Global Journal of Advanced Engineering Technologies and Sciences

ANALYSIS OF TWO WHEELER EXHAUST SILENCER WITH THERMODYNAMICS PRINCIPLE AND COMPUTER AIDED DESIGN

Ravindra S .Girge, Nitesh Rane

Department of Mechanical Engineering, Dr. APJ Abdul Kalam University, Indore (M.P.)

ABSTRACT

Exhaust system of an automobile consist of three parts such as exhaust manifold, catalytic converter and silencer out of those silencer having very short life span as there is lot of restriction provided to the flow of hot gases due to complex geometry in order to reduce the stress, noise level hence gases staying more time in this section as compare to other two part of exhaust system. There are important areas to be focused during design phase of silencer in order to achieve the uniform distribution of heat over the entire exhaust system which consequently enhanced life of the elements in the exhaust system. The problem recognized for this proposed study is to assess the heat flow during the passage of hot gases and design the passage in such a way that it will minimize the destructive effects of hot-spots and localized thermal stress over the length of the silencer, especially at the front end mating with the exhaust manifold.

Keywords: Exhaust silencer, hear transfer, heat exchanger Exhaust system, Local heat transfer, coefficient, Effective heat transfer area.

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=Longitud in a lPitch Between Two Dimples,
DX= Raidius Of Dimple Along X- Axies,
DY=RaidiusofDimpleAlongY-Axies
PD/H=ProfileStandRatio,
PD/P=PrintDiameterToPitchRatio.

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 A It is possibl etoem ploy in case of silencer. 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.

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.

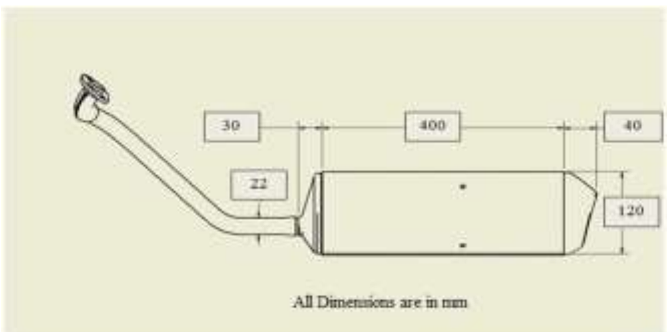


Figure1: Dimension of silencer

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.

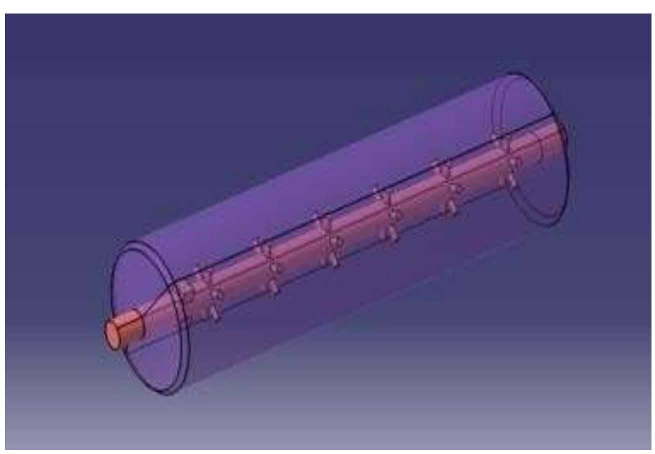


Figure2: Fluid domain model of silencer

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

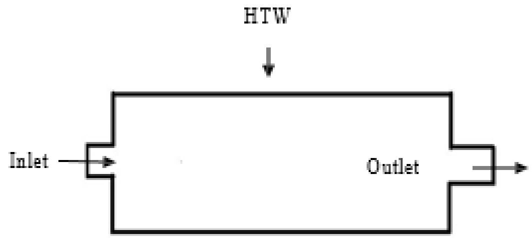


Figure3: Schematic diagram of first part of analyses

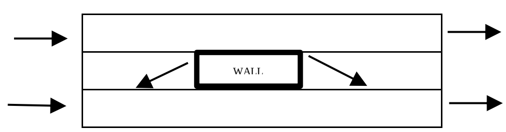


Figure4: Schematic diagram of second part of analysis

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 -velocity inlet

Inlet flow velocity (m/s)	70
Inlet temperature (K)	853
	10

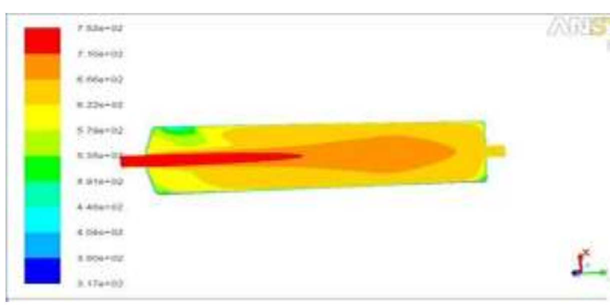
ISSN 2349-0292
Impact Factor (PIF): 3.802

Turbulence intensity (%)	
Hydraulic diameter (m)	0.12

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	42
Free Stream Temperature(k)	304

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. 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.



Inlet flow velocity (m/s)	11.3
Inlet temperature(K)	304
Turbulence intensity (%)	10
Hydraulic diameter(m)	0.13

For outlet-pressure outlet

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

ISSN 2349-0292
Impact Factor (PIF): 3.802

For wall-wall ,HTW(heat transfer wall)

Material name	aluminium
Convective heat transfer	0,120
temperature	300,300
Free streaemperaturem t	300,650

Different technique of investigation

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. 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 . So we can make our design assumption on that basic

Author name	Test structure	Changing parameter	Constant parameter
ChinrukThianpong et.al.[44]	Dimpletube (spherical dimple)	P/D=0.7,1	PD,D,H
SandeepS.Kore etal.[4]	Squarechannel (spherical dimple)	PD/H=5,3.33,2.5	PD
N.Katkhawetal. [8]	Squarechannel (ellipsoidal dimple)	ST/SL	PD
IftikarahamadH. Pateletal.[5]	Squareduct (spherical dimple	Inlineand Staggeredpattern	P,H,P,D

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)

Figure7: Dimension of dimple(PD/Hratio)

ISSN 2349-0292
Impact Factor (PIF): 3.802

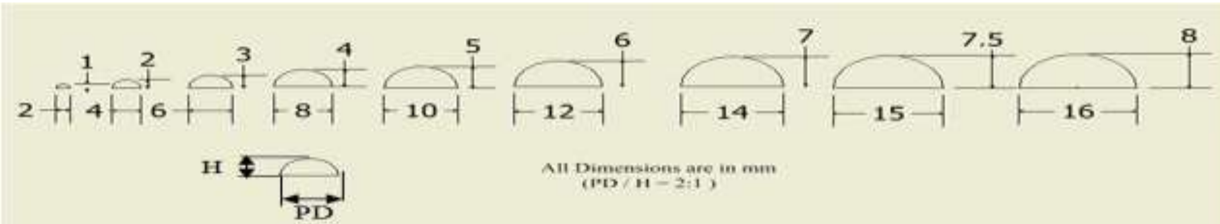
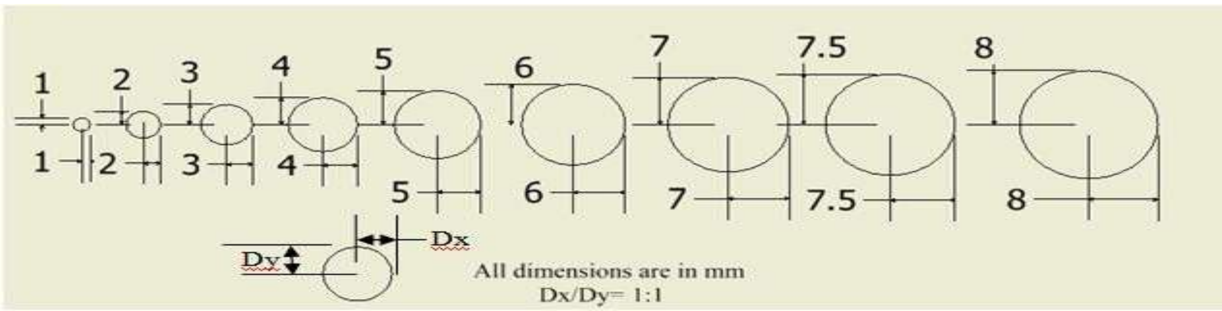


Figure8: Dimension of dimple (Dx/Dyratio



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 inc reasing 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 m/s
Inlet temperature (K)	303 (30^0C)
Turbulence intensity (%)	10
Hydraulic diameter (m)	0.12

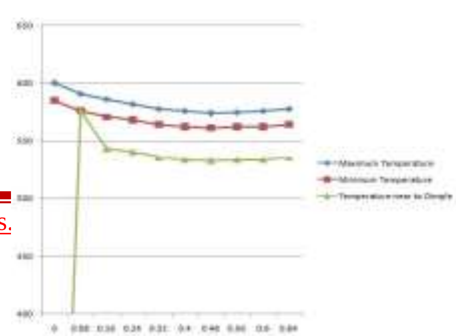
For outlet: - Pressure outlet

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 (27^0C),300 (27^0C)
Free Stream Temperature (K)	300(27^0C), 660(387^0C)

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



ISSN 2349-0292
Impact Factor (PIF): 3.802

Figure9: 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 in verse 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

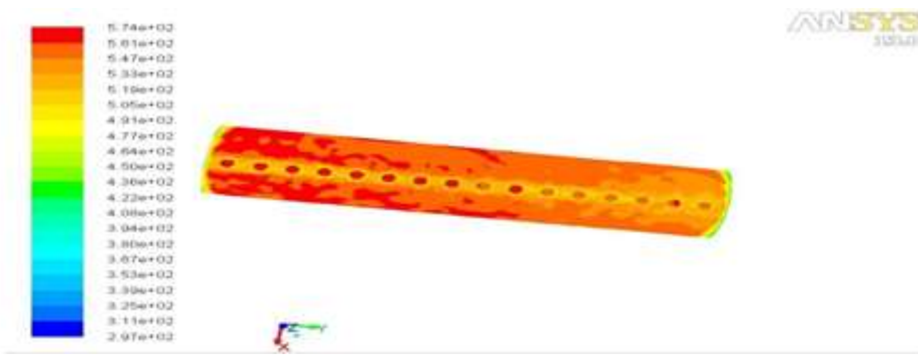


Figure 4.21 Analysis of silencer body (with dimple of 12 mm diameter)

Observation on experimental setup (prototype) of plain and modified silencer body

On the basis of final modification experimental setup are prepare. This is shown in figure 4.29. 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. The dimension of aluminized mild steel sheet from which silencer body will generate and final structure for both dimple and plain tube are shown in figure 4.30 and 4.31 respectively.

ISSN 2349-0292
Impact Factor (PIF): 3.802

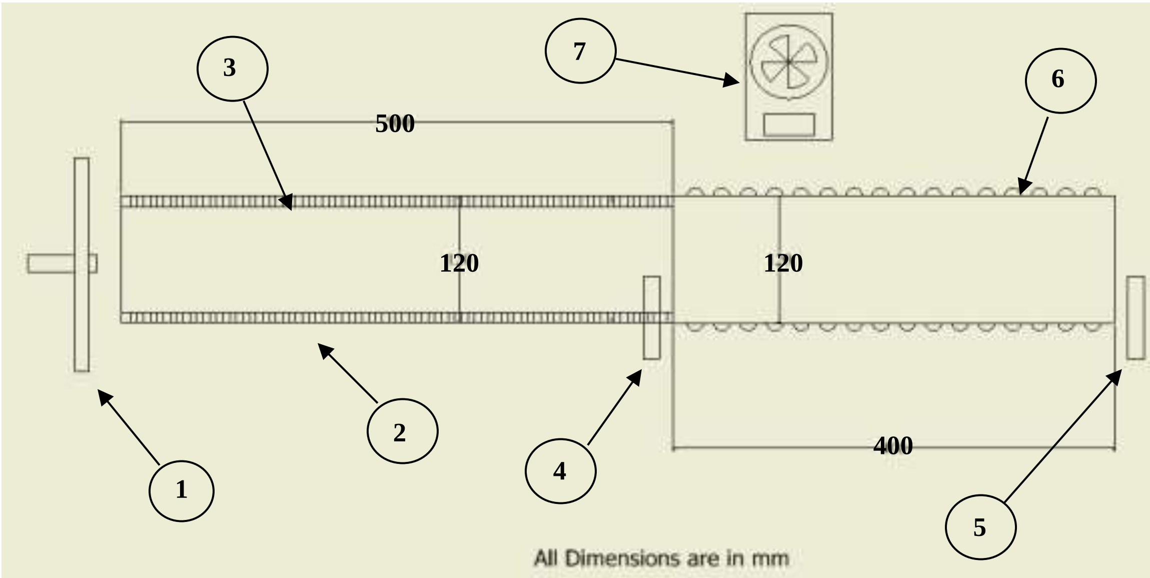


Figure 4.29 Experimental setup

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)

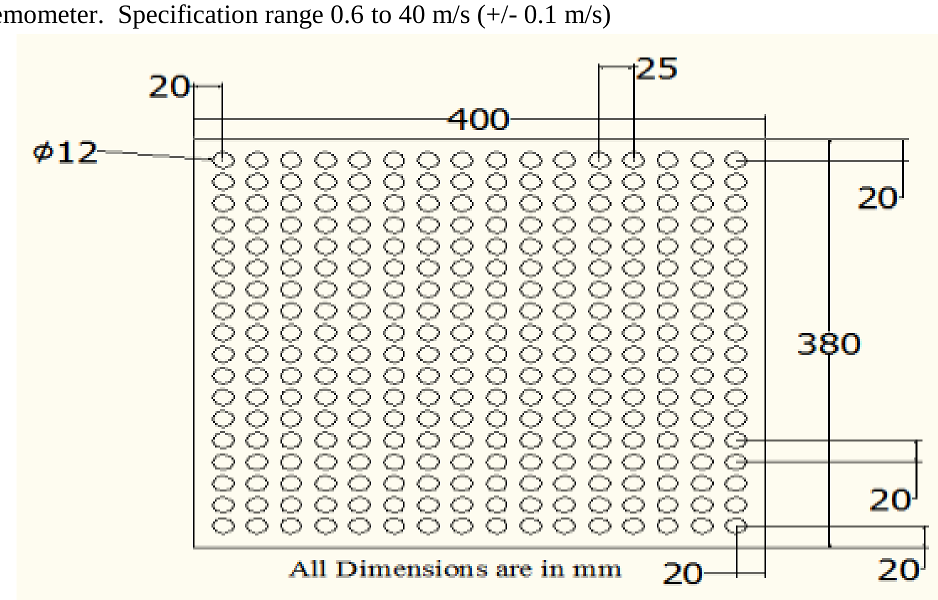


Figure 4.30 Dimension of aluminized mild steel sheet

ISSN 2349-0292
Impact Factor (PIF): 3.802

Figure 4.31 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. This is shown in figure 4.32, 4.33, 4.34 and 4.35.

Experimental procedure: the layout of experimental setup shown in figure 4.29. 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 [62] which can be expressed as:

$Q_o = Q_{conv}$……………………………………………………I

Where, $Q_o = m\, C_{p,a}(T_i - T_o)$ …………………………………………….....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

$h= m C_{p,a} (T_i-T_o) / A (T_b-T_w)$ ………………………………………………...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. All the calculation are summarized in table number 3 and 4.

Table 3a-observation parameters for plain body

Velocity inlet	Temperature	Outlet temperature	Reynold number	Local heat transfer
3m/s	402k	385k	23077.4	3.88
4m/s	402k	373	3088.64	4.08
3m/s	415k	381k	23088	2.82
4m/s	415k	374	30770.21	9.26

Table4b-Observation parameters for dimple body

Velocity inlet	Temperature	Outlet temperature	Reynold number	Local heat transfer
3m/s	402k	380k	23077.5	7.88
4m/s	402k	363k	3088.64	14.08
3m/s	415k	371k	23088.22	1.82
4m/s	415k	363k	30770.23	10.26

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.

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.

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].EugenioAulisa, AntonioBarletta, MassimoGallipoli, AlessandroTerenzi, EnzoZanchini,"CFD analysis and overheating control of aturbine", International Journal of Thermal Sciences4 3(2004) 1119–1124.

[2].Articlefrom HR Stechnology.

[3].Mehmet Avcu, Şadi Kopuzetal., "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].If tikarahamad H.Patel, SachinL.Borse "Experimental Investigation Of Heat Transfer Enhancement Over The Dimpled Surface" IJEST 2012.

[6].R.L. Edlabadkar, N.K.Sane, G.V. Parishwad "Computational Analysisof Natural Convection with SingleV-Type Partition Plate " 5th European Thermal-Sciences Conference, 2008.

[7].Satish G.Kandlikar, Shailesh Joshi, Shurong Tian "Effect of channe tlroughness on heat transfer and fluid flow characteristicsa tlow Rey nold snumberinsmall tube" 35th National Heat Transfer Conference, California, ASME.

[8].Nopparat Katkhaw, Nat Vorayos, Tanongkiat Kiatsiriroat, Yottana Khunatorn, Damorn Bunturat, Atipoang Nuntaphan, "Heat transfer behavior off latplatehaving 45˚ellipsoidal dimpled surfaces" Case Studies in Thermal Engineering 2 (2014) 67–74.

[9].Veeresh Fuskele, R.M. Sarviya, "Experimental investigation of heat transferenhance mentindou b lepipe heat exchanger usingt wiste d meshtape", IJARS/vol.1/issuII/Jan200905-09.

[10].R.Zimmerman, M.Gurevich, A.Mosyak, R.Rozenblit, G.Hetsroni, Heat transfer to air–water annular flow i n a horizontal pipe, (elsevier) International Journal of Multiphase Flow 32 (2006)1–19.

[11]. Dr. A.G.Matani, Swapnil A.Dahake, "Experimental Study On Heat Transfer Enhancement In A Tube Using Counter/Co-Swirl Generation", (IJAIEM)ISSN2319–4847.

[12].C. Nithiyesh Kumar, P. Murugesan, "Reviewon Twisted Tapes Heat Transfer Enhancement, International Journal Of Scientific & Engineering Research", Volume 3,Issue 4, April-2012.

[13].S. Liu, M. Sakr, "A comprehensi verve iew on passive heat transferenhancement si npipe exchangers" ,(Elsevier) Renewable and Sustainable Energy Reviews 19 (2013) 64–81.

[14].Li Zhang, Hongmei Guo, Jianhua Wu, Wenjuan Du "Compound heat transferenhancementforshell sideofdouble-pipeheatexchangerbyhelicalfinsandvortexgenerators"Heat Mass Transfer o9o(2012) 48:1113–1124.