# ANALYSIS OF EXHAUST MANIFOLD OF MULTI-CYLINDER SI ENGINE USING ANSYS

G. Sree Bhavani Charan[1] | Dr G. Nagamalleswara Rao[2]
[1](BTech Student, Department of Mechanical Engineering, G.PullaReddy Engineering College, Kurnool, India)
[2](Department of Mechanical engineering, Eswar College of Engineering, Narsaraopet, India)

***Abstract—*** *In present century, spark ignition engines have become a non-separable part of the society, and are used in many sectors of energy. They act as backbone for transportation systems, but, as a bitter truth they behave like a major source of air pollution. There are basically three types of emissions, emerged from an SI engine; exhaust emissions, evaporative emission, and crankcase emission, and the major pollutants emerged from these engines are CO, CO2, SOX, NOX. Present work aims at reducing emissions. It is a well-established fact that smooth combustion minimizes the emissions, and exhaust process contributes a lot in accomplishing smooth combustion process. In present work, different designs of exhaust manifold for a multi cylinder spark ignition engine are optimized for reducing emissions, by evaluating back pressures. For this purpose four different designs, namely, short bend center exit, short bend side exit, long bend center exit with reducer, and long bend side exit with reducer are considered, and their performance is evaluated for different loading conditions. As a result performance scores of different models based on back pressure are evaluated, and on the basis of these scores, overall performance score is investigated. In next step, on the basis of overall performance score, ranking of different models is carried out. The results show the suitability of short bend center exit model for the purpose, as it scores better rank in the analysis. The analysis is carried out on virtual models of manifolds. Models of manifolds are developed on CATIA v5 modeling software, and for the purpose of analysis ANSYS Workbench is used.*

***Keywords— Exhaust Manifold; Back Pressures; Exhaust Velocities; CATIA v5; ANSYS Workbench***

## 1. INTRODUCTION

In automotive engineering, an exhaust manifold collects the exhaust gases from multiple cylinders into one pipe. Exhaust manifolds are generally simple cast iron or stainless steel units which collect engine exhaust gas from multiple cylinders and deliver it to the exhaust pipe. In a normal engine, once the exhaust gases exit the cylinder they end up in the exhaust manifold. In a four-cylinder or eight-cylinder engine, there are four cylinders using the same manifold. From the manifold, the exhaust gases flow into one pipe toward the catalytic converter and the muffler.

Fig: 1. Four cylinder engine exhaust manifold

The most common types of aftermarket headers are made of mild steel or stainless steel tubing for the primary tubes along with flat flanges and possibly a larger diameter collector made of a similar material as the primaries. They may be coated with a ceramic-type finish (sometimes both inside and outside), or painted with a heat-resistant finish, or bare. Another form of modification used is to insulate a standard or aftermarket manifold. This decreases the amount of heat given off into the engine bay, therefore reducing the intake manifold temperature.

### *1.1 Back Pressure*

Engine exhaust back pressure is defined as the exhaust gas pressure that is produced by the engine to overcome the hydraulic resistance of the exhaust system in order to discharge the gases into the atmosphere.

It should be noted that the term “back pressure” is counter-intuitive and may interfere with a proper understanding of the exhaust gas flow mechanics. The word back seems to suggest a pressure that is exerted on a fluid against its direction of flow.

### *1.2 Effects of Increased Back Pressure*

Increased exhaust pressure can have a number of effects on the diesel engine, as follows:

- Increased pumping work
- Reduced intake manifold boost pressure
- Cylinder scavenging and combustion effects
- Turbocharger problems

At increased back pressure levels, the engine has to compress the exhaust gases to a higher pressure which involves additional mechanical work and/or less energy extracted by the exhaust turbine which can affect intake manifold boost pressure [1]. This can lead to an increase in fuel consumption, PM and CO emissions and exhaust temperature. The increased exhaust temperature can result in overheating of exhaust valves and the turbine. An increase in NOx emissions is also possible due to the increase of engine load [2].

Other effects on diesel combustion are possible, but depend on the type of engine. Increased back pressure may affect the performance of the turbocharger, causing changes in the air-to-fuel ratio—usually enrichment—which may be a source of emissions and engine performance problems. The magnitude of the effect depends on the type of the charge air systems. Increased exhaust pressure may also prevent some Exhaust gases from leaving the cylinder (especially in naturally aspirated engines), creating an internal exhaust gas recirculation (EGR) responsible for some NOx reduction. Slight NOx reductions reported with some DPF systems, usually limited to 2-3% percent, are possibly explained by this effect [3].

It is generally accepted by automotive engineers that for every inch of Hg of back pressure (that's Mercury - inches of Hg is a unit for measuring pressure) approximately 1-2 HP is lost depending on the displacement and efficiency of the engine, the combustion chamber design etc.

## 2. OBJECTIVES OF THIS WORK

i. This work focuses upon study of pressure distribution inside an exhaust manifold of different geometries and to conclude best possible geometry from emissions point of view.
ii. It is flaunted with symmetric and asymmetric designs and have flocked concepts of having either long or short bends (inlet for exhaust manifold).
iii. These calculations were carried out at loading condition i.e. 2kg.

## 3. DESIGNING, MODELING AND ANALYSIS OF EXHAUST MANIFOLD

Large numbers of design and analysis software are available in the market for designing and analysis of parts such as PTC creo, Solid works, CATIA, ANSYS, Hypermesh, and Inventor and out which CATIA and ANSYS were chosen for design and analysis of exhaust manifold

### *3.1 Introduction to CATIA*

CATIA started as an in-house development in 1977 by French aircraft manufacturer Avions Marcel Dassault, at that time customer of the CADAM software to develop Dassault's Mirage fighter jet. It was later adopted in the aerospace, automotive, shipbuilding, and other industries.

Computer Aided Three dimensional Interactive Application (CATIA) is well known software for 3-D designing and modeling for complex shapes. Commonly referred to as 3D Product Lifecycle Management software suite, CATIA supports multiple stages of product development (CAX), including conceptualization, design (CAD), engineering (CAE) and manufacturing (CAM). CATIA facilitates collaborative engineering across disciplines around its 3DEXPERIENCE platform, including surfacing & shape design, electrical, fluid and electronic systems design, mechanical engineering and systems engineering [4].

CATIA facilitates the design of electronic, electrical, and distributed systems such as fluid and HVAC systems, all the way to the production of documentation for manufacturing.

### *3.2 Meshing and analysis*

#### *3.2.1 Introduction to ANSYS*

The ANSYS program allows engineers to construct computer models or transfer CAD models of structures, products, components, or systems, apply loads or other design performance conditions and study physical responses such as stress levels, temperature distribution or the impact of lector magnetic fields [5].

In some environments, prototype testing is undesirable or impossible. The ANSYS program has been used in several cases of this type including biomechanical applications such as high replacement intraocular lenses. Other representative applications range from heavy equipment components, to an integrated circuit chip, to the bit-holding system of a continuous coal-mining machine.

ANSYS design optimization enables the engineers to reduce the number of costly prototypes, tailor rigidity and flexibility to meet objectives and find the proper balancing geometric modifications.

Competitive companies look for ways to produce the highest quality product at the lowest cost. ANSYS (FEA) can help significantly by reducing the design and manufacturing costs and by giving engineers added confidence in the products they design. FEA is most effective when used at the conceptual design stage. It is also useful when used later in manufacturing process to verify the final design before prototyping.

#### *3.2.2 Procedure for ANSYS Analysis*

Static analysis is used to determine the displacements, stresses, strains and forces in structures or components due to loads that do not induce significant inertia and damping effects. Steady loading in response conditions are assumed. The kinds of loading that can be applied in a static analysis include externally applied forces and pressures, steady state inertial forces such as gravity or rotational velocity imposed (non-zero) displacements, temperatures (for thermal strain). A static analysis can be either linear or nonlinear. In our present work we consider linear static analysis.

The procedure for static analysis consists of these main steps:

i. Building the model.
ii. Obtaining the solution.
iii. Reviewing the results.

#### *3.2.3 Build The Model*

In this step we specify the job name and analysis title use PREP7 to define the element types, element real constants, material properties and model geometry element types both linear and non-linear structural elements are allowed. The ANSYS element library contains over 80 different element types. A unique number and prefix identify each element type.

E.g. BEAM 94, PLANE 71, SOLID 96 and PIPE 16

#### *3.2.4 Material Properties*

Young's modulus (EX) must be defined for a static analysis. It is to apply inertia loads (such as gravity), material properties such as density (DENS). Similarly to apply thermal loads (temperatures) define coefficient of thermal expansion (ALPX).

*3.2.5 Obtain the Solution*

In this step we define the analysis type and options, apply loads and initiate the finite element solution. This involves three phases:

- Pre – processor phase
- Solution phase
- Post-processor phase

The following table-1 shows the brief description of steps followed in each phase:

TABLE1. STEPS INVOLVED IN ANALYSIS

| PRE-PROCESSOR PHASE | SOLUTION PHASE | POST-PROCESSOR PHASE |
|---|---|---|
| Geometry definitions | Element matrix formulation | Post solution operations |
| Mesh generation | Overall matrix triangularization | Post data print outs(for reports) |
| Material Definitions | (wave front) | Post data |
| | | Scanning post data displays |
| Constraint | Displacement. | |
| Definitions | Stress, etc. | |
| Load definition | Calculation | |

*3.2.6 Introduction to Fluid Flow (FLUENT)*

ANSYS Fluent fluid flow systems in ANSYS Workbench to set up and solve a three-dimensional turbulent fluid-flow and heat-transfer problem in a mixing elbow. It is designed to introduce you to the ANSYS Workbench tool set using a simple geometry. Guided by the steps that follow, you will create the elbow geometry and the corresponding computational mesh using the geometry and meshing tools within ANSYS Workbench. You will use ANSYS Fluent to set up and solve the CFD problem, then visualize the results in both ANSYS Fluent and in the CFD-Post post processing tool. Some capabilities of ANSYS Workbench (for example, duplicating fluid flow systems, connecting systems, and comparing multiple data sets) are also examined in this tutorial.

*3.2.7 Steps involved in the ANSYS fluid flow*

- Launch ANSYS Workbench.
- Create a Fluent fluid flow analysis system in ANSYS Workbench.
- Import the geometry to ANSYS.
- Create the computational mesh for the geometry using ANSYS Meshing.
- Set up the CFD simulation in ANSYS Fluent, which includes:

– Setting material properties and boundary conditions for a turbulent forced-convection problem.
– Initiating the calculation with residual plotting.
– Calculating a solution using the pressure-based solver.
– Examining the flow and temperature fields using ANSYS Fluent and CFD-Post.

- Create a copy of the original fluent fluid flow analysis system in ANSYS Workbench.
- Change the geometry in ANSYS Design Modeller, using the duplicated system.
- Regenerate the computational mesh.
- Recalculate a solution in ANSYS Fluent.
- Compare the results of the two calculations in CFD-Post.

*3.2.8 Material Fluid Properties*

Exhaust gas will be considered as an incompressible fluid operating at 230- 280°C. The material properties under these conditions are:

TABLE 2: MATERIAL FLUID PROPERTIES

| Material | Air + Gasoline |
|---|---|
| Density (kg/m3) | 1.0685 |
| Viscosity (Pa-s) | 3.0927 x 10-5 |
| Specific heat (J/kg-K) | 1056.6434 |
| Thermal conductivity | 0.0250 |

*3.2.9 Boundary Conditions*

The inlet mass flow rates for different models at six different loading conditions are given below using these mass flow rates the pressure and velocity contours were obtained.

TABLE 3: INLET MASS FLOW RATE

| Load | Inlet 1 | Inlet 2 | Inlet 3 | Inlet 4 |
|---|---|---|---|---|
| 2 KG | 0.000424 Kg/s | 0.000424 Kg/s | 0.000424 Kg/s | 0.000424 Kg/s |

TABLE 4: INLET MEAN HYDRAULIC DIAMETER

| Boundary | Mean Hydraulic Diameter |
|---|---|
| **INLET 1** | 1 0.00877m |
| **INLET 2** | 2 0.00877m |
| **INLET 3** | 3 0.00877m |
| **INLET 4** | 4 0.00877m |

Outlet pressure was taken as 0 atm (Gauge) for all models. The mean hydraulic diameters for outlets of different models are shown below:

TABLE 5: OUTLET MEAN HYDRAULIC DIAMETER

| Model | Mean Hydraulic Diameter |
|---|---|
| Short Bend Center Exit (SBCE) | 0.01302m |
| Short Bend Side Exit (SBSE) | 0.01302m |
| Long Bend Center Exit (LBCE) | 0.01302m |
| Long Bend Side Exit (LBSE) | 0.01302m |
| Short Bend Center Exit with Reducer (SBCER) | 0.0095m |
| Short Bend Side Exit with Reducer (SBSER) | 0.0095m |
| Long Bend Center Exit with Reducer (LBCER) | 0.0095m |
| Long Bend Side Exit With Reducer (LBSER) | 0.0095m |

*3.2.10 Engine Specifications*

Following engine parameters are considered for calculation of mass flow rate at different loading conditions. The flow through exhaust manifold was considered density Based.

TABLE 6: ENGINE SPECIFICATION

| Engine | 4 Stroke 4 Cylinder SI Engine |
|---|---|
| Make | Maruti-Suzuki Wagon-R |
| Calorific Value of Fuel (Gasoline) | 45208 KJ/Kg-K |
| Specific Gravity of Fuel | 0.7 gm/cc |
| Bore and Stroke | 69.05 mm X 73.40 mm |
| Swept Volume | 1100 cc |
| Compression Ratio | 7.2 :1 |

## 4. RESULTS

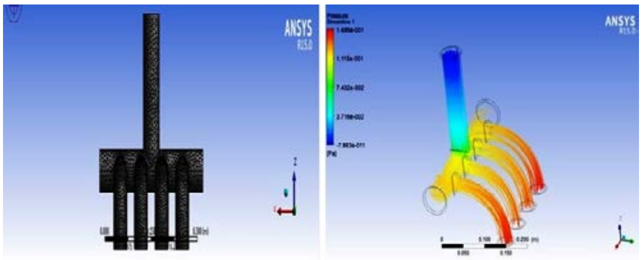

**Fig 2**: Long bend center exit

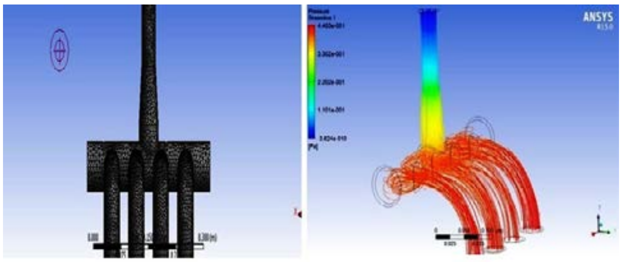

**Fig 3:** Long bend center exit with reducer

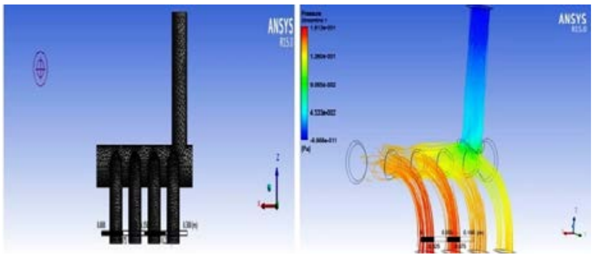

**Fig 4:** Long Bend Side Exit

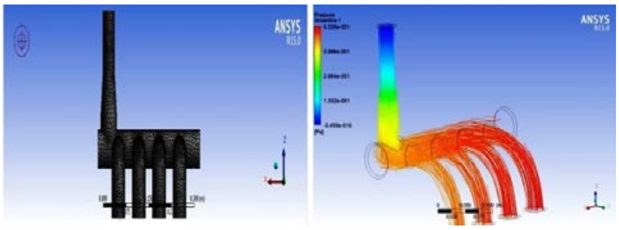

**Fig 5:** Long Bend Side Exit with Reducer

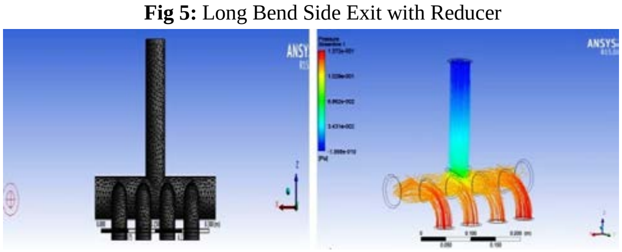

**Fig 6:** Short bend center exit

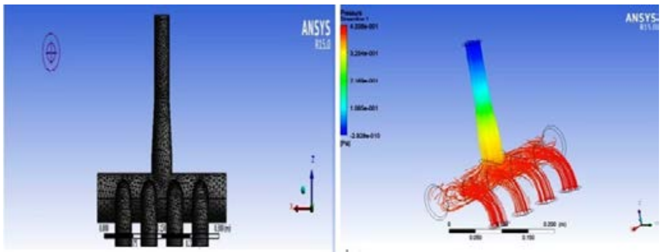

**Fig 7:** Short Bend Centre Exit with Reducer

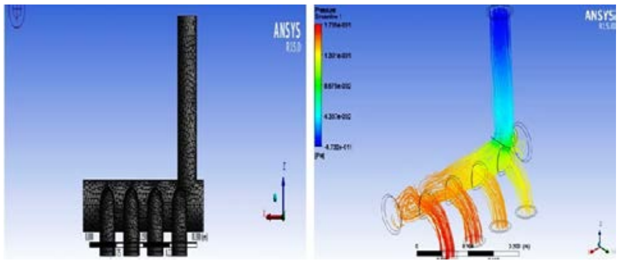

**Fig 8:** Short Bend Side Exit

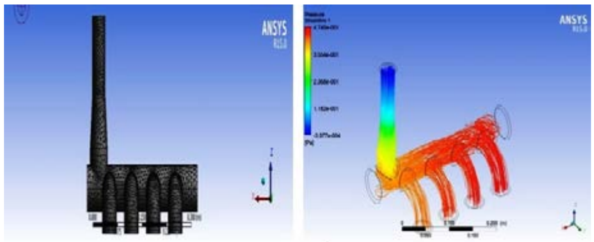

**Fig 9:** Short Bend Side Exit with Reducer

The back pressure for all the models at all loading conditions are listed as under table 7.

TABLE 7: BACKPRESSURE FOR DIFFERENT MODELS IN PASCAL

| | 2KG | 4KG | 6KG | 8KG | 10KG | 12KG |
|---|---|---|---|---|---|---|
| LBCE | 850 | 863 | 894 | 923 | 984 | 1012 |
| LBCER | 1180 | 1214 | 1222 | 1222 | 1272 | 1303 |
| LBSE | 973 | 1005 | 1039 | 1076 | 1099 | 1125 |
| LBSER | 1138 | 1174 | 1219 | 1219 | 1276 | 1271 |
| SBCE | 940 | 976 | 1002 | 1036 | 1079 | 1111 |
| SBCER | 984 | 1012 | 1047 | 1077 | 1114 | 1154 |
| SBSE | 1020 | 1071 | 1098 | 1113 | 1132 | 1172 |
| SBSER | 1037 | 1080 | 1112 | 1112 | 1187 | 1201 |

## 5. CONCLUSIONS

Present work is devoted to the evaluation of different models of exhaust manifold for the purpose of reducing exhaust emissions from a four cylinder SI engine. For this purpose, a set of eight alternatives are chosen, and modelled with the help of CATIA V5 modeling software. In next stage, CFD of different models are carried out on the basis of k-ε model, which finally yield the values of back pressures, and exhaust velocities at different loading conditions. After that performance score is calculated for both the parameters, and as the last step of this work overall performance score for different types was calculated. The following are the conclusions may be drawn during different conduction of CFD, and ranking procedures in the work:

i. Forces exerted by gas particles in the manifold effect the values of back pressure, due to which overall performance score on the basis of these two parameters changes.
ii. Due to increased length, differences in overall performance score in long bend models are greater than that of short bend models.
iii. Out of available set of alternatives, long bend centre exit (LBCE) model of manifold may be the best one because it has scored rank first for overall performance score.

## REFERENCES

[1] K.S. Umesh, V.K. Pravin and K. Rajagopal "CFD Analysis and Experimental Verification of Effect of Manifold Geometry on Volumetric efficiency and Back Pressure for Multi-cylinder SI Engine" International Journal of Engineering & Science Research IJESR/July 2013/ Vol-3/Issue-7/342.

[2] Kulal et al.(2013) "Experimental Analysis of Optimal Geometry for Exhaust Manifold of Multi-cylinder SI Engine for Optimum Performance" Global journal of researches in engineering A: Classification (FOR) 091304,091399, Vol.10 Issue 5 (Ver1.0) October 2010.

[3] Benny Paul, V. Ganesan, "Flow field Development in a Direct Injection Diesel Engine with Different Manifolds" International Journal of Engineering, Science and Technology, Vol-2, No.1, 2010;

[4] Gopaal, M.M.M Kumara Varma, L. Suresh Kumar, "Exhaust Manifold Design – FEA Approach" International Journal of Engineering Trends and Technology, Volume-17, Number 10, November 2014;

[5] MohdSajid Ahmed, Kailash B.A.,Gowreesh,"Design and Analysis of a Multi-Cylinder Four Stroke SI Engine Exhaust Manifold Using CFD Technique" International Research Journal of Engineering and Technology, volume-2, Issue-9, December 2015;