Research Journal of Recent Sciences ISSN 2277-2502
Vol. **1(7),** 39-44, July **(2012)** *Res.J.Recent Sci.*

Numerical study of fluid flow and effect of inlet pipe angle In catalytic converter using CFD

Thundil Karuppa Raj R. and Ramsai R.
School of Mechanical and Building Sciences, VIT University, Vellore– 632 014, Tamil Nadu, INDIA

Available online at: www.isca.in
(Received 27th March 2012, revised 5th April 2012, accepted 9th April 2012)

Abstract

Catalytic converter has become a necessity to achieve low emissions in all the vehicles. The design of catalytic converter has become critical which requires a thorough understanding of fluid flow inside the catalytic converter. In this paper, an attempt has been made to study the effect of fluid flow due to geometry changes using commercial CFD tool. The study has been conducted assuming the fluid to be air. A section of catalytic converter has been solved for analysis due do its rotational symmetry. The substrate region is modeled as a porous medium. The governing equations namely conservation of mass, momentum will be solved for analysis. The predicted numerical results are validated with those available in literature. The analysis involved determining back pressure across the converter system for mass flow rates and inlet pipe angle. The numerical results were used determine the optimum geometry required to have a uniform velocity profile at the inlet to the substrate.

Keywords: Catalytic converter, rotational symmetry, CFD.

Introduction

As global automotive emission standards become more stringent, several efforts have been taken to determine the source of emissions and development of new technologies for controlling regulated and non-regulated emissions. An Automotive catalytic converter usually consists of a round, or oval shaped, monolith reactor encased in a metallic shell, and connected to the exhaust system through inlet and outlet cones. Mainly, NOx,CO, and unburned hydrocarbons (HC). Significant effort has been invested into the design of a converter that will lead to maximum use of the catalyst volume. It is known that this maximum utilization of the catalyst volume would be achieved by having a uniform flow distribution through the monolith substrate.

Therefore, most modern catalytic converters have long, tapered inlet and outlet headers to smooth the flow between sections of different cross-sectional areas. This tapered header provides a uniform flow distribution across the monolith inlet face. A non-uniform flow across the substrate leads to uneven residence time distribution and non-uniform poison accumulation during the catalyst aging.

In the past, some papers have studied the flow in round cross-section monolith converters with conical inlet and outlet headers. The study found that the monolith flow field to be extremely maldistributed[1]. The effect of truncating the inlet and outlet diffusers of a monolith catalytic converter was found in to be insignificant[2]. Another study, confirmed these findings through water-visualization tests on full-scale transparent model of a double-brick converter with tapered inlet and outlet headers[3]. An experimental work has shown that dynamic flow characteristics were different from those under steady flow conditions in the catalytic converter[4]. Other researchers have looked at the effect of engine operating conditions on the converter temperature[5,]. Several recent studies, have investigated the effect of the flow on chemical reactions using one-dimensional unsteady models three-dimensional transient models[6,7]. Others have studied the effect of the substrate cell size and shape[8,9].

An experimental optimization of the design parameters of a catalytic converter is extremely expensive and time consuming. The design process involves building several prototypes with different geometries for experimental testing[10,11]. These models must be absolutely exact, since the flow inside a catalytic converter is extremely sensitive to geometric deviations. Stereo-lithographic manufacturing of plastic models from CAD data has proved to be an exact method and a useful tool for experimental investigation of internal flow devices. However, it is also an expensive and time-consuming method. Hence, a computational approach to the design optimization of catalytic converters is more feasible[12,13].

This paper involves numerical study to perform three-dimensional calculations of turbulent flow in an inlet pipe, inlet cone, catalyst substrate (porous medium), outlet cone, and outlet pipe using computational fluid dynamics (CFD). Very often, the designer may have to resort to offset inlet and outlet cones, or angled inlet pipes due to space limitations. Hence, it is very difficult to achieve a good flow distribution at the inlet cross section of the catalyst substrate[14]. Therefore, it is important to study the effect of the geometry of the catalytic converter on flow uniformity in the substrate.

Material and Methods

Experimental Work: The catalytic converter geometry considered for study is shown in figure-1. The dimensions are shown in table- 1.The straight section of the system contains the monolith (catalyst) substrate. A typical catalytic converter consists of a catalyst substrate, mat-insulation material, and an outer metallic shell. The monolith substrate consists of a large number of small channels with 350 cells per square inch or cpsi. The cells are originally square ducts. However, after a washcoat is applied, the cells cross section becomes more circular. The experimental work includes using hotwire anemometry to measure the velocity profile at the outlet of the catalyst substrate, and pressure drop measurements across the system using air as working fluid at 873K[10]. table-2 shows the plot of superficial velocity and pressure drop for different mass flow rates.

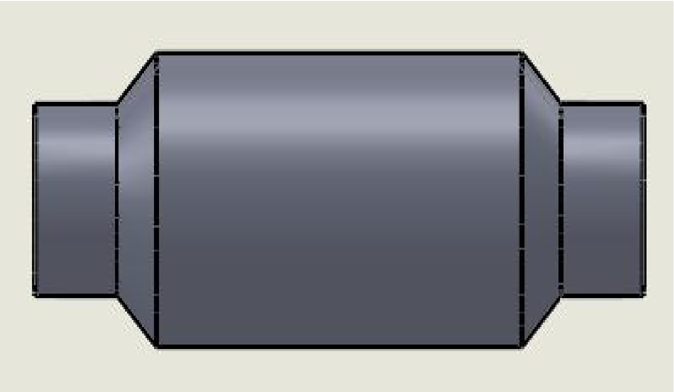

Figure-1
Catalytic converter

Table-1
Catalytic converter dimensions

Inlet pipe diameter	1.875 in.
Substrate diameter	3 in.
Substrate length	4.5 in.
Cone angle	45 degrees
Pipe length	1.24 in.

Table-2
Superficial velocity and pressure drop for different mass flow rates[10]

Flow rate(g/s)	Superficial Velocity(m/s)	Pressure drop(Pa)
10	5.4	690
20	10.77	1400
40	21.38	2900
80	42.1	6160
110	57.16	8800
150	76.59	12550
190	95.27	16500
220	108.8	19560
250	121.9	22700

Computational modeling and grid generation: The 3-D model is modelled in ICEM CFD pre-processing tool. A 90 degree sector of catalytic converter is modelled for analysis using ICEM CFD due to its rotational symmetry. In order to capture both the thermal and velocity boundary layers the entire model is discretized using hexahedral mesh elements which are more accurate and involve less computation effort. Fine control on the hexahedral mesh near the wall surface allows capturing the boundary layer gradient accurately. The catalytic converter is divided into four domains inletpipe, inlet cone, substrate and outlet for the sake of parameters study. The discretized model is checked to have a minimum angle of 22°and min determinant quality of 65 %. The fluid domains are shown in figure-2.

Governing equations and boundary conditions: The 3-dimensional heat flow through the cylinder and fins are simulated by solving the appropriate governing equations viz. conservation of mass, momentum using ANSYS CFX code. The equations are shown in equations 1,2, 3 and 4 respectively. The simulations are conducted in a three-dimensional geometry under steady-state flow conditions. Convergence of the solution is achieved when the normalized absolute residual sum drops below a user-specified value, typically 10^{-4}. Heat transfer from the fluid is not considered and hence the fluid is considered to be isothermal. Turbulent flow is assumed in the inlet and outlet pipes and cones. The hydraulic diameter of a channel is of the order of 1.167mm for 350 cpsi, respectively, the corresponding Reynolds number results in a laminar flow in the channels. The linear and quadratic resistances are found using the Darcy's relation from the experimental data shown in figure-3. Δp is the pressure drop across the substrate. U is the superficial velocity. L is the length of the substrate. It was used as input for porous medium.

The standard k-epsilon turbulence model is selected to calculate the turbulent flow. The monolith substrate, though it consists of a large number of channels, is modeled as a porous medium to simplify the geometric model and numerical calculations.

Conservation of mass: $\nabla \cdot (\rho \vec{V}) = 0$ (1)

Conservation of x-momentum:

$$\nabla \cdot (\rho u \vec{V}) = -\frac{\partial \rho}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + \rho g \tag{2}$$

Conservation of y-momentum:

$$\nabla \cdot (\rho u \vec{V}) = -\frac{\partial \rho}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} + \rho g \tag{3}$$

Conservation of z-momentum:

$$\nabla \cdot (\rho u \vec{V}) = -\frac{\partial \rho}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho g \tag{4}$$

The walls are assumed to have smooth surface. For the analysis, buoyancy and radiation effects are neglected. Grid independence study started with a coarse mesh and gradually refined to finer mesh. Number of nodes used is around 4,50,000. Figure-4 shows the catalytic converter created in ANSYS CFX 12.1 pre processor tool after applying the boundary conditions.

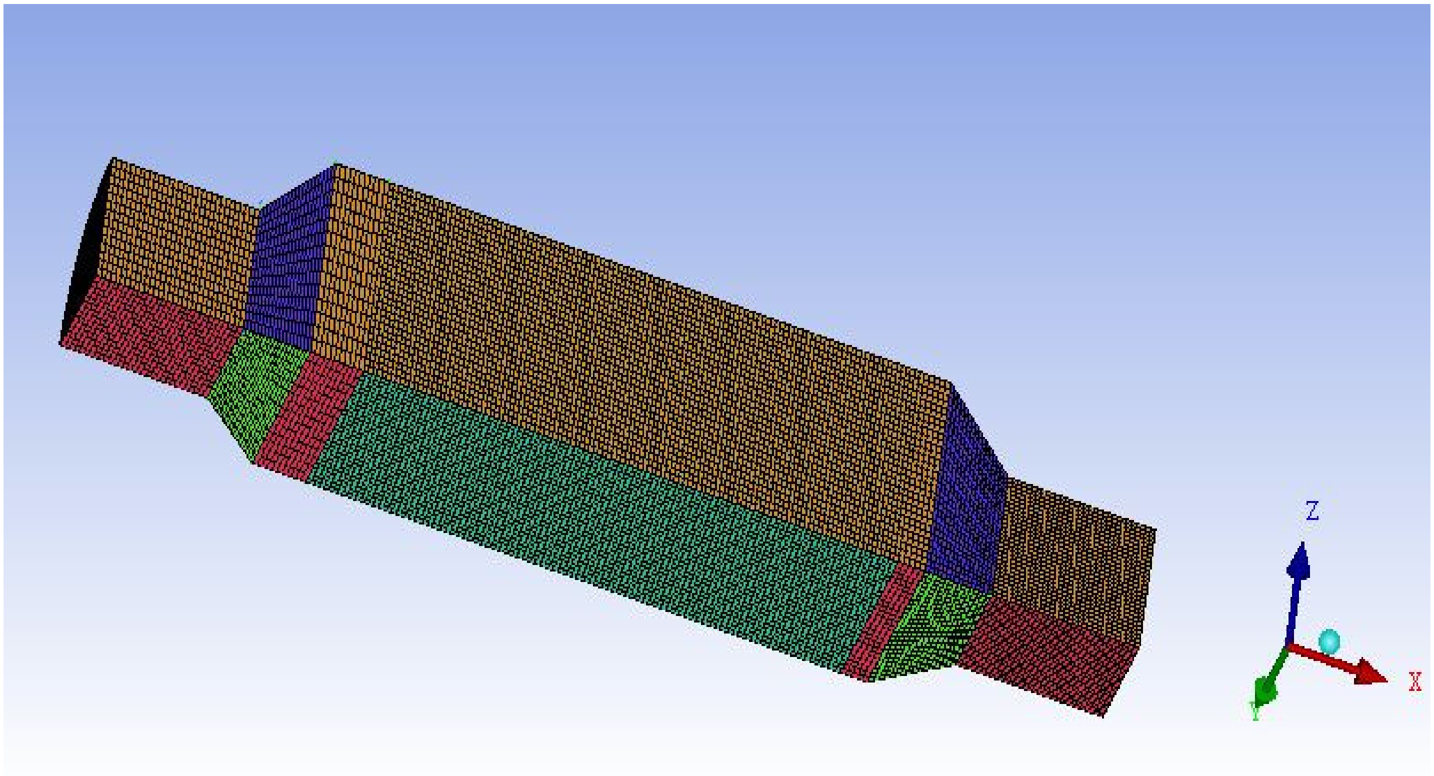

Figure-2
90 deg sector after discretisation

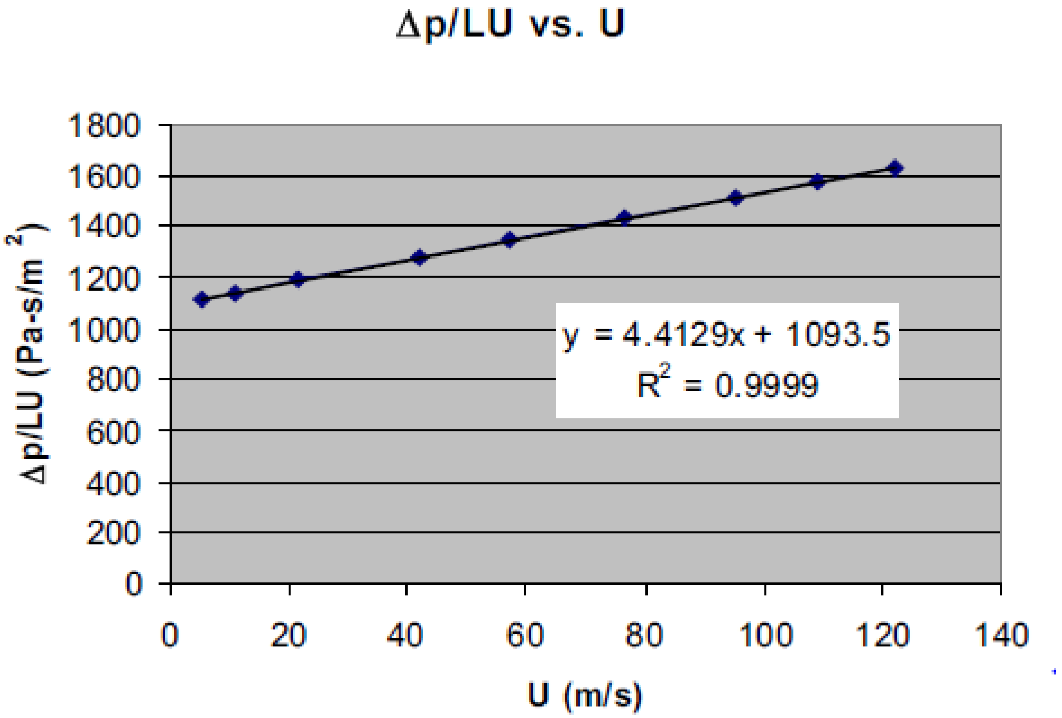

Figure- 3
Plot of Δp/LU vs U

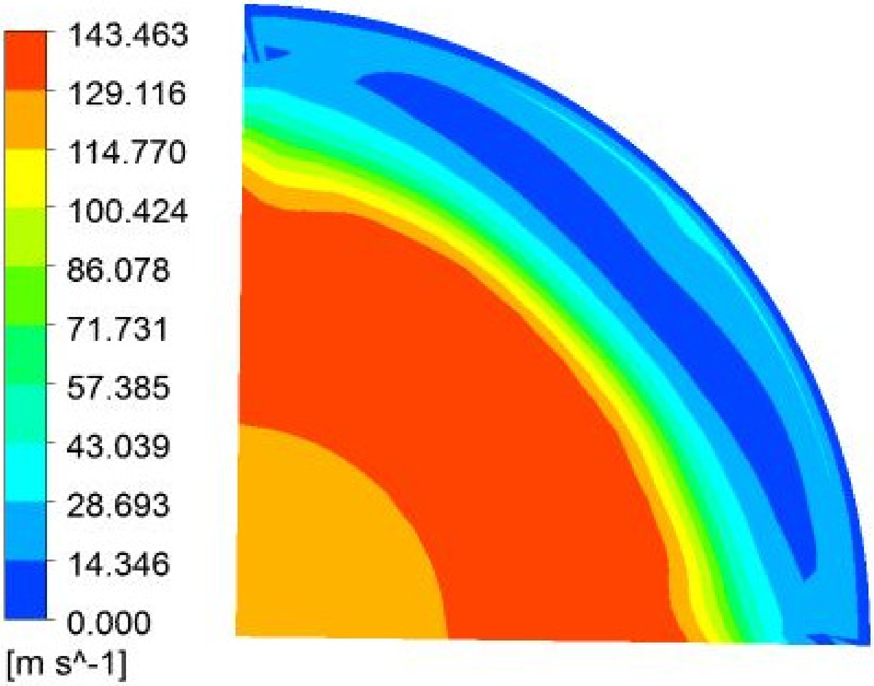

Figure-4
Velocity distribution on the substrate inlet for 0 degree angle inlet pipe

Results and Discussion

For the validation, the pressure drop across the substrate is measured for different mass flow rates. The model showed a good conformance with the experimental results with the maximum deviation around 7%.The numerical results are shown in table-3. Thus numerical model is used for the study purpose.

Table-3
Numerical pressure drop for different mass flow rates

Flow rate(g/s)	Pressure drop(Pa)
10	710
20	1508
80	6566
150	13370

The velocity distribution on the inlet of substrate is shown in figure-5.The velocity is less near the walls. The velocity is gradually increases and reaches maximum and again drops. There is also a patch of low velocity section in between the substrate layers. This is a indication of misdistribution. This makes the flow in the substrate non-uniform. This can be corrected by changing the cone angle, diameter of inlet pipe and angle of inlet pipe. Since the designer has to confront space constraints, the study is conducted for fixed inlet diameter and cone angles. The velocity profile is studied for different inlet pipe angles. The inlet pipe angles used are 30,45 and 60 degrees. The discretized model with 45 degree inlet pipe is shown in figure-4.

The mass flow rate considered for study is 150 g/s. The velocity distribution for 30degree inlet pipe is shown in figure-6.The velocity is highly non uniform on the substrate inlet.

The velocity distribution for 45 degree inlet pipe is shown in figure-6.The velocity distribution is uniform on the substrate inlet. Since the cone angle is also 45 degrees, it guides the fluid to the substrate. The peak velocity of air is moderate.

The velocity distribution for 60 degree inlet pipe is shown in figure-7. The velocity distribution is uniform than 60 degree. But the peak velocity is very high. This local peak velocity will be detrimental for the substrate. Also, in this case an extra back pressure is observed as the higher angle acts as a flow restriction. The non-uniformity increases in the substrate with the increase in mass flow.

Conclusion

The procedure of modeling the catalyst substrate as a porous medium in ANSYS CFX was successful. The numerical simulations performed on the catalytic converter internal flow agree with the experimental values of pressure drop across the substrate. The results show that the converter geometry has a significant effect on flow distribution in the monolith substrate. Moreover, the flow in the catalytic converter with appears to be less uniform for lower angles. The flow tends to create some additional backpressure for higher angles. The flow tends to be more uniform if the angles are closer to inlet cone angles. The results show an increase in flow non-uniformity in the substrate with an increase in mass flow rate. These results will aid the designer when using truncated and angled inlet and outlet cones.

References

1. Howitt J.S. and Sekella T.C., Flow Effects in Monolithic automotive Catalytic Converters, *SAE paper* 740244, **(1974)**
2. Wendland D.W., and Matthes W.R., Effect of Head Truncation on Monolith Converter Emission-Control Performance, *SAE* paper 922340, **(1992)**
3. Hwang K., Lee K., Mueller J., Stuecken T., Schock H. and Lee J.C., Dynamic Flow Study in a Catalytic Converter Using LDV and High Speed Flow Visualization, *SAE* paper 950786, **(1995)**
4. Lee S., Bae C., Lee Y. and Han T., Effects of Engine Operating Conditions on Catalytic Converter Temperature in an SI Engine, *SAE* paper 2002-10-1677, **(2002)**
5. Cho Y.S., Lee Y.S., Kim D.S., Jung S.Y. and Ohm I.Y., An Alternative Method for Fast Light-Off of Catalysts – Cranking Exhaust Gas Ignition, *SAE* paper 2002-01-1678, **(2002)**
6. Gregory D., Read M., Campbell B., Inman G., Nice G., Hims R., Rabinowitz H., Tauster S., and Collin T., Emissions Implications of a Twin Close Coupled Catalyst System Designed for Improved Engine Performance on an In-line 4 Cylinder Engine*, SAE* paper 2001-01-1092**, (2002)**
7. Onorati A., Ferrari G., and Derrico G., 1 D Unsteady Flows with Chemical Reactions in the Exhaust Duct-System of SI Engines: Predictions and Experiments, SAE paper 2001-01-0939, **(2001)**
8. Jeong S.J., and Kim W.S., Three-Dimensional Numerical Study on the Use of Warm-up Catalyst to Improve Light-Off Performance, *SAE* paper 2000-01-0207, **(2000)**
9. Jinke Gong, Longyu Cai, Weiling Peng and Jingwu Liu, Yunqing Liu, Hao Cai and Jiaqiang E, Analysis to the Impact of Monolith Geometric Parameters on Emission Conversion performance Based on an Improved Three-way Catalytic Converter Simulation Model, *SAE* paper 2006-32-0089 **(2006)**
10. Bassem H. Ramadan and Philip C**.** Lundberg,Characterization of a Catalytic Converter Internal Flow, *SAE paper*, 2007-01-4024 **(2007)**
11. Karthikeyan S., Hariganesh R., Sathyanandan M., Krishnan S., Computational and Experimental Investigation on After-Treament Systems to Meet Future Emission Norms for Truck Applications, *International Journal of Engineering Science and Technology*, **3(4)**, 3314-3326 (**2011**)
12. Kumar Krishan and Aggarwal M.L., A Finite Element Approach for Analysis of a Multi Leaf Spring using CAE Tools, *Research Journal of Recent Sciences,* **1(2), 92-96**, **(2012)**
13. Dev Nikhil, Attri Rajesh, Mittal Vijay, Kumar Sandeep, Mohit, Satyapal and Kumar Pardeep, Thermodynamic Analysis of a Combined Heat and Power System, *Research Journal of Recent Sciences,* **1(3),**76-79 **(2012)**
14. Wu Guojiang,Tan Song, CFD Simulation Of The Effect Of Upstream Flow Distribution on the Light-Off Performance of a Catalytic Converter, *Elsevier*, **46(13),** (**2005**)

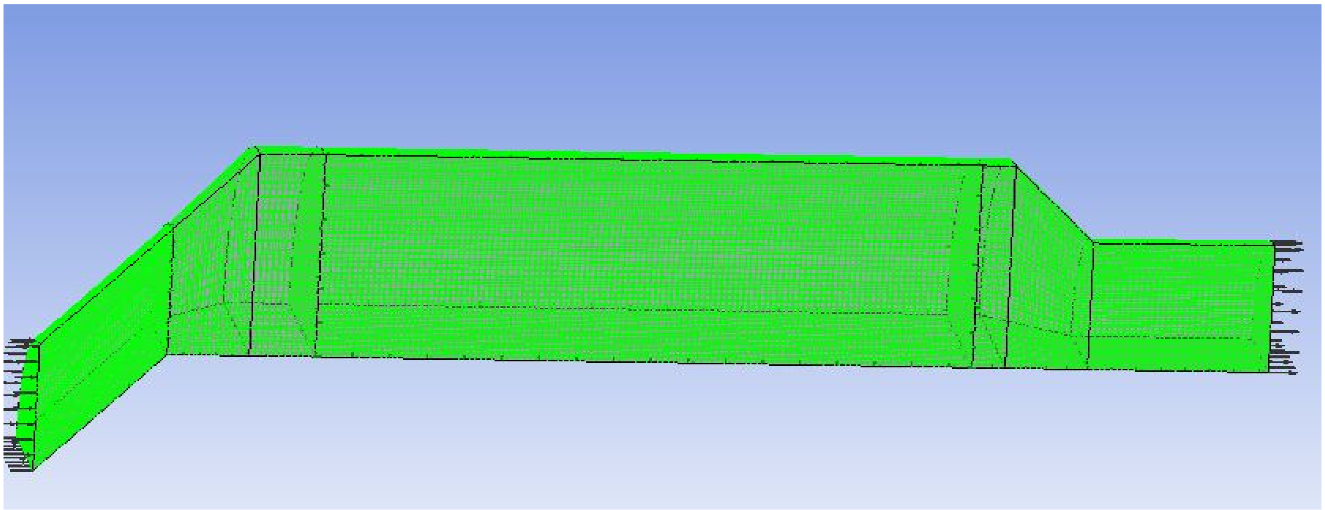

Figure-5
Catalytic converter with 45 degree inlet pipe

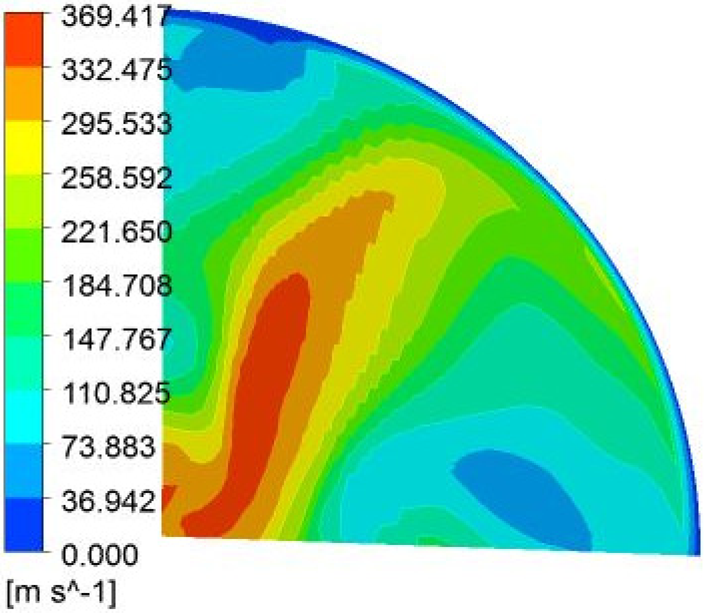

Figure 6
Velocity distribution on the substrate inlet for 30 degree angle inlet pipe

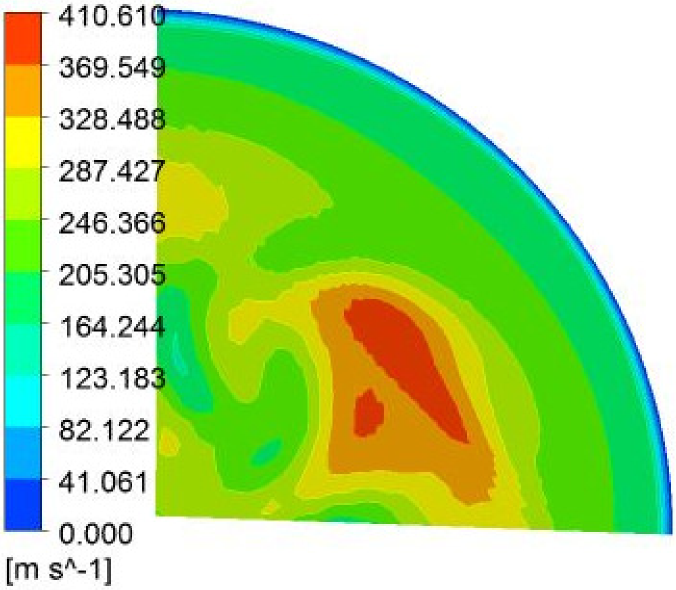

Figure -7
Velocity distribution on the substrate inlet for 45 degree angle inlet pipe

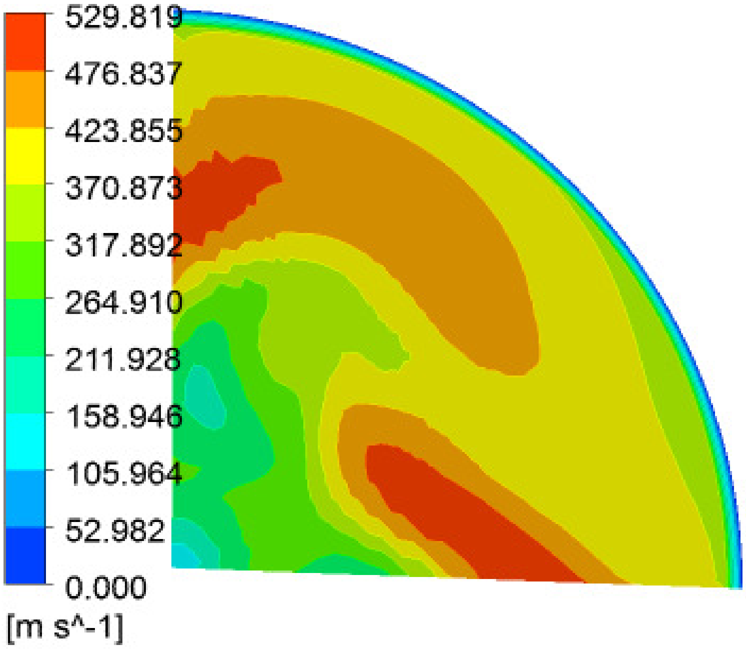

Figure-8
Velocity distribution on the substrate inlet for 60 degree angle inlet pipe