Design And Optimization of Inlet Manifold Used In 4-Stroke Engine

Basawaraj S Hasu[1], K Venkatesh[2], Kandula Gopi[3]
Department of Mechanical Engineering
[1]Professor & HOD, [2]Assistant Professor, [3]PG Student
[1, 2, 3]AVN Institute of Engineering & Technology, Hyderabad, India

Abstract-- *The fueling system is one of the crucial variables that must be focused on, to achieve good fuel efficiency and low engine out emissions. Fuel injection system seems a promising technology as a medium to supply suppressed fuel because of its high fuel delivery efficiency, enhanced fuel economy and reduced engine out emission. Inlet manifolds have a major effect on engine's performances and emission pollutants. In automotive engineering, an inlet manifold or intake manifold is the part of an engine that supplies the fuel/air mixture to the cylinders. The primary function of the intake manifold is to evenly distribute the combustion mixture to each intake port in the cylinder head. At the time of distribution, the pressure (2 Bar) applied internally in the inlet manifold. Even distribution is important to optimize the efficiency and performance of the engine. Locally, common intake manifolds are usually made of aluminium alloys.*

In this project 3D model of an inlet manifold used in the 4-stroke SI engine shall be developed. It is used to analyze the inlet manifold for flow analysis and finite element analysis. In this project, to verify the deflections and stress values due to the air pressure load applied internally in inlet manifold during the distribution of air in cylinders. The dynamic analysis shall be done to verify the deflections and stress values due to vibrations produced in the inlet manifold. In this study NX_CAD software shall be used for 3Dmodeling of the inlet manifold, ANSYS software shall be utilized for the finite element analysis of an inlet manifold.

I. INTRODUCTION

In automotive engineering, an inlet manifold or intake manifold is the part of an engine that supplies the fuel/air mixture to the cylinders. The intake manifold considered here is Aluminum, steel or plastic cover that sits on top of the engine. It is attached to the top of the engine by an intake manifold gasket that is made of rubber and plastic. The intake manifold directs the air-fuel mixture in the engine to the corresponding cylinder here it is burned to produce power. The coolant is made to flow through the intake manifold to the cylinder which leads to reducing engine temperature. Expansion, contraction, and heat from the engine cause the gasket to fail. This will cause coolant passing through the intake manifold to leak. A leaky manifold gasket can also result in air being sucked through the leaky gasket, resulting in a vacuum leak and poor engine performance.

Engines having more than one cylinder need a way of distributing air and fuel, in the case of carbureted engines from a single inlet to the various cylinders. Regularly, air enters through a single hole and moves through a throttle valve plate that can open and close to have a control on the volume of air that enters inside. After the valve, the air and sometimes fuel goes into a central holding chamber called the "plenum," it acts as kind of a reservoir.

II. LITERATURE REVIEW

Studies On Improvement Of Intake Manifold ForCompressed Natural Gas Engine by Devender Kumar, in this paper, the Geometrical development of intake manifold is very prominent for the better performance of an I.C. Engine. Indifferent velocity distribution of intake air at runner's outlets of intake manifold makes it less productive. The reported work aims to make this unequal distribution of velocity in the nearly equal manner with an increase of velocity at outlets without any major modification in the design of intake manifold. Let us consider 'Maruti Wagnor (petrol version)' engine intake manifold is used for experimental testing, to examine the variation of the velocity of air flow at the outlet of four runners. To modify the intake manifold first a 3-D model of the actual manifold is made in design software (PRO-E) and developed in it, and then validation of the designed model is done in itself by using commercial CFD software preferably FLUENT. To achieve the desired and mostly improved results two other models of the same intake manifold with different design configuration are made in software then examine the result of these two models with original one to find out pressure and velocity losses. After analysis of models, it noticed that the hidden projections of nut projected stiffeners and depth cut at the extreme of plenum cause pressure losses due to which uneven distribution takes place at runner's outlet manifold only. The target of nearly equal velocities in all the mentioned four runners is reached in the inlet manifold by the redesign of the plenum of the intake manifold and set it free from unwanted hidden projection inside the plenum. The results show the nearly equal distribution in all four runners with an increase in velocity of air flow by 14% in outlet-1 and 5% to 7% approx. In other three runners of the inlet manifold.

Finite Element Analysis Of Automotive Intake Manifold Using CAE Software by G.Venkata Punna Rao and M.Vimal Teja, This project focused on simulation testing of automotive intake manifold design using Computer aided engineering software. The finite element random vibration analysis is conducted on excavator intake manifolds designs for material cast iron and aluminium alloy. The main motto of this work is to study the maximum computational stress on the model due to the effect of engine vibrations and pressure pulsation loads. The software's used are PRO-E 4.0, HYPERMESH-10 and ANSYS-11. Based on the simulation results obtained, the maximum Stress of both materials is compared to distinguish which is better in resisting the vibration Loads applied.

III. PROBLEM DEFINITION & SOLUTION METHODOLOGY

In this project 3D model of an inlet manifold used in the 4-stroke engine is developed by using the NX-CAD software. In this project, Structural analysis of inlet manifold is also carried out to verify the deflections and stress values due to the pressure applied internally in inlet manifold during the distribution of air in cylinders.

In this study NX-CAD software shall be used for 3Dmodeling of the inlet manifold, ANSYS software shall be used for the finite element analysis of an inlet manifold.

THE METHODOLOGY FOLLOWED IN MY PROJECT IS AS FOLLOWS

Create a 3D model of the inlet manifold is developed by using the NX-CAD software.Perform steady state analysis on inlet manifold using ANSYS software and obtain the deflections and Von Miss stresses values produced in the inlet manifold.Perform modal analysis of the inlet manifold with aluminum alloy and obtain natural frequencies and their mode shapes.Perform harmonic analysis of the inlet manifold and obtain the deflections and von misses stresses at critical frequencies.Optimize the inlet manifold if required for better results.

IV. 3D MODELLING OF INLET MANIFOLD

The CAD model of the inlet manifold is shown below:
2D drawing:

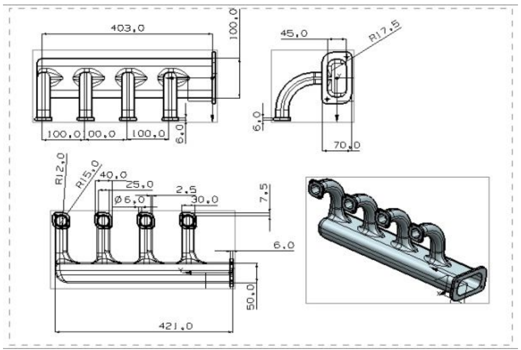

Fig: Shows the drafting of the inlet manifold

3D modeling:

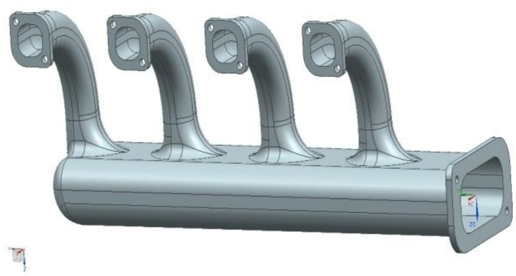

Fig: Shows CAD model of the inlet manifold

Isometric view of inlet manifold:

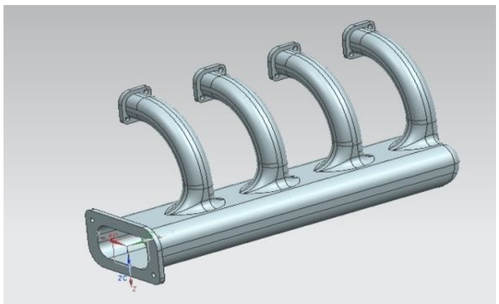

Fig. Shows the Isometric view of inlet manifold

V. FINITE ELEMENT ANALYSIS OF INLET MANIFOLD

Structural Analysis of Inlet Manifold

Finite Element Analysis (FEA) and worlds best Finite Element Modeling (FEM) are two most popular mechanical engineering applications offered by existing CAE systems. This is attributed to the fact that the FEM is perhaps the most popular numerical technique for solving engineering problems. The method is general enough to handle any complex shape of geometry (problem domain), any material properties, any boundary conditions and any loading conditions. The generality of the FEM fits the analysis requirements of today's complex engineering systems, and designs, where closed form solutions are governing equilibrium equations, are not available. Also, it is an efficient design tool by which designers can perform parametric design studying various cases (different shapes, material loads etc.) analyzing them and choosing the optimum design.

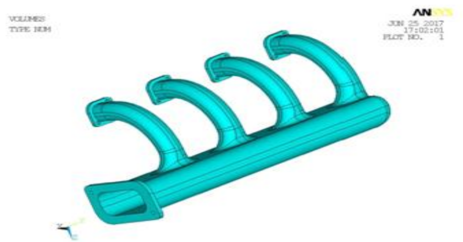

Fig. Shows the geometric model of the Inlet Manifold

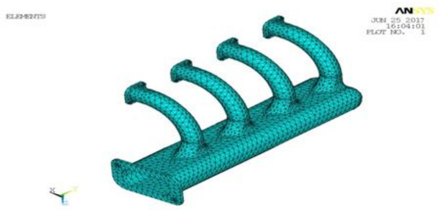

Fig: shows the meshed model of Inlet Manifold

RESULTS

DEFLECTION

The Max. Displacement vector sum observed 0.014269mm on Inlet Manifold.

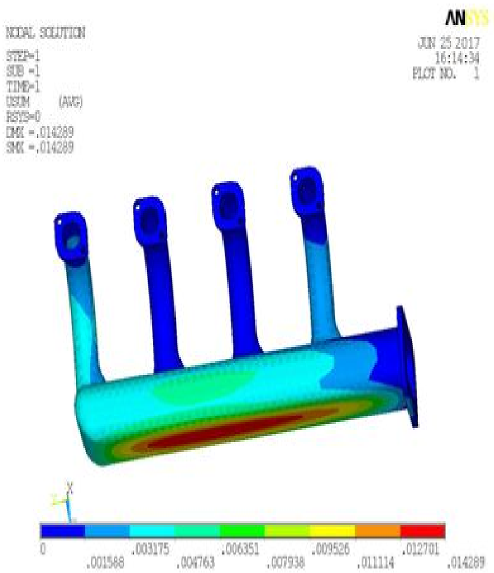

Fig: shows the Max. Displacement of Inlet Manifold

STRESS

The 1stprincipal Stress observed 4.998MPa on Inlet Manifold.

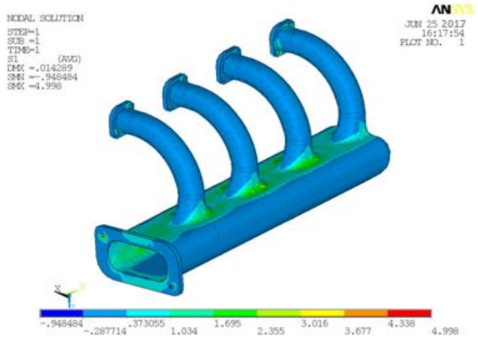

Fig. Shows the 1st principal Stress of Inlet Manifold

The Max.Von Miss Stress observed 7.20MPa on Inlet Manifold.

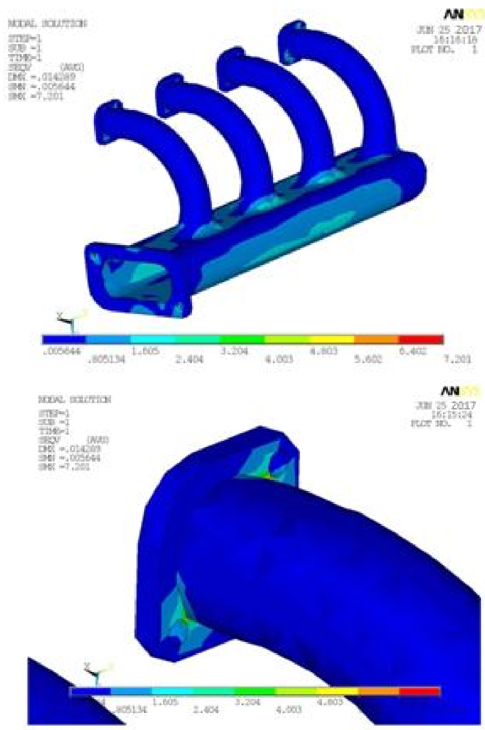

Fig: shows the Von Misses stress of Inlet Manifold

MODAL ANALYSIS OF INLET MANIFOLD

MODAL ANALYSIS

Modal analysis is used to govern the vibration characteristics (both natural frequencies and mode shapes) of a simple or complex structure or even a machine component while it is being designed. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic response analysis, or a seismic analysis.

The mode shapes for the above frequencies are plotted below.

Results –Mode1 @ 363.097 Hz

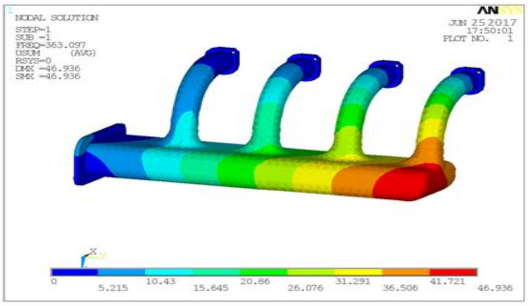

Fig. Shows Mode shape 1@363.097Hz for Inlet Manifold

Results –Mode2 @ 1022.31Hz

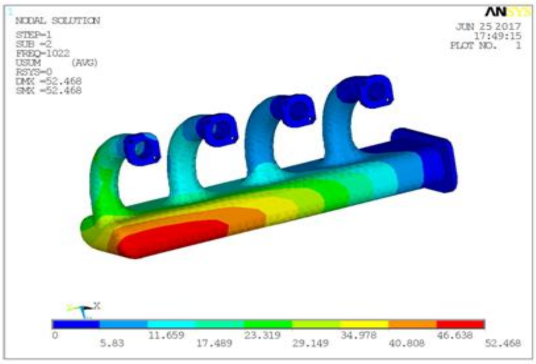

Fig Shows Mode shape 2@ 1022.31Hz for Inlet Manifold

From the modal analysis:

The total weight of the inlet manifold is 0.0019Tone.

It is observed that the peak mass participation of 0.0005Tone in X-direction for the frequency of 363.097 Hz.

It is observed that the maximum mass participation of 0.00001Tone in Y-direction for both the frequencies of 363.097Hz.

It is observed that the maximum mass participation of 0.00065Tone in Z-direction for the frequency of 363.097Hz.
To check the structure governing result at the mentioned rate due to the operating loads, harmonic analysis is carried out on the Inlet Manifold.

VI. HARMONIC ANALYSIS OF INLET MANIFOLD

Harmonic response occurs at forcing frequencies that match the natural frequencies of your structure. Before obtaining the harmonic solution, you should first determine the natural frequencies of your structure by obtaining a modal solution.

Harmonic analysis, by definition, assumes that any applied load varies harmonically (sinusoidal) with time. To completely specify a harmonic load, three pieces of information are usually required: the amplitude, the phase angle, and the forcing frequency range

VonMises stress:

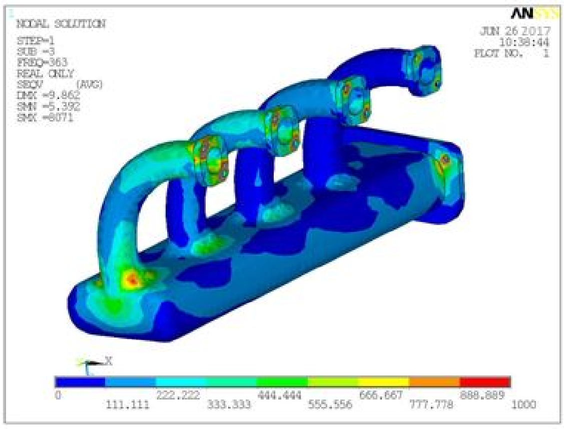

Fig: shows the VonMises stress of Inlet Manifold

Von Mises stress:

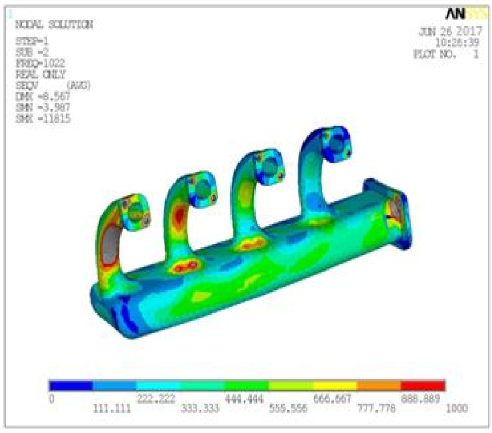

Fig: shows the Von Mises stress of Inlet Manifold.

VII. FINITE ELEMENT ANALYSIS OF MODIFIED INLET MANIFOLD

STATIC ANALYSIS OF MODIFIED INLET MANIFOLD

Static analysis can, however, include steady inertia loads and time varying loads that can be approximated as static equivalent loads. The 3d model of the modified Inlet Manifold is created in NX-CAD and converted into Parasolid. The Parasolid file is imported into ANSYS, and finite element analysis is carried out using ANSYS software.

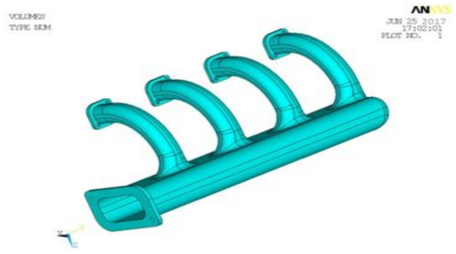

Fig. Shows the geometric model of the modified Inlet Manifold

RESULTS
DEFLECTION

The Max. Deformation observed 0.00111mm on modified Inlet Manifold in X-dir.

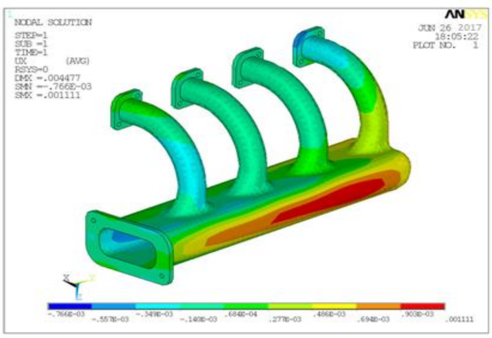

Fig. Shows the deformation of Inlet Manifold in X-dir

STRESS
The 1stprincipal Stress observed 3.408MPa on modified Inlet Manifold.

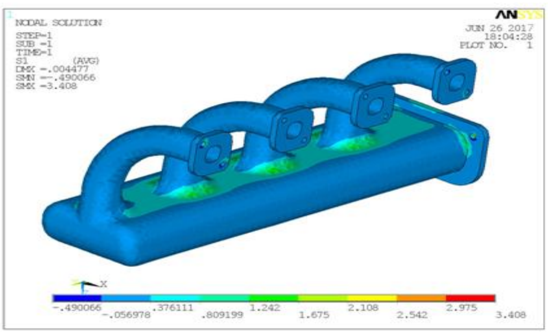

Fig. Shows the 1st principal Stress of modified Inlet Manifold

VIII. RESULTS AND DISCUSSIONS

There are three different cases to study the Inlet Manifold:

Static analysis
Modal analysis
Harmonic analysis

STATIC ANALYSIS

The Max Deflection and the Max Avg. VonMises Stress observed on the Inlet Manifold for applied pressure is 0.0142mm and 7.201MParespectively. Hence according to the Maximum Yield Stress Theory, the VonMises stress is less than the yield strength of the material aluminum is 110MPa. The design of Inlet Manifold is safe for the above operating loads.

MODAL ANALYSIS

From the modal analysis, a total of 2 natural frequencies is observed in the frequency range of 0-1200 Hz.

The total weight of the inlet manifold is 0.0019Tone.

It is observed that the maximum mass participation of 0.0005Tone in X-dir for the frequency of 363.097 Hz.

It is observed that the maximum mass participation of 0.00001Tone in Y-dir for both the frequencies of 363.097Hz.

It is observed that the maximum mass participation of 0.00065Tone in Z-dir for the frequency of 363.097Hz.

HARMONIC ANALYSIS

The harmonic analysis was carried out on the Inlet Manifold to determine the deflections and stress of a structure in the frequency range of 300-1100 Hz. The total numbers of sub steps defined for the analysis are 16.

Table. From the Harmonic analysis,

S.NO	FREQUENCY(Hz)	DEFLECTIONS (mm)	VON MISES STRESS (MPa)
1	363	9.862	1000
2	1022	8.567	1000

From the above results, it is observed that the critical frequencies 363Hz and 1022Hz are having stresses of 1000MPa and 1000MPa respectively. Hence according to the Maximum Yield Stress Theory, the VonMises stress is greater than the yield strength of the material Aluminium is 110 MPa. The design of Inlet Manifold is not safe under above-mentioned frequencies

MODIFIED INLET MANIFOLD ANALYSIS RESULTS
STATIC ANALYSIS

The Max Deflection and the Max Avg. VonMises Stress observed on the Inlet Manifold for applied pressure is 0.0044mm and 2.541MPa respectively. Hence according to the Maximum Yield Stress Theory, the VonMises stress is less than the yield strength of the material aluminium is 110MPa. The design of Inlet Manifold is safe for the above operating loads.

MODAL ANALYSIS

From the modal analysis, a total of 1 natural frequency is observed in the frequency range of 0-1200 Hz.

The total weight of the modified inlet manifold is 0.00282Tone.

It is observed that the maximum mass participation of 0.0007Tone in X-dir for the frequency of 346.819 Hz.

It is observed that the maximum mass participation of 0.00001Tone in Y-dir for both the frequencies of 346.819 Hz
.

It is observed that the maximum mass participation of 0.0009Tone in Z-dir for the frequency of 346.819 Hz.

HARMONIC ANALYSIS

The harmonic analysis was carried out on the Inlet Manifold to determine the deflections and stress of a structure in the frequency range of 300-1100 Hz. The total numbers of sub steps defined for the analysis are 5.
From the Harmonic analysis,

Table shows the deflections and Von Mises stress for operating frequencies

S.NO	FREQUENCY(Hz)	DEFLECTIONS (mm)	VON MISES STRESS (MPa
1	346	0.388	100

From the above results, it is observed that the critical frequencies 346Hz are having stresses of 100MPa respectively. The Yield strength of the material metal Aluminum is 110 MPa. Hence according to the Maximum Yield Stress Theory, the Von Mises stress is less than the yield strength of the material Aluminum is 110MPa. The design of Inlet Manifold is safe for the above operating loads.

REFERENCES

[1] Gordon P Blair, "Design and Simulation of Four Stroke Engines", Society of Automotive Engineers, 1998, SAE reference R-186,
[2] William Harry Crouse, Donald L. Anglin"Automotive Mechanics", McGraw-Hill College, 1993.
[3] John Heywood, "Internal Combustion Engine Fundamentals", McGraw-Hill Education, April, 1988.
[4] Corky Bell, "Maximum Boost, Designing, Testing and Installing Turbocharger Systems", Benthey Publishers, 1997.
[5] Wes Ruebman and Travis Butler, "Data Analysis and Selection of IHI and Garrett Turbo Systems", February 2015.
[6] Sagar KC, Pablo Lopez, Robert Lopez, Tobias Overdiek, Sebastian Peters, Luis Ulloa, "CBR 250 Turbo Package-Improvements And Revisions", August 2012.
[7] Chris Craig, "Dynamic Analysis of FSAE Intake Manifolds Using WAVE Engine Simulation Software", 2006.
[8] Jacob Bell, Akula Aditya, Subhash Seshadri, "Honda CBR 250R (MC14E) Engine Rebuild", August 2014.
[9] Jacob Bell, Travis Butler Lauren Middleton, Wes Ruebman, Austin Williams, "Design of 250CC Four Cylinder Racing Engine", Maverick Performance And Engineering, May 2015.
[10] Gordon P Blair, W. Melvin Cahoon, "Best bell, Special Investigation: Design of an Intake Bellmouth", September 1, 2010.