Research Paper
Volume : 3 | Issue : 7 | July 2014 • ISSN No 2277 - 8179

Redesign of Catalytic Converter

Engineering

KEYWORDS : catalytic converter, fluid flow, temperature distribution, the mass flow rate, heat and mass transfer

T. Sekar	Final Year P.G Student, Department of Mechanical Engineering, Saveetha School of Engineering, Saveetha University, Chennai
Mr. P. Kumaran	Associate Professor, Department of Mechanical Engineering Saveetha School of Engineering, Saveetha University, Chennai. Tamilnadu, India
S. Mohana Murugan	Professor, Department of Mechanical Engineering Saveetha School of Engineering, Saveetha University, Chennai. Tamilnadu, India
A. Sivasubramanian	Assistant Professor, Department of Mechanical Engineering Saveetha School of Engineering, Saveetha University, Chennai. Tamilnadu, India.

ABSTRACT

An improvement of catalytic converter design requires better fundamental understanding of complex processes taking place involving fluid flow, heat and mass transfer, and chemical reactions. The project deals with the study of fluid flow inside the catalytic converter and the study of temperature distribution and back pressure in catalytic converter. Based on the outcome of the study, simulation has been conducted to redesign the catalytic converter to provide better efficiency, ease reusability and replaceable ceramic instead of replacing the complete unit, using ANSYS-13 on fluid dynamics (CFX). CATIAV5R17 and SOLIDWORKS have been used for geometric modeling of catalytic converter which intern used in ANSYS designing. Flow field in the catalytic converter is influenced by the flow resistance of the substrate for a given geometric configuration. As the mass flow rate increases, the pressure drop also increases. At lower temperature, the catalytic converter will be inactive.

1.Introduction

In this project we would like to designed and through CFD analysis, a compromise between two parameters namely, more filtration efficiency with limited back pressure was aimed at. In CFD analysis, various models with different wire mesh grid size combinations were simulated using the appropriate boundary conditions and fluid properties specified to the system with suitable assumptions. The back pressure variations in various models and the flow of the gas in the substrate were discussed .the honeycomb catalysts in to expose the exhaust gases to a large surface made of one or more noble metals: platinum, palladium and rhodium such a way that the exhaust is uniformly increases with uniform actuation. Since exhaust air is compressible, so it is not possible to use the basic momentum equation, for deriving the outlet discharge. Hence we are using ANSYS CFD FLOTRAN ,ANSYS CFX analysis for different shape of honeycomb catalysts.

2.Analysis

The nature of exhaust gas flow is a very important factor in determining the performance of the catalytic convertor. Of particular importance is the pressure gradient and velocity distribution through the substrate. Hence CFD analysis is used to design efficient catalytic converters. By modeling the exhaust gas flow , the pressure drop and the uniformity of flow through the substrate can be determined. ANSYS FLUENT is used to model the flow of nitrogen gas through catalytic converter geometry, so that the flow field structure is analyzed.

This analysis is the following:

a. Set up porous zone for substrate with appropriate resistances.
b. Calculate a solution for gas flow through the catalytic converter using the pressure-based solver.
c. Plot pressure and velocity distribution on specified planes of the geometry.
d. Determine the pressure drop.

3.Calculations

3.1Design Calculation

As per design methodology we benchmarked same kind of engine models to set the target of transmission loss of Catalytic conveter.

Engine data: Bore(D)= 80 mm, Stroke (L)= 98 mm, No.Cylinders (n) = 3,Engine power=65hp,Max RPM(N)=3500rpm,

3.2 Shape of Catalytic Converter

The cylindrical shape was considered due to ease of fabrication, minimum assembly time, rigidity and easier maintenance.

3.3Volume of Catalysts

$$\text{Space Velocity} = \frac{\text{Velome flow rate}}{\text{Catalysts Velume}}$$

Volume flow rate =Swept volume x Number of Intake Stroke p hr

$$= \frac{\pi}{4} + (0.0875)2x(0.110)x\frac{1500}{2}x60 = 29.31\text{m}^3$$

$$\text{Catalysts Volume} = \frac{\text{Velome flow rate}}{\text{space Velocity}}$$

$$= \frac{29.31}{20000} = 0.001465\text{m}^3 = 1465\text{ml}$$

The shell is the central cylindrical part between the inlet and outlet cones. This part contains circular discs with coated pellets.

$$V_{catalyst} = \frac{\pi}{4} x D^2 x L \text{ mm}^3$$

Where, D=Diameter of the catalyst

L=Length of the catalyst (assume L=2D)

$$0.001465 = \frac{\pi}{4} x2xD^3$$

D=0.0977m = 97.7mm@100mm
L=2*100 =200mm, Length of the shell =200mm

3.4The Porous Domain.

The catalyst-coated honeycomb structure will be modelled using a porous domain with a directional source of resistance, as described in the problem description. The stream wise directional resistance is aligned with the Z axis.

For quadratic resistances, the pressure drop is modelled using.

$$\frac{\partial p}{\partial xi} = KqIUIUi$$

Where Kqis the quadratic resistance coefficient,
Ui is the local velocity in the direction,

$\frac{\partial p}{\partial x_i}$ is the pressure drop gradient in the direction.

3.5 Initial Values.

The inlet velocity: 25 [m s^-1]

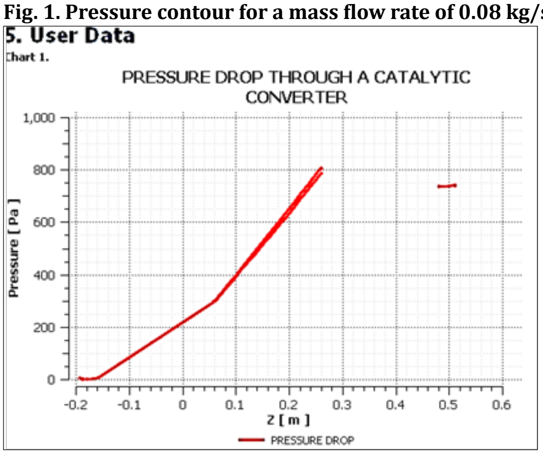
The cross sectional area of the inlet and housing, which can be determined using the function calculator in CFD-Post: 0.001913 m^2 and 0.024039 m^2 respectively The porosity of the honey-comb structure: 70%

3.6 Parameters in the simulation

a. AreaDen : 360 m-1
b. HTC : 50 Wm-2K-1
c. HTCoutside : 20 Wm-2K-1
d. L : 0.4 m
e. Porosity : 0.7
f. Tinlet : 500K
g. Toutside : 40C

Location	Type	Mass Flow	Momentum		
			X	Y	Z
Housing Default (Housing)	Boundary	0.0000e+00	-2.4574e-01	-4.0942e-02	4.1939e-02
InletSide Side 1 (Pipes)	Boundary	0.0000e+00	1.4937e-05	9.7940e-06	1.1202e-01
InletSide Side 2 (Housing)	Boundary	0.0000e+00	0.0000e+00	0.0000e+00	0.0000e+00
Inlet (Pipes)	Boundary	3.4010e-02	1.5999e+00	1.0969e-07	-1.5999e+00
OutletSide Side 1 (Pipes)	Boundary	0.000`0e+00	-1.8826e-07	2.7598e-08	-4.4426e-02
OutletSide Side 2 (Housing)	Boundary	0.0000e+00	0.0000e+00	0.0000e+00	0.0000e+00
Outlet (Pipes)	Boundary	-3.4010e-02	-6.0802e-01	-3.4666e-04	6.1096e-01
Pipes Default (Pipes)	Boundary	0.0000e+00	-9.7722e-01	3.3372e-03	-1.1646e+01

Fig. 1. Pressure contour for a mass flow rate of 0.08 kg/s



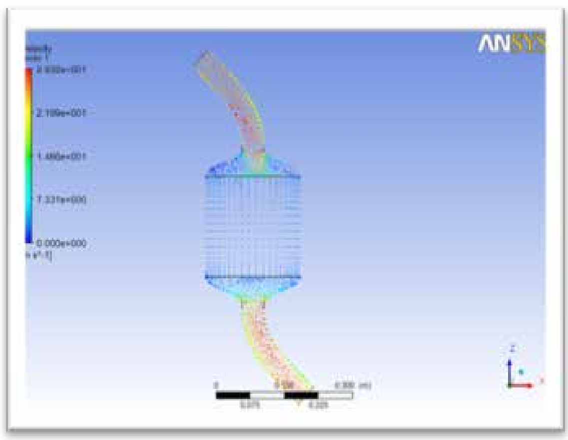


Fig.2Velocity Vectors

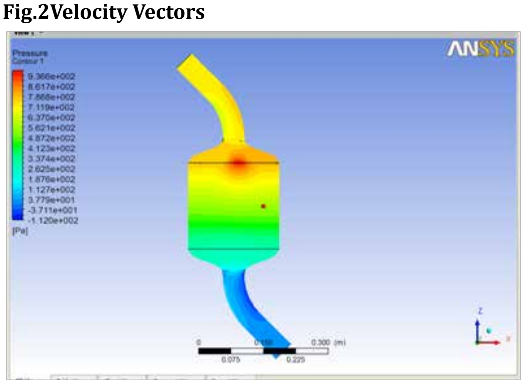


Fig. 3. Honey Comb

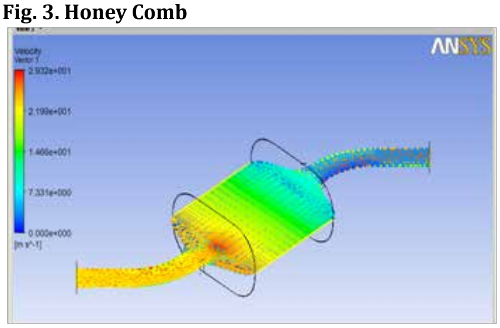


Fig. 4. Fabricated Catalytic converter

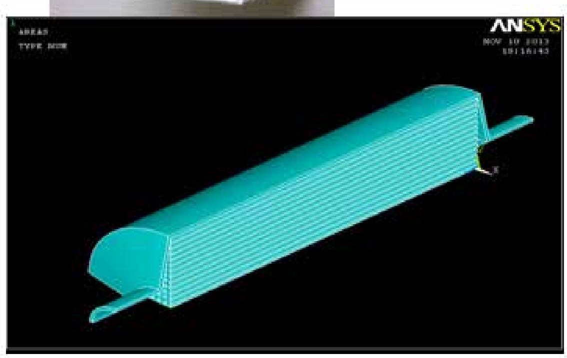


Fig. 5 Example of an image with acceptable resolution

4. Results and Discussion

The generation of catalytic converter geometry involved solid modeling. The Specification of catalytic converter along with the porous media is shown. The porosity value applied was 0.7 Geometric modeling was carried out using ANSYS FLUENT and fluid domain was extracted. The nature of exhaust gas flow is a very important factor in determining the performance of the catalytic convertor. Of particular importance is the pressure gradient and velocity distribution through the substrate. Hence CFD analysis is used to design efficient catalytic converters. By modeling the exhaust gas flow , the pressure drop and the uniformity of flow through the substrate can be determined. ANSYS FLUENT is used to model the flow of nitrogen gas through catalytic converter geometry, so that the flow field structure is analyzed.

Catalytic converter has been designed in order to determine the pressure drop and heat transfer. Ref Fig:4. Through it, when air enters the inlet at 25 m/s and 500 K, and exits the outlet at a static pressure of 1 atm. For simplicity, chemical reactions are not focused here. A mesh is used for the passageways inside a pipe-and-flange structure along with that a honeycomb structure is used with 70% porosity and the rest 30% is occupied by solid. The honeycomb structure is lined up with the flow to prevent flow travel in the transverse direction.

To model resistance to the flow, you will apply a streamwise quadratic resistance coefficient of 650 kg m^-4. To reduce the amount of transverse flow, apply a quadratic resistance coefficient of 6500 kg m^-4 in the transverse direction. These given resistance coefficients are based on the superficial flow velocity, rather than the true flow velocity. The Inlet boundary has a static temperature of 500 K. You will model heat transfer through the solid material in the porous domain. The heat transfer between the air and steel within the porous domain is modeled using an interfacial area density of 360 m^-1 and a heat transfer coefficient of 50 W m^-2 K^-1. Thermal energy is lost to the environment through the midsection walls of the catalytic converter; the rate of heat loss is defined by the heat transfer coefficient (20 W m^-2 K^-1).

Results obtained in the simulation leads to a pathway to think, by extending the Contact area of honeycomb structure without changing existing outer dimensions, will result in better efficiency without affecting any back pressure or any other existing parameters and also reduces the manufacturing complexity. The idea of extending contact area of the honeycomb can be achieved by arranging multiple layers honeycomb structure in a certain angle OR modifying the pathway of each porous into certain degree of angle by twisting the honeycomb structure itself.

5.Conclusions

The catalytic converter was successfully designed and fabricated. Through CFD analysis the vortices and backpressure of various catalytic converter models were studied. The increase in inlet cone length reduces the backpressure and also reduces the recirculation zones. Installation of the catalytic converter reduces the brake thermal efficiency and increase the brake specific fuel consumption, fuel flow rate. This study investigated the flow characteristics, the temperature distribution and conversion efficiency of catalytic converter. The computational tool CFD was used to study the behavior of fluid flow and the conversion rates of emissions as a function of exhaust gas temperature. The analysis shows that the flow field in the catalytic converter is influenced by the flow resistance of the substrate for a given geometric configuration. As the mass flow rate increases the pressure drop also increases. The conversion efficiency depends upon the substrate temperature and composition of the inlet. By increasing the temperature the conversion efficiency also increases. At lower temperature the catalytic converter will be inactive. The heat release due to chemical reaction does not play a significant role.

REFERENCE

[1] Francisco Payri, Jesus Benajes, and Jose Galindo (1999), One-dimensional Fluid – Dynamic Model for catalytic converter in automotive engines, SAE-1999-01-0144 | [2] Cathy Chung, SivanandiRajadurai and Larry GEE(1999), CFD Investigation of Thermal fluid flow and conversion characteristics of the catalytic converter, SAE 1999-01-0462. | [3] Daniel Chatterjee, Olaf Deutschmann and JurgenWarnatz(2001), Detailed surface reaction mechanism in a three way catalytic converter, Interdisciplinary Centre of Scientific Computing (IWR), Heidelberg University, 2001. | [4] Joachim Braun, Thomas Hauber, and Julia Windmann(2004), Three-Dimensional Simulation of the Transient Behaviour of a Three-Way Catalytic Converter, SAE 2004-01-0148. | [5] Julia Windmann, Joachim Braun and Peter Zacke(2003), Impact of the inlet flow distribution on the light off behaviour of a 3-way catalytic converter, SAE-2003-01-0937. | [6] G.Gaiser, J.Oesterie, and J.Barun(2003), The progressive spin inlet-homogeneous flow distributions under stringent conditions, SAE 2003-01-0840. | [7] Ming Chen, Joe Alexio, and Thierry Leprince(2004), CFD Modelling of 3-way catalytic converter with detailed catalytic surface reaction mechanism, SAE 2004-01-0148. | [8] SoojinJeong and TaehunKim(1997), CFD investigation of the 3-Dimensional Unsteady flow in the catalytic converter, SAE 1997-971025. | [9] William Taylor III(1999), CFD Prediction and experimental validation of high temperature thermal behaviour in catalytic converters, SAE-1999-01-0454. | [10] FLUENT 6.1 help manual | [11] Automobile Engineering by Dr.Kirpalsingh-Standard publications | [12] Automobile Engineering by G.B.S Narang-Kanna publications . |