Harinivas Bala, 2022, 10:1
ISSN (Online): 2348-4098
ISSN (Print): 2395-4752

The Impact of Turbo Swirl Effect on Gas Flow Uniformity in Catalytic Converter and Modeling Methodology

Vice President Dnyanesh Dandekar, Asst. Manager Harinivas Bala, S Senior Engineer Veera Vishnu
Flow Thermal department,
Sharda motor industries,
Tamil Nadu, India,
d.dandekar@shardamotor.com, harinivas@shardamotor.com, veera@shardamotor.com

Abstract- At present time, global warming and air pollution are the big global issues in the environment. One of the polluting factors is emission out of internal combustion engines. Catalytic converter plays a vital role in reducing harmful toxic gases and delivering non-toxic gases to the environment. The automobile industry uses computational fluid dynamics (CFD) software to analyze the flow properties and gas distribution inside the convertor catalysts extensively. Uniformity index is a critical parameter for catalytic converter performance, which ensures the uniform exhaust gas flow distribution over the monolith. BS-VI emission legislation norms for light & heavy commercial vehicles are set to meet the uniformity index target of over 95% - 98% through monolith brick. In the current scenario, Catalytic converters are closely mounted to the turbocharger. Turbocharger creates a swirl, which greatly affects the flow distribution over the catalytic converter. The flow uniformity analysis procedure needs to evaluate and provide a precise result with testing. In this paper, we present a uniformity index calculation methodology to study the influence of turbocharger swirls. LNT, DOC, and TWC configurations methodology are simulated by segregated and coupled principles. To find the difference in, Flow uniformity; Pressure drops.

Keywords- Turbo inlet, Turbocharger swirl, Uniformity index, CFD, Pressure drop, Flow distribution, BS-VI emission Norm, Swirl effect, LNT, DOC, TWC.

I. INTRODUCTION

Catalytic converter performance is a critical objective in the automobile industry especially in the BS-VI emission regulation. To maximize the efficiency of after-treatment exhaust systems, it is preferred that the flow distribution over the catalyst convertor should be more uniform over the entire catalyst.

Non-uniformity of flow leads to over-utilization or under-utilization of catalyst in the catalytic converter and it creates local thermal points which will reduce the catalyst efficiency. At present catalytic converters are designed close to a turbocharger, for utilizing the temperature of the exhaust gas and accommodating additional catalyst or filter for BS-VI norms. Since turbocharger operates at higher RPM, it creates non-controllable turbulent velocity in the shorter distance between turbo out to the frontal face of a substrate which will affect the gas flow uniformity index. Due to the continuous engine rotational displacement transfer to the turbocharger and EATS.

Therefore, the analyst needs to predict the turbocharger influence over the catalyst convertor frontal face gas distribution. In this paper, we are discussing the impact of the turbocharger swirl on the catalytic converter uniformity index.

We consider one exhaust after-treatment system TWC+TWC gasoline configuration with a turbocharger. The segregated flow model solver solves the momentum and continuity equation separately. The linkage between the momentum and continuity equations is achieved with a predictor-corrector approach.

The segregated flow solver controls the solution update for the segregated flow model according to the SIMPLE algorithm. The Coupled Flow model solves the conservation equations for mass, momentum, and energy simultaneously using a pseudo-time- marching approach. One advantage of this formulation is its robustness for solving flows with dominant source terms, such as rotation.

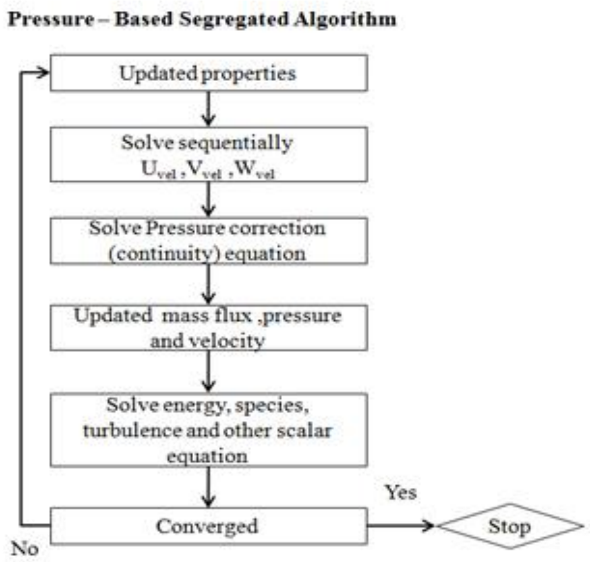

Fig 1. Segregated Algorithm Process.

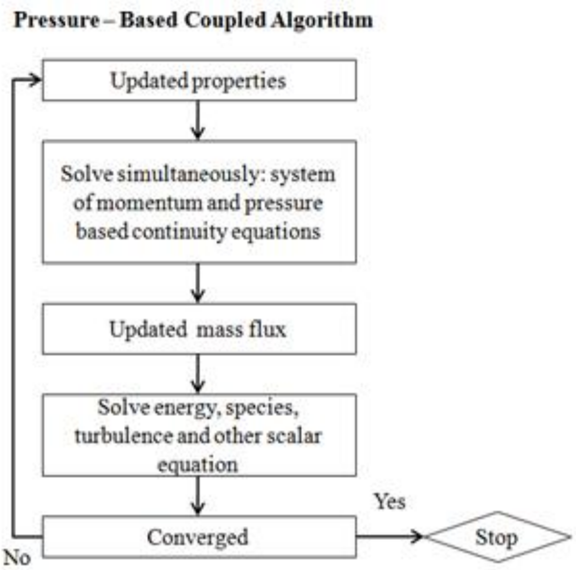

Fig 2. Coupled Algorithm Process.

II. TURBO CHARGER AFTER TREATMENT SYSTEM DETAILS

In this paper, we have considered a four-stroke turbocharged gasoline engine with TWC + TWC EATS configuration.

Table 1. Engine Details.

Parameter	Specification
Engine type	4 stroke, Turbocharged GDI
Number of Cylinders	Three-cylinder, inline engine
Displacement	1.2L
Max power & torque	120 Ps & 215Nm
Max exhaust temperature	1050°C
Engine RPM	5500rpm

Table 2. Turbocharger Details.

Parameter	specification
type	Waste gate turbocharger
Turbo RPM	266760rpm
Turbine wheel diameter	34mm
Waste gate opening angle	5 deg
Waste gate flow	130kg/h

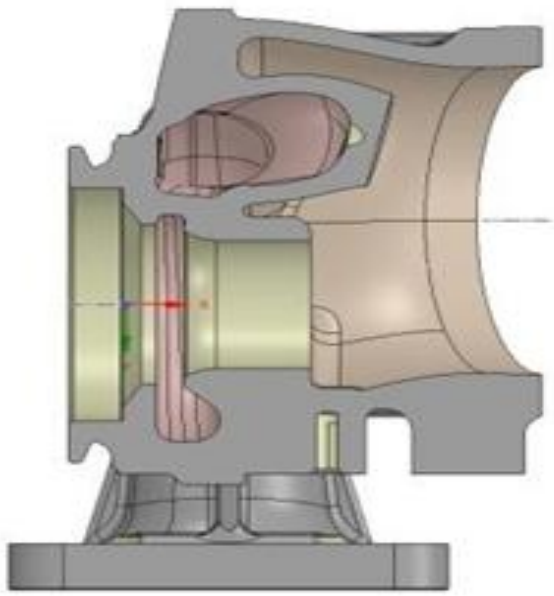

Fig 3. Turbocharger cut section.

A three-way catalytic converter is used to reduce engine emissions. 1.6L volume ceramic monolith is used as a catalyst convertor and the convertor dimensions are diameter 105mm and length 182mm. For the manufacturing feasibility and easy handling, brick is made of two halves at each 91mm.

The distance between two bricks is 10 mm. Cell densities are 600 CPSI and wall thickness is 2MIL. Inlet pipe diameter of 60mm and outlet pipe diameter 55.56mm. oblique type of cone were used for inlet cone and straight type for outlet cone.

Exhaust after-treatment system completely covered by insulation material for reducing the temperature losses, For the flow analysis only exhaust gas fluid domain is considered as computational domain. All other solid thicknesses of pipe, cone, and shell are neglected.

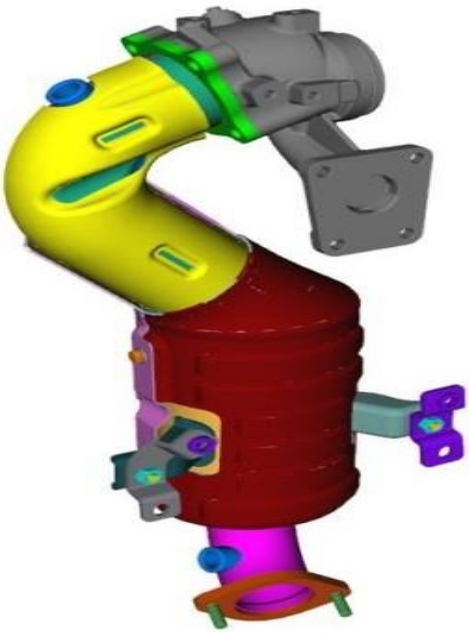

Fig 4. EATS System with a turbocharger.

III. CFD FLOW MODEL AND BOUNDARY CONDITION

Flow domain export from final CAD model through Star CCM+ software. The flow domain encloses turbo fluid, fluid in, fluid fid, fluid out, and two monoliths. The inlet and outlet region of the two flow models is defined. The inlet is in the turbo domain, the outlet is defined in the fluid out domain.

Make all interfaces in place and interfaces all should be in the internal flow interface.

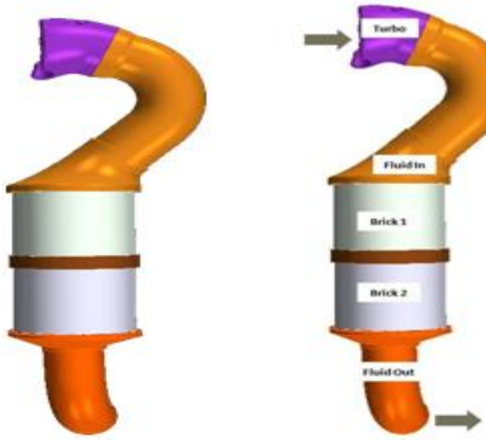

Fig 5. Flow fluid domain of the EATS.

2. Mesh:

Create a mesh continuum in the continua for creating mesh parameters. Select surface remesher to prepare the fluid mesh surface before creating volume mesh. The polyhedral mesh for volume mesh and prism layer approach for capturing velocity near-wall effect is added.

The mesh cell base size is 4mm and the minimum mesh size is 0.5mm, the number of prisms is 8 layers, and the prism layer thickness is 0.5mm. Localized meshing operating is performed to capture all geometrical regions wherever it's required. All y+ wall treatment is used to capture the near-wall effect.

Fig 6. EATS volume mesh profile and mesh profile in cross-section.

Two CATCON are modeled as porous domain, and porosity value and porous coefficient value to porous media are defined. Darcy–Forchheimer equation is used to generate the resistance co-efficient inside the catalyst brick to replicate the catalyst brick effect.

Exhaust gas flow direction is also specified to ensure uni-direction flow inside the wall through the filter and in this flow direction, porous coefficients are specified.

On other lateral directions, 1000 times of coefficient value is introduced. Model volume element has met all mesh standards like maximum skewness angle (<85degree) and no invalid cell present in the mesh model.

IV. NUMERICAL MODELING

STAR CCM+, commercial computational fluid dynamics (CFD) software, was used for numerical modeling. The governing equations relating to continuity, momentum, and energy are given in Equations (1)– (3).

1. Continuity Equation:

$$\nabla\vec{\vartheta} = 0 \quad \text{- (1)}$$

Momentum conservation equation:

$$\frac{\partial\vec{\vartheta}}{\partial t} + \left(\vec{\vartheta}\nabla\right)\vec{\vartheta} = -\frac{\nabla p}{\rho} + \frac{\mu}{\rho}\nabla^2\vec{\vartheta} \quad \text{- (2)}$$

Energy conservation equation:

$$\rho Cp\left(\frac{\partial E}{\partial t} + \vartheta x\frac{\partial E}{\partial x} + \vartheta y\frac{\partial E}{\partial y} + \vartheta z\frac{\partial E}{\partial z}\right) = k_T\left(\frac{\partial^2 E}{\partial x^2} + \frac{\partial^2 E}{\partial y^2} + \frac{\partial^2 E}{\partial z^2}\right) \quad \text{- (3)}$$

Reynolds average Navier stokes equation for fluid flow

$$\rho\left(\frac{\partial V}{\partial t} + V.\nabla V\right) = \nabla P + \rho g + \mu\nabla^2 V$$

V. UNIFORMITY INDEX CALCULATIONS

Gamma value is an industry-standard index, which integrates the velocity distribution profile over a section plane halfway between the front and rear end of the substrate. It varies between 0 to 1. Gamma value near 0 indicates fully non-uniform flow concentrated on a single point and 1 indicates maximum distributed flow.

Gamma value formulae is as per below:

$$\gamma = \frac{\int(u-\bar{u})dA}{2\bar{u}A}$$

Were,
An area of the substrate (m2)
u Substrate face cell velocity (m/s)
u Substrate face average cell velocity (m/s)

VI. UNIFORM FLOW WITHOUT TURBO EFFECTS

Traditional gas flow analysis methodology was carried out in the analysis. The turbocharged effect has been neglected. However, the turbocharger internal flow path was included in this analysis to maintain the turbo flow path over the analysis.

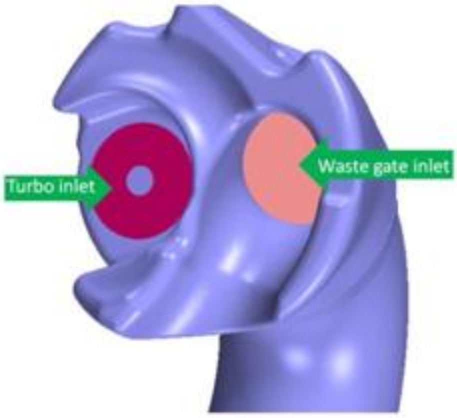

Fig 7. Turbo model with inlets.

Inlet is considered as a mass flow inlet, in which constant mass flow rate and temperature are applied. The outlet is considered as a pressure outlet, in which downstream parts like muffler back pressure is applied.

Table 3. Uniform flow without turbo effect inputs.

Domain	Type	Value
Turbo Inlet	Mass Flow Rate Temperature	266.11kg/h 910ºC
Waste gate Inlet	Mass Flow Rate Temperature	151.35kg/h 910ºC
Porous TWC 1	Porosity Inertial Resistance Viscous Resistance	0.85 12.0 kg/m^4 1960.0 kg/m^3-s
Porous: TWC 2	Porosity Inertial Resistance Viscous Resistance	0.85 12.0 kg/m^4 1960.0 kg/m^3-s
Pressure outlet	Pressure	1 atm +25kPa

Even though low Mach number and not much gas recirculation occur in this type of simulation for comparison both segregated and coupled methodology are used to perform to compare the result.

Flow analysis initiated with appropriated mesh and physics model, from this analysis the inlet and outlet temperature converge, inlet and outlet mass flow convergence should be monitored. Once mass and temperature are balanced, the analysis converged and the results can be post-processed. Uniformity index calculated at 10mm from the brick frontal face. Velocity index calculated at brick front face.

Pressure drop is calculated across the system and the results need to meet the following targets:

- The target of flow velocity uniformity index is above 0.95 for BS-VI EATS.
- The target of velocity index is less than 0.7 for BS-VI EATS.
- Pressure drop target should meet as per layout target value.

VII. RESULT AND DISCUSSION

1. Segregated solver:

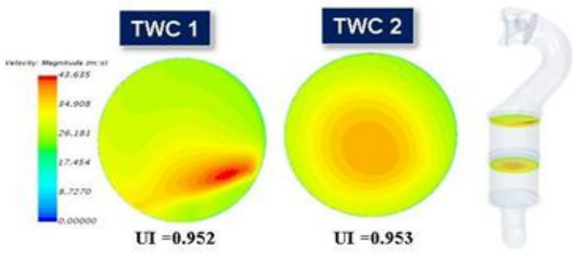

Fig 8. Uniformity index.

Uniformity index measured @ 10mm down from brick frontal face.

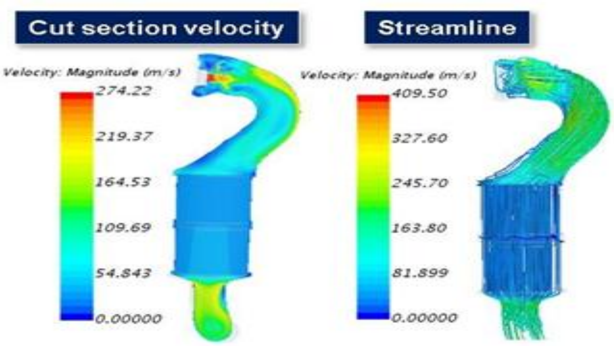

Fig 9. Velocity magnitude.

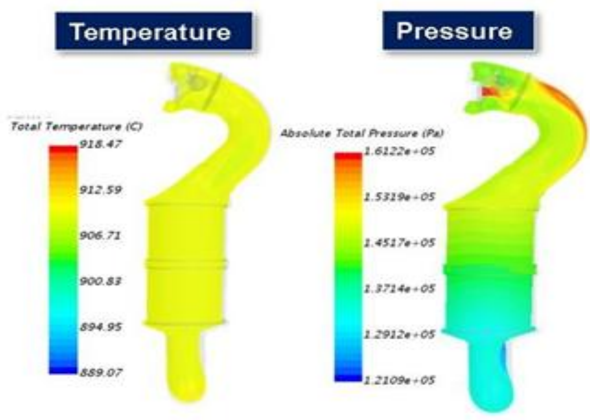

Fig 10. Total temperature and pressure contour.

Total EAT system pressure drop (except Turbo fluid) is 206.10mbar.

2. Coupled Solver:

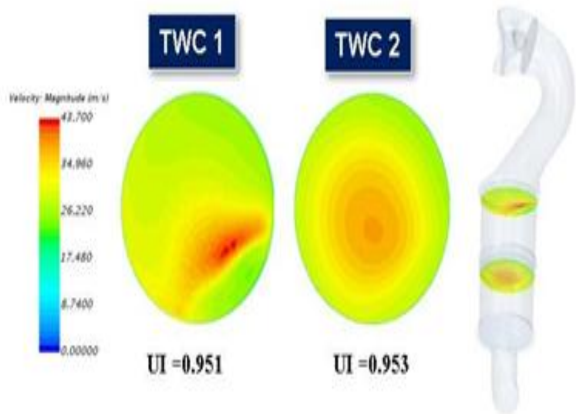

Fig 11. Uniformity index.

Uniformity index measured at 10mm down from the brick frontal face.

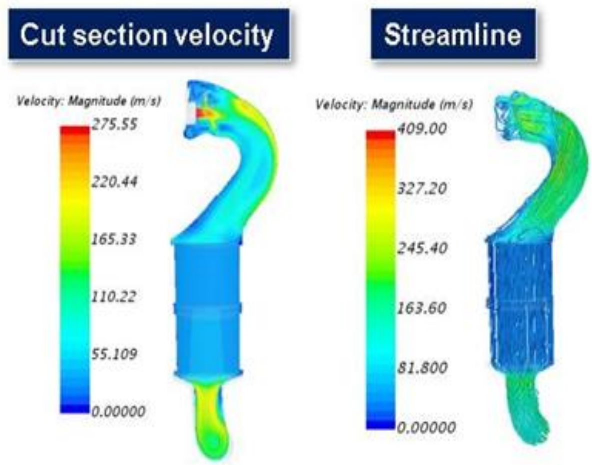

Fig 12. Velocity magnitude.

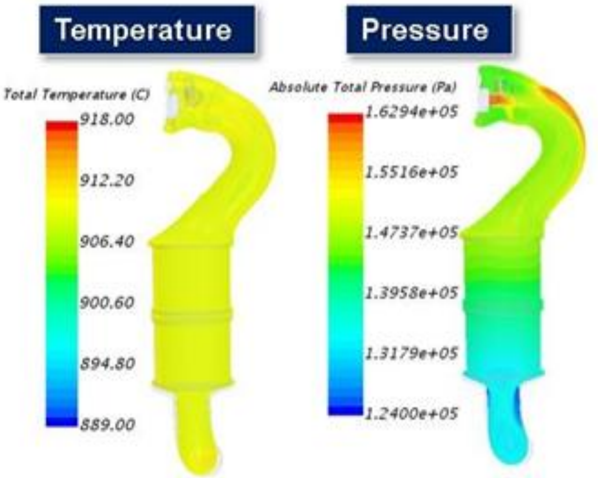

Fig 13. Total temperature and pressure contour.

Total EAT system pressure drop (except Turbo fluid) is 207.75mbar.

VIII. FLOW ANALYSIS WITH TURBO EFFECTS (TURBO VELOCITY PROFILE)

Performed flow analysis with turbocharger swirl effect. Velocity components after turbocharger swirl blades are taken as an inlet boundary condition. For creating the same boundary condition, a separate analysis was carried out considering the turbocharger blade and rotation.

After the blade profile, a plane is created for exporting boundary conditions like axial velocity, tangential velocity, radial velocity, mass flux, and total temperature concerning the plane area as a tabular input.

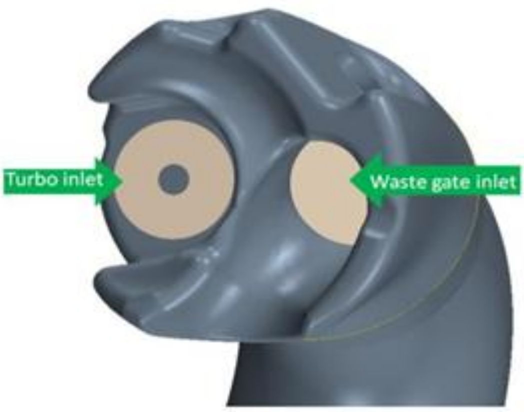

Fig 14. Turbocharger Inlets.

In Turbocharger inlet is defined as mass flow inlet a negative sign of radial velocity indicates turbo swirl direction. In which mass flux table imported.

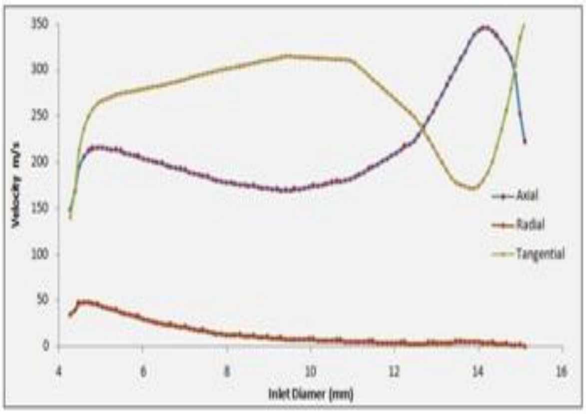

Fig 15. Inlet diameter(mm) vs Velocity(m/s).

Turbocharger blade rotation analysis results are mapped into the turbo inlet. Wastegate inlet considers mass flow inlet and outlet considered as pressure outlet.

Table 4. Turbo flow with turbo effect inputs.

Domain	Type	Value
Turbo Inlet	Mass flux Temperature	Tabular value w.r.t radius
Waste gate Inlet	Mass Flow Rate Temperature	151.35kg/h 910°C
Porous TWC 1	Porosity Inertial Resistance Viscous Resistance	0.85 12.0 kg/m^4 1960.0 kg/m^3-s
Porous: TWC 2	Porosity Inertial Resistance Viscous Resistance	0.85 12.0 kg/m^4 1960.0 kg/m^3-s
Outlet	Pressure condition	1 atm +25kPa

For flow analysis, three different velocities axial, radial, and tangential are used to define the flow direction, flow angles, and flow rotation directions, to create the real turbo flow paths.

Flow analysis initiated to appropriate mesh model and from this analysis it was observed that inlet and outlet temperature with the mass flow is balanced. Once mass and temperature are converged then inlet boundary conditions are cross-checked with inlet boundary conditions then results are extracted from the analysis.

Results need to meet the following targets:

- The target of uniformity index is above 0.95 for BS-VI EATS.
- Pressure drop target is to meet customer target value.

IX. RESULT AND DISCUSSION

1. Segregated solver:

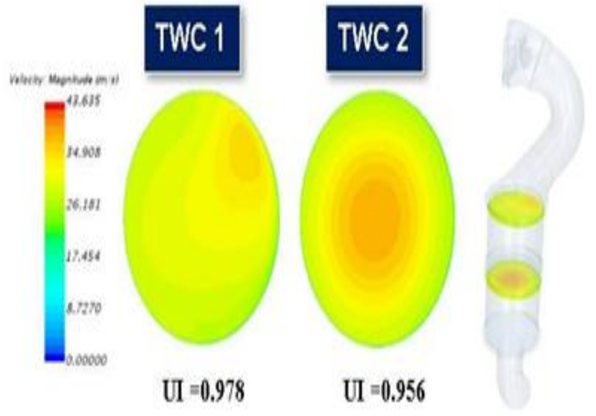

Fig 16. Uniformity index.

Uniformity index measured at 10mm from brick frontal face.

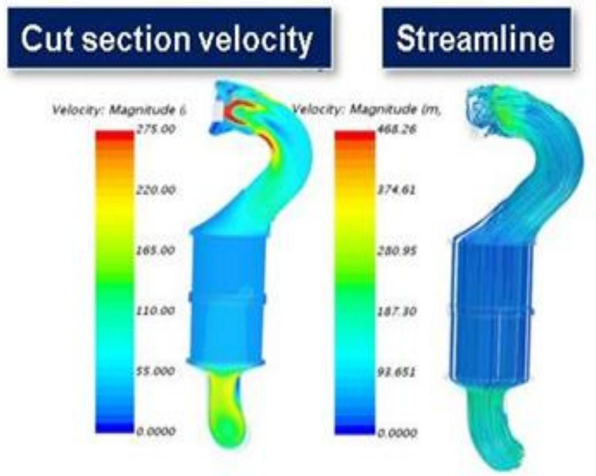

Fig 17. Velocity magnitude.

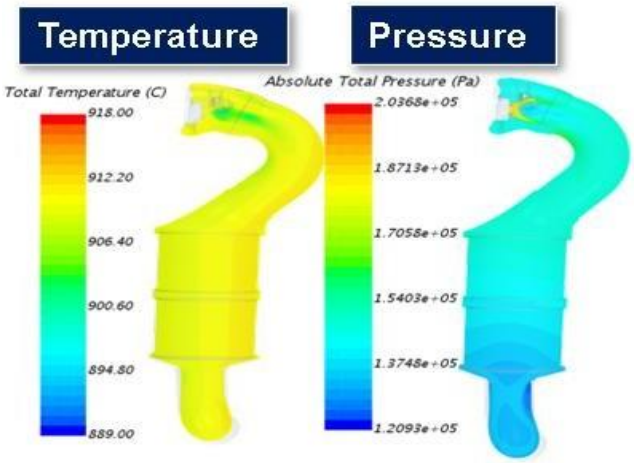

Fig 18. Total temperature and pressure contour.

Total EAT hot end system pressure drop (except Turbo fluid) is 212.15mbar.

2. Coupled flow:

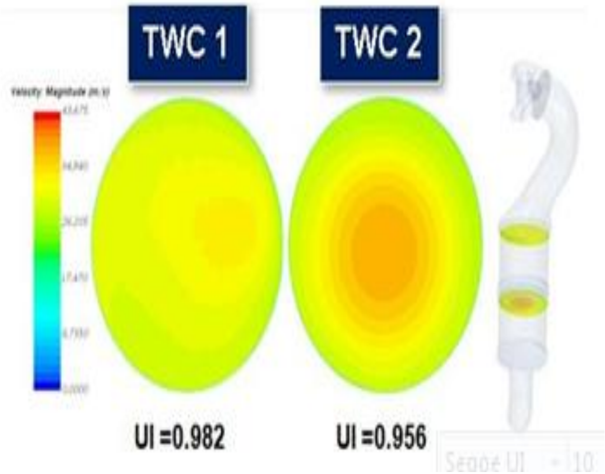

Fig 19. Uniformity index.

Uniformity index measured at 10mm from brick frontal face

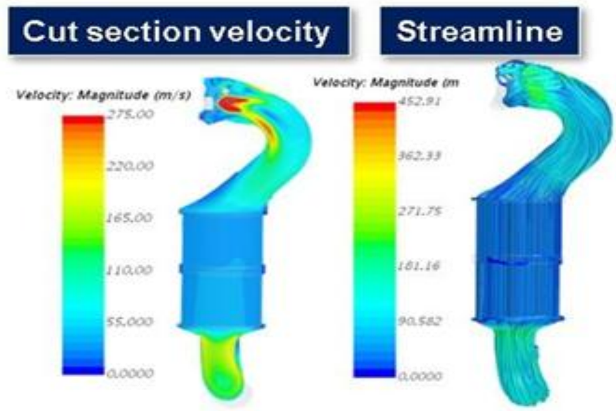

Fig 20. Velocity magnitude.

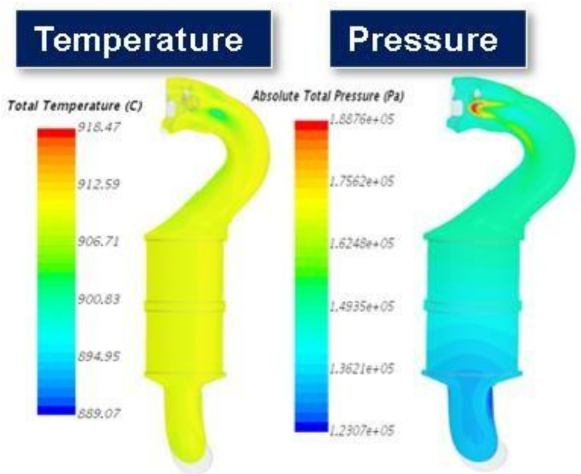

Fig 21. Total temperature and pressure contour.

Fig 22. MACH number.

Total EAT system pressure drop (except Turbo fluid) is 220.16mbar.

X. CONCLUSION

From the above study, two different analyses normal flow and turbo swirl flow with two different physics methodologies (segregated and coupled) were performed and the results are summarized. In normal flow-condition, segregated flow analysis and coupled flow analysis have no significant change in the results of flow characters, Uniformity Index and Pressure drop. In normal flow conditions, segregated and coupled flow analyses have less than a 1% difference in result values which is acceptable. The coupled flow analysis convergence time is 43.3% higher than the segregated flow analysis for normal flow condition.

In turbo swirl conditions, segregated and coupled flow analyses have less than a 1% difference in result values. The coupled flow analysis convergence time is 50.3% higher than the segregated flow analysis for turbo swirl flow condition.

Table 5. Results comparison of normal flow condition.

Parameter	Normal Flow Condition		
	Segregated	Coupled	% of change
Time per iteration(s)	7.25	12.08	43.3
TWC 1 UI	0.952	0.951	0.1
TWC 2 UI	0.953	0.953	0
Total PD (mbar)	206.1	207.75	0.79

The results of four different simulations, Normal flow (segregated and coupled) versus Turbo flow (Segregated and coupled) are compared in the below table.

Table 6. Results comparison of turbo flow condition.

Parameter	Turbo Flow Condition		
	Segregated	Coupled	% of change
Time per iteration (s)	12.79	25.74	50.3
TWC 1 UI	0.978	0.982	0.4
TWC 2 UI	0.956	0.956	0
Total PD (mbar)	212.5	220.16	3.4

According to the latest emission regulations BS – VI and Euro – VI exhaust system is closely mounted with turbo charged engine for this configurations turbo swirl flow consideration as exhaust system inlet is required to obtain accurate CFD prediction of TWC uniformity index and pressure drop of system. From this study CFD methodologies with coupled and segregated physics for turbo swirl flow was studied and validated.

XI. ACKNOWLEDGMENTS

Authors would like to special thanks to Mr. Bineesh Benny for his valuable inputs and guidance while working on this task.

REFERENCES

[1] Pressure drops and flows uniformity analysis of complete exhaust systems for diesel engines André Bergel Edson L. Duque General Motors – Global Propulsion Systems – South America.

[2] Heywood, John, Internal Combustion Engine Fundamentals, McGraw Hill, New York, NY, 1988.

[3] Kutaiba J.M. AL-Khishali, Mahmoud A. Mashkour& Ehsan Shamil Omaraa, "Analysis of Flow Characteristics in Inlet and Exhaust Manifolds of Experimental Gasoline Combustion in A VCR Engine" Eng. & Tech. Journal, Vol. 28, No. 7, 2010.

[4] Three-dimensional CFD simulation of a turbocharger turbine for motorsport applications.

International Journal of Science, Engineering and Technology

An Open Access Journal

[5] Implementation of Bharat Stage VI norms for small and medium-duty CI engines Gagandeep Singh Theti, Rupesh J. Yadav, Kiran More and Ashish S. Utage Department of Mechanical Engineering, MAEER's MITCOE, Pune 411038, Maharashtra, India Accepted 12 March 2017, Available online 16 March 2017, Special Issue-7 (March 2017).
[6] CFD Analysis of Exhaust Manifold of SI Engine and Comparison of Back Pressure using Alternative Fuels Kanupriya Bajpai1, Akash Chandrakar, Akshay Agrawal, Shiena Shekhar Mechanical Engineering Department, BIT Durg, India.