International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Design and Analysis of Intake Manifold using Ansys Fluent

Lakhan Buke[1], Prof. Aditya Pratap Singh[2]

[1]*PGStudent, Department of Mechanical Engineering, Sagar Institute of Research & Technology Indore, India*

[2]*Professor, Department of Mechanical Engineering, Sagar Institute of Research & Technology Indore, India*

ABSTRACT:-

This study focuses on the design and analysis of an intake manifold using ANSYS Fluent. The intake manifold plays a crucial role in optimizing engine performance by delivering air to the combustion chambers. Utilizing computational fluid dynamics (CFD) capabilities, a 3D CAD model is created and imported into ANSYS Fluent for mesh generation. The computational model is set up with appropriate material properties and boundary conditions. Numerical simulations are performed to analyze flow behavior, velocity profiles, pressure distribution, and mass flow rate. This process enables engineers to optimize intake manifold designs before physical prototyping, leading to more efficient and effective designs for internal combustion engines.

Keywords: Intake manifold, ANSYS Fluent, Computational Fluid Dynamics (CFD), CAD model, Mesh generation, Flow analysis, Performance optimization, Internal Combustion Engine.

Introduction:-

The intake manifold is a critical component in internal combustion engines, responsible for distributing the air-fuel mixture to the individual cylinders for combustion. Optimizing the design of the intake manifold is crucial for enhancing engine performance, including factors such as airflow distribution, pressure drop, and volumetric efficiency.

To improve intake manifold designs, engineers often utilize computational fluid dynamics (CFD) analysis with tools like ANSYS Fluent. CFD allows for a detailed understanding of the flow behavior within the manifold, providing valuable insights into velocity profiles, pressure distribution, and other relevant parameters.

By employing ANSYS Fluent, engineers can simulate and analyze various intake manifold designs prior to physical prototyping and testing. This approach offers cost and time savings while enabling performance optimization. The ability to visualize and quantify flow characteristics aids in identifying design improvements to enhance airflow uniformity, reduce pressure drop, and improve volumetric efficiency.

The objective of this study is to design and analyze an intake manifold using ANSYS Fluent. Through CFD techniques, we aim to gain insights into the flow behavior and performance of different intake manifold configurations. Analyzing velocity profiles, pressure distribution, and mass flow rate allows us to optimize the design and improve engine performance.

The use of ANSYS Fluent for intake manifold analysis provides engineers with a powerful tool to evaluate, refine, and validate designs virtually. This approach contributes to the development of more efficient and effective intake manifold designs, ultimately leading to enhanced engine performance.

Literature Survey:-

Dilip Kumar Sonar, and et.al [1]Explain that, The processes that cause damage stem from a wide variety of sources, the most common of which are wear, temperature, and fatigue. Both thermal fatigue and mechanical fatigue, which may occur at ambient temperature or at elevated temperatures, are important contributors to fatigue failures. In this assignment, we use CATIA V5R20 to design a piston. The final blueprint is put into ANSYS 14.5 for analysis.

P.Viswabharathy [2]Describes how to use Hyper mesh's optistruct model to fine-tune the piston model. AUTOCAD is utilised to produce the CAD model, and hyper mesh is used to mesh the final product. The findings demonstrate that the piston's increased mass is well within acceptable design parameters, since the revised layout results in lower fuel usage.

Gopal et al. [3]looked at how a 4-wheeler's petrol engine's piston, connecting rod, and crankshaft work in detail. The assembly's ingredients should be rigid, and it should function as a transport mechanism. It was recommended to switch out the meeting's additives with a fresh batch of materials and put them through their paces in terms of static, dynamic, and thermal assessment in order to see how well they hold up. The primary components of the engine (piston, connecting rod, and crankshaft) were modelled and constructed in ANSYS for FEA in accordance with the specified design. Hyper Mesh was used to complete the meshing.

Krishnan et al. [4]lightweight materials, such as high-performance ultra-high tensile steels, aluminium and magnesium alloys, polymers, and carbon-fiber reinforced composites, have been explored approximatively. Here, the piston's longevity is increased by using a newly developed composite matrix of aluminium with particulates of silicon carbide, which has the highest wear factor and the same normal performance apart from a slight variation in properties. Aluminium and silicon carbide, in a 2:3 ratio, are used in the piston's design and analysis. Using the 3D modelling software Autodesk Inventor, a parametric model of a piston is created.

Objective:-

The objective of this study is to address the above challenges and achieve the following:

- Develop a systematic approach for the design and analysis of intake manifolds using ANSYS Fluent.
- Gain a comprehensive understanding of the flow behavior within the intake manifold by accurately modeling the complex geometry and boundary conditions.
- Optimize the intake manifold design to improve performance metrics, such as airflow distribution, pressure drop, and volumetric efficiency.
- Validate the computational model by comparing simulation results with experimental data or established benchmarks.
- Provide engineers with a user-friendly and efficient workflow to explore different intake manifold configurations and assess their impact on engine performance.

Design of Intake Manifold:-

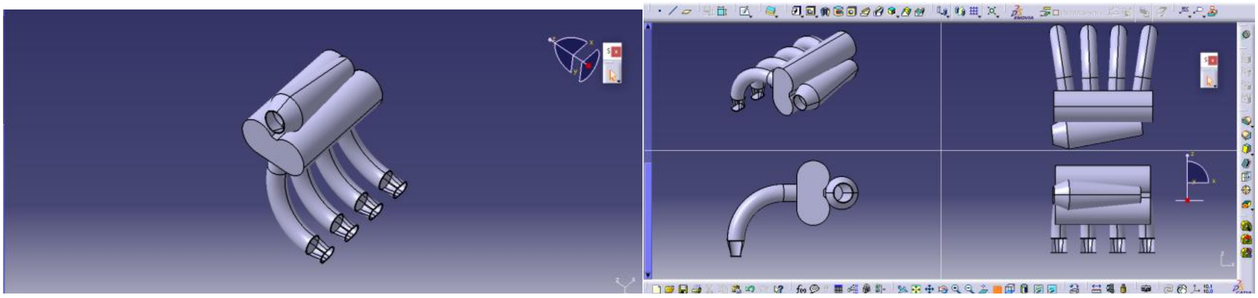

Figure 1: Intake manifold type-1

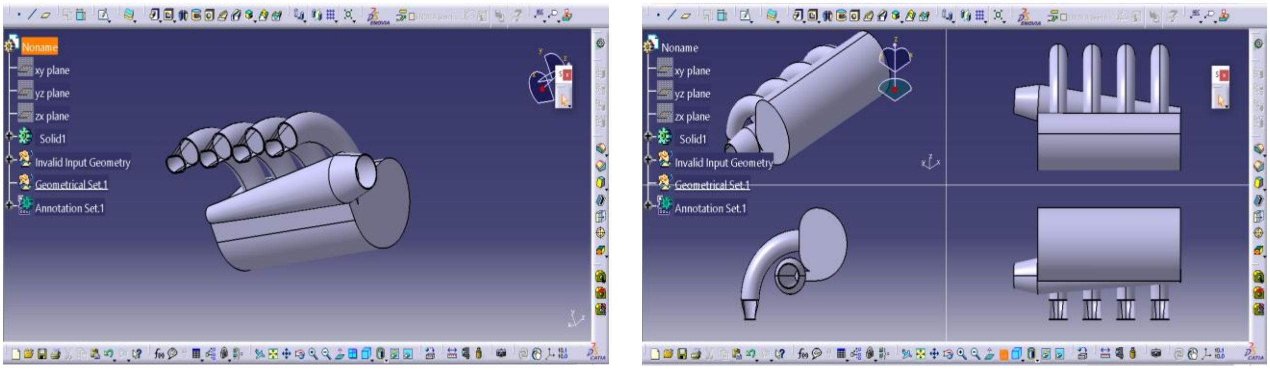

Figure 2: Intake manifold type-2

CFD analysis result of intake manifold:-

Meshing of Type-1 intake manifold

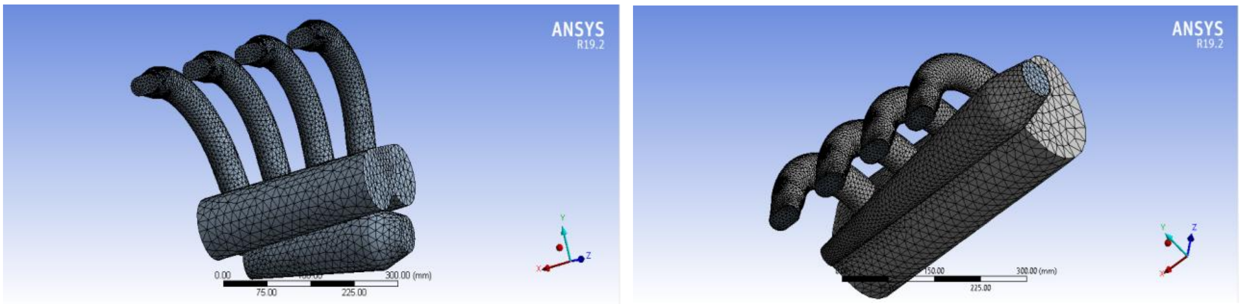

Figure 2: Meshing of intake manifold Type-1 & Type-2

Table 1

Name	Type-1	Type-2
Node	29480	144128
Element	53247	222803

Analysis of Type-1 intake manifold

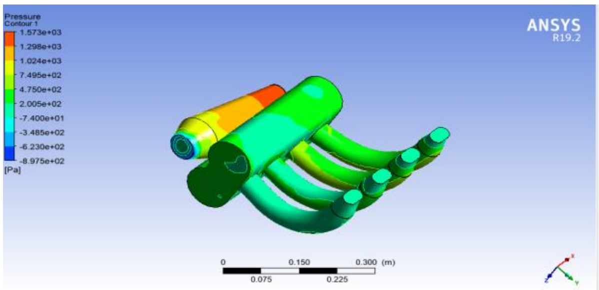

Figure 3: Velocity Contour for type-1 Intake manifold

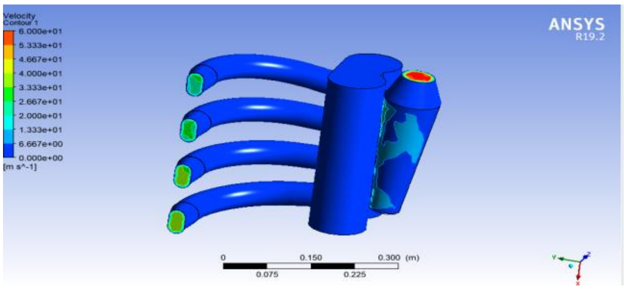

Figure 4: Pressure contour of type-1 manifold

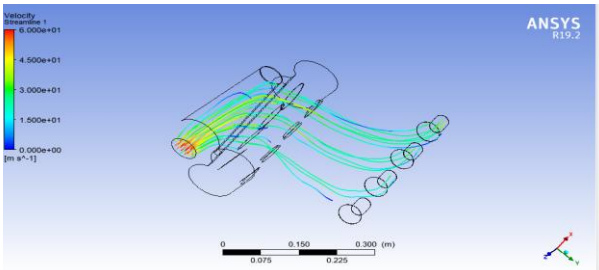

Figure 5: Streamline velocity of type-1 intake manifold

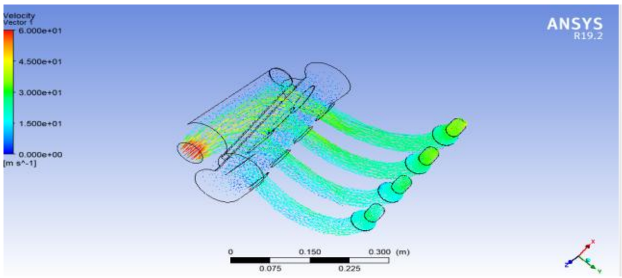

Figure 6: Velocity vector of intake manifold

Analysis of Type-1 intake manifold

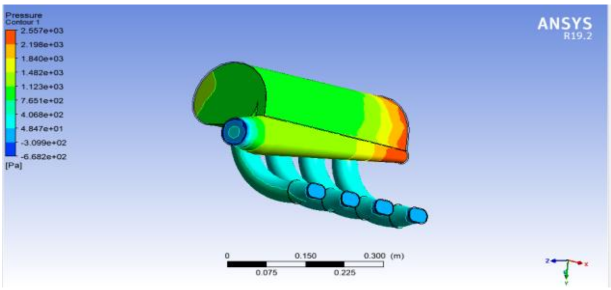

Figure 7: Pressure Contour for type-1 Intake manifold

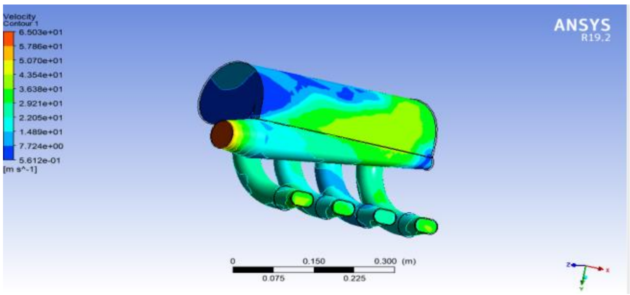

Figure 8: Pressure contour of type-1 manifold

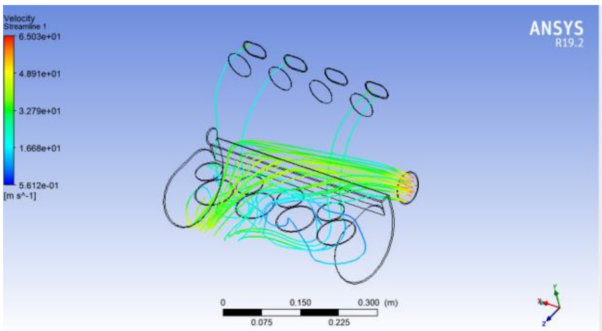

Figure 9: Streamline velocity of type-1 intake manifold

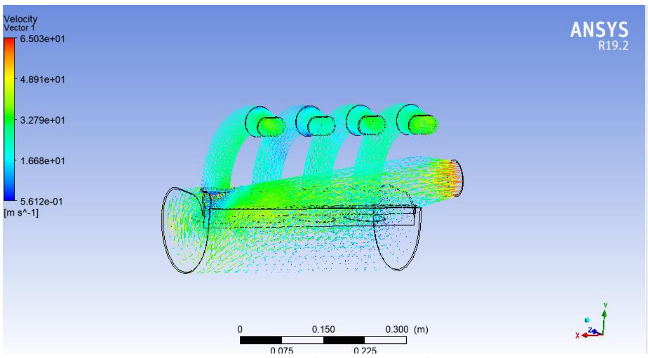

Figure 10: Velocity vector of intake manifold type-2

Conclusion:-

The analysis of the intake manifold using ANSYS Fluent yielded important insights into flow behavior and performance. Key findings encompassed velocity profiles, pressure distribution, and turbulence characteristics. Performance metrics such as airflow distribution, pressure drop, and volumetric efficiency were evaluated. Here we have two type of intake manifold plenum design type-1 and type-2 we put into analysis of both the profile and compare their results we found that type-1 manifold shows better result than type-2 manifold.

Comparative analysis assisted in identifying optimal designs. Validation against benchmarks ensured accuracy, while acknowledging limitations guided future research. These results deepen our understanding of the impact of intake manifold design on engine performance, facilitating design improvements and optimization.

References:-

[1] Ryan I., Christopher B. W., "Design and Manufacture of a Formula SAE Intake System Using Fused Deposition Modeling and FiberReinforced Composite Rapid Prototyping", Journal, Vol. 16 Is: 3, pp.174 – 179.

[2]Karthikeyan S, Hariganesh R, Sathyanadan M, Krishnan S, ― Computational Analysis Of Intake Manifold Design And Experimental Investigation On Diesel Engine For LCV‖, ISSN: 0975-5462, vol. 3 no. 4, (March 2011).

[3] Benny Paul, Ganesan V, ―Flow Field Development In a Direct Injection Diesel Engine With Different Manifolds‖, IJEST: Vol. 2, No. 1, pp. 80-91, (2010).

[4]Jemni M .A, Kantchev G, Abid M. S, ―Intake Manifold Design Effect On Air Fuel Mixing And Flow For An LPG Heavy Duty Engine‖, IJEE:Vol. 3, Issue 1, pp.61-72, , (2011).

[5] Heavy Duty Engines – MTZ 6/2005. 4. Jacobs, T., Chatterjee, S., Conway, R., Walker, A., Kramer, J. and K. Mueller-Haas, Development of a PartialFilter Technology for Hdd Retrofit, Sae Technical Paper 2006-01-0213.

[6] C. Lahousse, B. Kern, H. Hadrane and L. Faillon, "Backpressure Characteristics of Modern Three-way Catalysts",Benefit on Engine Performance, SAE Paper No. 2006011062,2006 SAE World Congress, Detroit, Michigan ,April 36, 2006

[7] M.B. Beardsley et al.,Thermal Barrier Coatings for Low Emission, High Efficiency Diesel Engine Applications" SAE Technical Paper 1999-01-2255.

[8] Rajesh Biniwale , N.K. Labhsetwar, R. Kumar and M. Z. Hasan, "A non-noble metal based catalytic converter fortwostrokes, two-wheeler applications", SAE Paper No. 2001011303, 2001. [7] G. Muramatsu, A. Abe, M. Furuyama, "Catalytic Reduction of Nox in Diesel Exhaust", SAE 930135, 1993.

[9] Automobile Engines - In Theory, Design, Construction, Operation, Testing and Maintenance by A.W. Judge

[10] Hartman, Jeff, "How to Tune and Modify Engine Management System",2003