: - Pressure outlet

Gauge pressure (Pa)	0
Back flow turbulent intensity (%)	10
Back flow hydraulic diameter (m)	0.12

For wall: - HTW (Heat transfer wall)

Material Name	Aluminum
Convective Heat Transfer Coefficient (w/m2-k)	40
Free Stream Temperature (k)	303

Analysis result for hot spot detection is shown in figure 5. As per the result the maximum temperature attained by the silencer body is 660 K and minimum temperature attained by the body is 491 K.

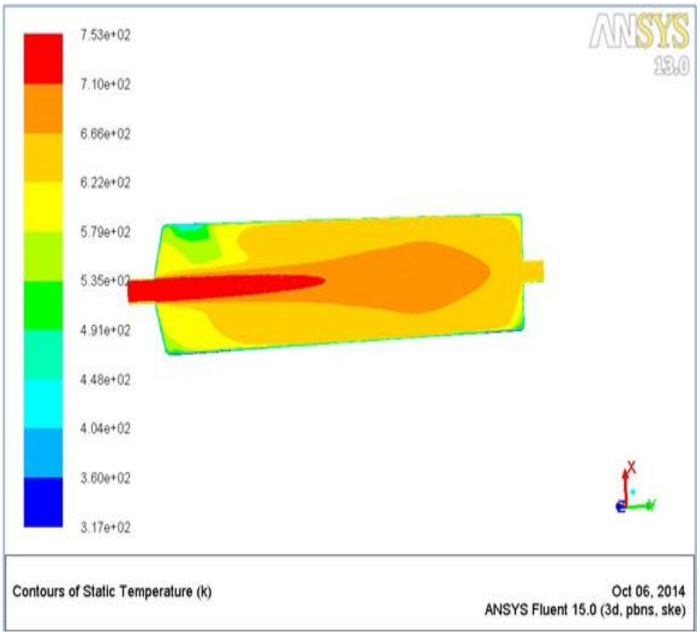

Figure 5: Analysis of complete silencer

Hence now our aim is to reduce the maximum temperature which generate on the silencer body. But first of all we have to identify without any modification how much that temperature cools down on the silencer body. For that only consider silencer outer body on which mainly hot spot is generate. And this is a second part of analysis. The boundary condition for this part is given below.

For inlet: - Velocity inlet

Inlet flow velocity (m/s)	11.2
Inlet temperature (K)	303
Turbulence intensity (%)	10
Hydraulic diameter (m)	0.12

For outlet: - Pressure outlet

Gauge pressure (Pa)	0
Back flow turbulent intensity (%)	10
Back flow hydraulic diameter (m)	0.12

For wall: - wall, HTW (Heat transfer wall)

Material Name	Aluminum, Aluminum
Convective Heat Transfer Coefficient (w/m2-k)	0, 120
Temperature (K)	300,300
Free Stream Temperature (k)	300, 650

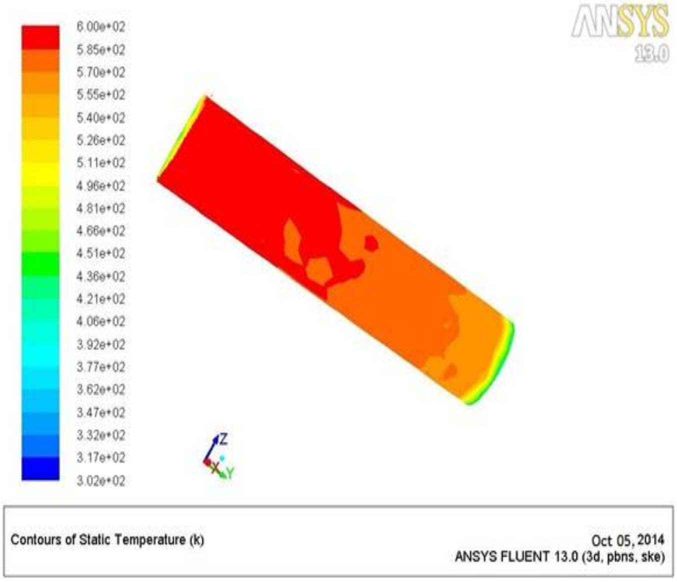

Figure 6 : Analysis of outer body without modification

The analysis of outer body of silencer is shown in figure 6. From the analysis it is visible that hot spot generated on the silencer body is not affected and it present on their position. The maximum and minimum temperature generate on the silencer outer body are 600 and 585 K respectively.

4. Decided modification in the geometry and recommendation of best suitable solution :

As mention in table 1 we decided to use dimple surface at outer body of silencer. But before modification the work of different author done on dimple feature is mention in table 2. So we can make our design assumption on that basis.

Table 2: Different technique of investigation

Author name	Tested structure	Changing parameter	Constant parameter
Chinruk Thianpong et. al. [44]	Dimple tube (spherical dimple)	P/D = 0.7,1	PD, D, H
Sandeep S. Kore et al. [4]	Square channel (spherical dimple)	PD/H = 5, 3.33, 2.5	PD
N. Katkhaw et al. [8]	Square channel (ellipsoidal dimple)	S_T/S_L	PD, H
Iftikar ahamad H. Patel et al. [5]	Square duct (spherical dimple)	Inline and Staggered pattern	P, H, PD

Assumption before design modification:
1) Pitch between dimples should be constant.
2) The ratio of print diameter to height of dimple should be maintaining 2:1.
3) Use of inline pattern of 16 numbers of dimples.
4) Use maximum circumference fitting on silencer body.
5) Surface area of dimple along the x and y axis should be uniform (i.e. $D_x/D_y = 1$)

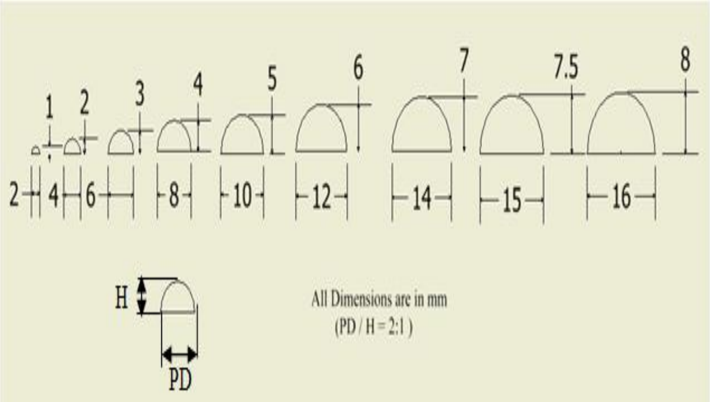

Figure 7: Dimension of dimple (PD/H ratio)

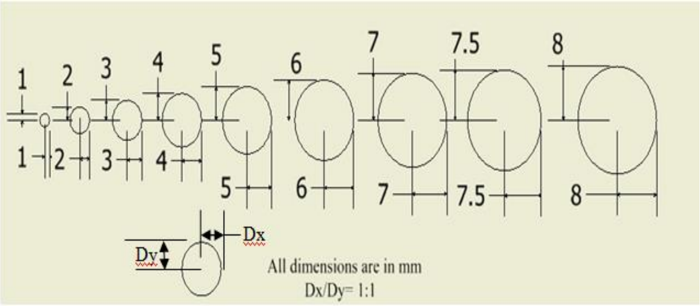

Figure 8: Dimension of dimple (D_x/D_y ratio)

As shown in figure 7 the ratio of print diameter to height (PD/H) of dimple is 2:1 for all dimples. But as per the assumption if we keep the pitch between two dimple constant then diameter to pitch ratio (PD/P) for different dimple features are goes on increasing as diameter increase. The number of dimples used for different trial is shown in figure 8. For comparison purpose the boundary condition for all those configuration are same. Those are mention below.

For inlet: - Velocity inlet

Inlet flow velocity (m/s)	11.2
Inlet temperature (K)	303
Turbulence intensity (%)	10
Hydraulic diameter (m)	0.12

For outlet: - Pressure outlet

Gauge pressure (Pa)	0
Back flow turbulent intensity (%)	10
Back flow hydraulic diameter (m)	0.12

For wall: - Wall, HTW (heat transfer wall)

Material Name	Aluminum, Aluminum
Convective Heat Transfer Coefficient (w/m2-k)	0, 120
Temperature (K)	300,300
Free Stream Temperature (k)	300, 660

Temperature recorded during the analysis of different diameter of dimple with same boundary condition is shown in figure 9.

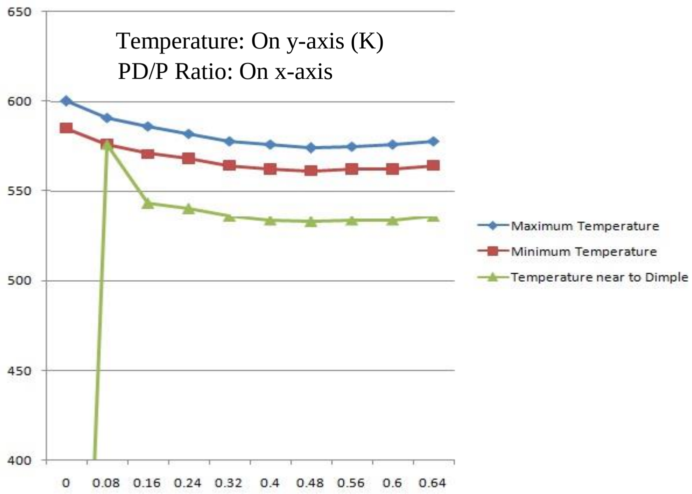

Figure 9: Comparison of result between plain and different dimple surface of silencer body from CFD

From the above analysis it is found that temperature near to dimple surface are goes on decreasing as the ratio of print diameter to pitch (PD/P) increases up to 0.48. But as the ratio of print diameter to pitch increases above 0.48 it start to show its inverse effect on temperature drop. Hence use of sixteen dimples of 12 mm diameter and 25 mm pitch will be effective modification for silencer body. To enhance complete silencer body we have to arrange dimple of effective features on the entire surface of it. As the diameter of silencer body is 120 mm then to form cylinder of that diameter we have to use metal sheet of 380 mm and maximum number of dimple row possible on that sheet is 18. Hence we are adding 18 number of row on the silencer body. Analysis of final geometry with same boundary condition is shown below.

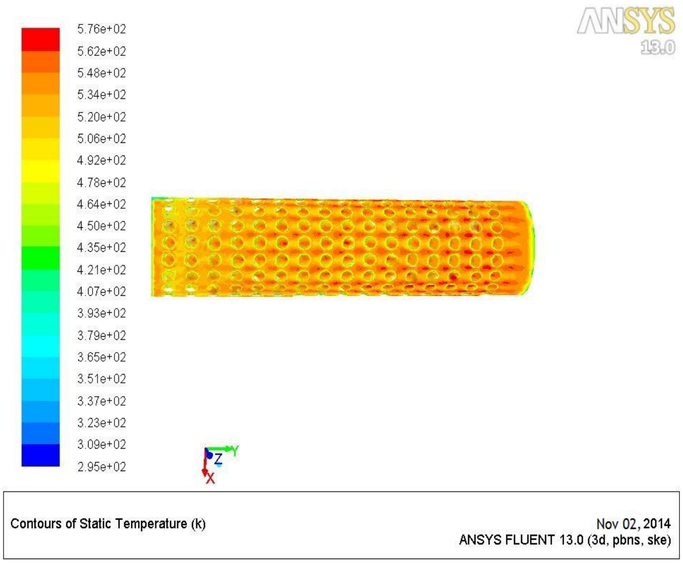

Figure 10: Analysis of silencer body (with dimple pattern of 12mm diameter)

5) Observation on experimental setup (prototype) of silencer plain and modified body:
On the basis of final modification experimental setup are prepare. This is shown in figure 11. In experimental setup we are only testing silencer outer body as tested during the analysis. The test section of experimental setup consists of two different bodies which made up of 1.2 mm thick aluminized mild steel sheet. On the one test body finalize dimple pattern are present and other body made as completely plain tube. These are shown in figure 12.

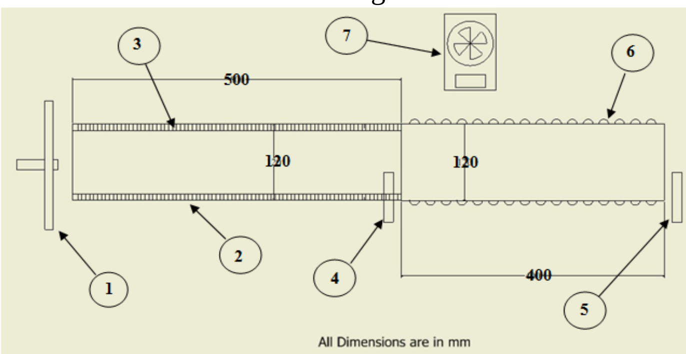

Figure 11: Experimental setup

1) Blower. 50 W. 2200 RPM Max.
2) 120 mm diameter and 500 mm length of aluminum tube for heater mounting.
3) Heater coil. 800 W
4) RTD PT 100 type temperature sensor (for inlet temperature). +/- 1˚C
5) RTD PT 100 type temperature sensor (for outlet temperature). +/- 1˚C
6) Test section (i.e. dimple tube with 120 mm diameter and 400 mm length).
7) Digital anemometer. Specification range 0.6 to 40 m/s (+/- 0.1 m/s)

Figure 12: Plain and Modified body of silencer

In this experimentation we are checking the surface temperature at center positions of both silencer bodies i.e. for plain and for modified one. In addition to that we are calculating heat transfer rate from the surface of both body. For the comparison purpose boundary condition of both test section are same. But it is difficult to achieve actual boundary condition used in CFD analysis because of prototype limitation. Hence to validate CFD result and modification we are taking two more run on ansys for modified and plain silencer body with prototype boundary conditions.

Experimental procedure: the layout of experimental setup shown in figure 11. It consist of two tubes one is supporting tube having diameter 10 mm and length 500 mm and second is test tube which may plain or with dimple surface having diameter 120 mm and length 400 mm. Those tubes are made from aluminized mild steel. The blower are fitted in frontal portion of tube which blow the air at inside as well on the outer surface of test tube to give the fill of free stream over the surface of silencer body. The air which enter inside the tube section are heated with the help of heating coil which present at inside the supporting tube. As that heated air enter inside the test section first temperature sensor sense the inlet temperature of it. With further moment of air inside the test section it starts to transfer heat with surrounding through the wall of it; and as it reaches near to end of test section second temperature sensor sense the outlet temperature of air. We have to repeat this experiment with both tubes at same boundary condition. As we are using dimmer circuit for both blower and heating coil we can take variable inlet condition of air in to test section.

Data Reduction:

The data reduction of the measured result is summarized in the following procedure. Heat transfers mainly take place via convection mechanism. Therefore, at the steady state, the rate of heat transfer absorbed by the fluid is assumed to be equal to the rate of convective heat transfer [43] which can be expressed as:

$Q_o=Q_{conv}$ ……………………………………………………………I

Where, $Q_o=mC_{p,a}(T_i-T_o)$ ……………………………………………………......II

The convection heat transfer from the test section can be written by:

$Q_{conv}=hA_{surf}(T_b-T_w)$ …………………………………………………...III

Where,

$T_b=(T_i+T_o)/2$ …………………………………………………...IV

Hence from equation I, II and III

$h=mC_{p,a}(T_i-T_o)/A(T_b-T_w)$ …………………………………………….........V

From equation V we can able to determine local heat transfer coefficient at outer surface of tube at different reynolds number on the base of outer air velocity. Hence for comparison purpose we are performing test on two different outer air velocities. The Reynolds number for two different tests is 23077.6 and 30770.2 respectively. The recorded data during the experiment are summarized in table 3 and 4.

Table 3a : Observer parameter for plain body

Experimental Data for Plain Tube				
Velocity	Inlet temperature	Outlet temperature	Reynolds number	Local heat transfer coefficient
3 m/s	403 K	385 K	23077.6	33.7885
4 m/s	403 K	373 K	30770.2	84.0362
3 m/s	416 K	390 K	23077.6	44.6697
4 m/s	416 K	381 K	30770.2	87.9080

Table 3b: Observed parameter for dimple body

Experimental Data for Modified Tube				
Velocity	Inlet temperature	Outlet temperature	Reynolds number	Local heat transfer coefficient
3 m/s	403 K	370 K	23077.6	67.1357
4 m/s	403 K	364.44 K	30770.2	113.35
3 m/s	416 K	381 K	23077.6	62.791
4 m/s	416 K	374 K	30770.2	109.25

Table 4 : Surface temperature of silencer body

Velocity of air near to surface of silencer body	Temperature near to inlet section (K)		Temperature at center portion from CFD results (K)		Temperature at center portion from Experimental setup (K)	
	Plain	Dimple	Plain	Dimple	Plain	Dimple
4 m/s	403	403	339	334	342	338
4 m/s	416	416	344	337	347	340

The CFD and experimental result as shown in table 4 are approximately matches. From that we can say dimple structure body will be better alternative for existing design.

III. CONCLUSION

Our work and the results obtained so far are very encouraging and reinforce the conviction that modified exhaust subsystems are practical, efficient, and economically potential to contribute more heat transfer from the surface of silencer body. And prevent the exhaust subsystem against the generation of hot spot which creates high temperature oxidation on silencer body which responsible for the mechanical breakage of silencer.

IV. FUTURE SCOPE

1. In case of our modified silencer body we are using spherical inline dimple pattern with dimple head count 288. We can also check the effect of different shape of dimple pattern on the surface of silencer body by changing the D_x / D_y ratio to achieve same or more efficient effect with less dimple head count.
2. We can also check the effect of finalize dimple diameter with different profile stand ratio in order achieve minimum and effective height of dimple.
3. We can develop correlation for dimple in terms of PD/P ratio which is function of prandlt and Reynolds number.

REFERENCES

[1]. Eugenio Aulisa , Antonio Barletta, Massimo Gallipoli, Alessandro Terenzi , Enzo Zanchini, “CFD analysis and overheating control of a turbine”, International Journal of Thermal Sciences 43 (2004) 1119–1124.
[2]. Article from HRS technology.
[3]. Mehmet Avcu, Şadi Kopuz et al., “Diesel Engine Exhaust System Design” , Journal of Naval Science and Engineering 2010, Vol. 6.
[4]. Sandeep S. Kore, Satishchandra V. Joshi, Narayan K.Sane “Experimental Investigations Of Heat Transfer Enhancement From Dimpled Surface In A Channel” IJEST 2011.
[5]. Iftikarahamad H. Patel, Sachin L. Borse “Experimental Investigation Of Heat Transfer Enhancement Over The Dimpled Surface” IJEST 2012.
[6]. R. L. Edlabadkar, N. K. Sane, G.V.Parishwad “Computational Analysis of Natural Convection with Single V-Type Partition Plate” 5th European Thermal-Sciences Conference,2008.
[7]. Satish G. Kandlikar, Shailesh Joshi, Shurong Tian “Effect of channel roughness on heat transfer and fluid flow characteristics at low Reynolds number in small tube” 35th National Heat Transfer Conference , California , ASME.
[8]. Nopparat Katkhaw, Nat Vorayos, Tanongkiat Kiatsiriroat, Yottana Khunatorn, Damorn Bunturat, Atipoang Nuntaphan, “Heat transfer behavior of flat plate having 45˚ ellipsoidal dimpled surfaces” Case Studies in Thermal Engineering 2 (2014) 67–74.
[9]. Article from orbit coating.
[10]. Veeresh Fuskele,R.M.Sarviya, “Experimental investigation of heat transfer enhancement in double pipe heat exchanger using twisted mesh tape” , IJARS/vol. 1/ issu II/Jan 2009 05-09.
[11]. R. Zimmerman, M. Gurevich, A. Mosyak, R. Rozenblit, G. Hetsroni, Heat transfer to air–water annular flow in a horizontal pipe, (elsevier) International Journal of Multiphase Flow 32 (2006) 1–19.
[12]. Dr. A. G. Matani , Swapnil A. Dahake, “Experimental Study On Heat Transfer Enhancement In A Tube Using Counter/Co-Swirl Generation”,(IJAIEM) ISSN 2319 – 4847.
[13]. C.Nithiyesh Kumar,P.Murugesan, “Review on Twisted Tapes Heat Transfer Enhancement, International Journal Of Scientific & Engineering Research” , Volume 3, Issue 4, April-2012.
[14]. S. Liu, M.Sakr, “A comprehensive review on passive heat transfer enhancements in pipe exchangers”,(Elsevier) Renewable and Sustainable Energy Reviews 19 (2013) 64–81.
[15]. Li Zhang , Hongmei Guo , Jianhua Wu ,Wenjuan Du “Compound heat transfer enhancement for shell side of double-pipe heat exchanger by helical fins and vortex generators” Heat Mass Transfer (2012) 48:1113–1124.

BIOGRAPHY

Dattatray Dilip Giripunje ME-CAD/CAD & Robotics- student, Datta Meghe College Of Engineering, Iroli, Mumbai

Dr. Vilas.B. Shindem Director, Amity School of Engineering & Technology (ASET) Registrar, Amity University, Mumbai,. On lien Professor in Mechanical Engineering, at DMCE, Airoli, Navi Mumbai. Total Experience: 24 Years

Swapnil.S. Kulkarni Associate, Advent Tool Tech India Pvt. Ltd., Pune. The Company offers Engineering Services and Manufacturing Solutions to Automotive OEM’s and Tier I and Tier II Companies. He is a Graduate in Industrial Engineering with PG in Operations Management. With around 20 years of working experience in the domain of R&D, Product Design and Tool Engineering, he has executed projects in the Automotive, Medical and Lighting Industry. His area of interest is Research and Development in the Engineering Industry as well as the emerging sector of Renewable Energy.