Estimation of Pressure Drop across Exhaust System by CFD Simulation

Madhavi Swamy[1], Renuka Kulkarni[2], Radhika Dharmadhikari[3], Suman Rajput[4]

[1-4]Dept of Mechanical Engineering, MKSSS's Cummins college of Engineering, Maharashtra, India

---***---

Abstract - *With the BS VI Emission Norms becoming stringent in India, there will be an extensive use of the After-treatment technology system. The After-treatment system primarily consists of the DPF (Diesel Particulate Filter) followed by the DOC (Diesel Oxidation Catalyst) and SCR (Selective Catalytic reduction). These systems reduce the emissions of greenhouse gases such as CO, CO2 and NOx. Every engine exhaust system has a certain amount of Backpressure, similarly the After-treatment systems also adds in some amount of Backpressure. Backpressure is normally defined as the resistance to the unidirectional flow of the exhaust stream. The Backpressure or Pressure drop across the inlet and outlet of after treatment system has some effects on engine performance parameters such as specific fuel oil consumption, exhaust gas flow and temperature. Therefore, it is important to know whether the recorded pressure drop fits in the allowable range or not. We have tried to project the flow of the exhaust gases through the DOC, mixing pipe and SCR using the Star ccm+ software.*

Key Words: Backpressure, Exhaust, Emission, After-treatment, Greenhouse gases, Porosity

1. INTRODUCTION

Backpressure is majorly caused due to installation of certain systems like silencer, muffler or aftertreatment system. Every such combination would develop backpressure. When this backpressure exceeds the allowable limit, it might cause a failure in the system. Therefore, through this paper, backpressure of a diesel engine is determined and the flow simulation of exhaust gas pipeline is also made. Porous Medium is characterized by various geometrical properties but for the purpose of mathematical modelling of the flow through them, we consider only specific parameters such as porosity, permeability and tortuosity. For establishing a relation between these parameters in case of a porous medium flow we made use of Darcy's Law that states the discharge (Q) is proportional to the dynamic viscosity (μ) and permeability of the fluid and the pressure drop across the inlet and outlet walls of the porous zone. The permeability of any porous wall is a function of its porosity, pore size distribution and the morphology of the porous system. The Law can be equated as:

$$Q = \frac{\kappa}{\mu} \quad \Delta P \tag{1}$$

Using Computational Fluid Dynamics software has become a popular trend for avoiding several iterations of tests that may also be time consuming and expensive. For our CFD analysis, we have used the Star ccm+ software. The results of any CFD simulation are in the form of pressure, velocity contours and streamline. In usual CFD practise the geometries are divided into small elements or cells, the collection of these subdomains is called a Mesh. We used the Finite Volume Method to develop a Three-Dimensional Model. Accordingly, the use of certain solvers was made to compute the differential equations.

The After-treatment system consists of the DOC, mixing pipe followed by the SCR. These geometries are quite complex and different from each other in terms of their cross-sectional areas. In such cases unstructured grids are used due to their greater adaptability to complex geometries. Unstructured meshes namely Polyhedral and Tetrahedral were majorly used in this study. For mesh generation our focus was to make finer mesh in order to achieve accurate approximate results [5]. SCR and DOC systems make use of a porous substrate. The porous medium is referred to as a solid body with pores. The nonsolid spaces are called pore space. Porosity is an index of the amount of fluids that can be stored within the medium or the pores. Ceramic particulate filters are used in DOC and in SCR to minimize the effect of pollutants by capturing the pollutant particles in porous medium. To study the flow through porous medium, a thorough study of Darcy's Law should be done. Darcy's law is used to calculate pressure drop across inlet and outlet walls in case of porous medium. As velocity increases flow becomes turbulent and flow resistance becomes nonlinear hence modified Darcy's Law known as Horkheimer extension is more appropriate. Following assumption were made while considering the physics conditions:

1. Steady flow (Time)
2. Turbulent flow
3. Air is considered as working fluid

For the Physics conditions certain solvers were used. First and foremost, we used the Reynolds Averaged Navier Stokes (RANS) model to solve the second order turbulent flow equations. Physics models solves the Navier-stokes equation. The local oscillations in the flow field are solved in a time averaged sense. RANS model consists of several turbulent solvers. The transport equations are solved using these solver models. We have used the (kinetic energy) k-Epsilon model.

2. MODELLING

The Geometry of the Aftertreatment system was built using CREO and converted into (.stl) format. This makes it easier to load into any software. Further on the Geometry is loaded on the software using the "Load" tool. The entire geometry was divided into on the software three main parts. Considering the complexity of the Aftertreatment geometry, we bifurcated the model, making it easy to compute result.

- Diesel Oxidation Catalyst (DOC)
- Mixer pipe
- Selective Catalytic Reduction (SCR)

3. SURFACE REPAIR

The Surface Repair tool diagnoses errors, and provides repair tools for surfaces and feature curves. Before generating the surface mesh, the validity of the surfaces on the geometry must be taken into consideration. For multiple regions having multiple parts, surfaces can be imprinted at the interface between them [1]. The Surface Repair tool is applicable only to selected surfaces, the changes do not affect the overall geometry. The Repair tool modes are presented in the Edit panel; display, selection, and diagnostics functions are included within the Graphics window. The diagnostics slide out is located on the right side of the Graphics window in STAR-CCM+. It contains a list of thresholds and issues that the Surface Repair tool identifies. The colored boxes in the Diagnostics slide represent different types of Errors. They are displayed in CAD Geometry which is present in surface repair tool. LIST of Errors in the Diagnostics Window [1]:

1. Pierced Faces

When a face is intersected by one or more edges, it is termed as a pierced face. It is represented by RED colour in the Geometry.

2. Face Quality [Yellow]

Face quality is a measure of similarity between a face which is an equilateral triangle. It is represented by YELLOW colour in the Geometry.

3. Face Proximity

The distance between two faces is represented by the face proximity. It is represented by ORANGE colour in the Geometry.

4. Free Edges

When an edge is joined to only one face, it is termed as a free edge. It is represented by GREEN colour in the Geometry.

5. Non-Manifold Edges

A non-manifold edge is an edge that three or more other faces share. It is represented by BLUE colour in the Geometry.

6. Non-Manifold Vertices

When the connected faces do not have any link to other faces, it is a non-manifold vertex. It is represented by BLUE colour in the Geometry.

A duplicate face is a face that shares three vertices with another face. The surface repair tool removes the duplicate faces and merges them.

4. MESHING

The conventional method of selection of the mesh type usually depends upon the complexity, size and shape of the geometry. This discretization of the model is a very important part of the computational analysis. Meshing is significant as it affects the rate of convergence, time required for simulation and solution accuracy. Through various reference and studies, we found out polyhedral mesh gives highest rate of convergence with acceptable solution accuracy over Tetra and Hexa mesh. The mesh adjacent to wall should be fine to capture the effect of turbulence thus we used prism layer thickness.

Table -1: Mesh sizes used for each part of he After-treatment system

Component	Base size(mm)	Element type
1.Diesel oxidation catalyst (DOC)	4	polyhedral
2. mixer pipe	4	polyhedral
3. selective catalytic reduction (SCR)	4	tetrahedral

4.1 Mesh Quality

After completing generation of mesh whether quality of mesh is within the acceptable range or not that has to be check.

1. *Skewness*- it is basically a geometrical orientation of cell. For perfect mesh it should be 0 and for the worst it is 1.

2. *Aspect ratio*- it is a measure of stretching of cell. It is the ratio of maximum edge length to minimum edge length of an element. Aspect ratio less than 5 is accepted. Having large aspect ratio result in interpolation error.

3. *Smoothness* (cell jump) (less than 20%)-there should not be sudden change in cell size that means it should be smooth.

1. Import Surface : Imports surface data for generating a mesh.

2. Launch Surface Repair : Opens the Surface Preparation dialog for regions or parts.

3. Clear Generated Meshes : Deletes meshes generated in STAR-CCM+.

4. Initialize Meshing : Provides the starting surface for the meshing models.

5. Generate Surface Mesh : Begins the generation of a surface mesh.

6. Generate Volume Mesh : Begins the generation of a volume mesh.

Fig -1: Figure demonstrating steps to generate Volume Mesh

5. REGIONS

Regions are assigned types that models within their associated continuum use to determine what specific conditions and values are required. Regions were established for those parts that required a boundary condition input.

6. SELECTION OF PHYSICS CONTINUUM

In any Computational Fluid Dynamics, the input of the physics and solver values decides the output of fluid application. Therefore, it is necessary to identify and define the physics structure of the flow such as the boundary conditions, reference values, initial conditions, fluid state, etc. [3]. After calculating the Reynolds' number, it was found out that the flow lies in the turbulent region. It was known considering the real-life conditions: the exhaust gas at the exhaust manifold is at high pressure

and high temperature resulting into a turbulent environment at the DOC system inlet. For most of engineering problems widely acceptable models are Spalart-Allmaras, SST and k-epsilon/k-omega which provide a great agreement between cost and accuracy. RANS have the many physical model for turbulence.

RANS Model- In Reynolds Averaged Naiver Stokes, the Naiver stokes equations for the instantaneous pressure and velocity field are decomposed into their fluctuating component and mean value. By doing this averaging process additional unknowns get introduced hence to model this additional term known as Reynolds stress tensor two approaches are available: A) Turbulent-viscosity models B) Reynolds -stress model

6.1. Turbulent-viscosity models

Single-equation model

a) Spalart-Allmaras

Two-equation model

a) Standard k-epsilon – In these two variables are to be solved one the turbulent kinetic energy (k), and the rate of dissipation of kinetic energy (€). It uses wall function unlike Spalart-Allmaras. This is simplest model available to model turbulence with the help of two transport equation to solve for characteristics length for length and viscosity. These scales represent the turbulent viscosity which considered to be isotropic that is ratio between Reynolds stress and mean rate of deformations is same in all directions.

Rate of change of kinetic energy and dissipation rate + Transport of both variables by advection= Transport of kinetic energy and dissipation rate by diffusion + Rate of production of kinetic energy and dissipation rate – Rate of destruction of k & e

b) Realizable k-epsilon

c) RNG k-epsilon

d) k-omega - aim is to model near wall function more accurately than k-epsilon models. it has a limitation in difficulty of convergence compared to K-epsilon.

e) Large eddy simulation (LES) and Detached-eddy simulation (DES)- uses to predict large eddy with fine mesh resulting in highest computational and programming cost.

By considering advantages and limitation of above-mentioned turbulence models we have selected two- equation K-Epsilon model for our simulation. The k-epsilon model shows good result for wall function and free shear flows. Star cmm+ provides various available k-epsilon model we have used realizable k-epsilon two-layer model which is different than standard k -epsilon model that it has new formulation of turbulent viscosity and new transport equation for dissipation rate with two-layer approach. this two-layer approach has the flexibility of an y+ wall treatment.

7. POROUS MEDIA

Regions that are assigned to gas or liquid continua can be designated with the porous type. The physical models, such as flow, energy, and turbulence, make the appropriate modelling assumptions governing porous media flow in these regions. Following models can be used in combination:

1. Three-Dimensional Gas

2. Ideal Gas

3. Turbulent

4. Steady

5. Reynolds Averaged Navier Stokes

6. Reynolds K-Epsilon Two Layer

7. Segregated Flow

Segregated Fluid Temperature

8. ALGORITHMS TO SOLVE FLOW EQUATIONS

Coupled flow- solves the flow equation in coupled manner that is simultaneously for velocity component and pressure. Gives accurate result for compressible flow in the present off shocks. Requires high CPU time and computational cost.

Segregated flow- solves the flow equation separately one for each velocity components and one for pressure. Suitable for incompressible flow. By keeping merits and demerits and degree complexity of problem we chose segregated solver which provide great trade-off between computational time and solution accuracy.

Boundary conditions are necessary as they are a part of CFD. Boundaries are assigned types that models within their associated continuum use to work out and identify what conditions and values are required.

9. CONCLUSIONS

The study is based on two different temperature conditions and hence following can be concluded:

A 3D physics model that describes the porous flow, temperature dependent cold flow and the compressible gas flow is established. Expected pressure drop was obtained with the help of Star ccm+ software. Convergence was obtained successfully by this model.

The pressure drop across the after-treatment system was computed and compared to Bharat Standard (BS) VI engine test bed. Comparing the developed pressure drop experimental data and the analytical data obtained through simulation, a percentage difference of 5-12% was obtained and this lies between the acceptable range.
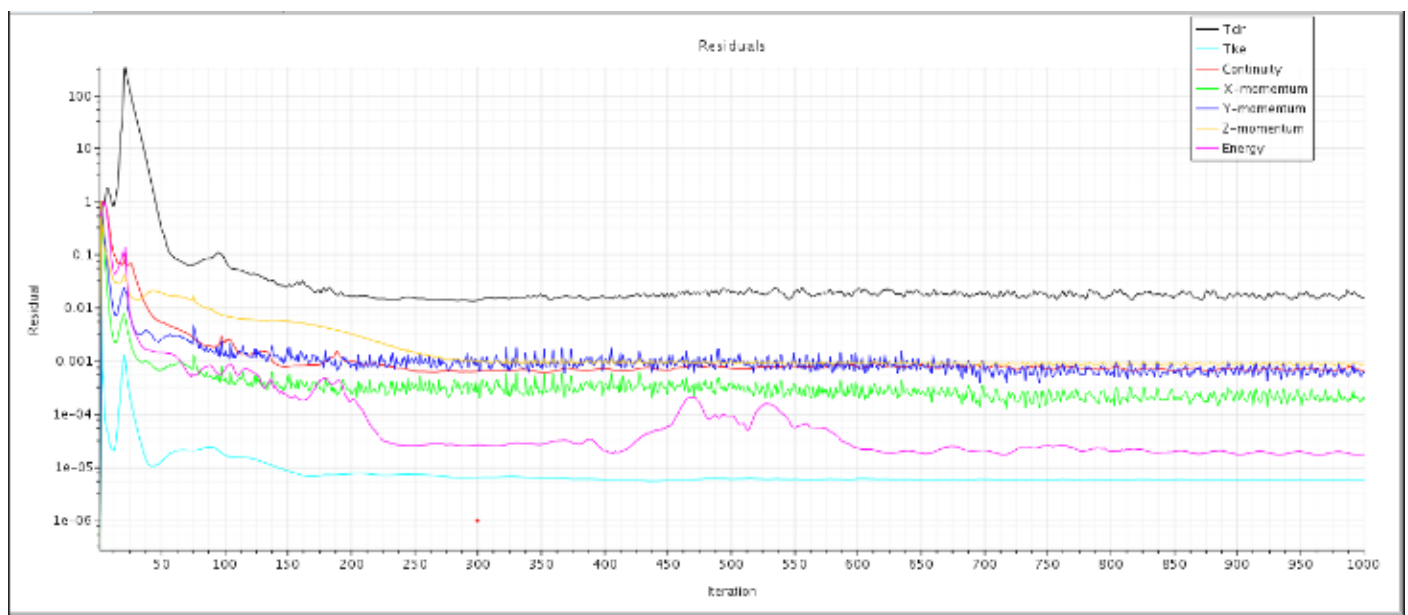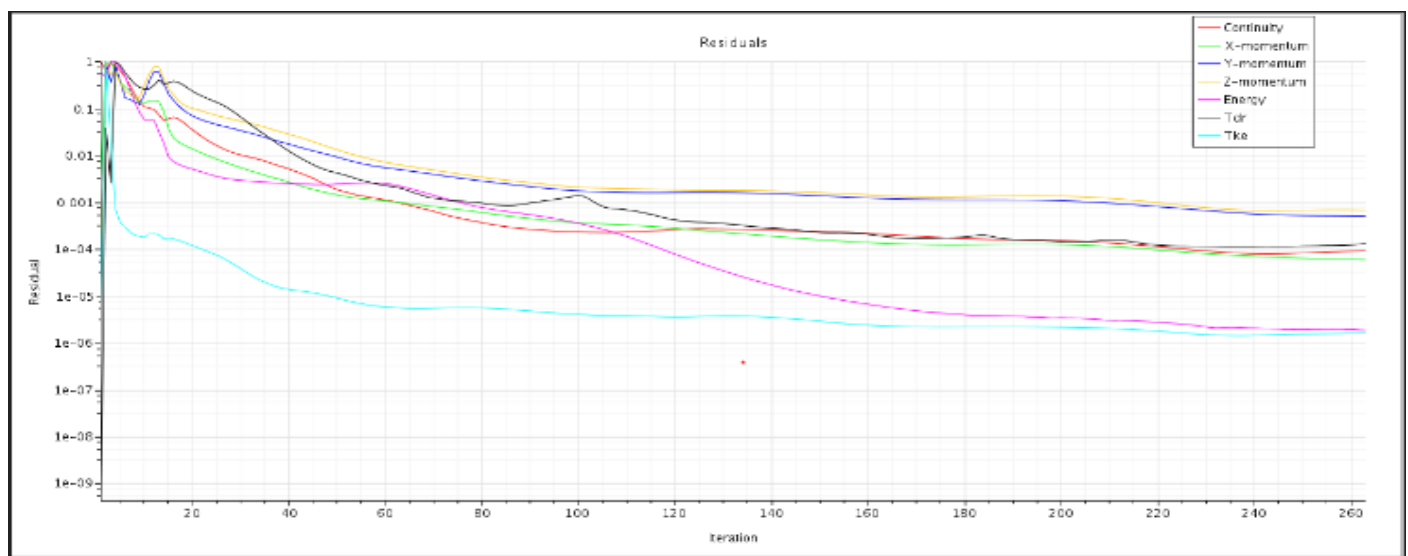

Chart -1: Residual Graph of DOC
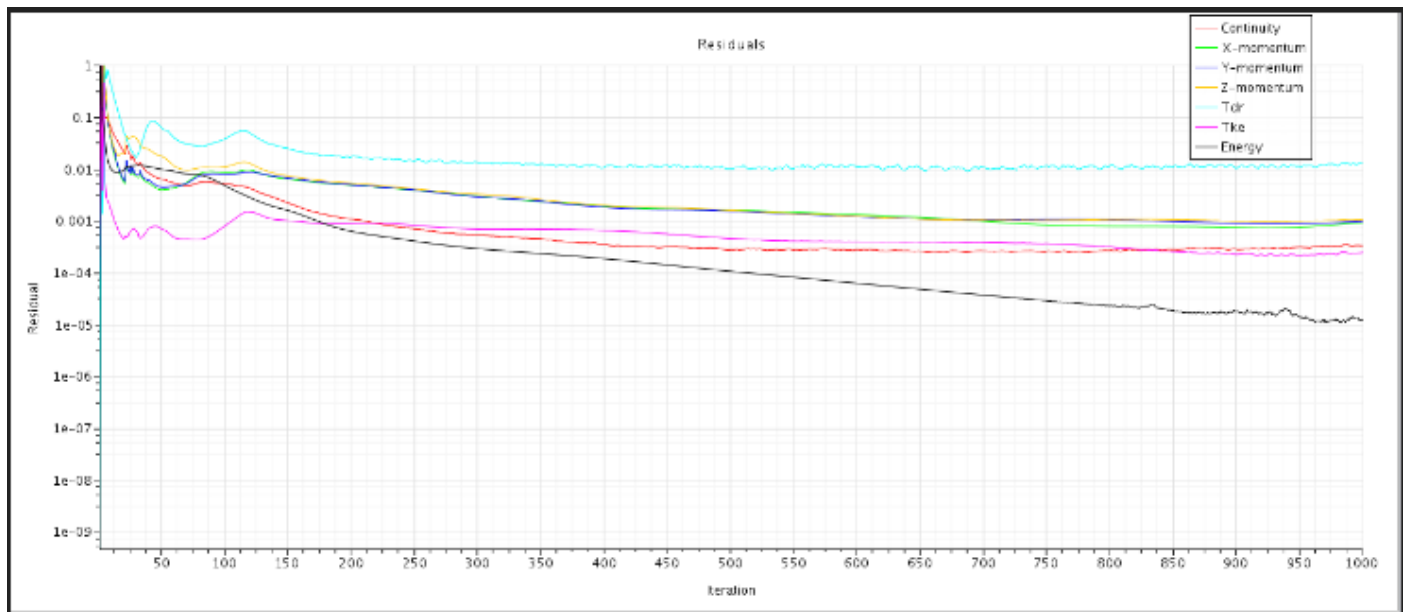

Chart -2: Residual Graph of Mixer

Chart -3: Residual Graph of SCR

REFERENCES

[1] User Guide, Star ccm+, Siemens PLM.

[2] "Ministry of Road Transport & Highways". Morth.nic.in. Retrieved 17 September 2019.

[3] Ridwan Saputra Nursal, Abdul Hadi Hashim, Nor Isha Nordin, Mohd Afandi Abdul Hamid and Mohd Redzuwan Danuri., "CFD analysis on the effects of exhaust back pressure generated by four-stroke marine diesel generator after modification of silencer and exhaust flow design" in ARPN JOURNAL OF ENGINEERING AND APPLIED SCIENCES Vol. 12,' no. 4, February 2017, ISSN 1819-6608

[4] F. Ekstrom and B. Andersson, "Pressure drop of monolithic catalytic converters experiments and modeling", SAE Internationals, 2002-01-1010, 2002

[5] PL.S. Muthaiah, Dr.M. Senthil Kumar, Dr. S. Sendilvelan – "CFD Analysis of catalytic converter to reduce particulate matter and achieve limited back pressure in diesel engine", Global journal of researches in engineering – A: Classification (FOR) 091304,091399, Vol.10 Issue 5 (Ver1.0) October 2010.

[6] Balakrishna, B., & Mamidala, S. (2014). "Design Optimization of Catalytic Converter to reduce Particulate Matter and Achieve Limited Back Pressure in Diesel Engine by CFD", International Journal of Current Engineering and Technology, 2, 651-658.

[7] Henk Kaarle Versteeg, Weeratunge Malalasekera (2007). "An Introduction to Computational Fluid Dynamics", The Finite Volume Method. Pearson Education Limited. ISBN 9780131274983

BIOGRAPHIES

Madhavi received a Research Fellowship at Teesside university, Middlesbrough, UK. Wherein she worked in a DASA, UK sponsored project under the title "Expanded Metal Foam – Strength improvements through sintering".

Renuka Kulkarni did her internship in Cummins India Limited. Where she worked on 'Value Stream Mapping' of engine blocks of various on-road as well as on off-road engines. Also

worked on the repetitive issues regarding engine blocks.

Radhika Dharmadhikari has been a core part of ASME Cummins Student Section and lead the team as Chair, for ASME EFx in her final year. She has been selected for the Student Leadership Training Conference held at IMECE 2019, USA.

Suman Cummins Diploma: Suman Rajput has been a core member of SAE AERO DESIGN team of Cummins college if engineer and lead the team as a vice captain. She interned at DEETECH industry, Pune