International Journal of Research in Engineering and Science (IJRES)
ISSN (Online): 2320-9364, ISSN (Print): 2320-9356
www.ijres.org Volume 11 Issue 6 ǁ June 2023 ǁ PP. 552-560

Design and Analysis of Engine Inlet Manifold

Mr. MD. Azeem[1], B. Vamshi[2], D. Vamshi[3], T. Ranjeeth
*[*1]Department of Mechanical Engineering, Guru Nanak Institute of Technology, Hyderabad, Telangana*
[2]Department of Mechanical Engineering, Guru Nanak Institute of Technology, Hyderabad, Telangana
[3]Department of Mechanical Engineering, Guru Nanak Institute of Technology, Hyderabad, Telangana
[4]Department of Mechanical Engineering, Guru Nanak Institute of Technology, Hyderabad, Telangana

Abstract
Present project work aims at reducing emissions. It is a well-established fact that smooth combustion minimizes the emissions, and inlet process contributes a lot in accomplishing smooth combustion process. In present project work, different designs of inlet manifold for a multi cylinder spark ignition engine are optimized for reducing emissions, by evaluating back pressures and inlet velocities. For this purpose, four different designs, namely, short bend center exit, short bend side exit, long bend centre 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 and inlet velocity 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 centre 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 modelling software, and for the purpose of analysis ANSYS Workbench is used.

Keywords: *Combustion, Optimised, Multi-Cylinder, CATIA, ANSYS Workbench, Manifold.*

Date of Submission: 13-06-2023 Date of acceptance: 28-06-2023

I. INTRODUCTION

An internal combustion engine (ICE) is a heat engine where the combustion of a fuel occurs with an oxidizer (usually air) in a combustion chamber that is an integral part of the working fluid flow circuit. In an internal combustion engine the expansion of the high- temperature and high-pressure gases produced by combustion apply direct force to some component of the engine. The force is applied typically to pistons, turbine blades, or a nozzle. This force moves the component over a distance, transforming chemical energy into useful mechanical energy. The first commercially successful internal combustion engine was created by Etienne Lenoir around 1859 and the first modern internal combustion engine was created in 1864 by Siegfried Marcus. The term internal combustion engine usually refers to an engine in which combustion is intermittent, such as the more familiar stroke and two-stroke piston engines, along with variants, such as the six-stroke piston engine and the Wankel rotary engine. A second class of internal combustion engines use continuous combustion: gas turbines, jet engines and most rocket engines, each of which are internal combustion engines on the same principle as previously described. Firearms are also a form of internal combustion engine. Typically an ICE is fed with fossil fuels like natural gas or petroleum products such as gasoline, diesel fuel or fuel oil. There's a growing usage of renewable fuels like biodiesel for compression ignition engines and bi-ethanol for spark ignition engines. Hydrogen is sometimes used, and can be made from either fossil fuels or renewable energy

II. LITERATURE REVIEW

PL. S. Muthaiah [1] He has analyzed the inlet manifold in order to reduce the backpressure and also to increase the particulate matter filtration. He has modified the different inlet manifold by varying the size of the conical area of the inlet manifold and varying the size of the grid wire mesh packed throughout the inlet manifold. When size of the grid mesh packed decreased the backpressure increases which leads to lower the performance of the engine due to more fuel consumption and hence low volumetric efficiency. When size of the grid mesh packed increased the backpressure decreases the filtration of the particulate matter also reduces which will not satisfy the standards of the pollution control. Computational fluid dynamics is used for the study of the inlet manifold and best possible design of the inlet manifold with minimum backpressure and maximum particulate matter filtration efficiency is suggested.

K.S. Umesh, V.K. Pravin and K. Rajagopal [2] In this work eight different models of inlet manifold were designed and analyzed to improve the fuel efficiency by lowering the backpressure and also by changing the position of the outlet of the inlet manifold and varying the bend length. The eight different modified models are short bend center exit (SBCE), short bend side exit (SBSE), long bend center exit (LBCE), long bend side exit (LBSE), short bend center exit with reducer (SBCER), short bend side exit with reducer (SBSER), long bend center exit with reducer (LBCER), long bend side exit with reducer (LBSER). After analysis they included that the inlet manifold with long bend center exit with reducer (LBCER), gives the highest overall performance.

Kulal etc. (2013) [3] wo rk comprehensively analyzes eight different models of inlet manifold and concluded the best possible design for least fuel consumption. CFD is the current trend on automotive field in reducing the cost effect for analysis of various models on the basis of fluid flow. A multi-cylinder Maruti - Suzuki Wagon-R engine with maximum speed of 1500 rpm is taken for the analysis. The load and performance test is conducted. From the experiment back pressure and inlet temperatures are measured. The mass flow rate and velocities are calculated. Flow through the inlet manifold is analyzed using commercially available software with mass flow rate and pressure as boundary conditions.

Vivekananda Navadagi and Siddaveer [4] Sangamad they analyzed the flow of inlet gas from two different modified inlet manifold with the help of Computational fluid dynamics. To achieve the optimal geometry for the low back pressure they have analyzed two different inlet manifold, base geometry inlet manifold and the modified geometry inlet manifold. In the base model of the inlet manifold the outlet is at side of the first inlet where as in the modified model of the inlet manifold the outlet is at the center of the inlet manifold. Analysis has been done for the two different inlet manifolds. The results were compared for the two models and it is found that the modified model gives low back pressure in comparison with other base model which ensures the improvement in the efficiency of the engine.

III. PARTS OF IC ENGINE

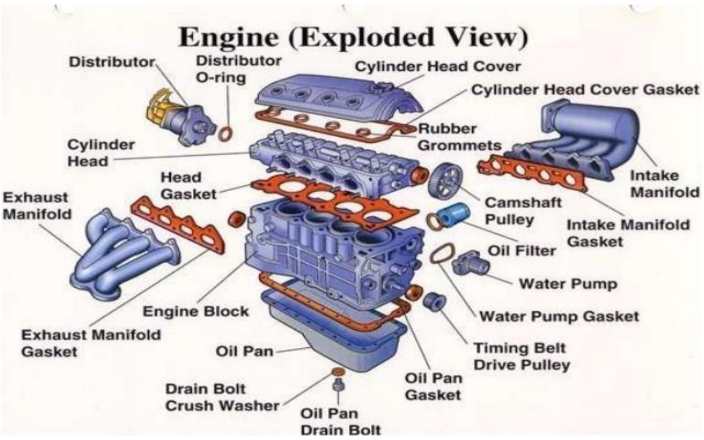


Figure 3.1: Engine (Exploded View)

IV. DESIGNING AND ANALYSIS OF INLET MANIFOLD

Large numbers of design and analysis software are available in the market for designing and analysis of parts. In that we had chosen CATIA

4.1 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. 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.

4.2 Design:

CATIA offers a solution to shape design, styling, surfacing workflow and visualization to create, modify, and validate complex innovative shapes from industrial design to Class-A surfacing with the ICEM surfacing technologies. CATIA supports multiple stages of product design whether started from scratch or from 2D sketches.

4.3. Designed Models of Inlet Manifold

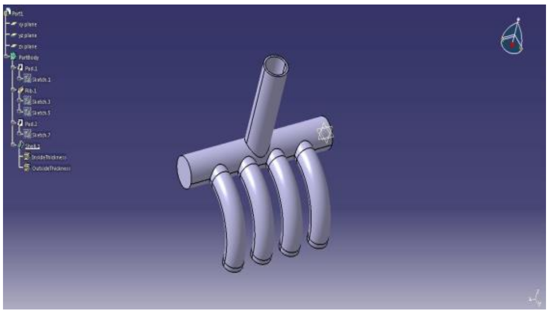


Figure 4.1: Long Bend Center Exit

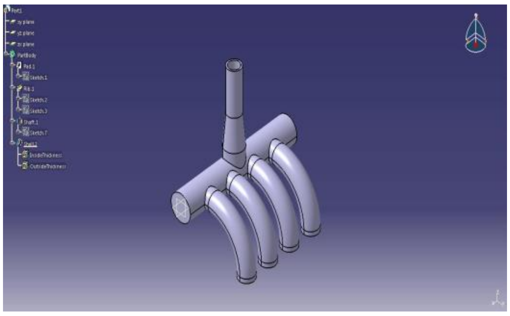


Figure 4.2: Long Bend Center Exit with Reducer

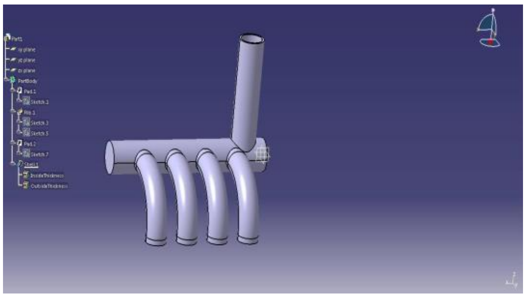


Figure 4.3: Long Bend Side Exit

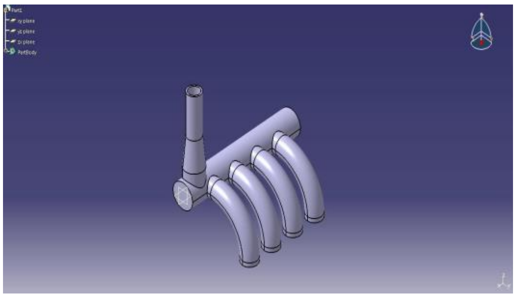


Figure 4.4: Long Bend Side Exit with Reducer

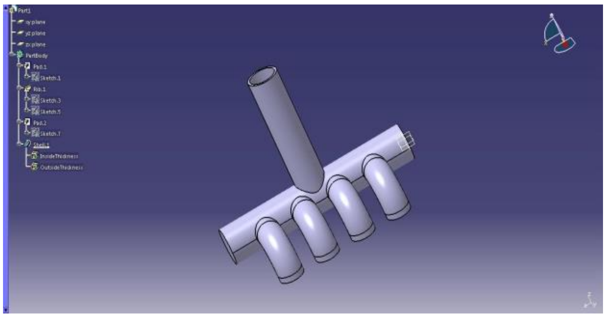


Figure 4.5: Short Bend Center Exit

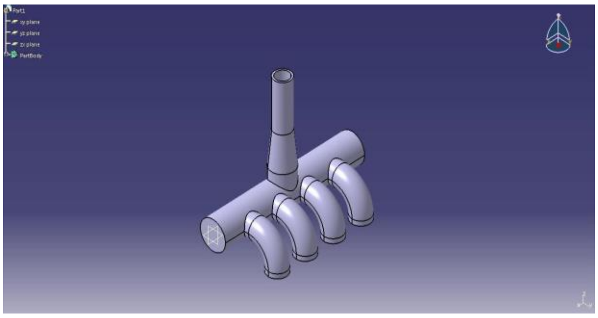


Figure 4.6: Short Bend Center Exit with Reducer

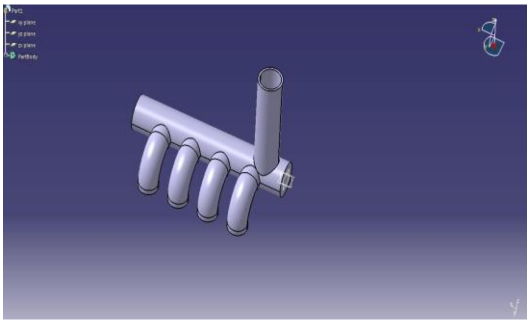


Figure 4.7: Short Bend SideExit

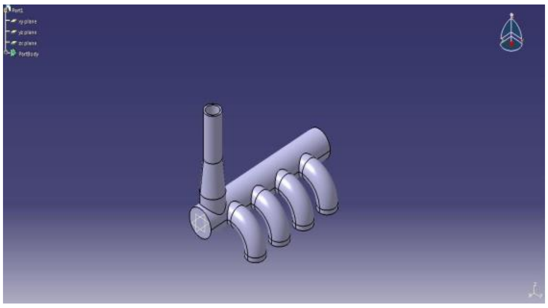


Figure 4.8: Short Bend Side Exit with Reducer

V. ANALYSIS

5.1 Types Of Analysis:

1. Structural Static Analysis
2. Structural Dynamic Analysis.
3. Structural Buckling Analysis.
    - ➢ Linear Buckling
    - ➢ Non-Linear Buckling
4. Structural Non-Linearity's
5. Static And Dynamic
6. Kinematics Analysis.
7. Thermal Analysis.
8. Electromagnetic Field Analysis.
9. Electric Field Analysis
10. Fluid Flow Analysis
    - ➢Computational Fluid Dynamics
    - ➢Pipe Flow
11. Coupled-Field Analysis
12. Piezoelectric Analysis.

5.2 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.

5.3 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 Modeler, using the duplicated system.
Regenerate the computational mesh.
Recalculate a solution in ANSYS Fluent.
Compare the results of the two calculations in CFD-Post.

5.4 Material Fluid Properties

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

Table 5.1: 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

5.5 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 5.2: 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 5.3: 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 0atm (Gauge) for all models. The mean hydraulic diameters for outlets of different models are shown below:

Table 5.4: 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

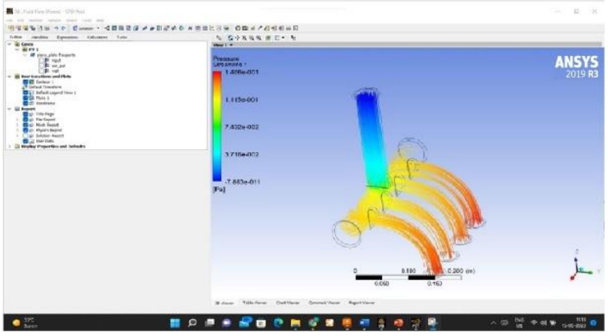
5.6 Engine Specifications

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

Table 5.5: 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

VI. RESULTS AND DISCUSSIONS



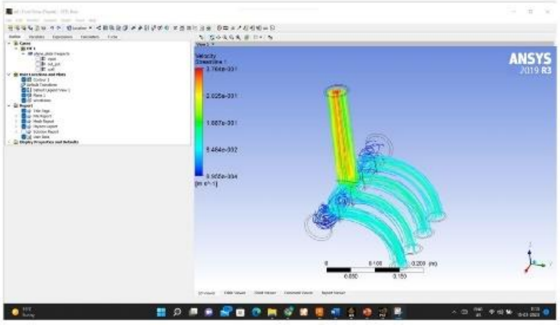


Fig 6.1: Long bend center exit Pressure

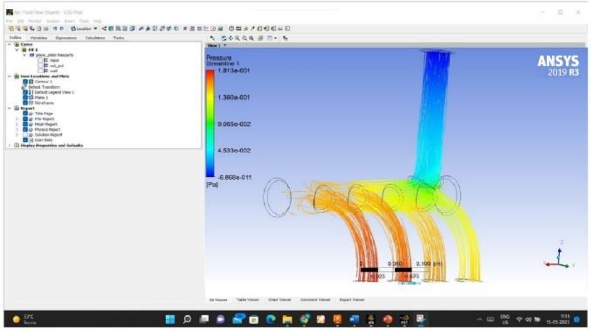
Fig 6.2: Long bend center exit velocity

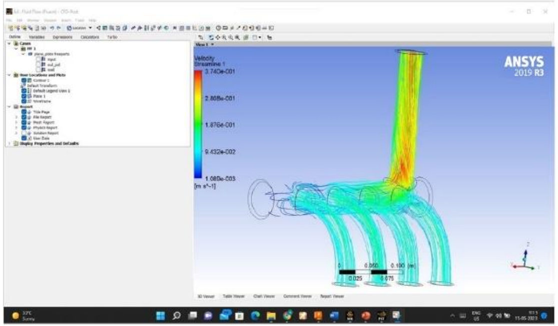
Fig 6.3: Long bend center inlet with reducer Pressure

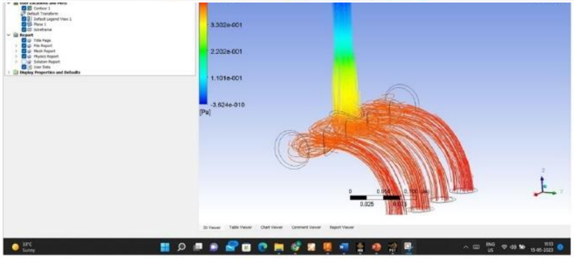
Fig 6.4: Long bend center inlet with reducer Velocity

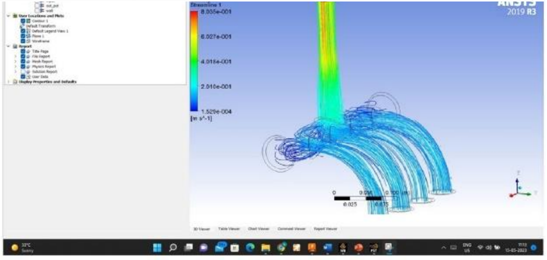
Fig 6.5: Long bend side inlet Pressure

Fig 6.6: Long bend side inlet -

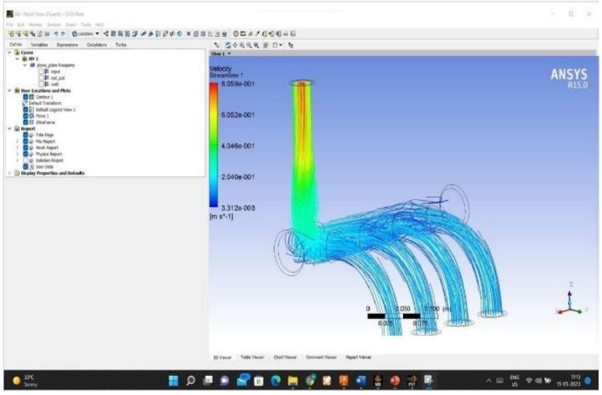








Velocity



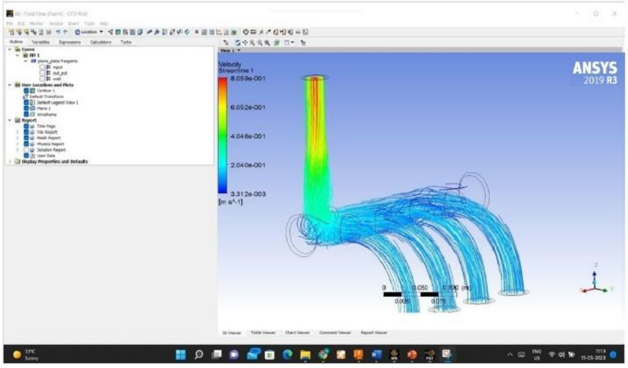
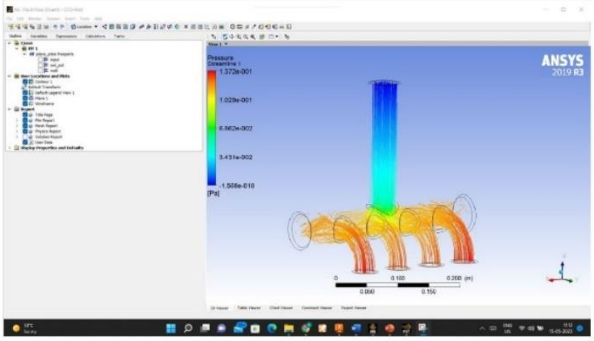


Fig 6.7: Long bend side inlet with reducer Pressure

Fig 6.8: Long bend side inlet with reducer Velocity



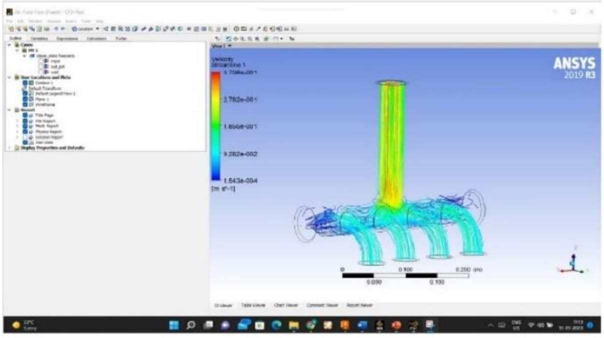


Fig 6.9: Short bend center inlet Pressure

Fig 6.10: Short bend center inlet Velocity

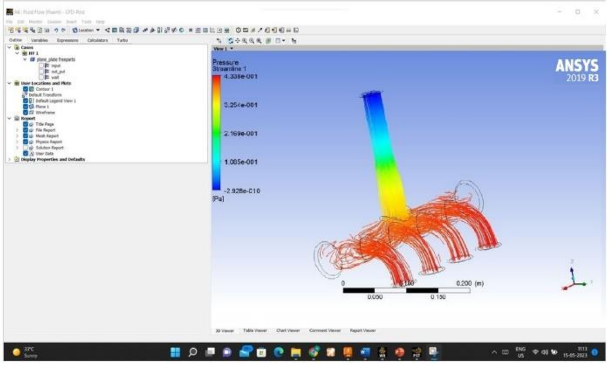


Fig 6.11: Short bend center inlet with reducer Pressure

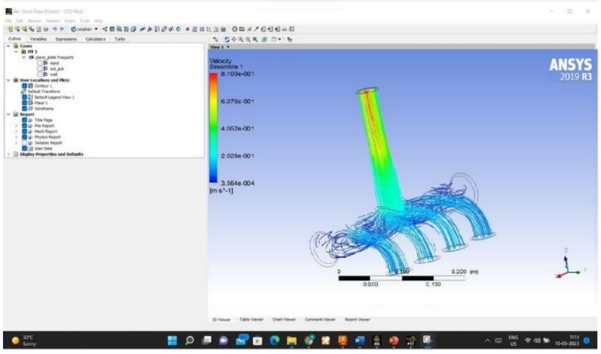


Fig 6.12: Short bend center inlet with reducer Velocity

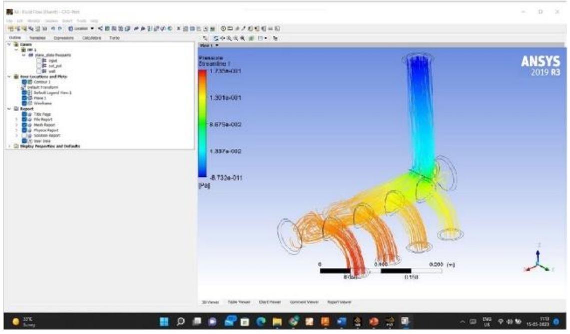


Fig 6.13: Short bend side inlet Pressure

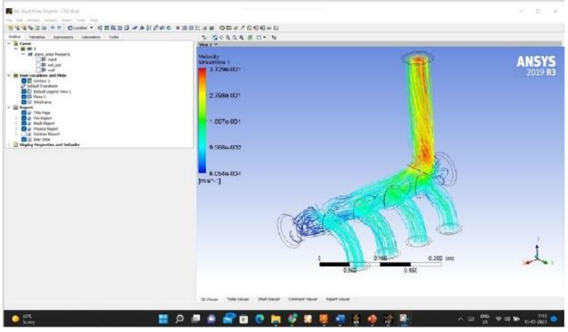


Fig 6.14: Short bend side inlet Velocity

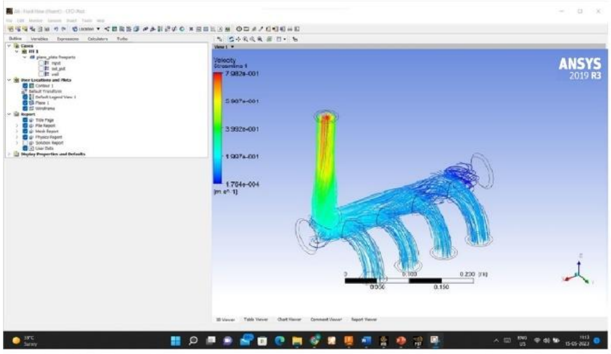


Fig 6.15: Short bend side inlet with reduce pressure

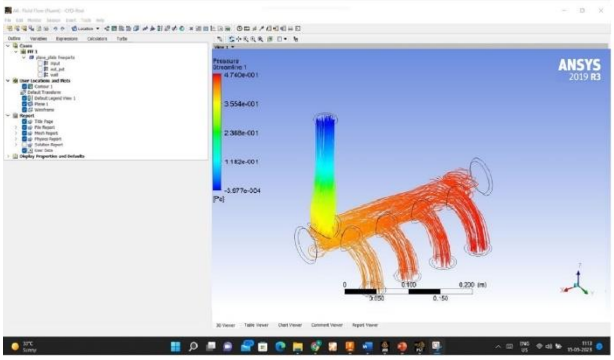


Fig 6.16: Short bend side inlet with reducer velocity

The back pressure and inlet velocity for all the models at all loading conditions are listed below:

Table 6.1: Backpressure for Different Models
Models in
in Pascal

	2KG	4KG	6KG	8KG	10KG	12KG
SBCI	940	976	1002	1036	1079	1111
SBSI	1020	1071	1098	1113	1132	1172
LBCI	850	863	894	923	984	1012
LBSI	973	1005	1039	1076	1099	1125
SBCIR	984	1012	1047	1077	1114	1154

Table 6.2: Inlet Velocity for Different
Meter per Second (m/s)

	2KG	4KG	6KG	8KG	10KG	12KG
SBCI	17.03	18.1	18.7	19.52	21.45	23.01
SBSI	18.1	18.6	19.1	20.2	21.6	23.5
LBCI	20.2	21.33	22.07	23.52	23.98	24.77
LBSI	18.71	18.92	19.23	20.12	22.21	23.65
SBCIR	17.7	17.79	18.23	19.86	21.1	23.89

SBSIR	1180	1214	1222	1222	1272	1303
SBSIR	1037	1080	1112	1112	1187	1201
LBSIR	1138	1174	1219	1219	1276	1271

SBSIR	16.8	17.12	18.6	19.9	21.76	23.92
SBSIR	17.3	18.67	19.54	21.96	23.65	24.71
LBSIR	17.9	18.01	19.1	20.65	21.86	23.98

VII, CONCLUSION

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

1. Forces exerted by gas particles in the manifold effect the values of back pressure and exit velocity, due to which overall performance score on the basis of these two parameters changes.
2. Short bend models show better performance, as compared with long bend models.

Due to increased length, differences in overall performance score in long bend models are greater than that of short bend models. And Out of available set of alternatives, long bend center exit (LBCE) model of manifold is the best one because it has scored rank *first* for overall performance score.

ACKNOWLEDGEMENT

We are delighted to present the initial project report on 'Design And Analysis Of Engine Inlet Manifold'. Our heartfelt thanks go to our internal guide Mr. MD. Azeem for their unwavering assistance and guidance throughout the project. We are truly appreciative of their valuable suggestions, which proved to be extremely beneficial. We would also like to express our gratitude to the HOD, Dr. B. Vijaya Kumar, Guru Nanak Institute of Technology, Hyderabad, India, for their invaluable support and recommendations, which have been instrumental in the success of this project

REFERENCES

[1]. PL. S. Muthaiah, "CFD Analysis of catalytic converter to reduce particulate matter and achieve limited back pressure in diesel engine", Global induced during suction and compression stroke l journal of researches in engineering A: Classification (FOR) 091304,091399, Vol.10 Issue 5 (Ver1.0) October 2010

[2]. 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/Issue7/342.

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

[4]. Vivekananda Navadagi and SiddaveerSangamad Development of a Partial FilterTechnology for Hdd Retrofit, Sae Technical Paper 200601-0213.Jacobs, T., Chatterjee, S., Conway, R., Walker, Development of a Partial Filter Technology for Hdd Retrofit, Sae Technical Paper 2006-01-2013.

[5]. Jafar M Hasan, Wahid S Mohammad, Thamer A Mohamed, Wissam H Alawee, "CFD Simulation for Manifold with Tapered Longitudinal Section" International Journal of Emerging Technology and Advanced Engineering, Volume 4, Issue 2, February 2014.

[6]. M. Usan, O. de Weck, D. Whitney, "Inlet System Manifold Development Enhancement through Multi-Attribute System Design Optimization", American Institute of Aeronautics and Astronautics.

[7]. HessamedinNaeimi, DavoodDomiryGanji, MofidGorjivadirad and MojtabaKeshavarz, "A Parametric Design of Compact Inlet Manifold Junction in Heavy Duty Diesel Engine Using Computational Fluid Dynamics Codes" Thermal Science, Volume-15, No. 4, 2011; Kyuang-Sang Cho, Kyung-Bin Son, Ue-Kan Kim, "Design of Inlet Manifold for Pulse.

[8]. Converters Considering Fatigue Strength due to Vibration", Journal of the Korean Society of Marine Engineering, vol-37, No.7, 2013;

[9]. Masahiro Kanazaki, Masashi Morikawa, Shigeru Obayashi and Kazuhiro Nakahashi, "Inlet Manifold Design for a Car Engine Based on Engine Cycle Simulation" International Conference.

[10]. Parallel Computational Fluid Dynamics, Japan, May 2002;

[11]. Hong Han-Chi, Huang Hong-Wu, Bai Yi-Jie, "Optimization of Intake and Inlet System for FSAE.

[12]. Car Based on Orthogonal Array Testing" International Journal of Engineering and Technology, Volume 2, No. 3, March 2012..

[13]. Xueyuan Zhang, Yu Luo, Jianhua Wang, "Coupled Thermo-Fluid-Solid Analysis of Engine Inlet Manifold Considering Welding Residual Stresses" Transactions of JWRI, 2011L;

[14]. Simon Martinez-Martinez, Ruben D. Leal-Garza, Fausto A. Sanchez-Cruz, Esteban Baez Villareal, "CFD Analysis of the effect of the Inlet Manifold Design on the Close-Coupled Catalytic Converter Performance" Journal of KONES Powertrain and Transport, vol-17, No.4, 2010; [13] S. N. Ch. Dattu. V, M. PradeepVarma, B.

[15]. ShyamSundar, "Thermal Analysis on 4-1 Tubular Type IC-Engine Inlet Manifold through Analysis" International Journal of Advanced Mechanical Engineering, volume-4, Number 7, 2014;

[16]. 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;

[17]. Gopal, M.M.M Kumara Varma, L. Suresh Kumar, “Inlet Manifold Design – FEA Approach” International Journal of Engineering Trends and Technology, Volume-17, Number 10, November 2014;

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

[19]. I.P. Kandylas, A.M. Stamatelos,“Engine Inlet System Design based on Heat Transfer Computation” Energy Conservation & Management, 1999;