SHODH SANGAM -- A RKDF University Journal of Science and Engineering

"Modeling Improves Monolithic Catalytic Converter Efficiency With Geometric Changes In Substrate Length Using CFD"

Jawed Aquebal[1], Dr.Sohail Bux[2]

[1] *M.Tech Research Scholar ,Agnos college of Technology, Bhopal, MP, INDIA*

[2] Professor, *Agnos college of Technology, Bhopal, MP, INDIA*

[1,2]*Department of Mechanical Engineering*

dreamsanchi@gmail.com[1] , buxsohail@gmail.com[2]

***Abstract*- The number of vehicles is expected to rise to about 1600 million by 2036. Due to the low temperature in the engine, there are many combustible products such as Carbon monoxide, Nitrous oxides, hydro carbons, harsh substances, etc. Some toxins have an effect. in the air, atmosphere and human body conditions that lead to the complex processes of emissions. Catalytic converter is a vehicle emission control device that converts toxic fumes from exhaust gas into less toxic waste by creating a redox reaction (oxidation or reduction).**

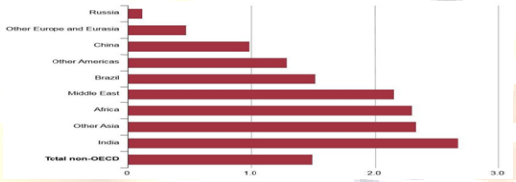
Optimization can no longer be based on traditional approaches, which are intensive in hardware use and laboratory testing. The CFD is in high demand for the analysis and design in order to reduce developing cost and time consuming in experiments. This work describes the conversion efficiency by changing the substrate length of automotive three-way catalytic converters, which are employed to reduce engine exhaust emissions. It is found that the CFD model in simulating the performance of three-way catalytic converter. There is a difference of 2.4% for oxide of nitrogen, 2.1% for propane and 1.8 % for carbon monoxide increase in conversion efficiencies by increasing the substrate length by 10mm while by reducing the substrate length by 10mm conversion efficiency reduced. The result also shows that the increase in substrate length leads to reduce emission concentration.

Keywords - Catalyst, CFD modeling, chemical reaction, conversion efficiency, simulation, Substrate length.

I. INTRODUCTION

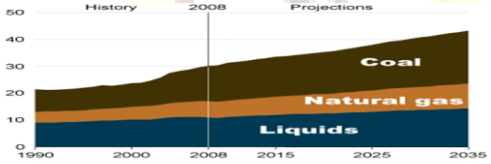
The internal combustion engine, either rotary or piston cylinder, is the primary power plant of today's world. Although it is used in both an on- and off-road capacity, automobiles are typically their primary customer. Because of their versatility, flexibility and low initial cost, automobile direct the market for customer and transport freight. still, inexpensive growth in developing countries such as india, china, brazil, and Chile has resulted in an increase in the number of vehicles on the road.

the combustion of carbonaceous fuel in these engines leads to the formation of various products, which are free from air like exhaust gases. as heat occurs at high temperatures and pressures in the second half, separation and incomplete heat lead to incomplete and complete fire products. eliminate forms such as carbon monoxide (co), carbon dioxide (co2), particulate matter (pm), water (h2o), nitrogen oxides (nox), sulfur oxide (sox) and Hydrocarbon (hc) from heat pollutes the air and has a detrimental effect on human health and environment. when these pollutants are near the earth surface, they cause various health issues like asthma, respiratory problems, and irritation to the eyes and damage to the lungs. away from the



surface, they are directly liable for acid rain and global warming.

For example, Fig 1 and Fig 2 illustrate how the



mounting number of automobiles around the globe has increased the total carbon dioxide footprint in the atmosphere. As a result, before lasting damage is done to nature and mother earth, greenhouse gasses must be controlled.

Figure 1.1: Average Annual Growth in Carbon Dioxide Emissions in percent per year 2012-2040

Figure 1.2: Carbon. Dioxide Emissions by Fuel Type1990-2035(in Billion metric tons)

II. LITERATURE REVIEW

Due to the rapid increase in public transport each year, the pollution control units of the bakkie intensified the annual pollution level. Ultra low emission or zero emission emissions are intended and preferred in the future. Researchers therefore use a more sophisticated method to reduce emissions. Oil refurbishment, engine design modifications and performance parameters and treatment of fossil fuels are some of the major alternatives. Most researchers are focused on fuel conversion, as well as engine construction, but very few researchers are focused on reducing engine emissions.

Yug.al Kishore.al. (2017) "3D CFD study was perform on three colossal conversion methods on the basis of various conclusions. The change rate of NO, CO, C3H6 is a thermal task". At mounting temperature the reaction rate on the outdoor with catalyst first rises and then stabilize. [1]

SP. Vnkatesan et.al (2017) "has a observe to govern catalytic converter using copper oxide. the principle reason of this paintings is to build a device, in which the concentration of poisonous gases is managed by chemical reactions to a more constant level. "So there is no want to separate the mute". The whole assembly is connected to the exhaust pipe from the engine. on this paintings, a catalytic converter containing copper oxide as a catalyst, by incorporating legit catalysts along with platinum, palladium and rhodium is synthetic and integrated into engine exhaust. [2]

Vladimir Lozhkinet.al (2017) "The purpose of the investigation was to develop a mathematical model of the catalysis process in the form of a heat dissipation (extraction) phase switching device and to ensure efficient cleaning of hazardous materials and possible device configuration in the bus engine instead of sound pressure using a calculation method converter on the city cycle. [3]

Ak Sharma et.al (2016) A monolith catalytic converter usually has hundreds or thousands of channels. Modeling of arithmetic that seeks to solve the complexity of combined transport, pressure, types and temperatures on different scales is a challenging task of calculation. The calculation penalty for the reduced model is very small compared to the full model, making it suitable for detailed monolith simulation. [4]

Young-Deuk Kim et.al (2009) in this case, the effective distribution of steel over the length of the converter can affect its performance. The ideal design for the decent metal distribution of the constant value of the catalyst is being investigated to determine the best performance of the dual monolithic catalytic converter. The appropriate design for the optimal axial distribution of catalyst was determined by solving multi-objective development problems in order to reduce both the accumulated CO emissions during the FTP-75 cycle, and the difference between the main value of catalyst distribution work over monolith volume. and a complete control area above the total monolith value. [5]

Thundil Karuppa Raj.Ret.al (2008) analyzes that the formation of a catalytic converter has become vital that requires a whole expertise of the flow of fluid inside the catalytic converter. in this paper, an strive has been made to have a look at the effect of fluid go with the flow due to geometric adjustments the use of the commercial CFD tool. research has been done on the assumption that the liquid is air. The numerical consequences had been used to determine the most geometry required to have a profile of the equal velocity at the doorway to the substrate. [6]

Mingchenet.al, (2008) it's been analyzed that a version is added within the use of the layout of positive transformers. the first step in optimization is the assisted version for measuring catalysts. the second one step concerns the glide performance of the catalyst converter beneath the given geometric limits. The substrate is classified as porous media, where viscous and in all

battle is different by an artistic formula. With the aid of the CFD tool, the converter flow can be made by means of the appropriate layer control method. [7]

Chenet.al (2004) used a 3D CFD drift version as well as a diverse catalytic converter version. They calculate the strain and velocity discipline with the aid of incorporating flow resistance in the monolith substrate. They concluded that the flow subject is motivated with the aid of resistance to the monolith substrate of a selected geometry and the wide variety of Reynolds. further, the uniformity of the go with the flow on the front surface multiplied with the increase in the weight of the monolith and decreased with the increase in the flow rate of the Reynolds. [11]

Chakravarthyet.al (2003) Uses a multi-sided channel model. It was recorded that the ignition behavior may be significantly affected by the re-flow of the flow in the entry area of the substrate leading to poor distribution of high flow rates especially at low exhaust temperatures. The study concluded that the flow imbalance of the results was significantly more important with the increase in flow temperature. In addition, the distribution of the pressure drop remained unchanged and depended on the rotation pattern on the surface of the monolith. [8]

From the above work it can be seen that the following study can be followed:

(i) In the there work the thermal behavior model of a monolithic catalytic converter by numerical change in the lower coldness using cfd to decrease emission will be calculated as a catalytic converter by geometric changes.

(ii) At present work the length of the substrate is increased and reduced by 10mm to strengthen heat dissipation.

(iii) The results will be calculated to reflect the accumulated effect on the reduction of emissions due to geometric changes in relation to the reduction of contaminants by different lengths of the substrate.

III. METHODOLOGY

The present study i.e. "Modeling Of Thermal Behavior Of The Monolithic CATALYTIC Converter By Geometrical Changes In Substrate Length Using CFD" is a simulation study on pre-defined system which is similar to the one in the previous study i.e. "CFD Analysis OF Three Way Monolithic Catalytic Converter Using ANSYS 14.5R" with similar boundary conditions and material, selection also dimension of catalytic converter remains same. The geometry creation and meshing operation is done in simulation software Ansys fluent 14.5 Validation Procedure

As it is stated earlier that the model which is pre-defined in previous study would be followed in this study, Hereby there lies an occasion of validating the new model urban in recreation software ,as the variation of emission for previous model is already known. Similar Plot of secretion drop will be plotted for present model, which would be compared with previous study, since all the boundary conditions as well as height remain same in both model the outcome is expected to remain same hence the existing model would be validated

MODELING AND ANALYSIS

To initiate this study ahead a Three dimensional modal is developed with the help of simulation software used, i.e. Ansys 14.5.The geometrical model thus developed undergoes process of meshing, which is a process of dividing the complete model into small elements so that we could study the variation of various properties like temperature, pressure etc. at various points in the model for a given set of boundary condition which is well known in advance. In this study the crucial parameters under observation are variation of temperature from inlet to outlet.

3.4-COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics (CFD) is a computer-based simulation method for analyzing fluid flow, heat transfer, and conditions related to chemical reactions. This project uses CFD to analyze flow and heat transfer. Some examples of application areas are: gravity increase and drag

(i.e. aircraft or air machine wings), mechanical heating, chemical processes, heating / ventilation, and biomedical engineering (simulating blood flow in arteries and veins). CFD analysis carried out in various industries using R&D as well as the manufacture of aircraft, fire engines, and many other industrial products It would be useful to use CFD over traditional test-based analysis, because testing has a value that is directly proportional to the amount of preparation required for testing, in contrast to CFD, where most results can be produced at no additional cost. In this way, parametric studies of resource development are more expensive for CFD compared to experiments.

The present study i.e. "Modeling Of Thermal Behavior Of The Monolithic CATALYTIC Converter By Geometrical Changes In Substrate Length Using CFD" is a simulation study on pre-defined system which is similar to the one in the previous study i.e. "CFD Analysis OF Three Way Monolithic Catalytic Converter Using ANSYS 14.5R" with similar boundary conditions and material, selection also dimension of catalytic converter remains same. The geometry creation and meshing operation is done in simulation software Annoys fluent 14.5

Validation Procedure

As it is stated earlier that the model which is pre-defined in previous study would be followed in this study, Hereby there lies an occasion of validating the new model urban in recreation software ,as the variation of emission for previous model is already known. Similar Plot of secretion drop will be plotted for present model, which would be compared with previous study, since all the boundary conditions as well as height remain same in both model the outcome is expected to remain same hence the existing

model would be validated

MODELING AND ANALYSIS

To initiate this study ahead a Three dimensional modal is developed with the help of simulation software used, i.e. Ansys 14.5.The geometrical model thus developed undergoes process of meshing, which is a process of dividing the complete model into small elements so that we could study the variation of various properties like temperature, pressure etc. at various points in the model for a given set of boundary condition which is well known in advance. In this study the crucial parameters under observation are variation of temperature from inlet to outlet.

3.4-COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics (CFD) is a computer-based simulation method for analyzing fluid flow, heat transfer, and conditions related to chemical reactions. This project uses CFD to analyze flow and heat transfer. Some examples of application areas are: gravity increase and drag (i.e. aircraft or air machine wings), mechanical heating, chemical processes, heating / ventilation, and biomedical engineering (simulating blood flow in arteries and veins). CFD analysis carried out in various industries using R&D as well as the manufacture of aircraft, fire engines, and many other industrial products It would be useful to use CFD over traditional test-based analysis, because testing has a value that is directly proportional to the amount of preparation required for testing, in contrast to CFD, where most results can be produced at no additional cost. In this way, parametric studies of resource development are more expensive for CFD compared to experiments.

Fig 3.1 Flowchart showing. research methodology

IV. GEOMETRY SETUP AND MODELLING

4.1 DISAGREES

Catalytic Converter is a auto manufacturing control device that converts toxic burning merchandise from the inner combustion engine into less toxic resources in the shape of power-triggered chemical reactions. In catalytic converters, the design of the inlet cone has a great which means, as it determines the distribution of fuel glide over the substrate (a shape located in the converter wherein the response of the extraction response occurs.) The substrate incorporates catalyst lively elements together with Palladium, I -Rhodium and Platinum provoke chemical reactions to convert pollution into less dangerous substances. This substrate can be a metal or ceramic substrate and is modeled as a bold media in CFD.

draw media can be used to model a variety of business application, including flow-through beds, filters, perforated plates, flow distributors, and tube banks. It is generally enviable to specify a pressure drop across a bold path and to conjecture a flow field in order to improve a given design.

Porous middle features are usually defined in the solution according to the following structure:

- **Forgiveness**
- **Viscous resistance**
- **Inertial resistance**

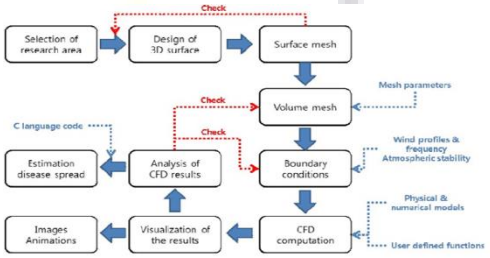
If we have test data for Total Pressure Drop vs. Mass Flow Rate, where we can easily find the viscous and inertial resistance of those perforated media. Strong resistance and internal flow in the inlet can also be calculated from the Ergun equation, which provides a complete reduction of pressure through the inlet. Here,

Total Pressure Depression =

Viscous Loss + Incoming Loss

4.2. Model Description

The geometry of the catalytic converter performing simulation studies is considered to be one



of the paper Yugal Kishore et.al 2017 scientist with a precise size without the size of the substrate scale. Part of the model is built on the ANSYS (Fluent) workbench14.5 software. The geometric dimensions of the catalytic converter are shown in Table4.1. To mimic the monolithic catalytic convertor ANSYS 14.5 a limited control element control method has been used. Where the CAD model of the catalytic converter is divided into 105 orders of nodes

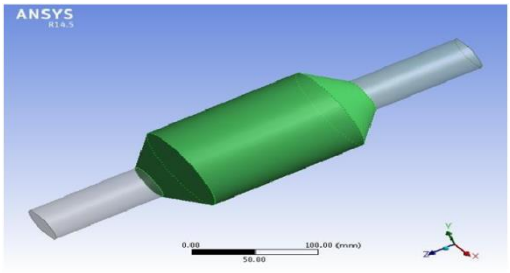


Fig. 4.i Geometry for substrate distance end to end of 190mm

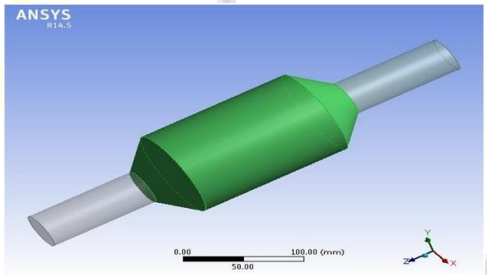


fig. 4.ii Geometry for substrate distance end to end of 200mm.

BOUNDARY CONDITION

Table-Physical Properties for simulation & Composition of combustion gases at the inlet

PARAMETER	VALUE
Cells per square inch, CPSI	400
Substrate volume fraction	0.26
Wash coat volume fraction	0.12
Fluid volume fraction (OFA)	0.62
Hydraulic diameter, Dh, mm	1
Geometric surface area, GSA	2740
Active metal surface	27-28m^2/g
Ratio of active metal surface	70
Wash coat Material	Ceria Stabilized-alumina
Velocity temperature dependent	1.35 m/s at 25°C
Substrate material	Cordierite

group	MASS CONCENTRATION
O_2	0.007%
N_2	80.0 %
HC	0.007%
CO_2	17.9 %
CO	1.1 %
NO_2	0.006%

V. RESULTS

After placing the boundary putting, the solution is initiated and used repeatedly so that the values of all of the parameters are contemplated inside the comma or line graph. After the quit of the generation the final effects may be visible.

To begin with a grid autonomy check was achieved to cast off mistake because of grid imbalances; the look at become achieved with a unique quantity of cells. The CFD model changed into tested using preceding capacity beneath strict conditions. The prediction of models under stable world conditions was as compared with preceding estimates. The fuel supply settings used in this situation are listed in table four. three. The full-size majority of species are categorized as a status feature of the catalytic converter duration from the doorway to the go out. The information display that the effect of the CFD version are in line with preceding estimate.

Contours for substrate length. Of 90 mm

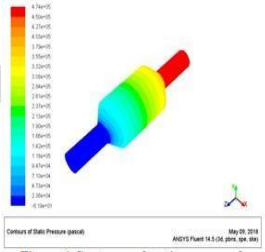


Figure 4 Contours of static pressure for substrate length of 190mm

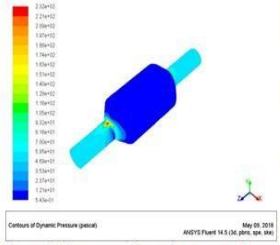


Figure 5 Contours of dynamic pressure for substrate length of 190mm

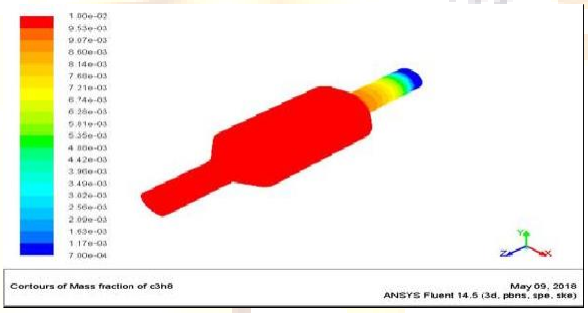


Figure Contours of mass fraction of hydrocarbonfor substrate length of 210mm

CALCULATION OF CONVERSION EFFICENCY

Table I .Conversion rate of oxide of Nitro gen

Inlet Temperature(In K)	Conversion Rate of Oxide of Nitrogen		
	For substrate length of 190mm	For substrate length of 200mm	For substrate length of 210mm
800	0.918	0.932	0.952

Table I I Conversion rate of propane

Inlet Temperature(In K)	Conversion Rate of Propane		
	For substrate length of 190mm	For substrate length of 200mm	For substrate length of 210mm
800	0.889	0.9	0.921

Table III Conversion rate of carbon monoxide

Inlet Temperature(In K)	Conversion Rate o f Carbon Monoxide		
	For substrate length of 190mm	For substrate length of 200mm	For substrate length of 210mm
800	0.839	0.85	0.868

VI. CONCLUSIONS

From the model effects it's far observed that:

- The drift distribution in a catalytic converter assembly is governed through the geometry configurations of inlet and outlet cone segment, the substrate and exhaust gasoline compositions and therefore a higher layout of the catalytic converter is very vital.
- there's a difference of 2.4% for oxide of nitrogen, 2.1% for propane and 1.eight % for carbon monoxide increase in conversion efficiencies by way of mounting the substrate length with the aid of 10mm
- the prevailing proposed model has 2.four % better NOx change rate as compared to other species.
- The order of conversion price is as follows NO>C3H6>CO.
- For the present day layout of 210 mm substrate length configuration, exhaust gas conversion efficiency was discovered to be top-rated.

REFERENCES

[1] Yugal Kishore et. al CFD analysis of three way monolithic catalytic converter using Ansys 14.5r, International Journal of Mechanical And Production Engineering, Volume- 5, Issue-11, Nov.-2017.

[2] S.P. Venkatesan et al I.C. Engine emission reduction by copper oxide catalytic converter. Materials Science and Engineering 197 (2017) 012026 doi:10.1088/1757- 899X/197/1/012026

[3] Vladimir Lozhkinet al Catalytic Converter with Storage Device of Exhaust Gas Heat for City Bus. Transportation Research Procardia 20 (2017) 412 – 417

[4] A.K. Sharma et al Validity and scalability of an asymptotically re duce d single-channel model for full-size catalytic monolith converters Applied Mathematics and Computation 281 (2016) 186–198

[5] Young-Deuk Kim Optimal design of axial noble metal distribution for improving dual monolithic catalytic converter performance Chemical Engineering Science 64 (2009) 1373 – 1383

[6] ThundilKaruppaRaj.R and Ramsai.R "Numerical Study of Fluid Flow And Effect of Inlet Pipe Angle In Catalytic Converter Using CFD", School of Mechanical and Building Sciences, VIT University, Vellore– 632 014,Tamil Nadu, INDIA.

[7] Ming Chen, Karen Schirmer "A modeling approach to the design optimization of catalytic converters of I.C. Engines" PROCEEDINGS OF ICEF03: 2003 Fall Technical Conference of the ASME Internal Combustion Engine Division, Pennsylvania,USA, September 7-10, 2008.

[8] Chakravarthy, V. K., Conklin, J. C., Daw, C. S., &D'Azevedo, E. F. (2003). Multi- dimensional simulations of cold-start transients in a catalytic converter under steady inflow conditions. Applied Catalysis A: General, 241(1–2), 289–306.

[9] Young-Deuk Kim Optimal design of axial noble metal distribution for improving dual monolithic catalytic converter performance Chemical Engineering Science 64 (2009) 1373 – 1383

[10] Thundil Karuppa Raj.R and Ramsai.R "Numerical Study of Fluid Flow And Effect of Inlet Pipe Angle In Catalytic Converter Using CFD", School of Mechanical and Building Sciences, VIT University, Vellore– 632 014,Tamil Nadu, INDIA.

[11] Ming Chen, Karen Schirmer "A modeling approach to the design optimization of catalytic converters of I.C. Engines" PROCEEDINGS OF ICEF03: 2003 Fall Technical Conference of the ASME Internal Combustion Engine Division, Pennsylvania,USA, September 7-10, 2008.

[12] Chakravarthy, V. K., Conklin, J. C., Daw, C. S., & D'Azevedo, E. F. (2003). Multi-dimensional simulations of cold-start transients in a catalytic

converter under steady inflow conditions. Applied Catalysis A: General, 241(1–2), 289–306.

[13] Ekström, F., & Andersson, B. (2002). Pressure drop of monolithic catalytic converters experiments and modeling (No. 2002-01-1010). SAE Technical Paper.

[14] Karvounis, E., & Assanis, D. N. (1993). The effect of inlet flow distribution on catalytic

[15] A. Khanfekr, K. Arzani, A. Nemati, and M. Hosseini, "Production of perovskite catalysts on ceramic monoliths with nanoparticles for dual fuel system automobiles", International Journal of Environmental Science and Technology, vol. 6 (1), pp. 105-112, 2009.