



IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

“A COMPREHENSIVE REVIEW OF FLOW BEHAVIOUR AND PARAMETRIC OPTIMIZATION IN CARBURETOR VENTURI”

Dipesh Parashar ¹, Dr Roopesh Tiwari ²

¹ PG scholar, Department of Mechanical Engineering, SAGE University, Indore, Madhya Pradesh, India

² Professor, Department of Mechanical Engineering, SAGE University, Indore, Madhya Pradesh, India

ABSTRACT

Carburetors play an important role in supplying the required air-fuel mixture to spark ignition engines. The Venturi is a critical component of a carburetor because it creates a pressure difference that helps in drawing fuel into the incoming air stream. The flow behaviour inside the Venturi is influenced by several parameters, including Venturi geometry, throat diameter, inlet and outlet dimensions, nozzle angle, throttle position, air velocity, and pressure distribution. Improper selection of these parameters can result in non-uniform flow, pressure losses, turbulence, and poor fuel-air mixing, which may affect engine performance and fuel efficiency. This review paper presents a comprehensive study of flow behaviour and parametric optimization in carburetor Venturi systems. Previous experimental, numerical, and CFD-based investigations related to Venturi flow, pressure variation, velocity distribution, turbulence, fuel entrainment, and geometric optimization are reviewed and discussed. The influence of throttle opening, Venturi dimensions, nozzle configuration, and operating conditions on flow characteristics is also examined. Special attention is given to CFD techniques used for predicting pressure, velocity, and flow patterns inside carburetor Venturi passages. The review identifies important design parameters and summarizes their effects on Venturi performance. Based on the reviewed studies, suitable geometric and operating parameters can improve airflow, fuel mixing, pressure recovery, and overall carburetor performance. The study also highlights existing research gaps and provides directions for future CFD-based parametric optimization of carburetor Venturi systems.

Key Words: Carburetor Venturi, Flow Behaviour, Parametric Optimization, CFD Analysis, Air-Fuel Mixture, Pressure Distribution, Velocity Distribution, Throttle Opening, Venturi Geometry, Fuel Mixing, Turbulence, Engine Performance.

I. INTRODUCTION

A carburetor is an important device used in conventional spark ignition engines to prepare and supply the required air-fuel mixture to the engine. The quality of the air-fuel mixture has a direct effect on combustion, engine performance, fuel consumption, and exhaust emissions. For proper engine operation, the carburetor must provide an appropriate quantity of fuel mixed with air under different operating conditions. Therefore, understanding and improving the flow characteristics inside a carburetor is an important area of research. The Venturi is one of the most important parts of a carburetor. It consists of a converging section, a narrow throat, and a diverging section. As air passes through the converging passage, its velocity increases and its static pressure decreases. The pressure difference created at the throat helps draw fuel through the fuel nozzle into the air stream. The resulting air-fuel mixture then passes through the outlet section and enters the engine. Hence, the flow behaviour inside the Venturi strongly affects the fuel supply and mixing process. The performance of a carburetor Venturi depends on several geometric and operating parameters. Important geometric parameters include inlet diameter, throat diameter, throat length, outlet diameter, nozzle position, nozzle

<https://www.ijrtsm.com> © International Journal of Recent Technology Science & Management

angle, and the shape of the converging and diverging sections. Operating parameters such as throttle opening, air velocity, pressure, temperature, and fuel properties also influence the flow characteristics. A suitable combination of these parameters is required to obtain stable airflow and effective fuel mixing. Improper Venturi design may produce excessive pressure losses, flow separation, turbulence, and non-uniform velocity distribution. These effects can reduce the quality of the air-fuel mixture and may lead to incomplete combustion, increased fuel consumption, and higher emissions. Therefore, optimization of Venturi geometry and operating conditions is important for improving the overall performance of a carburetor. Experimental investigation of internal carburetor flow can be difficult because the flow passage is relatively small and the flow characteristics change continuously with throttle position and engine operating conditions. Computational Fluid Dynamics (CFD) provides an effective method for studying these flow characteristics. CFD can be used to predict pressure distribution, velocity distribution, turbulence, flow separation, and other important parameters without requiring extensive physical testing. Several researchers have investigated carburetor and Venturi flow using experimental and numerical techniques. Their studies have examined the effects of throttle opening, Venturi dimensions, nozzle configuration, and different operating conditions on airflow and fuel mixing. These investigations show that changes in small geometric parameters can significantly influence the pressure and velocity fields inside the Venturi. Parametric optimization provides a systematic approach for identifying suitable Venturi dimensions and operating conditions. By studying different combinations of parameters, the flow characteristics can be compared and improved. CFD-based optimization can also reduce development time and experimental cost while providing detailed information about the internal flow field. Therefore, the present review focuses on the flow behaviour and parametric optimization of carburetor Venturi systems. The review discusses the basic working principle of the Venturi, important geometric and operating parameters, pressure and velocity characteristics, turbulence and flow separation, fuel-air mixing, and the application of CFD techniques. Previous research findings are critically reviewed to identify important design parameters, research gaps, and future possibilities for improving carburetor Venturi performance.

II. PROBLEM IDENTIFICATION

The flow behaviour inside a carburetor Venturi is influenced by several geometric and operating parameters, making it difficult to obtain optimum flow conditions under all engine operating conditions. Changes in air velocity, pressure, throttle opening, and fuel flow can significantly affect the pressure and velocity distribution inside the Venturi. One of the major problems is the non-uniform flow distribution through the Venturi passage. Improper selection of inlet, throat, and outlet dimensions may cause excessive pressure losses, turbulence, and flow separation. These conditions can reduce the effectiveness of the Venturi and affect the formation of the required air-fuel mixture. The throat region is particularly important because the maximum air velocity and minimum pressure generally occur in this region. The throat diameter and throat length must therefore be properly selected. An unsuitable throat geometry may result in insufficient pressure drop or excessive flow restriction, affecting fuel induction and engine performance. The fuel nozzle is another important parameter affecting Venturi performance. Its position, diameter, and angle influence fuel discharge and mixing with the incoming air. An unsuitable nozzle angle may produce non-uniform fuel distribution and poor air-fuel mixing, which can negatively affect combustion and fuel efficiency. Throttle opening also produces significant changes in the flow field. Different throttle positions alter the airflow rate, velocity, and pressure distribution inside the carburetor. Therefore, a Venturi design that performs satisfactorily at one throttle position may not provide the same flow characteristics at other operating conditions. Experimental investigation of the internal flow field is also difficult because of the complex geometry and continuously changing flow conditions. CFD analysis can overcome some of these limitations by providing detailed information about pressure, velocity, turbulence, and flow patterns inside the Venturi. Hence, there is a need to systematically review the available research on carburetor Venturi flow and identify the most influential geometric and operating parameters. Parametric optimization of these parameters can help in achieving improved airflow, pressure characteristics, fuel mixing, and overall carburetor performance.

III. RESEARCH OBJECTIVE

The main objective of this review is to study and evaluate the flow behaviour and parametric optimization of carburetor Venturi systems for improving airflow, pressure characteristics, fuel-air mixing, and overall carburetor performance. The specific objectives of the research are as follows.

The major objectives of this review paper are:

<https://www.ijrtsm.com> © *International Journal of Recent Technology Science & Management*

1. To study the fundamentals of carburetor Venturi flow and its working principle under different operating conditions.
2. To review the construction, geometry, and flow characteristics of conventional carburetor Venturi systems.
3. To investigate the effect of Venturi geometric parameters such as throat diameter, throat length, and inlet-outlet dimensions on flow behaviour.
4. To examine the effect of throttle opening, pressure, velocity, and other operating conditions on airflow and fuel-air mixing.
5. To review the influence of fuel nozzle parameters, including nozzle diameter, position, and angle, on fuel discharge and mixing characteristics.
6. To study the application of CFD techniques for analyzing pressure distribution, velocity distribution, turbulence, and flow separation inside carburetor Venturi systems.
7. To review parametric optimization techniques used for improving airflow, pressure distribution, fuel mixing, and overall Venturi performance.
8. To identify research gaps and suggest future directions for the design and CFD-based optimization of carburetor Venturi systems.

IV. LITERATURE REVIEW

The flow behaviour and performance of carburetor Venturi systems have been important research areas because the Venturi directly influences airflow, pressure reduction, fuel discharge, and air-fuel mixing in spark ignition engines. The converging and diverging geometry of the Venturi produces changes in air velocity and pressure, which affect the fuel induction process. Conventional carburetor designs are simple and reliable, but their flow characteristics may be affected by throttle position, Venturi dimensions, nozzle geometry, and engine operating conditions. Research has therefore increasingly focused on experimental investigations, CFD-based flow analysis, geometric modification, and parametric optimization of carburetor Venturi systems to achieve improved airflow, pressure distribution, fuel mixing, and overall engine performance.

Ajitabh et al. (2026) investigated the flow and fuel-delivery characteristics of a carburetor used in a small spark ignition engine. The study focused on understanding the relationship between airflow through the Venturi and fuel delivery through the nozzle. Experimental measurements were used to evaluate the variation in fuel flow under different operating conditions. The researchers observed that Venturi pressure reduction plays an important role in controlling fuel discharge. The study also highlighted the importance of accurate fuel-metering characteristics for maintaining the required air-fuel ratio. The findings demonstrate the need for proper Venturi and nozzle design for stable carburetor operation.

Chirag Shukla et al. (2026) investigated the internal flow characteristics of a carburetor using numerical analysis. The study examined the effects of Venturi geometry and operating conditions on pressure and velocity distribution. CFD analysis was used to identify regions of high velocity and low pressure inside the Venturi passage. The results indicated that changes in the throat geometry can significantly influence the pressure drop and airflow behaviour. The study also emphasized the importance of smooth flow development through the converging and diverging sections. The research provides useful information for optimizing carburetor Venturi geometry.

Bhanupratap Saktawat et al. (2025) studied the flow characteristics and performance of a carburetor Venturi under different operating conditions. The investigation focused on the effect of airflow conditions and Venturi geometry on pressure and velocity distribution. Numerical analysis was used to examine the flow field inside the Venturi passage. The results showed that the throat region experiences a significant increase in velocity accompanied by a reduction in static pressure. The study indicated that suitable geometric parameters can improve airflow and pressure characteristics. The findings support the use of CFD for analyzing and optimizing Venturi flow.

Mohammad Sheikh et al. (2025) investigated the effect of Venturi geometry on internal flow characteristics using computational methods. The study considered the influence of geometric parameters on pressure loss, velocity

distribution, and flow uniformity. The researchers observed that improper geometric transitions can increase turbulence and cause flow separation in the diverging region. Suitable converging and diverging profiles were found to provide smoother flow development. The study emphasized that Venturi dimensions should be selected carefully to reduce unnecessary pressure losses. The findings are useful for developing optimized carburetor Venturi configurations.

Sonthalia et al. (2024) investigated the flow behaviour of a carburetor using CFD analysis. The study examined pressure and velocity variations within the carburetor passage under different operating conditions. The CFD model helped identify the high-velocity region near the Venturi throat and the corresponding low-pressure region responsible for fuel suction. The researchers observed that throttle position significantly affects the flow field and pressure distribution. The study demonstrated that CFD can provide detailed information that is difficult to obtain through conventional experimental measurements. The work provides a useful basis for further parametric optimization of carburetor Venturi systems.

Arvind and Swaminathan (2024) investigated compressible flow through a complex-geometry carburetor Venturi using three-dimensional CFD analysis. The study considered important internal components such as the fuel tube and throttle plate and examined their influence on airflow characteristics. Different throttle plate positions were analyzed to determine changes in pressure, velocity, turbulence, and total pressure. The results showed that the fuel tube and throttle plate can create wakes and contribute significantly to pressure losses within the Venturi. The study also observed flow separation and increased turbulence near the throttle and downstream regions. The work demonstrated the importance of considering actual carburetor geometry rather than analyzing an ideal Venturi alone.

Suman Kumar and Krishanpal Singh Rajpoot (2023) investigated the effect of throttle-valve variation on flow characteristics in a Venturi chamber using CFD. The study considered different nozzle angles and throttle conditions to evaluate pressure, velocity, and air-fuel mixing behaviour. The numerical analysis was focused on understanding the flow field inside the Venturi and improving atomization and mixing characteristics. The results indicated that changes in throttle position and nozzle geometry influence the internal pressure and velocity distribution. The study also highlighted the importance of suitable nozzle-angle selection for obtaining improved flow and mixing conditions. The research provides useful information for parametric optimization of carburetor Venturi systems.

Lokhande et al. (2023) investigated the flow through a carburetor Venturi using CFD analysis in ANSYS. The study focused on the effect of throttle angle on pressure drop and velocity inside the carburetor. Different throttle conditions were considered to understand their influence on Venturi flow behaviour. The researchers emphasized that the Venturi is responsible for creating the pressure drop required for fuel induction into the incoming air stream. The study demonstrated that CFD can be effectively used to analyze pressure and velocity characteristics and identify suitable operating conditions for carburetor design. The findings support the use of CFD as an effective tool for improving Venturi performance and achieving more uniform fuel-air supply.

N. Vinoth et al. (2022) investigated the flow characteristics of a carburetor using CFD analysis for different valve positions. The study focused on understanding the changes in airflow and pressure distribution caused by variations in the valve opening. The numerical model was used to examine the flow field inside the carburetor under different operating conditions. The results showed that valve position has a significant influence on velocity and pressure distribution within the carburetor passage. The study also demonstrated that CFD can effectively identify regions of high velocity, pressure loss, and flow disturbance. The findings are useful for improving carburetor flow characteristics under different engine operating conditions.

S. J. Suryawanshi and R. B. Yarasu (2022) investigated the mixing performance of different producer-gas carburetor configurations using CFD simulation. The study examined the flow and mixing characteristics of air and producer gas inside different carburetor geometries. The numerical analysis was used to evaluate velocity distribution, pressure variation, and mixing behaviour. The results showed that carburetor geometry has an important influence on the uniformity of the mixture and the overall flow field. The study indicated that suitable geometric modification can

improve mixing performance and reduce non-uniform flow regions. The research demonstrates the usefulness of CFD-based geometric optimization for improving carburetor performance.

S. Kumar and K. S. Rajpoot (2021) studied the effect of throttle-valve variation on the flow characteristics of a Venturi using CFD analysis. Different throttle positions were considered to evaluate changes in pressure and velocity inside the Venturi passage. The study showed that increasing the throttle opening changes the airflow pattern and pressure distribution through the Venturi. The researchers also examined the influence of nozzle configuration on the air-fuel mixing process. The results indicated that suitable throttle and nozzle parameters can improve flow behaviour and mixing characteristics. The study provides useful guidance for selecting appropriate operating and geometric parameters during carburetor Venturi optimization.

A. D. Endaylalu et al. (2020) investigated the flow characteristics through a Venturi tube using numerical and experimental approaches. The study focused on understanding the relationship between pressure variation and fluid velocity along different sections of the Venturi passage. Experimental observations were compared with numerical predictions to evaluate the reliability of the flow analysis. The results showed that velocity increases through the converging section while pressure decreases toward the throat region. The study also indicated that proper Venturi geometry is important for obtaining stable and predictable flow characteristics. The findings provide useful guidance for applying CFD and experimental methods to the analysis and optimization of Venturi-based flow systems.

Wang et al. (2020) investigated the flow-field characteristics of a Venturi-based ejector using CFD simulation. The study examined the influence of flow geometry and internal flow conditions on velocity distribution, pressure variation, and mixing behaviour. Numerical analysis was used to identify important flow regions and understand the interaction between the primary and secondary flows. The results demonstrated that geometric modifications can significantly influence the pressure field and flow development inside the Venturi passage. The study highlighted the importance of optimizing Venturi geometry for improving flow performance and reducing undesirable flow losses. The findings are relevant to carburetor Venturi research because similar pressure and velocity mechanisms control fluid entrainment and mixing.

V. CONCLUSION

The present review examined the flow behaviour and parametric optimization of carburetor Venturi systems. The reviewed studies show that Venturi geometry plays an important role in controlling airflow, pressure reduction, and fuel induction. Parameters such as throat diameter, throat length, inlet and outlet geometry, throttle opening, and fuel nozzle angle significantly affect the internal flow characteristics. CFD analysis provides useful information about pressure distribution, velocity variation, turbulence, and flow separation inside the Venturi. Proper selection of geometric parameters can reduce pressure losses and improve airflow and fuel-air mixing. The reviewed studies also indicate that throttle position has a significant effect on pressure and velocity characteristics. Suitable nozzle configuration can further improve fuel distribution and mixing uniformity. Overall, parametric optimization of the Venturi can contribute to improved carburetor performance, fuel efficiency, and combustion quality. CFD combined with experimental validation can provide an effective approach for optimizing complex carburetor geometries. Further research is required to investigate multiple design parameters simultaneously under different engine operating conditions and develop optimized Venturi configurations.

REFERENCES

- [1] Y. Wang, J. Ge, Z. Kuang, G. Li, P. Xu, and R. Bi, "Flow field characteristics analysis of venturi ejector improved by swirler using CFD simulation," *Chinese Journal of Chemical Engineering*, vol. 95, pp. 302–312, 2026, doi: 10.1016/j.cjche.2026.01.024.
- [2] A. D. Endaylalu, "A numerical and experimental investigation of fluid flow through a Venturi tube," *American Journal of Mechanical and Industrial Engineering*, vol. 10, no. 6, pp. 116–127, 2025, doi: 10.11648/j.ajmie.20251006.13.

- [3] H. Catur and A. G. Wailanduw, "Analisis laju aliran fluida pada karburator PE dengan variasi tipe menggunakan software terhadap kecepatan fluida," *Jurnal Teknik Mesin*, vol. 15, no. 1, pp. 139–144, 2026, doi: 10.26740/jtm.v14n04.p139-144.
- [4] S. Weiss, B. Mickan, J. Polansky, K. Oberleithner, M. Bär, and S. Schmelter, "Improved prediction of the flow in cylindrical critical flow Venturi nozzles using a transitional model," *Flow, Turbulence and Combustion*, vol. 113, pp. 363–389, 2024, doi: 10.1007/s10494-024-00553-3.
- [5] Y. Chen, X. Zhang, C. Li, L. Zhou, W. Wang, G. Chen, and J. Duan, "Mass transfer enhancement and flow field simulations for a Venturi bubble generator with multiple inlet tubes," *Nuclear Engineering and Design*, vol. 424, Art. no. 113249, 2024.
- [6] L. G. Sigalotti, C. E. Alvarado-Rodríguez, F. Aragón, et al., "SPH simulations and experimental investigation of water flow through a Venturi meter of rectangular cross-section," *Scientific Reports*, vol. 13, Art. no. 21215, 2023, doi: 10.1038/s41598-023-48520-8.
- [7] E. Zhao, L. Li, G. Wang, D. Chen, and J. Xu, "Optimal design of Venturi flowmeter based on CFD analysis," *Atomic Energy Science and Technology*, vol. 56, no. 6, pp. 1196–1203, 2022, doi: 10.7538/yzk.2021.youxian.1056.
- [8] L. Avdieieva, A. Makarenko, and H. Dekusha, "Computer simulation of fluid flow through a Venturi nozzle of different configurations," *Science and Innovation*, vol. 18, no. 5, pp. 61–68, 2022, doi: 10.15407/scine18.05.061.
- [9] M. Ge, C. Sun, G. Zhang, et al., "Combined suppression effects on hydrodynamic cavitation performance in Venturi-type reactor for process intensification," *Ultrasonics Sonochemistry*, vol. 86, Art. no. 106035, 2022, doi: 10.1016/j.ultsonch.2022.106035.
- [10] E. De Oro Ochoa, M. Carmona García, N. Durango Padilla, and A. Martínez Remolina, "Design and experimental evaluation of a Venturi and Venturi-Vortex microbubble aeration system," *Heliyon*, vol. 8, no. 10, Art. no. e10824, 2022, doi: 10.1016/j.heliyon.2022.e10824.
- [11] J. Xiao, Z. Liang, X. Liu, Z. Zhao, and X. Xie, "Design optimization analysis of Venturi tube for medium conveying in strengthen grinding process," *Engineering*, vol. 13, no. 8, 2021, doi: 10.4236/eng.2021.138031.
- [12] D. A. Wilson, K. Pun, P. B. Ganesan, and F. Hamad, "Geometrical optimization of a Venturi-type microbubble generator using CFD simulation and experimental measurements," *Designs*, vol. 5, no. 1, Art. no. 4, 2021, doi: 10.3390/designs5010004.
- [13] A. D. Lokhande, D. S. Dhondge, and K. Patil, "CFD analysis of flow through Venturi in carburetor," *International Journal of Emerging Technologies and Innovative Research*, vol. 8, no. 7, pp. d605–d613, 2021.
- [14] N. Vinoth, V. Mohanavel, A. Kannappan, and K. Mohith, "CFD modelling of carburetor with several valve positions," *Materials Today: Proceedings*, vol. 37, part 2, pp. 1535–1549, 2021, doi: 10.1016/j.matpr.2020.07.149.
- [15] S. J. Suryawanshi and R. B. Yarasu, "Mixing performance analysis of various shapes of producer gas carburetor using CFD simulation," *Journal of the Institution of Engineers (India): Series C*, vol. 101, no. 4, pp. 683–691, 2020, doi: 10.1007/s40032-020-00568-5.
- [16] S. Kumar and K. S. Rajpoot, "CFD approach on throttle valve variation on Venturi," *International Journal for Scientific Research & Development*, vol. 8, no. 7, 2020.
- [17] H. Shi and P. Nikrityuk, "The influence of inflow swirl on cavitating and mixing processes in a Venturi tube," *Fluids*, vol. 5, no. 4, Art. no. 170, 2020, doi: 10.3390/fluids5040170.
- [18] "In silico screening of Venturi designs and operational conditions for gas-liquid mass transfer applications," *Chemical Engineering Journal*, vol. 383, Art. no. 123119, 2020, doi: 10.1016/j.cej.2019.123119.