International Journal of Future Generation Communication and Networking
Vol. 13, No. 2s, (2020), pp. 875–881

Numerical Investigation of Fluid Flow Through Catalytic Converter using Topological Changes.

Sarvesh M. Hajare, Vishal T. Gaikwad, Ashitosh B. Kamathe, Abhishek N. Adak, Prof. Renu Yeotikar

[1]*Department of Mechanical Engineering, SKNCOE, Vadgaon, Pune 411046, India*
[2]*Department of Mechanical Engineering, SKNCOE, Vadgaon, Pune 411046, India*
[3]*Department of Mechanical Engineering, SKNCOE, Vadgaon, Pune 411046, India*

Abstract:

In the current scenario Air pollution and global warming are the major issues in the world. And the emissions from internal combustion engine contribute more amount of air pollution. Catalytic converter plays an important role in reducing these harmful emissions, but the presence of catalytic converter increases the exhaust back pressure. This increase in back pressure causes increase in fuel consumption and decrease in the efficiency of engine. In this study, Numerical investigation is carried out in order to understand the flow through the catalytic converter using different topological changes. An attempt has been made to design and get the optimized back pressure of catalytic converter by using CFD software (Fluent). A CFD model K-epsilon turbulence model and porous media are utilized to simulate the flow through monolithic ceramic substrate. The analysis involved determining back pressure across the converter system for a given mass flow rate. In CFD analysis various models with different topological changes such as Inlet cone diameter (90mm,95mm,100mm), inlet Cone angle (90^o,45^o,23^o) Cone length (25mm,65mm,65mm) were simulated using the appropriate boundary conditions and fluid properties specified to the system with suitable assumptions. The numerical results were studied to get the optimized back pressure at inlet of the converter and it is found to be at 100mm cone diameter and cone angle 23^o.

Keywords: *Catalytic Converter, Exhaust back pressure, Cone angle, Cone diameter, Cone length, CFD.*

INTRODUCTION

Internal Combustion engines are the_main source of releasing harmful emissions which has led to the development of emission control systems to treat exhaust gases and convert them to less harmful products called catalytic converters. Catalytic Converter is a cylindrical unit installed into the exhaust system of vehicle, between exhaust manifold and the silencer. Due to presence of catalyst in the converter toxic gases are exhaust, i.e. CO, HC and NOx are converted into harm less CO2, H2O, and N2 commonly used catalyst are platinum and rhodium. Monolith substrates are the main component of exhaust line after treatment systems found in automobiles today. They provide superior performance in comparison to the pellets support type. Monolith substrates are typically characterized by their cell density and channel wall thickness. Due to the large total surface area of the channels and the small thermal mass of the substrate, heat transfer is greatly enhanced, which improves the conversion efficiency indicating an improved thermal performance. The need for better automotive technologies to improve fuel economy, while meeting more stringent global vehicle emission standards, continues to grow with the increasing demand for environmental protection and rising fuel prices. Therefore, improving the thermo-fluid performance of catalytic converters is necessary to meet emission regulations. Improving the performance of catalytic converters requires intensive experimental investigations to study flow behavior inside the catalytic converter requires accurate fluid flow and pollutant concentration measurement instrumentation and a safe environment for researchers to perform these tests. Those restrictions decrease the feasibility of such studies and a more feasible approach is needed. There are many investigations have been performed the study on flow characteristics of catalytic converter Hesham A. Ibrahim , Wael H. Ahmed , Sherif Abdou , Voislav Blagojevic','Experimental and numerical investigations of flow through catalytic converters. In

ISSN: 2233-7857 IJFGCN

this study both experimental and numerical investigations are carried out in order to understand the flow through the catalytic converter, using different monolith cell densities. A dynamically scaled-down model for a typical flow through catalytic converter was utilized for this study.They concluded that the flow becomes more uniformly distributed when the inlet pipe is shorter in length and the bending angle is smaller. Moreover, they investigated the effect of brick properties on flow distribution concluding that the higher the brick resistance the more uniform the flow distribution observed. They also added that, with multi-brick catalytic converters, the second brick tends to show more uniform flow distribution than the first one due to the gap between the two bricks which allows flow. They utilized CFD models to investigate the effect of the inlet Reynolds number on flow distribution and they reported that flow uniformity decreases when increasing the inlet flow velocity and Reynolds number to redistribute more uniformly. Both experimental investigation and numerical simulation were carried out in order to evaluate the thermal and hydraulic performance of catalytic converters with monolith cell densities of 400CPSI and 900 CPSI. A.M. Leman1,a, Fakhrurrazi Rahman dealt with 'Emission Treatment towards Back Pressure in Internal Combustion Engine against Performance of Catalytic Converter. The presented is a review of back pressure problems together with several alternatives taken by not affecting the performance of vehicles engine and fuel consumption. It is observed that the back pressure for the converters that have larger diameter. The result from the research shows that increase in inlet cone length will reduces the backpressure and also reduces the recirculation zones in the catalytic converter. The work carries by Cornejo, Petr Nikrityuk, Robert E. Hayes reports theoretical studies of flow behavior in a monolith outlet zone for different Reynolds numbers covering laminar and transitional/turbulent flow regimes. Simulations are performed using different discrete geometry to study and quantify the velocity fluctuations of flow leaving a monolith. Parametric studies are carried out to illustrate the influence of the Reynolds number on the appearance of flow fluctuations at the outlet zone of the monolith. The results were analyzed in terms of the flow regime and average and standard deviation of the velocity. Robert E. Hayes,Petr Nikrityuk, Ivan Cornejo dealt with 'A New Approach for the Modeling of Turbulent Flows in Automotive Catalytic Converters'. The proposed model implemented together with the turbulence damping emulates the fluid-flow interaction observed at the beginning and the end of a monolith, despite it being represented as a perfectly homogeneous porous medium. Compared to other available models of the converter, the proposed methodology represents the flow thought an actual honeycomb type substrate better, achieving a realistic behavior along the entire converter.

The task of presented work is to analyze the effect of Catalytic converters which restricts the free flow of exhaust, which negatively affects vehicle performance and fuel economy and also to reduce back pressure on the engine by providing streamline or laminar flow into the inlet of silencer by changing angel of inlet and outlet of catalytic converter to increase efficiency of the vehicle.

Different modules with varying inlet and outlet are to studied to remove the recirculating zone created near the entrance of converter the results of all modules are compared with each other and optimum solution is identified also using computational fluid dynamic the flow pattern, velocity thermal and pressure counters are studied.

Geometrical configuration:

The three-dimensional model, depicted in Figure 1, represents a Catalytic Converter used in Royal Enfield bike with a length of 715mm, while its catalyst diameter of 100mm with cone angle of 23^{o}.

ISSN: 2233-7857 IJFGCN

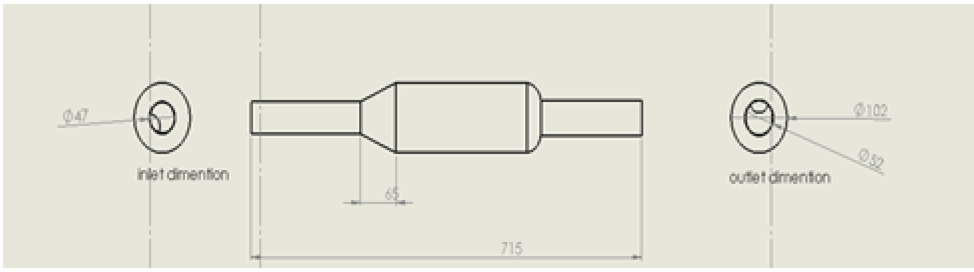

Fig.1 Block diagram

.

Test Geometry Model:

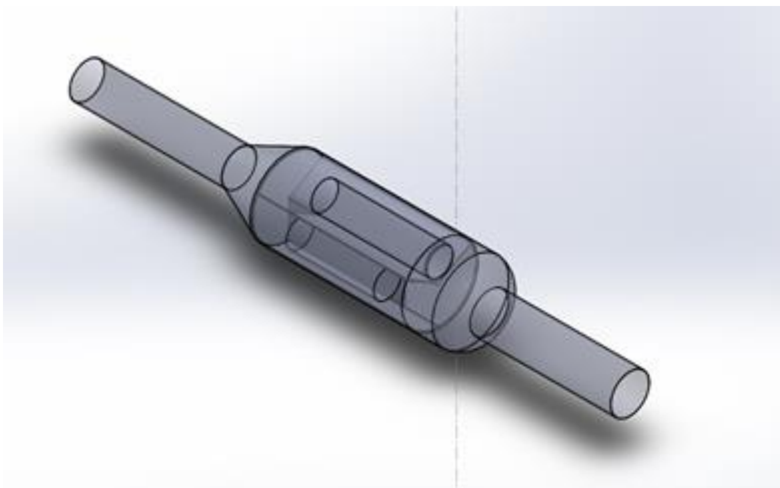

Fig. 2 Test Geometry model

Above figure shows the geometry which has been considered in this simulation to study effect of exhaust back pressure so that it will reduce emission

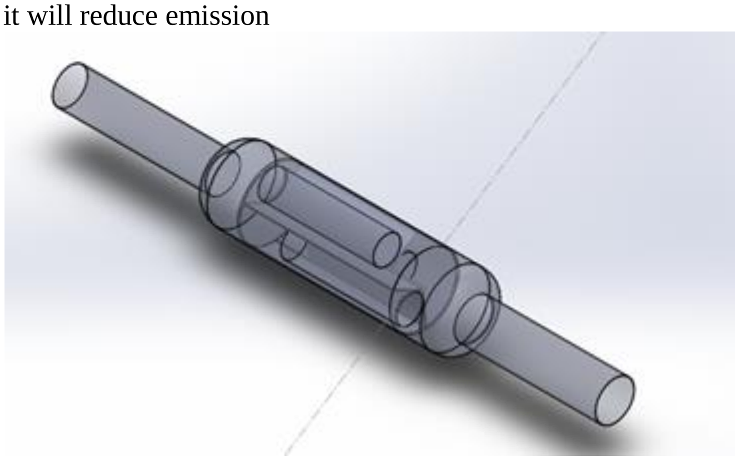

Fig. 3 Original Geometry

ISSN: 2233-7857 IJFGCN

NUMERICAL STUDY:

The commercial CFD package, ANSYS Fluent is used to simulate the fluid flow and heat transfer through the experimental domain. Steady state, 2D, axisymmetric simulations are performed with air as the working fluid. The solution domain is divided into three regions as shown in Fig. Regions I & III include the inlet and outlet sections where turbulent flow occurs. The third region is the monolith substrate region that is modeled as a porous media region. This region is dominated by laminar flow due to the small hydraulic diameter of the channels.
A computational fluid dynamic analysis of a three-dimensional exhaust catalytic model, provided with topological changes, as reported in Figure 4, is considered in order to evaluate its backpressure and fluid-dynamic behaviors and study the pressure and velocity fields. Different inlet diameter and cone length are considered in the ranges of turbulent regime

The working fluid is exhaust gases and the assigned boundary conditions are the following

- Inlet: Velocity inlet boundary condition [velocity, pressure].
- Outlet: Pressure Outlet [atmospheric pressure).
- Test geometry and Meshing is created using ANSYS FLUNT
- 3-Dimensional, CFD simulations are performed using ANSYS FLUENT.
- The pressure-based solver is chosen for the simulations with the consideration to the exhaust, flows in this problem

Boundary Condition

- Inlet : Velocity inlet boundary condition [velocity, temperature].
- Outlet: Pressure Outlet: [atmospheric pressure].
- Can Surface : No-slip, stationary outlet surface.

CFD Modeling - Solver Settings:
The mesh was imported in to ANSYS FLUENT or setting up as well as performing CFD simulations. The pressure based solver we chosen for the simulation as this project work, the gravitational effects are not included in the flow physics.
The following server settings are applied while solving this project work

- 3-Dimensional. Steady-state CFD simulations are performed using ANSYS FLUENT.
- The pressure solver is shown for the simulation with the consideration to the exhaust flow problem.
- This was done using the single channel model, where flow through a single channel was modeled at different flow rates and temperatures that represent the conditions available during the experiment.

ISSN: 2233-7857 IJFGCN

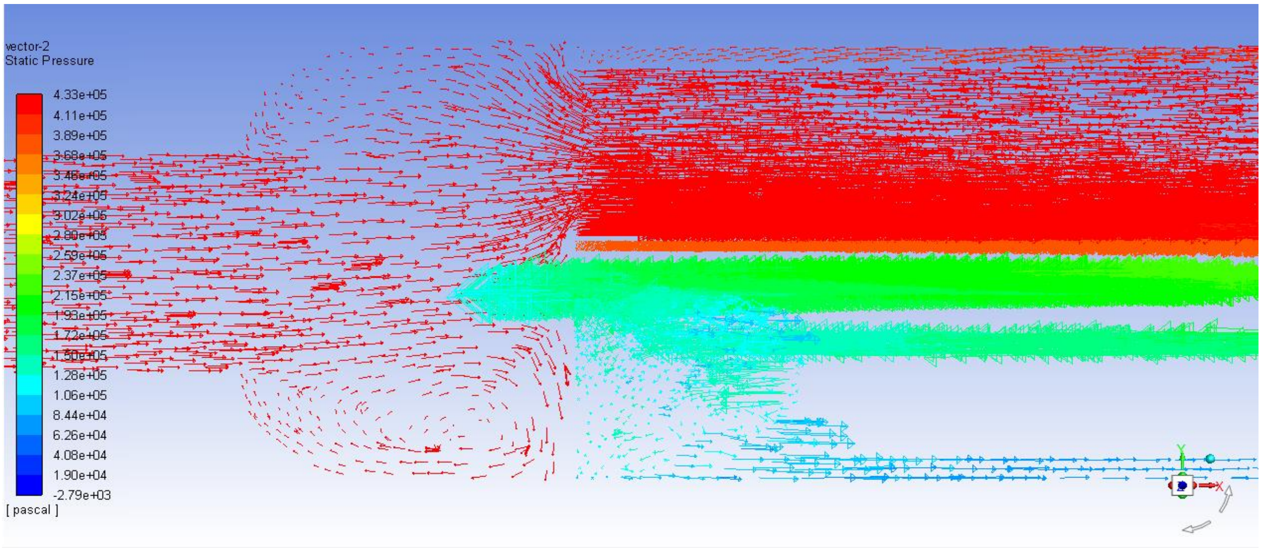

Fig.4 Pressure contour with Vector for cone diameter as 90mm and Cone angle 90°

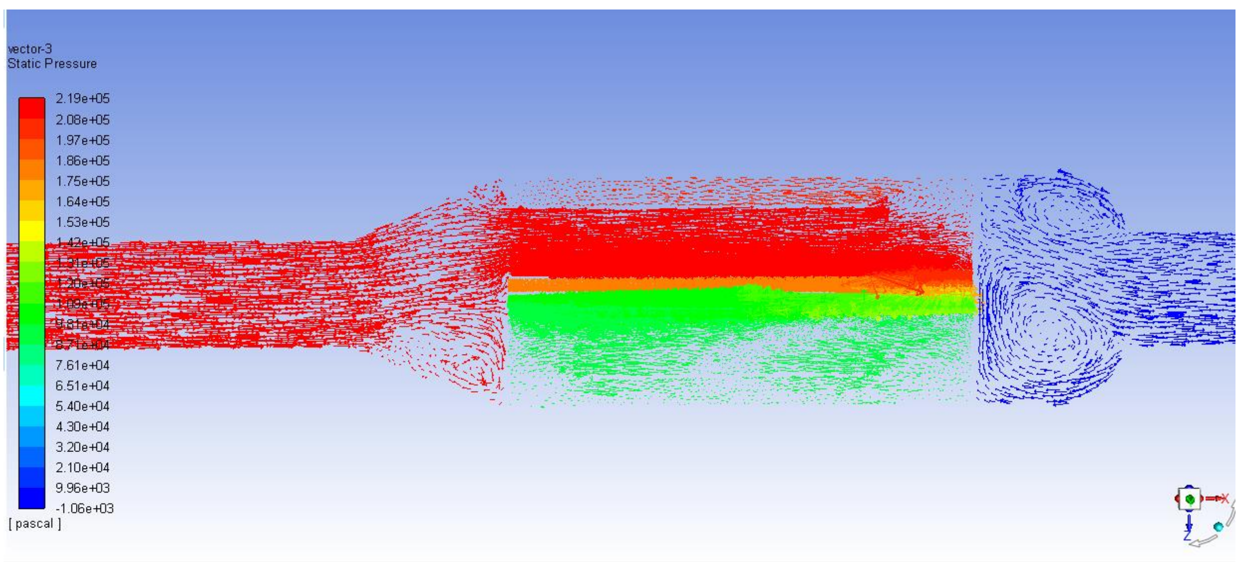

Fig.5 Pressure contour with Vector for cone diameter as 95mm and cone angle of 45°

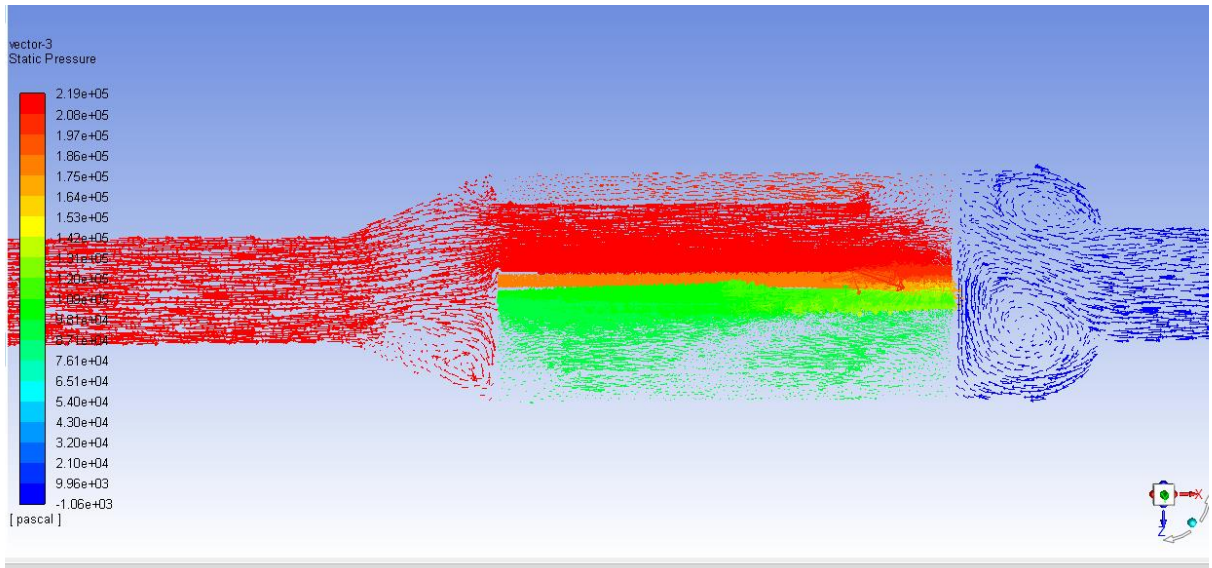

Fig. 6 Pressure contour with Vector for cone diameter as 100mm and cone angle of 23°

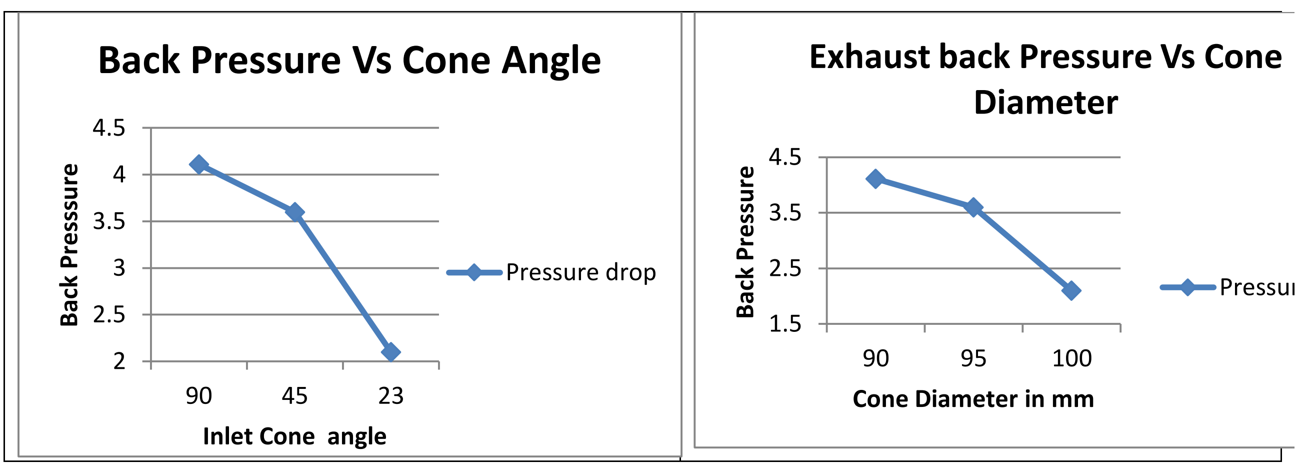

Fig7. Back Pressure Vs Cone angle and Fig.8 Back pressure Vs Cone Diameter

CONCLUSION:

1. Among all the types of technologies developed so far use of catalytic converter is the best way to control exhaust emission.
2. The increase in diameter of inlet, inlet cone length, angle of inlet and outlet of catalytic converter reduced the backpressure by 2.2 bar and also reduced the recirculation zone.
3. The turbulent flow pattern at inlet is converted into laminar flow
4. Modified catalytic converter is eco -friendly, because it reduces the emission of vehicles.

REFERENCES

1. H. A. Ibrahim, W. H. Ahmed, " Blagojevic Experimental and numerical investigations of flow through catalytic converters", (2018)
2. A.M. Leman1, Fakhrurrazi Rahman, "Feriyanto Emission Treatment towards Cold Start and Back Pressure in Internal Combustion Engine against Performance of Catalytic Converter: A Review" (2016)
3. Ivan Cornejo, Petr Nikrityuk, "Turbulence generation after a monolith in automotive catalytic converters ([2018)
4. Robert E. Hayes,Petr Nikrityuk, Ivan Cornejo A New Approach for the Modeling of Turbulent Flows in Automotive Catalytic Converters(2018)
5. Om Ariara Guhan C.P.Dr.Nagarajan G, "Numerical study of fluid flow and effect of catalytic converter volume in optimization of diesel oxidation catalyst in a CI engine using CFD" (2015)

ISSN: 2233-7857 IJFGCN