Numerical Analysis and Modification of Engine Exhaust Pipe for Performance Improvement

Sandeep M Chauhan[1], Abhishek R Pandya[2], Kuldeep C Parmar[3], Pruthvesh A Padariya[4], Noel Mecwan[5]

[1,2,3,4] BE, Mechanical Dept. (ITM Universe), Vadodara, India

[5] ME, Assistant Professor, Automobile Engineering (ITM Universe), Vadodara, India

***Abstract* –: Now a days the global warming and air pollution are big issue in the world. The more amount of air pollution is due to emissions from an internal combustion engine. The exhaust pipe is an important part of IC engine. it's structure and performance have direct impact on the engine power ,economy and emissions. Exhaust system plays a vital role in reducing harmful gases, but the presence of after treatment systems increases the exhaust back pressure. The back pressure at the exhaust port thus losing power, increasing fuel consumption and piston effort to exhaust the gases out. The substrate has been modeled as porous medium for analysis purpose. These models have been imported in CFD tool for analysis. After importing the CAD data inside the CFD software, with proper boundary conditions, the CFD analysis has been carried out. Based on the study, individual system contributions to the total pressure drop and flow uniformity have been analyzed and improvement areas of the existing system for better flow uniformity have been suggested. The design is reasonable and can achieve the design goal.**

***Index Terms* – Exhaust Pipe, Computational Fluid Dynamics (CFD), Heat Transfer Analysis, Exhaust system components, Computer Aided Design.**

I. INTRODUCTION

We have done Modeling and simulation of exhaust pipes. There are many different issues the research is focused on studies of the dynamics of the temperature in the Exhaust Pipe and also there is a lot of literature concerning the dynamics of the temperature in the exhaust pipe. However, the mathematical literature on this topic is not very large. The influence of the geometric structure of the exhaust pipe on the heating process is studied and the present work involves CFD study of the exhaust system of Light commercial Vehicle on pressure drop and uniformity index. Based on this study we can find out the pressure drop in the Exhaust Pipe and flow Uniformity. We used AUTO CAD software for design, purpose and system was computationally analyzed the pressure drop and uniformity Index by using ANSYS 15.0 software.

II. LITERATURE REVIEW

Mesut DURAT, Zekeriya PARLAK, Murat KAPSIZ, Adnan PARLAK ve Ferit FICICI [2] The model presented in the study could be used for the determination of optimal location of a catalyst along with the exhaust pipe of any gasoline engine in terms of minimum cold start HC emissions. In addition, the model can be used for any type motor with exhaust system requiring HC emissions.

Atul A. Patil, L.G. Navale, V.S. Patil [4] The Exhaust system is successfully designed. Through CFD analysis, the backpressures of various Exhaust diffuser systems are studied. The increase in inlet cone angle increases the pressure of the flow which leads to reduce the recirculation zones. Installation of the EDS–II increases the brake thermal efficiency and decreases the backpressure.

Om Ariara Guhan C Pa, Arthanareeswaren G a, Varadarajan K N b [3] There is a reverse flow observed in close coupled catcon near to inlet cone enlarging area,The inlet cone should be optimized to avoid this eddy formation. Flow is more uniform in under body catcon .The Uniformity index is in under body catcon. Close coupled catcon geometry to be optimized to obtain more uniformity index.

Jithendra Kumar B, Venugopala Reddy Kussam [6] The present work illustrates meshed model of exhaust system satisfied all quality criteria's hence the results are accurate. Typical road conditions are considered for loading. Static 1g load is preferred for smooth road, whereas road load for rough road which is included with pot holes, bumps etc., Loads and boundary conditions are accurately simulated to obtain the realistic loading conditions.

Pengyun Xu, Haiyong Jiang and Xiaoshun Zhao [1] the velocity index of the main and the pre catalytic converter meets the design requirements.The maximum flow velocity of the pre and main catalytic converter is less which meets the design requirements. The flow velocity of the oxygen sensor is higher and the oxygen sensor place is more reasonable.

M.P.Tambe, Saifali Sanadi, Chaitanya Gongale,Suraj Patil,Surajkumar Nikam [5] Sound attenuation in normal driving. Reduce in back pressure to optimize air outflow when throttle starts to wide open. Maximum utilization of the power generated by engine.Fuel consumption is decreased due to decrease in the losses caused by back pressure.

Julie M Pardiwala, Femina Patel, Sanjay Patel [7] Environmental, ecological and health concern result in increasingly stringent emissions regulations of pollutant emissions from vehicle engines. Among all the types of technologies developed so far, use of catalytic converters is the best way to control auto exhaust emission.

III. OVERVIEW OF EXHAUST SYSTEM.

Exhaust system refers to a group of independent but interrelated automotive components used to direct the waste exhaust gases out of the combustion chamber of an engine. Based on its design the exhaust system comprises several different parts such as cylinder head and exhaust manifold, catalytic converters for air pollution reduction.

3.1. Main Components of exhaust system are

1. Catalytic Converter
2. Exhaust Manifold
3. Mufflers
4. Tail pipe

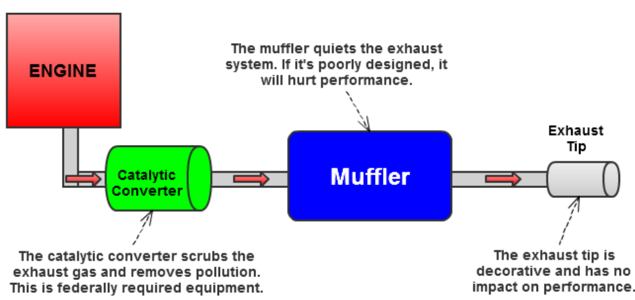

Fig. 1 Typical Diagram of Engine Exhaust System

IV. OVERVIEW OF DESIGN, ANALYSIS AND IMPLEMANTATION STRATEGY

4.1 CAD Software

Computer aided design is the use of computer systems to aid in the creation,modification,analysis or optimization of a design.CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation and to create a data base for manufacturing.CAD software for mechanical design uses vector-based graphics to depict the objects of traditional drafting or may also produce raster graphics showing the overall appearance of designed objects. We use the CAD software in our project for design the exhaust pipe for this we required the diameter (inlet, outlet) of the pipe, length, height, width etc**.**

4.2 CFD Analysis

CFD is a numerical method that can be solved real fluid flow in the presented study, CFD analysis of the exhaust pipe was performed with CFX in ANSYS Workbench V12.1 with three dimensional CFD numerical codes. Solid pipe and fluid volume in the exhaust pipe is specified as computational domain. The ansys fluid flow tool (CFX) generates mesh automatically using CAD model. The algorithm generates the mesh based on the geometry increasing the number of cells in the proximity of wall of the pipe.

- Development of the high speed CFD has a great influence on the engineering design and analysis of IC engine.
- Compare to theoretical approach to solve fluid flow problems, the CFD approach has the advantage that it can provide a solution to more complex problem.

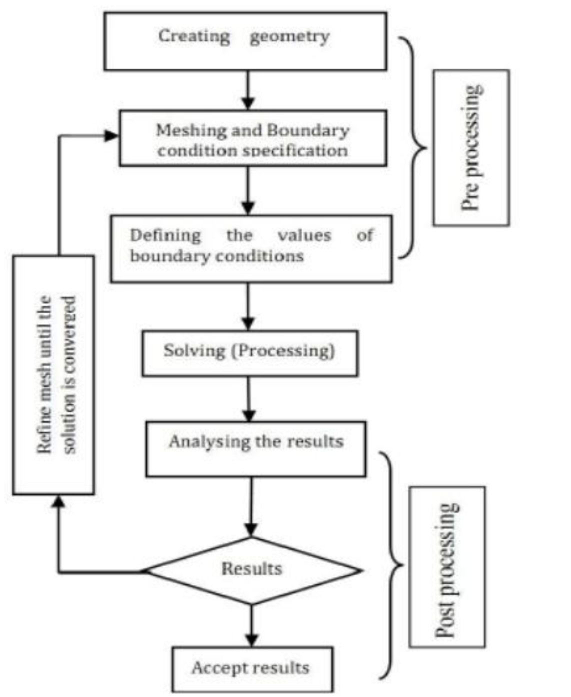

Fig. 2 CFD Analysis Flow Chart

V. STATEMENT OF THE PROBLEM

The emission caused by vehicles is an import issue in these times. In order to reduce the exhaust pollution and improve performance of engine governments are prescribing technical requirements for the production of Vehicles, especially in the exhaust system. The results will be compared for different geometry of pipe, and modification will be suggested for internal combustion engine. Exhaust system durability based on computer simulation rather than physical vehicle testing. This is due to the cost, time and availability of prototype vehicles and test track. Computer aided design is the use of computer systems to aid in the creation, modification, analysis or optimization of a design enables to assure the structural integrity of the exhaust system and also contribute to better understanding of the system behavior in the various operating conditions and evaluation of structural strength.

5.1. Objectives of the Project

The main Objectives of the Projects are:

- To generate a finite element model of the Exhaust system
- To carry out the all necessary checks on the model to simulate the flow in the exhaust pipe of an internal combustion engine.
- To improve performance of an internal combustion engine.
- To modify design of exhaust pipe of an internal combustion engine.

5.2. Scope of the Project

- The simulation of exhaust pipe flow for an internal combustion engine is to be done.
- The different geometries of exhaust pipe will be used for simulation.
- The results will be compared for different geometry of pipe, and modification will be suggested for internal combustion engine.

VI. METHODOLOGY

The analysis has been carried out on two designs an existing one that is EDS (Exhaust Diffuser System) with 0° inlet cone angle and modified one that is EDS with 90° inlet cone angle, results are subsequently compared. It was observed that the brake thermal efficiency improved drastically upon modification in exhaust geometry. Physical models of the same these two systems are subsequently manufactured and exhaustive experiments are carried out on them. The results obtained through CFD analysis are experimentally confirmed. In CFD analysis two major flow characteristics (back pressure and engine performance) are studied.

Study I:

In study I, the change in pressure of structure was studied. This study offered to find the change in pressure difference in inlet and outlet of exhaust diffuser system. The models which produce the higher pressure difference are selected for further studies.

Study II:

In study II, the models which had the higher pressure difference are studied for the flow pattern. The back- pressure characteristics of the models are modeled and the model having the lesser backpressure was taken for experimental study of engine performance.

VII. MATHEMATICAL MODELING

7.1 Exhaust Pipe Design Calculation:

1. Design Calculations:

***A.** Benchmarking:*

The first step in any design is to set a target by doing the benchmarking. The same will be applicable for the silencer here, to set a target in terms of transmission loss.

***B.** Target Frequencies*

After doing benchmarking exercise, there are needs to calculate the target frequencies to give more concentration of higher transmission loss. For calculating the target frequencies engine max power rpm is required and calculation follows,

Theoretical Computation:-

The exhaust tones are calculated using the following Formulae:-

CFR = Engine Speed in RPM/60.. For a two stroke engine (1)

CFR= Engine Speed in RPM/120-For a four-stroke engine (2)

EFR = n X (CFR) (3)

***C.** Muffler Volume Calculation Volume Of the muffler (Vm):-*

Vm= Vf X [/4(d2 X l)] X (No. of Cylinders/2) (4)

STEP 1: Benchmarking

As per design methodology we benchmarked same kind of engine models to set the target of transmission loss of muffler.

Engine data: Hero Honda Splendor

Bore (D) = 50 mm

Stroke (L) = 49.5 mm

No. Cylinders (n) = 1

Engine power (P) = 6.15kw (8.36ps) @ 8000rpm

Max. RPM (N) = 8500 rpm

Allowable back pressure for muffler = Not available (in H_2O) Transmission Loss Noise target

STEP 2: Target Frequencies

(Muffler) = 30 dB.

To find fundamental frequency

Cylinder Firing Rate:-

CFR to be calculated as per the equation –2,

CFR =8000/120 = 66.66

Engine firing rate

EFR to be calculated as per the equation –3, EFR = 1 x 66.66 = 66.66 Hz

STEP 3: Muffler Volume Calculation

Swept volume (Vs): (π x d2 x L)/4 = (3.14x502 x49.5)/4 = 97143.75 x 10^-6 Lit.

Volume to be consider for calculation,

Volume = (n) x Vs/2 = 0.48

Silencer Volume =Factor x Consider Volume = 2.26485Lit

Assumed Factor =4.7083

STEP 4: Internal Configuration of Muffler and Concept Design

Diameter of muffler calculated as:

Vm = (π/4) x d2x L

0.00226x10^-6 = (3.14/4) x D2 x 0.342

D=0.091750 m

D=91.75mm (Diameter of muffler)

STEP 5: Selection of Other Parameters

1. 3 chambers for good cancellation capacity
2. Inlet and Outlet extension: Not selected yet
3. Inlet and Outlet extension to be kept 180̊ reversal.
4. La= Length at which Perforation starts on pipe = Not selected.

Above calculation find in the research paper of "Analysis of Exhaust System-Semi Active Muffler". From using above calculations we can design the exhaust pipe of Hero Honda Splendor+.

7.2. Governing equations:

The description of flow is based on the three basic fundamental physical laws. These laws are: [3]

- Conservation of mass
- Conservation of momentum (Newton's Second Law)
- Conservation of energy

The first law gives continuity equation (Eq. 2.1). This equation represents conservation of mass in control volume for compressible flow

$$\frac{\partial p}{\partial t} + \varphi \frac{\partial U_i}{\partial x_{i \cdot}} = 0$$

In here ρ is the density and u_i is the velocity in direction i. The symbols t and x_i represents time and the position in the direction i respectively. The momentum equation is derived using Newton's second law. This equation is known as the Navier-Stokes equation.

$$\frac{\partial u_i}{\partial t} + uj \cdot \frac{\partial u_i}{\partial x_i} = -\frac{1}{\rho}\frac{\partial^p}{\partial x_i} + v\frac{\partial^2 u_i}{\partial x_j^2}$$

Where p represents the pressure and ϑ the kinematic viscosity. The second term on the left-hand side is the convective term and the second term on the right-hand side is the dilution term. It says that the net force on a fluid element equals the mass times the acceleration of the element. The forces acting on the fluid element can be divided into Body and Surface forces. The last fundamental physical law results in energy equation

$$\rho Cp\frac{\partial T}{\partial t} + \rho_{Cp} T\frac{\partial u_i}{\partial Xj} = \lambda\frac{\partial^2 T}{\partial x_j^2}$$

Where T is the temperature, cp the heat capacity coefficient at constant pressure and the thermal conductivity coefficient.

7.3. Boundary conditions

7.3.1. Loads

Because of frequency-varying nature, it is more complicated to apply dynamic loads than it is to apply static loads. Therefore, it is important to verify that the dynamic loads are correctly specified and that there are no discontinuous loads. The best way to verify proper dynamic load specification is to plot the loads as a function of frequency.

Major loads acting on the exhaust system is due to its self-weight and some engine vibrations also transferred through exhaust system. These loads led to breakage of clamping/hanger locations. Hence different values of gravity force are applied to analyze static and dynamic behavior.

7.3.2. Boundary conditions

The proper specification of boundary conditions is just as important for dynamic analysis as it is for static analysis. The improper specification of the boundary conditions leads to incorrect answers. One such improper specification of boundary conditions is forgetting to fully constrain the structure.

Inlet Boundary Condition

Inlet gas velocity	: 10.24 m/s
Gas temperature	: 633 K
Mass flow rate of gas	: 8.5 kg/hr

Outlet Boundary Condition

Outlet pressure	: Atmospheric pressure

VIII. TURBULENCE MODELING

8.1. Turbulence

All flows occurring in engineering practice both simple ones and complicated ones such as jet, wake, pipe, etc., become unstable above a certain Reynolds number. The flows at low Reynolds number are called laminar flows and the flows occurring at high Reynolds number are called turbulent flows.

In turbulent flow a chaotic and random state of motion develops in which the velocity and pressure change continuously with time. The results of experiments on fluid systems shows that at values below the critical Reynolds number the flow is smooth and adjacent layers of fluid slide past over each other in an orderly fashion. At values of the Reynolds number above critical Reynolds number a complicated series of events take place which eventually leads to a radical change of the flow character. In the final state the flow behavior is random and chaotic.

In an internal combustion engine extreme fluid velocities are involved. Due to these high velocities the Reynolds number is also high which shows that the flow field inside an internal combustion engine is turbulent. Turbulent flows are characterized by fluctuating velocity Fields. Transported quantities such as momentum, energy and species concentration are mixed due to these fluctuations. Presence of turbulence plays an important role in modeling combustion process, increase in turbulence results in better mixing of air and fuel in case of non-premixed combustion.

Modeling of turbulence is a fluid cult task when solving practical flow problems. To solve the governing equations exact to smallest scales more computational e ort is required. Due to which these equations are Reynolds averaged. There are various turbulence models available, but none of them is generally accepted to describe all the processes in turbulent flow.

8.2. Standard $k-\varepsilon$ Model

This is a two equation model and simplest one. It is generally used in simulation of turbulence due to its applicability, robustness and economy. The two transport equations for the kinetic energy and dissipation rate are solved to form a characteristic scale for both turbulent velocity and length. These scales represent the turbulent viscosity. The equations for the kinetic energy (2.4) and dissipation rate (2.5) are given below. [3, 4, 5]

$$\frac{\partial}{\partial t} pk_{+\frac{\partial}{\partial x_i}} pku_i = \frac{\partial}{\partial x_j} \mu_{+\frac{\mu 2}{\sigma_k} \cdot \frac{\partial k}{\partial^x j}} + \text{Gk} + \text{Gb} - \rho z - \text{Ym} + \text{Sk}$$

In these equations, the generation of turbulence kinetic energy because of means velocity Gradients, Gb is the generation of turbulence kinetic energy because of buoyancy; represents the contribution of the fluctuating dilation is represented by Ym in compressible turbulence to the overall dissipation rate;C1T , C2T and C3T are constants; αk and αt are turbulent Prandtl For k and E", respectively. Sk and ST are user defined source terms. The turbulent (or eddy) Viscosity, αt, is computed by combining k and E" as follows:

$$\mu_t = \rho C\mu \frac{k^2}{\varepsilon}$$

8.3. Standard k-w Model

It includes modifications due to effects of low Reynolds number, compressibility and shear flow spreading. It is a two equation semi-empirical turbulence model. The k □! Model is comparable with the equation of transport for the kinetic energy. The equation of dissipation rate is different. In k-w" model per unit mass dissipation is used while in k□! Model specific dissipation is used. It can be seen as ratio of "to k. The transport equations for kinetic energy and specific dissipation are [3, 4]

$$\frac{\partial}{\partial t}(pk) + \frac{\partial}{\partial X_i}(pk\varepsilon u_i) = \frac{\partial}{\partial x_j}[(u + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial^x j}] + G_k - y_k$$

Model	Constants				
Standard $k-\varepsilon$	$C_{1\varepsilon} = 1.44$	$C_{2\varepsilon} = 1.92$	$C_\mu = 0.09$	$\sigma_k = 1.0$	$\sigma_\varepsilon = 1.3$
RNG $k-\varepsilon$	$C_{1\varepsilon} = 1.42$	$C_{2\varepsilon} = 1.68$	$C_\mu =$ 0.0845	$\alpha_0 = 1.0$	$\eta_0 = 4.38$
	$\beta = 0.012$				
Standard $k-\omega$	$\alpha^* = 1.0$ ∞	$\alpha_\infty = 0.52$	$\alpha_0 = 1/9$	$\beta^* = 0.09$ ∞	$\beta_i = 0.072$
	$R_\beta = 8$	$R_k = 6$	$R_\omega = 2.95$	$\zeta^* = 1.5$	$M_{t0} = 0.25$
	$\sigma_k = 2.0$	$\sigma_\omega = 2.0$			

Where Gk is the generation of kinetic energy and G! is the generation of specific dissipation rate. Yk and Y! Shows dissipation of k and !. The turbulent viscosity _t is obtained by

$$\mu_t = \alpha\, \rho \frac{k}{\omega}$$

Where α is low Reynolds correction factor. The k-w Model offers good results for wall Bounded flows and free shear flows so this model is also very suitable for internal combustion Engine simulations. Table shows values of different constants used in turbulence models.

IX. SIMULATION OF EXHAUST PIPE USING SOFTWARE

Fluid flow problems can be solved using CFD codes which are structured on numerical algorithm. User interfaces are included in CFD packages to give input parameters of problem And to analyze results. CFD codes have main three components:

(i) Pre- processor,

(ii) Solver

(iii) Post- processor.

General program structure of GAMBIT and FLUENT packages.

- Modeling the exhaust pipe using GAMBIT

- Analyzing the exhaust pipe in FLUENT

9.1 Pre- processor

Pre-processing consists of giving input to CFD computer program by user interface and transform this input into a form suitable for use by the solver. The pre- processor involves following activities: [10]

- Definition of the geometry of the region of interest: the computational domain
- Grid generation: the sub division of the domain into a number of small sub domains
- Selection of the physical and chemical phenomenon that need to be modeled
- Definition of fluid properties Specification of appropriate boundary conditions

The solution of a flow problem (velocity, pressure, temperature etc.) is defined at nodes inside every cell. The accuracy of result obtained by CFD program depends on the number of cells. In general the larger the number of cells, the better the solution accuracy. Both the accuracy of a results and its cost in terms of necessary computer hardware and calculation time are dependent on the health of the grid. Optimal meshes are often non- uniform: finer in areas where large variations occur from point to point and coarser in regions with little change.

9.2. Solver

There are three distinct types of numerical solution techniques: finite difference, finite element And finite volume methods. Most CFD packages concerned central to the finite volume method. Numerical algorithm consists:

- Integration of the governing equations involved by fluid flow of all control volume of Domain
- Disctretisation- conversation of the obtained integral equations onto a system of algebraic equations
- Obtaining the solution of algebraic equations by iterative method

The resulting statements express the exact conservation of relevant properties for each finite size cell. This clear relationship between the numerical algorithm and the underlying physical conservation principle forms one of the main attractions of the finite volume method and makes its concept much simple to understand by engineers than the finite element and finite difference methods. The conservation of a general flow variable ', e.g. a velocity component or enthalpy, within a finite control volume can be expressed as a balance between the various processes tending to increase or decrease it.

9.3. Post- processor

As in pre- processing, a huge amount of development work has recently taken place in post- Processing field. Due to increased popularity of workstations, many of which have outstanding graphics capabilities, the leading CFD packages are now equipped with versatile data Visualization tools like:

- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- Particle tracking
- View manipulation (translation, rotation, scaling etc.)
- Colour PostScript output

More recently these facilities include animation for dynamic result display, and in addition to graphics all codes produce trusty alphanumeric output and have a data export facilities for further manipulation external to the codes.

X. CONCLUSION

The present work illustrates meshed model of exhaust system satisfied all quality criteria's hence the results are accurate. Typical road conditions are considered for loading. Static 1g load is preferred for smooth road, whereas road load for rough road which is included with pot holes, bumps etc., Loads and boundary conditions are accurately simulated to obtain the realistic loading conditions.

The results were plotted and analysed in inlet pipe, Close coupled catcon inlet , under body catcon and muffler inlets. Pressure contours were studied in porous inlet and outlet of close coupled and under body catcon as well. Flow Distribution at various locations flow distribution analysed with the help of streamline plots. We observed that there was a reverse flow near to cone area, because of formation of eddies and leads to the loss of head due to sudden enlargement and contraction of the exhaust system is explaining the streamline plot of close coupled catcon and is explaining about under body catcon. Flow distribution at across close coupled. Flow distribution at across under body the results.

From the structural design point of view the structure is considered safe as the stress levels are well below the ultimate stress. It was verified that numerical models validated with are a powerful tool during the development phases of vehicle, reducing project time and costs.

XI. SCOPE FOR FUTURE WORK

- Similar analysis approach can be followed for any type of Exhaust system, in general any hanging component.
- Dynamic analysis can be carried out for the Impact loading and random changes of the load on Exhaust system due to uneven road.
- Fatigue life estimation of each component can be performed for Exhaust system.
- Thermal analysis can be performed to know the thermal stress on individual components of Exhaust system.

- Acoustic analysis can be conducted for the muffler region to check the noise.

REFERENCES

[1]. "CFD Analysis of a Gasoline Engine Exhaust Pipe" Pengyun Xu*, Haiyong Jiang and Xiaoshun Zhao Mechanical and Electronical Engineering College, Agriculture University of Hebei, Baoding 071001, P.R. China.

[2]. "CFD AND EXPERIMENTAL ANALYSIS ON THERMAL PERFORMANCE OF EXHAUST SYSTEM OF A SPARK IGNITION ENGINE" Mesut DURAT, Zekeriya PARLAK, Murat KAPSIZ, Adnan PARLAK, ve Ferit FIÇICI Sakarya University, Faculty of Engineering, Department of Mechanical Engineering ,Istanbul.

[3]. "CFD Study on Pressure Drop and Uniformity Index of Three Cylinder LCV Exhaust System" Om Ariara Guhan C Pa, Arthanareeswaren G a, Varadarajan K N b Department of Chemical Engineering, National Institute of Technology, Tiruchirappalli -620015, India. B-Tech Mahindra Ltd., Electronic city, Bangalore-560100 ,India.

[4]. "Design, Analysis of Flow Characteristics of Exhaust System and Effect of Back Pressure on Engine Performance" Atul A. Patil, L.G. Navale, V.S. Patil Mechanical Department North Maharashtra University, Umavi Nagar, Jalgaon, Maharashtra, INDIA.

[5]. "Analysis of Exhaust System- 'Semi Active Muffler" M.P.Tambe, Saifali Sanadi, Chaitanya Gongale,Suraj Patil,Surajkumar Nikam Asst. Professor, Dept. of Mechanical Engineering, JSCOE, Hadapsar, Pune, India UG Student, Dept. of Mechanical Engineering, JSCOE, Hadapsar, Pune, India

[6]. "Structural Analysis of Passenger Car Exhaust System by Using Hypermesh" Jithendra Kumar B, Venugopala Reddy Kussam. Research Scholar, Nova College of Engineering and Technology, Jangareddy Gudem,WestGodava.

[7]. "Review paper on Catalytic Converter for Automotive Exhaust Emission" Julie M Pardiwala, Femina Patel, Sanjay Patel INTERNATIONAL CONFERENCE ON CURRENT TRENDS IN TECHNOLOGY, „NUiCONE – 2011‟INSTITUTE OF TECHNOLOGY, NIRMA UNIVERSITY, AHMEDABAD – 382 481, 08-10 DECEMBER, 2011