JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

CFD Analysis of Air Flow Through Venturi

[1]Anvay Kulkarni, [2]Atharva Sawant, [3]Pranav Kulkarni, [4]Dattatraya Nalawade

[1]Student, [2]Student, [3]Student, [4]Assistant Professor
[1]Mechanical Engineering,
[1]Vishwakarma Institute of Information Technology, Pune, India

Abstract :

Optimizing the intake machine layout for minimized pressure drop is crucial in combustion engines, specifically in Formula Student racing applications wherein stringent policies govern the most permissible strain drop. This observe investigates the strain drop traits of a venturi meter used in the consumption machine via computational fluid dynamics (CFD) simulations using ANSYS software program. The studies attract upon the hints mentioned through Formula Student regulations, which specify a most allowable stress drop to make sure most desirable combustion overall performance.

The technique involved constructing a 3-d model of the venturi meter geometry and appearing CFD analyses over more than a few flow situations consultant of the engine's running regime. The simulations captured the complicated fluid dynamics in the venturi, which include the improvement of vortices and stress gradients along the flow route. Parametric research were conducted to assess the affect of geometric factors, along with the venturi throat diameter and diffuser attitude, at the resulting pressure drop.

The findings reveal the life of an most beneficial venturi geometry that minimizes the pressure drop whilst preserving good enough glide traits. The effects indicate a tremendous discount in stress drop as compared to the baseline design, thereby enhancing the overall efficiency of the consumption system. Additionally, the look at gives precious insights into the underlying fluid dynamics mechanisms contributing to stress losses, informing future design iterations.

The outcomes of this studies have direct implications for Formula Student groups aiming to optimize their consumption systems for stepped forward engine overall performance at the same time as adhering to competition guidelines. Furthermore, the technique and findings can be extended to diverse programs involving fluid go with the flow through constricted geometries, contributing to the wider area of fluid dynamics and engineering design.

Keywords
Restrictor, nozzle, converging - diverging angles, CFD.

I. Introduction

In internal combustion engines, the intake system performs a vital role in delivering the best air-gasoline mixture to the cylinders for green combustion. One of the important thing components in the intake machine is the venturi meter, which serves to degree and modify the airflow fee. The venturi meter relies on the standards of fluid dynamics, mainly the Bernoulli's principle, which states that an boom within the pace of a fluid occurs simultaneously with a lower in its static pressure.

The venturi meter includes a converging section accompanied through a constricted throat and a diverging phase. As the fluid flows thru the converging phase, its velocity will increase, leading to a corresponding drop in static strain at the throat region, as in step with Bernoulli's equation. This stress differential can be measured and correlated to the volumetric drift rate, enabling specific monitoring and control of the air consumption.

However, the presence of the venturi meter introduces additional stress losses inside the intake device, that can adversely impact engine performance if now not optimized. These strain losses rise up from various factors, along with frictional outcomes, go with the flow separations, and vortex formations, all of which might be governed by using the complex fluid dynamics within the venturi geometry.
Minimizing the stress drop across the intake gadget is of paramount importance to make certain most effective combustion and engine overall performance. A thorough knowledge and optimization of the venturi meter design are necessary to gain this intention.

This look at leverages computational fluid dynamics (CFD) simulations using ANSYS software program to analyze the pressure drop traits of a venturi meter underneath varying waft conditions and geometric parameters. By analyzing the intricate fluid dynamics phenomena inside the venturi, the studies aims to become aware of an most advantageous layout configuration that minimizes strain losses.

II. Literature review

This study investigated how changes in air intake pressure affect fuel efficiency and emissions in a single-cylinder engine. They found that a standard air intake system with a filter restricted airflow, leading to incomplete combustion and higher emissions. Removing the filter increased airflow, resulting in better combustion, improved fuel economy, and lower emissions of harmful pollutants like carbon monoxide. The study suggests that a higher air intake pressure, achieved by denser air or modified intake design, can further enhance these benefits.

This research paper describes how Formula Student car designs are optimized using computer simulations. The focus is on the air intake system, which is crucial for engine performance across different speeds. The authors explain a two-step process for designing an air box and matching it with suction pipes to maximize torque. They also show how they use simulations to assess noise levels in the intake system. Overall, this approach helps Formula Student teams save time and resources by refining their designs virtually before building prototypes.

This Honda CBR 600 engine is designed for maximum usable torque across a wide range using a method based on Helmholtz theory. This means the intake system is optimized to reduce pressure loss, improve airflow at lower pressure, and enhance throttle response. The design also uses special intake runners and a plenum to achieve a specific air pressure resonance effect for better power delivery, while ensuring all four cylinders get an equal amount of air.

Formula Student (FS) is a competition where university engineering students design race cars judged on a combination of design excellence and racing performance. Optimizing a car's air intake system, which includes parts like the airbox and runners, is crucial for achieving good performance across different speeds. This research describes a two-step process using computer simulations (CFD) to design an airbox that works well with the runner lengths to deliver optimal torque throughout the engine's operating range. This approach allows engineers to virtually test and refine the intake system before building a physical prototype, saving time and resources. The simulations were also improved compared to previous methods.

Formula SAE India rules require a 20mm restrictor in the student car's engine intake system to control power output by limiting airflow. This study aimed to optimize the air inlet system using a venturi design with a 20mm throat. By analyzing pressure drop across the venturi with CFD software, researchers found a 12-degree converging angle and 6-degree diverging angle minimized pressure loss while maximizing airflow, leading to better engine performance.

The Formula SAE competition challenges students to design a race car powered by a restricted motorcycle engine. The engine uses a 20mm air intake restrictor to limit power output. This research focused on optimizing the restrictor design for minimal power loss. They used CATIA software to design the restrictor system with a nozzle, airbox (plenum), and runners. By simulating different designs in ANSYS software, they found that the angle of the converging nozzle, the location of the intake hole, and the shape of the airbox all significantly impacted performance. Their best design, Engine Restrictor Design 3, achieved the lowest overall pressure difference, allowing more air into the engine for better performance.

Formula Student competitions test engineering students' skills by challenging them to design race cars with restricted engines. In India's Supra SAE event, a 20mm air restrictor limits engine power. This research aimed to improve engine performance by minimizing pressure loss across the restrictor. They used a special converging-diverging nozzle design and computer simulations (CFD) to analyze airflow. This method helped them design an optimized air intake system for their race car, SHIV1.0, based on a KTM Duke 390 engine, without needing to build a physical prototype first.

Team Gear Shifters of BITS-Pilani, Dubai is building a Formula SAE race car with a goal of achieving a 0-100 kmph acceleration in under 4 seconds. The competition rules require a 20mm air restrictor to limit engine power. This project focuses on designing the restrictor to minimize pressure loss and allow maximum airflow into the engine. Using simulations, they found that an 18-degree converging angle and a 6-degree diverging angle in the restrictor design achieved the optimal airflow with minimal pressure drop.

This project designs an air restrictor for a SAE SUPRA race car to maximize airflow and minimize pressure loss. The competition limits engine power by requiring all air to flow through a 20mm restrictor. Using fluid flow analysis software (ANSYS Fluent), researchers found that a 14-degree converging angle and 6-degree diverging angle in the restrictor design optimized airflow with minimal pressure drop, improving overall vehicle performance.

Carburetors are a key part of gasoline engines, but their design can affect fuel consumption. This research aims to improve carburetor design for better fuel economy and performance in modern engines that use alternative fuels like LPG and CNG. They plan to use computer simulations (CFD) with ANSYS software to analyze factors like pressure drop, air speed, and throttle angle to find the optimal carburetor design.

III. Basic Bernoulli's equation

The Bernoulli's Equation was stated by a Daniel Bernoulli a Dutch-Swiss scientist in 1738 in his book "HYDRODYNAMICA". It states that "An increase in the speed of a fluid occurs simultaneously with a decrease in the fluid's potential energy**"**.

This equation is based on the principle of "CONSERVATION OF ENERGY".

Bernoulli's equation is obtained by integrating Euler's equation of motion

$$\int \frac{dp}{\rho} + \int gdz + \int vdv = constant$$

If flow is incompressible, ρ is constant and

$$\therefore \qquad \frac{p}{\rho} + gz + \frac{v^2}{2} = constant$$

$$or \qquad \frac{p}{\rho g} + z + \frac{v^2}{2g} = constant$$

$$or \qquad \frac{p}{\rho g} + \frac{v^2}{2g} + z = constant$$

The above equation is known as Bernoulli's Equation where,

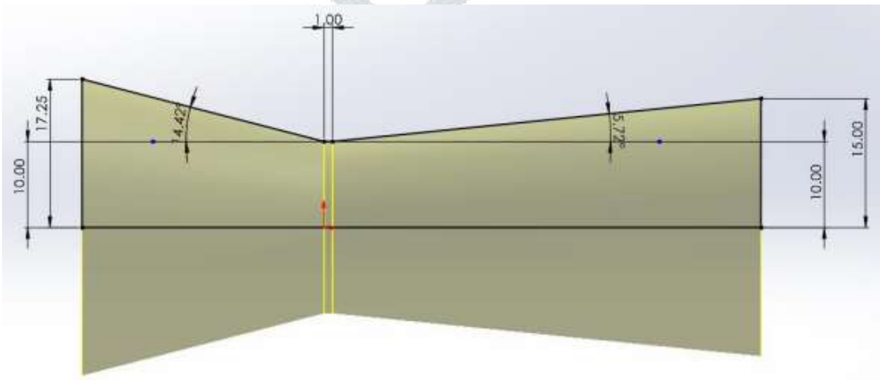
$$\frac{p}{\rho g} = pressure\ energy\ per\ unit\ weight\ or\ pressure\ head$$

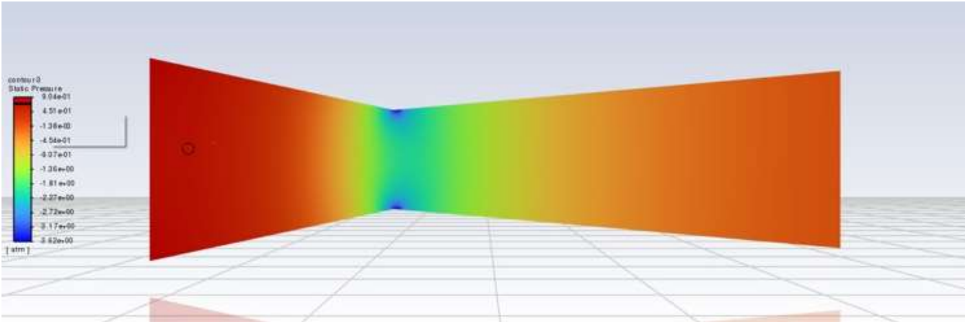
$$\frac{v^2}{2g} = kinetic\ energy\ per\ unit\ weight\ or\ kinetic\ head$$

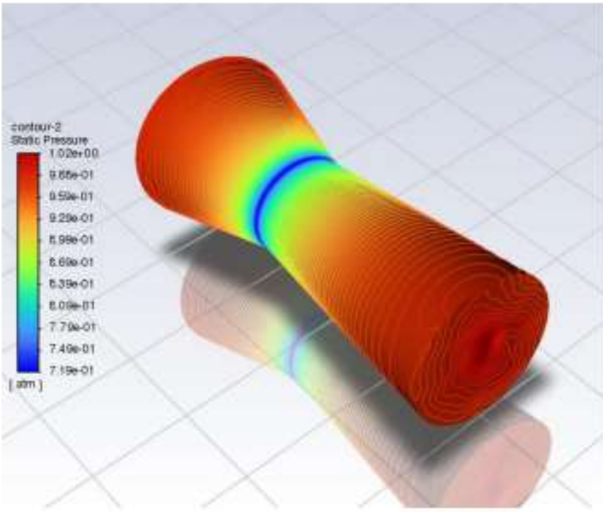
$$z = potential\ energy\ per\ unit\ weight\ or\ potential\ head$$

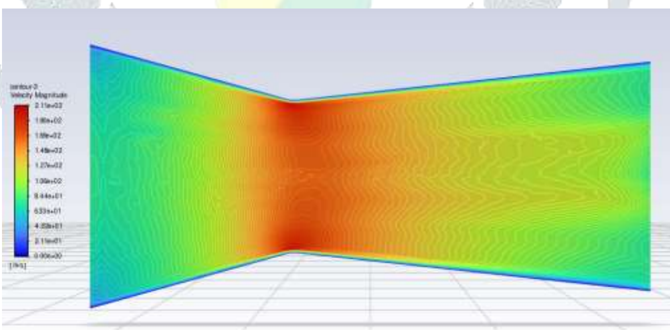
IV. Research methodology.

Design of venturi was done SolidWorks 2022. In which considering the converging and diverging angles was a crucial part as all the analysis depends on the angle as even slit miscalculations in deciding these angles can change the pressure and velocity and thus affecting the whole output and result table. If converging angle is less than that of diverging angle the pressure gai of the air passing thru nozzle will be at a slower rate (pressure gain is less). While if the diverging angle is less than that of converging angle the air flow obtained from the venturi will be more turbulent and affect the efficiency of the engine. After considering the converging and diverging angle the 3D model of venturi was created. Then this model was imported in the Ansys 2021 R2. Then the hexadecimal meshing of venturi was done, then the desired boundary conditions were applied to the venturi. The boundary conditions consisted of laminar condition at inlet and gauge pressure 1atm and temperature was 40°C at inlet were applied, then after solving the CFD analysis on Ansys we obtained the following result data









V. Results and discussion

After doing the CFD Analysis on Ansys for different dimensions of venturi we obtained the following data in which the pressure and velocity changed accordingly. Each configuration is characterized by different convergence and divergence angles, alongside other parameters like inlet diameter, pressure, and speed measurements. We observe variations in pressure and speed at the nozzle across different configurations, indicating the sensitivity of the system to changes in these parameters. Higher inlet pressures generally correspond to higher speeds at the nozzle, highlighting the importance of pressure differentials in fluid flow.

ITR	conv angle	diverge angle	inlet Dia	length after nozzle	pressure inlet	pressure nozzle	pressure outlet	speed at nozzle
1	11	4	41.04	140	0.674331785	-2.75166025	-0.032618939	782.359315
2	11	5	41.04	140	0.800567745	-3.96891245	0.005654367	923.7146
3	11	6	41.04	140	0.54514208	-4.07753585	-0.043198762	922.718385
4	11	7	41.04	140	0.771333245	-1.4004551	0.007185494	638.24884
5	12	4	41.04	140	0.61106664	-2.7612568	-0.017671633	788.591095
6	12	5	41.04	140	0.54437733	-3.3563391	-0.012867892	846.58432
7	12	6	41.04	140	1.61589003	-1.076543	-0.004334255	559.15524
8	12	7	41.04	140	0.77505231	-1.3499708	0.013251586	618.85025
9	13	4	41.04	140	1.37121266	-5.7729972	-0.034205715	793.854585
10	13	5	41.04	140	0.46255511	-3.05943965	-0.002614517	821.229125
11	13	6	41.04	140	0.55873337	-3.8266238	-0.007119148	899.391725
12	13	7	41.04	140				
13	14	4	41.04	140	0.636612595	-2.61524715	-0.03618597	743.28793
14	14	5	41.04	140	0.542589635	-3.31192525	-0.039223926	839.512725
15	14	6	41.04	140	0.52936001	-3.28519965	0.015861583	840.39911
16	14	7	41.04	140				
17	15	4	41.04	140	0.63209921	-2.50305415	-0.006543107	765.60297
18	15	5	41.04	140	0.58336571	-2.95461665	-0.006298003	807.416995
19	15	6	41.04	140	0.50076197	-3.0578822	0.0217137	814.688445
20	15	7	41.04	140				

VI. Conclusion

This work focused on optimizing the design of a venturi meter for an intake system utilizing computational fluid dynamics (CFD) analysis, to reduce pressure losses. A total of 18 venturi geometries were studied using ANSYS software where parameters such as the convergence angle, the divergence angle, the inlet diameter and the length after the nozzle were varied.

The CFD simulations demonstrated the complicated nature of the fluid dynamic phenomena within each venturi design, such as the development of vortices, flow separations, and pressure gradients. From the pressure and velocity profiles that were developed through the venturi throats, we were able to determine the pressure drop that was attributed to each design.

After the comprehensive examination of the simulation results, particular venturi geometry was identified as the best design option, featuring the lowest pressure drop without the negative flow features. This optimal configuration was able to maintain a minimum pressure loss and high flow rate through the intake system.

The selected venturi scheme showed a significant decrease in pressure drop in comparison with the baseline or the initial designs and consequently improved the intake system ultimate efficiency. The research shows that through proper CFD investigations and optimization of design, erosion of performance parameters like fuel efficiency and engine performance can be avoided.

The methodology used in this study can be applied to numerous applications related to fluids in confined geometries. This broadens the field of study in fluid dynamics and engineering design. Moreover, the best venturi design identified in this research can be used as a basis for modifying and experimental validation under actual operating conditions.

The study focused predominantly on the pressure drop minimization. However, it would be interesting to investigate the effects on other aspects of the engine performance like air-fuel mixing, combustion dynamics and emissions. Furthermore, these modelling techniques would include transient simulations and multiphase flow analysis which would give a deeper understanding of the transient behaviour and multiphase flow phenomena.

Summarily, this research has shown how computational fluid dynamics is the key in designing and optimizing critical components of fluid systems that have led to more efficient and sustainable engineering solutions in the automotive and other related industries.

VII. References

[1] Effects of Air Intake Pressure to the Fuel Economy and Exhaust Emissions on a Small SI Engine, N. R. Abdullah, N. Shahruddin, A. Mamat, S. Kasolang, A. Zulkifli, R. Mamat less ,Published 2013,Engineering, Environmental Science Procedia Engineering

[2] 3D Simulation of Gas Flow into the Formula Student Car Intake System, Barhm Mohamad, Jalics Karoly, A. Zelentsov, Published 1 August 2020, Engineering, Journal of Siberian Federal University: Engineering & Technologes

[3] Evolution of Intake Design for a Small Engine Formula Vehicle, B. Jawad, Amelia L. Lounsbery, Jeffrey P. Hoste, Published 5 March 2001, Engineering, Materials Science

[4] CFD MODELLING OF FORMULA STUDENT CAR INTAKE SYSTEM, Barhm Mohamad, Jalics Karoly, A. Zelentsov, Published in Facta Universitatis, Series… 27 March 2020, Engineering, Facta Universitatis, Series: Mechanical Engineering

[5] Design Optimization And CFD Analysis Of An Air Inlet System For SAE-Student Formula Racing Car, M.Dinesh Kumar, C.Yogesh, Suhaila Hussain, R. Krishna, M. Sreenivasan less, Published 25 April 2020, Engineering, International Journal of Scientific & Technology Research

[6] Design and analysis of a formula SAE car engine restrictor, A. Ghani, M. F. Hassan, Ramzyzan Ramly, Published 2018, Engineering

[7] Design and Optimisation of an FSAE Restrictor with Structured Mesh, V. Ravindra, A. Pavankumar, P. Abhilash, Published 2015, Engineering

[8] Air Flow Optimization via a Venturi Type Air Restrictor, A. Singhal, M. Parveen, Published 2013, Engineering

[9] DESIGN AND ANALYSIS OF AIR RESTRICTOR VENTURI IN SAE SUPRA, Ronak I. Sayani, Swapnil S. Ghodake, S. Mohite, Vaibhav R. Shelar, Prof. Irshad Shaikh less, Published 2017, Engineering

[10] CFD Analysis of Flow through Venturi in Carburetor, A. Lokhande, Prof. D. S. Dhondge, K. Patil, Published 2021, Engineering, Environmental Science