International Journal for Innovative
Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

Title: Thermal Analysis Of A V12 Engine.

Volume 06, Issue 04,Page No: 1303 - 1310.

Paper Authors

***TADIGADAPA MANIBABU, MR. B V VV B LAKSHMIPATHI RAO.**

* Dept of Thermal Emgineering, Pydah College of Engineering.

.

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

THERMAL ANALYSIS OF A V12 ENGINE

***TADIGADAPA MANIBABU, **MR. B V VV B LAKSHMIPATHI RAO**

*PG Scholar, Dept of Thermal Engineering, Pydah College Of Engineering
*Assistant Professor, Dept of Thermal Engineering, Pydah College Of Engineering

ABSTRACT

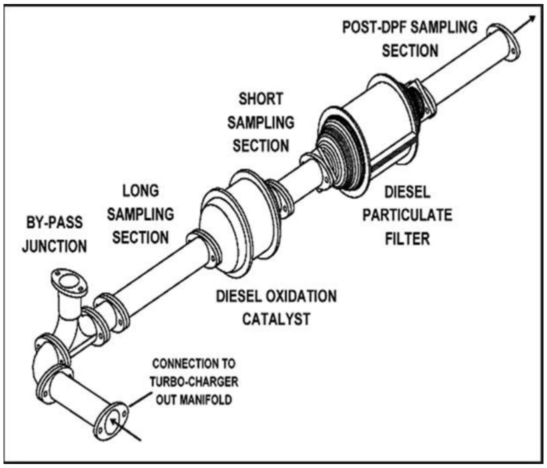
A V12 engine often just called a V12 is an internal combustion engine with 12 cylinders. The engine has six cylinders on each side called banks. The two banks form a "V" shaped angle. In most engines, the two banks are at a 60° angle to each other. All twelve pistons turn a common crankshaft. It can be powered by different types of fuels, including gasoline, diesel and natural gas. A V12 engine does not need balance shafts. A V12 angled at 45°, 60°, 120°, or 180° from each other has even firing and is smoother than a straight-6. This provides a smooth running engine for a luxury car. In a racing car, the engine can be made much lighter. This makes the engine more responsive and smoother. In a large heavy-duty engine, a V12 can run slower and prolonging engine life. The main objective of the project is how to develop the prototype of V 12 engine assembly using CAD tool SOLIDWORKS. These Engine assembly consists major components they are Piston, Connecting Rod Assembly, Crank Shaft, Cylinder head, Cam Shaft, Valves, crank case, oil tank and spark plug with required dimensions. The components which are developed in SOLIDWORKS are also analyzed in it using simulation tool. The thermal analysis of piston, crank shaft, cam shaft and valve is performed for 800k thermal loading and the results of temperature distribution of the components are shown. Finally the thermal analysis results of the components are compared and the best suited material is selected.

INTRODUCTION:

Energy efficient exhaust system development requires minimum fuel consumption and maximum utilization of exhaust energy for reduction of the exhaust emissions and also for effective waste energy recovery system such as in turbocharger, heat pipe etc. from C.I. engine. To analyse the exhaust energies available at different engine operating conditions and to develop an exhaust system for maximum utilization of available energy at the exhaust of engine cylinder is studied.Design of each device should offer minimum pressure drop across the device, so that it should not adversely affect the engine performance. Backpressure acting on engine is most important controllable factor which basically deteriorates the engine and emission control performance. So numbers of methods to increase the performance of the CI engine have been established. A better method of utilising the exhaust is being achieved by this paper. The method uses the exhaust gases from an optimal sized engine currently used, to convert available Kinetic energy and enthalpy of exhaust gases into the pressure energy for useful after treatment of exhaust gases. This

`

pressure is being used and sent through diffuser which reduces the back pressure.



1.1 Components that influence airflow in to the engine

1.1 Components that influence airflow *into* the engine are the:

- air filter
- intake air piping
- mass air sensor (if applicable)
- throttle body or carburettor
- intake manifold
- camshaft
- intake port and valve of cylinder heads
- turbo's compression, section, and supercharger (if applicable)
- Components that influence airflow *out of* the engine are the:
- exhaust valve and exhaust ports of the cylinder heads
- camshafts
- exhaust manifolds
- turbo's turbine (if applicable)
- exhaust tubing
- catalytic converters
- muffler

LITERATURE REVIEW

HCCI is an alternative and attractive combustion mode for internal combustion engines that offers the potential for high diesel-like efficiencies and dramatic reduction in NOx and PM. HCCI occurs as the result of spontaneous auto ignition at multiple points throughout the volume of the charge gas and each auto ignition may or may not produce a flame front. In order to control the energy release rate to acceptable levels the engine must be operated with high levels of dilution, exhaust or extra air, which results in significantly reduced pumping losses for SI engines and lower peak burned gas temperature. With appropriately higher compression ratio and less heat loss due to low combustion temperature; the thermal efficiency approaches the levels of CI engines.

[1] [Thring et,al.,1989] extended the work in a four-stroke engine using fully-blended gasoline and mapped the operating regime as a function of equivalence ratio and External EGR rate. The load range limitations of HCCI were noted

and an engine operating strategy was put forward, suggesting use of HCCI mode at part load and transitioning into SI flame mode at high load condition. HCCI research has continued over the past 20 years. Experiments have been conducted in four-stroke engines operating on fuels as diverse as gasoline, diesel, methanol, ethanol, LPG, natural gas, etc. with and without fuel additives, such as isopropyl nitrate, dimethyl ether (DME), di-tertiary butyl peroxide

[2] [Caton et al., 2005]. From these investigations and many others in the past five years it appears that the key to implementing HCCI is to control the charge auto ignition behavior which is driven by the combustion chemistry. Even more than in IC engines, compression ratio is a critical parameter for HCCI engines. Using high octane fuels, the higher the compression ratio the better in order to ignite the mixture at idle or near-idle conditions. However, compression ratios beyond 12 are likely to produce severe knock problems for the richer mixtures used at high load conditions. It seems that the best compromise is to select the highest possible CR to obtain satisfactory full load performance from SI fuels

[3] [Najt and Foster et,al., 1983]. The choice of optimum compression ratio is not clear; and it may have to be tailored to the fuel and other techniques used for HCCI control. For early direct-injection diesel-fueled HCCI engines compression ratios must also be limited to mitigate the problem of over advanced autoignition resulting from pre-ignition chemical reactions.

[4] [Zhao et al., 2002]. Effects of external EGR on auto ignition of the mixture are different from that of internal EGR even when both the EGR mixtures are at the same temperature [Law et al., 2002]. In four-stroke engines with flexible valve actuation, there are several strategies for internal EGR. One is the re breathing strategy of where the exhaust valve remains open throughout the intake stroke; another is the exhaust recompression strategy demonstrated that the variable valve timing strategy has a strong influence on the gas exchange process, which in turn influences the engine parameters and the cylinder charge properties, hence the control of the HCCI process. The EVC timing has the strongest effect followed by the IVO timing, while the EVO and IVC timing have the minor effects

3. METHODOLAGY:

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 a 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.

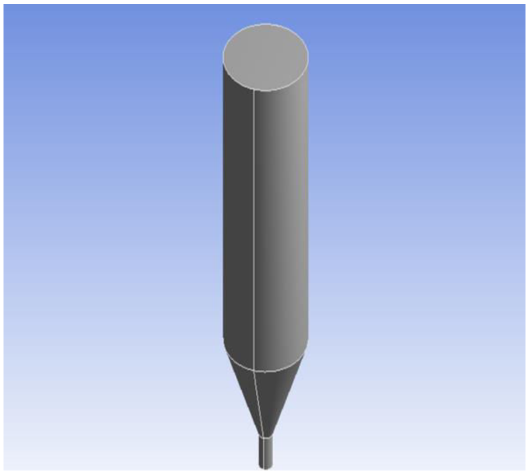


Figure 3.1 shows the structural design of exhaust system of engine

MESHING:

The Figure shown is the meshed model of rigid flange coupling in the ANSYS analysis for the static structural process. To analyse, the FEM triangular type of mesh is used for the rigid flange coupling in the ANSYS environment.

- The number of elements used in this meshing is 71441and the number of nodes is 122228.In this process regular type of meshing is done to analyse the process.
- Using the working condition of the coupling a relative rotational movement between the shafts comes into picture consequently.
- The determination of the shear stress along the contact region is essential. So, the model is meshed and then analysed to get the detail and authentic result of the stresses of the contact region.

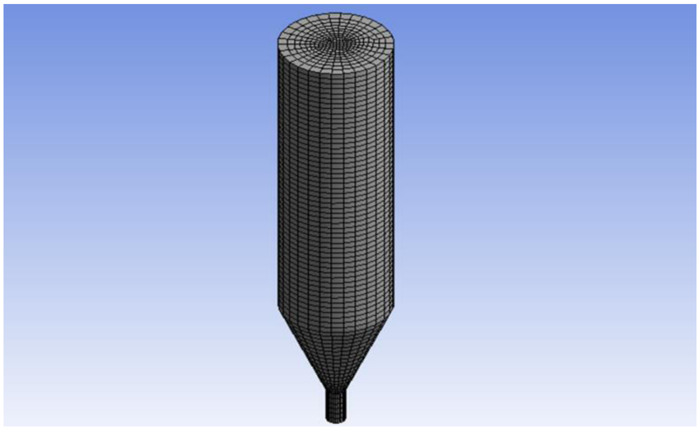


Figure 3.2 shows the total meshing of the engine exhaust system

4. ANALYSIS:

4.0 Introduction to ANSYS

4.1 Reducing the design and-manufacturing costs using ANSYS (FEA):

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.

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.

4.2 GEOMETRICAL DEFINITIONS:

There are four different geometric entities in pre -processor namely key points, lines, areas and volumes. These entities can be used to obtain the geometric representation of the structure. All the entities are independent of other and have unique identification labels.

4.3 BOUNDARY CONDITIONS AND LOADING:

After completion of the finite element model it has to constrain and load has to be applied to the model. User can define constraints and loads in various ways. All constraints and loads are assigned set 1D. This helps the user to keep track of load cases.

4.4 MODEL DISPLAY:

During the construction and verification stage of the model it may be necessary to view it from different angles. It is useful to rotate the model with respect to the global system and view it from different angles. Pre-processor offers this capability. By windowing feature pre- processor allows the user to enlarge a specific area of the model for clarity and details. Pre- processor also provides features like smoothness, scaling, regions, active set, etc for efficient model viewing and editing.

5. RESULTS:

The following are the analysis of the engine exhaust system which is done using ANSYS. In this the gases are emitted by the exhaust system at different angled systems are used like 220 300 on this analysis is done.

22^0 angles:

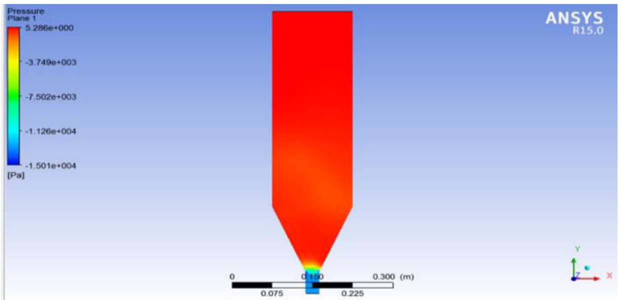


Figure 5.1 shows the pressure analysis inside the engine exhaust system

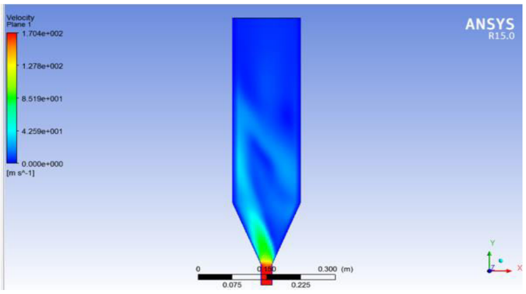


Figure 5.2 shows the Turbulence kinetic energy flow of gases inside exhaust system

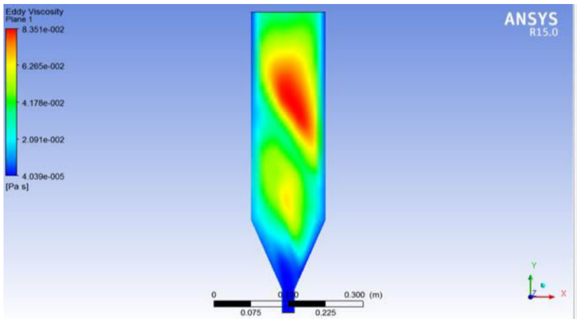


Figure 5.3 shows the Eddy Viscosity of the engine exhaust system

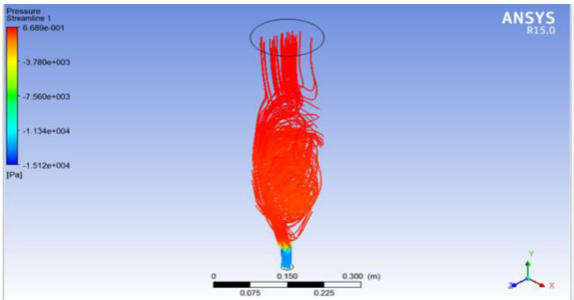


Figure 5.4 shows the pressure of flow inside the engine exhaust system

33^0 ANGLE:

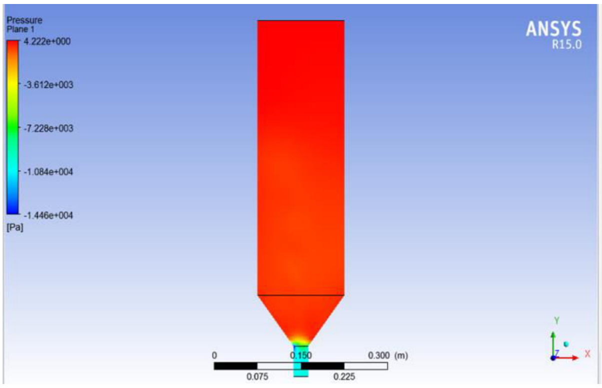


Figure 5.5 shows the pressure analysis of the exhaust system

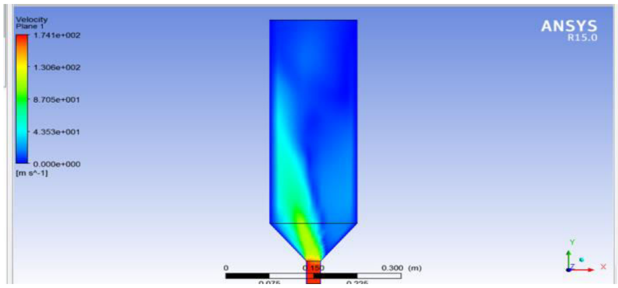


Figure 5.6 shows the velocity analysis of the exhaust system

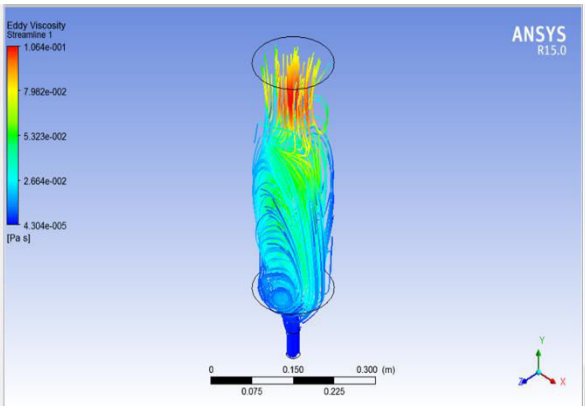
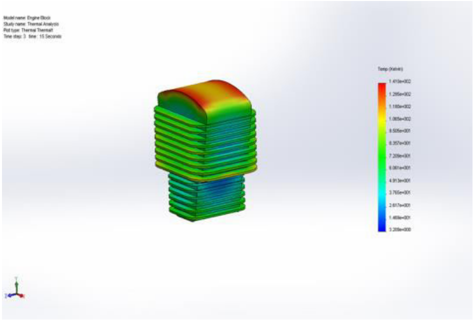


Figure 5.7 shows the Eddy Viscosity flow of the gases inside the exhaust system

5.2 Analysis of engine block



PARAMETERS

No. of nodes = 28644

No. of elements = 17188

No. of DOF = 28644

Total solution time = 00:01:24

5.3 THERMAL ANALYSIS OF ENGINE BLOCK

Aluminium Alloy 6061 Tap Water Without Coolant

Fig 5.3.1Tap Water without Coolant

Meshed Model Generate

Mesh Meshing is done by using size controls command of lines, the line of specimen is divided to get a good mesh. After that mesh area is selected as shown in Fig [Mesh] < [Pick All] < [Close] Warning.

Fig 5.3.2 After Apply Mesh

Apply Loads Loads – Define Loads – Apply – Thermal – Temperature Temperature –558K Loads – define Loads – Apply – Thermal – Convection – on areas Bulk Temperature – 313K

Film Coefficient – 0.005W/mm2K

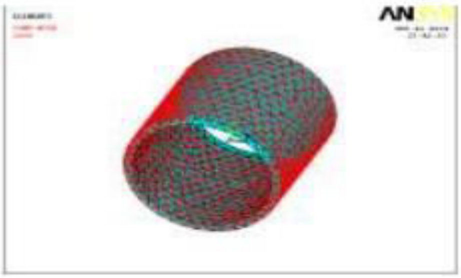


Fig 5.3.3 after Apply Loads

Solution Solution – Solve – Current LS – ok Post Processor General Post Processor – Plot Results – Contour Plot - Nodal Solution – DOF Solution – Nodal Temperature Vector sum

CONCLUSION

The decrease in back pressure is shown using contour and Pressure Vs Distance diagram. The flow is made efficient by decreasing the exhaust gas back pressure in the exhaust system design. The geometry, which gives minimum pressure drop and hence minimum backpressure, is the optimized geometry. Again for the optimized geometry ANSYS results would be finding out. Also the Non-dimensional stiffness rigidity is sufficiently high. Finally it can be stated that 3-D ANSYS simulation can be used as a strong and useful tool for design or optimization of Exhaust system.

In the present study, a series of experimental investigations have been conducted to explore the performance, combustion and emission characteristics with optimization of engine operation using diesel, thumba oil, thumba biodiesel and their specified blends with diesel fuel in direct injection single cylinder variable compression ratio multi fuel diesel engine. The present effort has contributed mainly in the following aspects

REFERENCE

[1] V. Ganeshan, ―Internal Combustion Engines", Tata McGraw-Hill. 2006.

[2] M. Safari & m. Ghamari, A. Nasirtosi, " Intake Manifold Optimization by using 3-D CFD Analysis", SAE paper 2003-32-0073.

[3] MedaLakshmikantha& Mathias Keck, " Optimization of Exhaust Systems ", SAE paper 2002-01-0059

[4] John D. Anderson, Jr., "Computational fluid dynamics - The basic with applications", McGraw - Hill international Editions 1995.

[5] Ansys ICEM CFD/ AI.Environment Help Manual 13.0

[6] H. K. Versteeg& W. Malasekera, " An introduction to Computational fluid dynamics - The finite volume method ―, Long man Scientific & Technical, Long man Group Ltd. 1995.

[7] David Tueddell, Clayton Sloss& Thomas Werner, " An Advanced CFD simulation strategy for Exhaust Manifolds with Close - Coupled Catalytic converters‖, SAE paper 2005-01-1922.

[8] PrabirDatta, Richard Liddy, " Correlation Study of Exhaust Manifolds - Lab Test Results Vs Customer fleet Results, SAE paper 2002-01-1317.

[9] Sophie Salasc, EdouardBarrieu& Vincent Leroy, " Impact of Manifold design on Flow distribution of a close - coupled catalytic converter, SAE paper - 2005-01-1626.

[10] Han-Scng Kim, Kyoung - Min Cho, " A study of flow characteristic Inside the two types of Exhaust Manifold & CCC Systems", SAE paper 1999-01-0457.