IOP Conference Series: Materials Science and Engineering

PAPER • **OPEN ACCESS**

Design and Analysis of Exhaust System for Ultra-fuel Efficient Vehicles

To cite this article: G Varun Ram *et al* 2021 *IOP Conf. Ser.: Mater. Sci. Eng.* **1033** 012019

IOP Conf. Series: Materials Science and Engineering 1033 (2021) 012019 doi:10.1088/1757-899X/1033/1/012019

Design and Analysis of Exhaust System for Ultra-fuel Efficient Vehicles

G Varun Ram[1], G Nithesh Sharan[2], T Praveen Kumar[3], P Tanish[4]

[1,2,4] Department of Mechanical Engineering, SRM Institute of Science and Technology, Chengalpattu, India
[3] Department of Automobile Engineering, SRM Institute of Science and Technology, Chengalpattu, India

Abstract. Exhaust systems for Internal Combustion (I.C) engines have been finding their significance in the performance of an engine from its inception. One of the detrimental factors for the performance of exhaust is the amount of backpressure building through the flow. This study focuses on reducing backpressure occurring in the exhaust and increasing the ease of manufacturability for a 50 cm^3 single cylinder Spark Ignition (S.I.) engine used in an ultra-fuel efficient vehicle. The flow of the exhaust gases from the engine are analyzed with the help of ram induction theory. Computational Fluid Dynamics has been employed to facilitate the study of turbulent flow in the exhaust and obtain results close to a streamlined flow. Parameters such as dimensions, operating pressures and velocities are calculated to understand the flow characteristics. The results strengthen the bilateral scopes of study here being, smooth flow of exhaust gases and greater manufacturability. The bend and end cone angles are defined in such a way that the wastage of material is least and complexity of the geometry is moderate. The exhaust has been designed keeping the chassis and engine complacency constraints in concern. The results of this study can be useful in manufacturing and integration with light gasoline engines, but of the same orientation as the one discussed here, for prototyping purposes.

1. Introduction

Exhaust System play a pivotal role in all the determinant factors of performance of an engine. Exhaust system traditionally consists of a pipe with varying diameters and bend angles. The purpose of an exhaust is generally to facilitate smooth flow of flue gases with reduced back pressure. A perfect exhaust would be a straight and quite a long pipe to efficiently push out all the gases from the exhaust valve/port to the atmosphere. This is not possible as such in an automobile as there are complacency constraints in integrating the exhaust pipe with the chassis and to contain within the shell/outer body of the vehicle. These constraints over time have led researchers to ponder deeper into the performance of exhausts and innovate ideas to strike a balance between the required flow characteristics and geometrical fit. Exhaust gases from the engine cylinders are collected by an exhaust manifold, passed into a single pipe and expelled out to the atmosphere.

Exhaust back pressure is defined as the exhaust gas pressure that is developed by an engine to overcome the hydraulic resistance of the exhaust system so as to discharge the exhaust gases into the atmosphere. The exhaust gas from the engine is raised to a higher pressure inside the combustion chamber in the exhaust stroke. Thereby, the mechanical work done by engine is increased. This mechanical work contributes to compression of flue gases which successively raises the fuel consumption of engine. A threshold value for backpressure is specified by the engine manufactures. An operating backpressure higher than the threshold value leads to significant efficiency drop. Exhaust gas velocity is defined as the velocity of exhaust gases from the outlet of the exhaust system. The velocity of exhaust gases will be high if engine backpressure is low. Backpressure on engine cylinder depends upon the exhaust system design, its operating condition and atmospheric pressure.

Exhaust system materials are exposed to a variety of harsh conditions; therefore, it must be resistant to high temperature oxidation, elevated temperature mechanical failure and stress corrosion cracking. The power needed to direct exhaust gases is called exhaust stroke loss. With the increase in speed, exhaust stroke loss also increases. The backpressure is directly proportional to the design of exhaust diffuser system. Effective utilization of exhaust energy is economically viable to limit the exhaust emissions from old engines rather than changing engine and fuel modifications.

For ultra-fuel efficient vehicles, the most common driving technique is called coast and burn (also known as kill and coast) method. In this method, the engine is accelerated constantly from the idling revolutions per minute (RPM) to the maximum permissible RPM to attain torque and power. As the engine reaches maximum RPM, saturation in the power obtained is attained. At this point, the Electronic Control Unit (ECU) is used to turn off the Electronic Fuel Injection (EFI) and engine thereby preventing the flow of fuel-air mixture into the combustion chamber. The stored power is then used to traverse the vehicle until a significant lower RPM is reached. In this stage, the vehicle is again propelled using the engine, controlled by the ECU and thereby this cycle continues. This is the most efficient method to save fuel thereby leading to higher efficiency. This study revolves around the characteristic of exhaust gases at peak RPM as the backpressure tends to increase with the load and speed of vehicle.

2. Literature Survey

During the year 2016, **Teja *et al*** [1] have published an extravagant detailing on the usage of Computational Fluid Dynamics (CFD) to establish results on nuances about designing an exhaust. The outcomes drawn from the paper include the study of various pressure drops, difference in theoretical & actual values of various boundary conditions, kinetic energy studies, velocity of gases, flow patterns generated due to the flow of the flue gases through the manifold. They also establish meshing coefficients. The authors follow a grass-root approach to design and analysis of an exhaust manifold taking into account all the possible areas to be explored by Computational Fluid Dynamics. The authors also explain the mathematical models used in the software ANSYS™ and the reason for existence of different models. This publication could be a step-wise guide for researchers aiming to design and analyze exhaust manifolds for any kind of engine using Computational Fluid Dynamics.

In 2002, **Ferrari *et al*** [2] published results on 1-D flow of flue gases including chemical reactions in the gas and solid phase. They have included pre catalysts, main catalysts and piping systems. The custom numerical model, GASDYN™ developed by the authors facilitates the study on vibrational variations of exhaust gases considering the gas phase changes. From their publication, it is evident that the pressure increases as the crank angle increases. This basically complements the fact that as the RPM of the engine increases, the pressure generated near the intake valve increases and thus peak pressure will be higher leading to faster emit of exhaust gases away from the engine. They were able to verify the experimentation with computational analysis.

In the year 2014, **Gopaal *et al*** [3] used ANSYS™ to perform the following analysis: dynamic analysis, modal analysis, couple field analysis, harmonic analysis, thermal analysis and structural analysis. They state about the nuances of using various kinds of materials in the integral parts of exhaust manifold. They establish results proving that their design is safe as the harmonics for various stresses end up being lesser than the threshold value for the particular material used to design an exhaust.

In the year 2018, **Venkatesan *et al*** [4] performed design and analysis on exhaust manifold for a SI Engine with respect to the emissions released by them on various iterations. The authors draw a conclusion that by modifying the perfectly straight length to a divergent-convergent nozzle, the exhaust gas velocities increase due to the increase in kinetic energy.

During 2014, **Navadagi *et al*** [5] established a comparison of exhaust manifolds and their geometry to reduce back pressure establishing various iterations. The best iteration contained the collective exhaust vent connecting to the catalytic converter dimensionally central to the pipes. The back-pressure comparison graphs show a uniform dip resulting a characteristic reduction in back pressure. Thus, the authors conclude that the volumetric efficiency of the engine is increased.

In the year 2017, **Aradhye *et al***[6] worked with a single cylinder 510 cm^{3} naturally aspirated SI Engine and established results after varying the length and diameters of the exhaust pipe. The unique aspect of this publication is that they explain the ram effect and the scavenging effect as to how essential they are for a smooth flow of the flue gases. The combined effects of varying length and diameter, simultaneously over the entire speed range, significant rise in torque and power at lower as well as higher RPMs.

In the year 2017, **Kanupriya Bajpai *et al*** [7] have compared the effects of using different fuels as a source for combustion in an Internal Combustion engine. The authors states that there is sufficient negative pressure at the intersection of various ducts zone. Under identical load and position of measure, authors conclude that Liquified Petroleum Gas (LPG) has the least back- pressure followed by gasoline and alcohol. The working pressure obtained in the cylinder is the maximum for Gasoline. This also suffices the fact that gasoline is being used widely under real life conditions to meet the requirement instant torques and power.

Manikandan *et al* [8] during the year 2017, have also tried to experiment with a similar kind of analysis taking Computational Fluid Dynamics as a base for the process running the engine at a lower speed and constant load. The authors claim that the iteration with central height being a mean of the tested values has the least back-pressure post analysis.

Kennedy *et al* [9] during the year 2011, published results on fabricating an intake and exhaust system for a Formula student racing car. As complying by the rules is the primitive goal and design complacency constraints exist, the twin-cylinder exhaust was later converted into a single exhaust system. Finite elemental analysis for intake was performed with SOLIDWORKS™ and ANSYS™ was used for exhaust. Silencer make of Suzuki was used here to comply by the rules. Also, the authors insist that twin exhaust design has to be taken into consideration for accelerating performances from the current output factors.

2.1. Problem Identification and Description

Previous research papers have substantial information on more than a single cylinder engine's exhaust system. Researchers have done appreciable study on the same and tried to analyze all possible parameters and optimize their designs. One of the areas which are not explored to a greater extent is the approach towards designing and optimizing a low powered, un-muffled exhaust system which can be used for a non- commercial ultra-fuel efficient prototype vehicle. These vehicles are generally built centric to the concept that the vehicle has to attain maximum fuel efficiency with a given set of physical conditions to apply.

Muffler as we know acts as the noise reducing component in the exhaust system. While doing this, one unnoticed effect it creates in the system is to increase the amount of back-pressure occurring through the tail. This disturbs the smooth flow of gases through the exhaust. As a cascading result, the amount of gas released i.e. the flow rate of gases reduces. This leads to a drop in efficiency of the vehicle. Being a single cylinder engine, the amount of noise and vibrations transmitted through the exhaust is not significant enough to cause an unpleasant sound except at high RPMs for an extended duration. Hence the usage of muffler as a design component has been avoided.

The main focus as stated is the single cylinder engine exhaust, the analysis of backpressure and the smooth flow of exhaust gases through the pipe. Backpressure occurs due to reasons such as:

- Variation in dimensions through the nozzle area
- Convergent – divergent nozzle distance variations
- Higher bend angles in the exhaust pipes
- Erratic inlet and outlet velocities of the gases
- Mismatch in valve timing and ram effect
- Impure gases and incomplete combustion

This is the problem we are trying to address in this study. Reduction in back-pressure does not go hand in hand with easier manufacturability. Manufacturability plays an extremely important role in an exhaust as welding structures and nature of welds are responsible for the misfits leading to disturbance of flows. Also, through the geometry of the exhaust, there are more of varying cross-sections which are hollow and sometimes might induct wastage of material. The exit angles of the exhaust which are sectioned with respect to plane to bring in swirl aiding faster exit flows. These angles are supposed to be analyzed from a manufacturing perspective as well. To form an outline, this study finds the most efficient, streamlined flow of exhaust gases through an exhaust with the least occurring backpressure for a single cylinder 50cm^3 displacement spark ignition engine.

3.Methodology Used

The fundamental approach towards the problem is centric around a CAD and analysis process flowchart.

Figure 1. Process flowchart

Initially, understand the need for an efficient exhaust system for a 50cm^3 single SI Engine. This division has been explained in detail in the problem identification and description section. Then CAD design of the exhaust system has been performed in the following manner briefly:

- Primarily a 2-D sketch consisting of various diameters of the exhaust pipe along the loop/path length is performed on SOLIDWORKS^{TM}.
- The 2-D sketch is converted to a 3-D model by providing thickness to the pipe.

This part requires us to calculate path length for initiating the design. The calculations will be explained in the subsequent parts. The 3D design is now imported into ANSYS FLUENT^{TM} software. The solver requires us to provide meshing elements, boundary conditions and establish outputs. This part again demands us to provide calculations for boundary conditions, decide the equation under concern and enable us to understand the results under comparison. Once the calculations are done and the results plotted, we need to compare different iterations which are obtained by changing certain decisive parameters for the least backpressure. If the result obtained is desirable, then we establish a solution concluding with the respective design. In any other case, we reiterate using different designs and keep the loop running until the most desirable solution is established.

3.1. Specifications of engine

Make : Honda
Model name : GY6
Displacement : 50 cm^3
Ignition type : Spark Ignition
Aspiration mode : Naturally aspirated

Table 1. Engine Specifications

Type	4- stroke SI
Starting	Electric/kick start
Cylinders	Single
Combustion chamber	Hemispherical
Valve train	Single OHC
Bore	39mm
Stroke	41.4mm
Displacement	49.5cc
Compression ratio	10.5:1
Compression pressure	15kg/cm 215psi

Figure 2. Honda GY6- side view

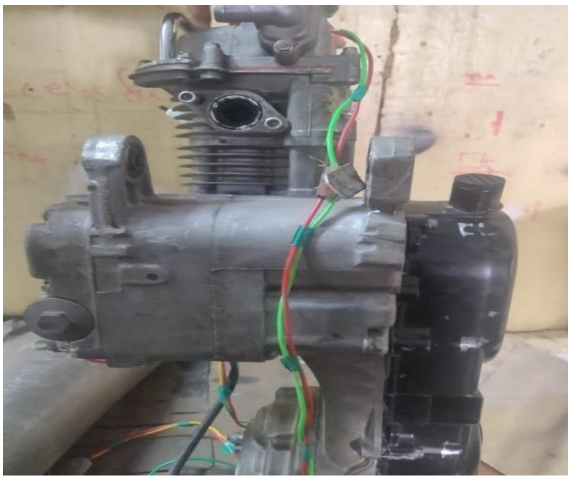

Figure 3. Honda GY6- exhaust outlet

Figure 2 depicts the orientation of positioning the engine in the vehicle to be powered. Figure 3 depicts the location of the exhaust outlet in the Honda GY6 engine.The exhaust to be designed and manufactured is in accordance to the engine shown here(Fig. 2 & Fig. 3)

3.2. Understanding the flow of exhaust gases

The exhaust pipe follows the ram-induction theory. The molecules through the pipe follow compression and rarefaction along the length of the exhaust. The efficiency of the exhaust highly depends on the time and number of molecules which exit the exhaust from the port to the pipe end. This is attained by altering inertial forces as the gas particles tend to stay without exiting the exhaust when the exhaust valve shuts down. The particles contain compression near the exhaust valve and rarefaction a bit further from the valve. This forms a pressure wave. The pressure wave as we know has a mixture of compressed and rarefacted waveform. The only way to reduce rarefaction and send all the molecules outside is when the compressed molecules pressurize the distanced ones at a given point in time attaining large quantities before opening the exhaust valve exiting port through tube. This ensures that the particles do not come back into the pipe due to inertia and thus reducing back-pressure. Generally, ram induction theory applies to intake manifolds of the SI engine. But in this case, there is only a single cylinder in the chamber and exhaust manifold is required to only combine gases from the present cylinder. Hence, we apply the ram-induction theory.

Table 2. Product Specifications and Technical data of Engine

Product Specifications		**Technical Data**	
Scooter Model	JL50QT-18 JL50QT-X1/8 JL50QT-14	Power	2.2 HP @ 8000 RPM
Length	1685mm (66.5 inches)	Torque	4.5 ft lbs @ 6000RPM
Width	660mm (26 inches)	Intake timing	6° BTDC-28° ATDC
Height	1070mm (42 inches)	Exhaust timing	38° BBDC-6° BTDC
Wheel base	1195mm (47 inches)	Valve lash	0.5mm
Engine type	4-stroke OHC	Idle speed	1850-1900 RPM
Displacement	49.5 cm	Oil system	Pressure/slash

The exhaust timing from the manual is the most important parameter for us to initiate the design calculations. The Exhaust valve opens at 38° before bottom dead center and closes 6° before top dead center.

3.3. Experimental Dimensions

The exhaust port exit contains 19.01mm internal diameter and 23.3mm outer diameter with weld limit

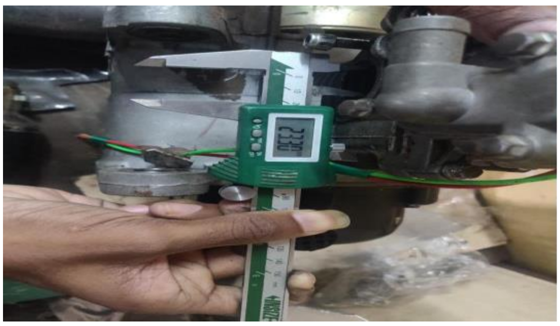

Figure 4. Pipe outer diameter with weld

Figure 5. Exhaust port size diameter

IOP Conf. Series: Materials Science and Engineering 1033 (2021) 012019 doi:10.1088/1757-899X/1033/1/012019

For accuracy sake this was confined to 20mm used in the calculations as the standard hole diameter is 20mm in the GY-6 50cm^3 engine.

4. Design calculations

To initiate the calculations, we need to understand the flow characteristics of the gases expelled by the engine. The length and diameters are constrained under conditions which strike a balance between physical complacency and theoretical mechanics. The assumptions done in the calculations are as follows:

- The working of the engine is ideally perfect.
- The efficiency is maximum possible theoretically.
- There is complete combustion taking place in the combustion chamber
- Air-Fuel mixture occupies full volume of cylinder.

The stoichiometric air/fuel ratio is ideally 14.7:1 with a flow rate of fuel from the injector at 128g/min. We used an electronic fuel injection kit from 'Ecotrons' to easily calibrate the intake characteristics.

4.1. Calculations for length of exhaust pipe

Let us assume for ideal conditions that the gases flow with a sonic velocity with Mach number = 1, at the average velocity of sound = 343m/s.

The engine has an exhaust opening timing for 212°. We know that for a complete stroke, the angle covered by a 4-stroke engine is 720°.Thus the angle that the exhaust valve is closed is 720° – 212°=508°.

At max rpm of engine, i.e. 8000 RPM the distance covered by the piston is 47998.8 $^\circ$/second.

8000 RPM converting into second scale is given by, 8000/60 = 133RPS=133*360 $^\circ$ = 47998.8° / second.

$$\text{Time} = \text{Distance} / \text{Speed} \tag{1}$$

Therefore, Time taken for one time the exhaust valve opens = 508 / 47998 = 0.0105 second.
Distance to be covered by molecules after complete combustion = Time taken per opening * Speed of sound; given by 0.0105 s x 343 m/s = 3.601m (2)

The previous value i.e. 3.6m does not take ram-induction theory into account.

Let us say that the exhaust gases form a pressure wave through the distance, then the gases need to travel only half the distance as the molecules undergo compression and rarefaction, i.e. 3.6/2 = 1.8m.

Now let us take the dimensions to be 1/3rd the maximum length for complacency sake to integrate with the chassis, i.e. 0.6m.

The time taken for the gases to flow through the given length and speed of sound = 0.6 m/343 m/s

$$= 0.00174 \text{ s} \tag{3}$$

For one bounce per second, the distance travelled by the wave is 0.00174*2= 0.00349m.
Thus, exhaust path length was finalized to be around 600mm thus iterated from 560- 620 mm to fine-tune the other parameters influencing the flow of gases.

4.2. Calculation for velocity of gases through the exhaust

At 8000rpm (maximum speed), the crankshaft rotates 8000 times a minute which gives 4000 exhaust strokes per minute. As assumed earlier, the gas flow is complete.

$$\text{Volume displacement} * \text{effective number of strokes} \tag{4}$$

$50cm^3$ * 4000 strokes = 2* 10^5 cm^3/min or 0.00333 m^3/s.

Velocity of piston:
At 8000 rpm, the strokes covered by the piston would be 16000.
The stroke length is 41.4mm and bore diameter is 39mm (from the engine manual)
Hence its velocity would be

$$41.4/ (1000) * (16000/60) = 11.04 m/s \tag{5}$$

The bore diameter is 39mm thus the area would be (3.14/4) * (0.039) * (0.039) = 0.00119m^2
The diameter of exhaust tip is 20mm, thus the area would be (3.14/4) * (0.02) * (0.02)
= 0.000314 m^2.

Using continuity equation at this juncture, A1*v1=A2*v2 (6)

Where A1 is the area of the cylinder, A2 is the area of the exhaust pipe at the tip, v1 is the velocity of piston during combustion at given rpm conditions and v2 is the velocity of the gases at exhaust post combustion.

Therefore, v2= (A1*v1)/(A2) from (eq. 6)

v2= (0.001193 * 11.04) / (0.000314) = 41.83 m/s.

5. Design iterations

As mentioned in the process flowchart and design calculations sections, we have varied the path length, convergent-divergent length, bend angle of the exhaust and exit cone angle. Four iterations were taken into considerations with the following dimensions:

Table 3. Dimensions of iterations of exhausts

Iteration No.	Inlet Diameter (in mm)	Outlet Diameter (in mm)	Length (in mm)	Convergent-Divergent length (in mm)	Bend angle (in degree)	Exit cone angle (in degree)
1	24	34	560	421.6	110	180
2	24	24	620	313.8 +/- 5	135	180
3	24	22	580	285 +/- 0.5	120	22.5
4	24	22	600	285+/- 0.5	115	22.5

Let us look at the CAD of the iterations mentioned above.

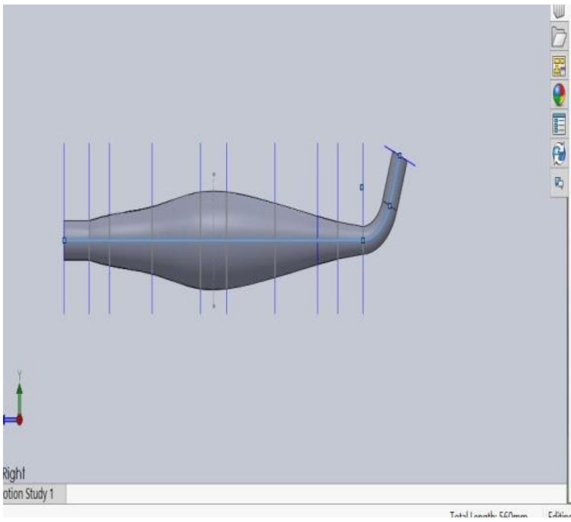

Figure 6. Exhaust – iteration 1

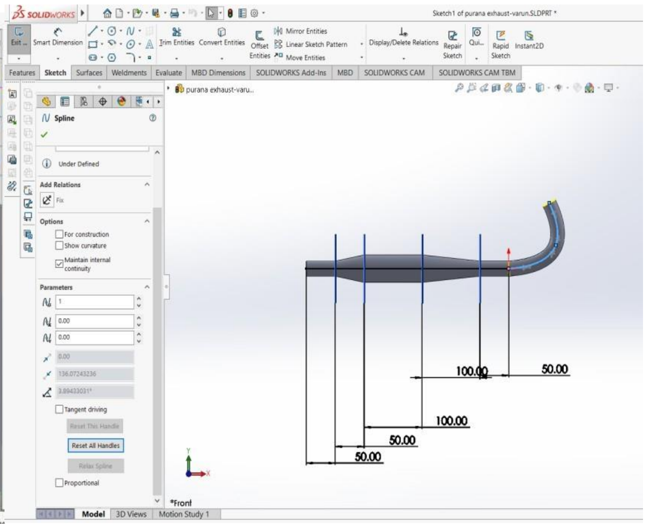

Figure 7. Exhaust – iteration 2

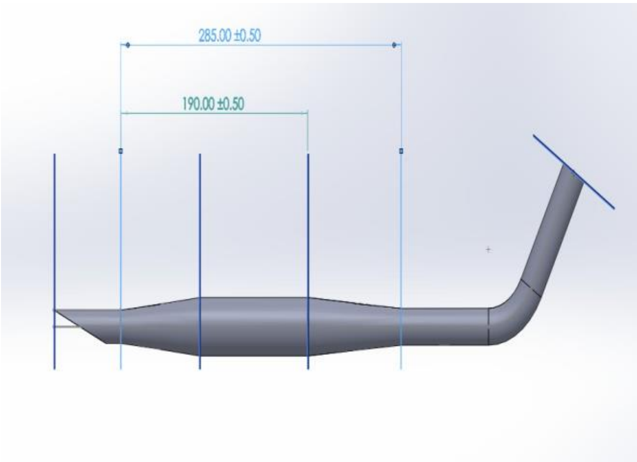

Figure 8. Exhaust – iteration 3

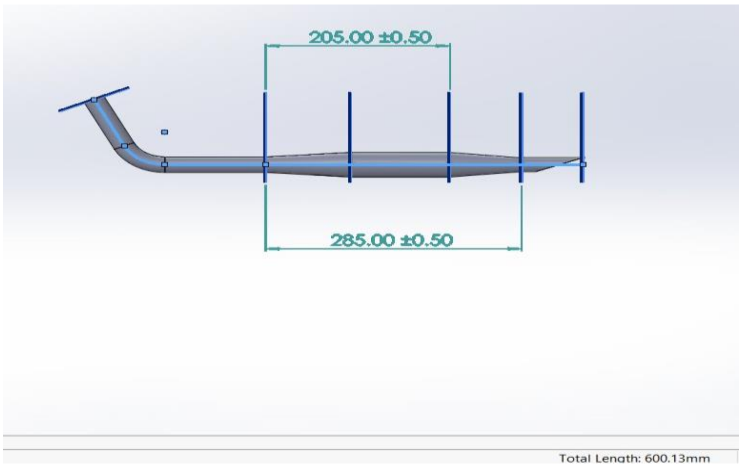

Figure 9. Exhaust – iteration 4

5.1. Analysis methodology

As mentioned earlier, Computational Fluid Dynamics in ANSYS FLUENT^{TM} 2019 R3 solver was used. The meshing size of the elements for iterations 1, 3 and 4 are 1.5e-3m and for iteration 2, 2e-3m has been used. This is due to the educational software license limitation as the number of elements for iteration 2 with an element size of 1.5e-3m was exceeding 512,000. The smoothing level is set to 'high'. This increases the quality of mesh leading to much more accurate results. With lower qualities, i.e. higher element sizes, the results were extremely misleading creating anomalistic behavior in comparison with physical calculations. Adaptive sizing mesh was not included as the geometry is symmetrical about at least one plane and is fairly straightforward.

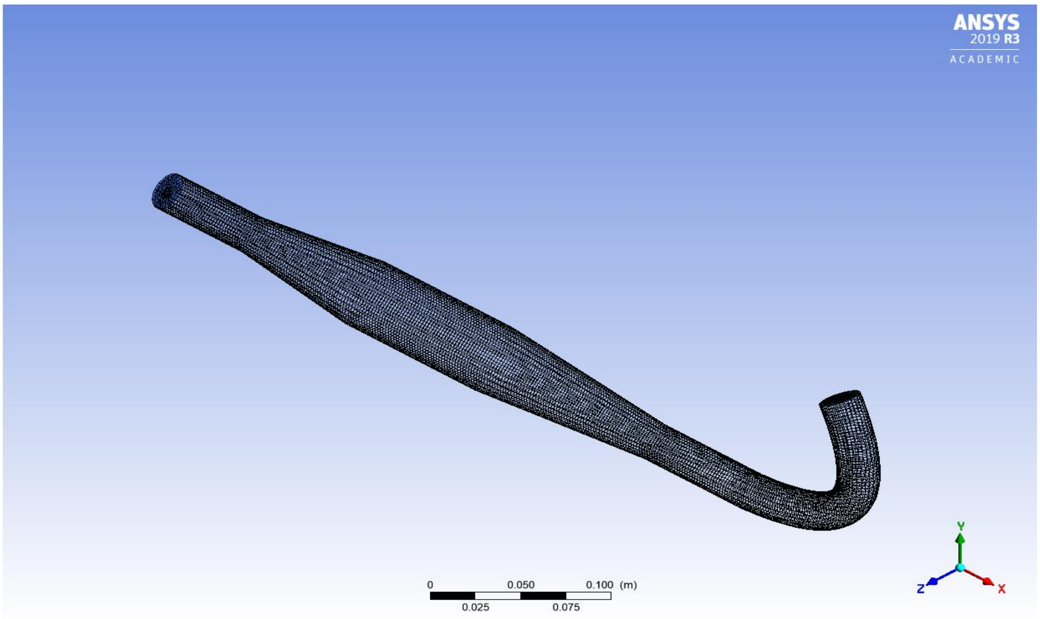

Figure 10. Mesh of iteration-2

The continuity, momentum and energy equations were solved in ANSYS™ 2019 R3 respectively:

$$\nabla \cdot (\rho v) = 0$$

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla \cdot (\rho\vec{v}\vec{v}) = -\nabla p + \nabla \cdot (\overline{\overline{\tau}}) + \rho\vec{g} + \vec{F}$$

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{v}(\rho E + p)) = \nabla \cdot \left(k_{\text{eff}}\nabla T - \sum_j h_j \vec{J}_j + (\overline{\overline{\tau}}_{\text{eff}} \cdot \vec{v}) \right) + S_h$$

The **K-** ϵ turbulence modelling was done using the following equations**:**

$$\frac{\partial}{\partial t}(\rho_m k) + \nabla \cdot (\rho_m \vec{v}_m k) = \nabla \cdot \left(\frac{\mu_{t,m}}{\sigma_k} \nabla k \right) + G_{k,m} - \rho_m \epsilon$$

$$\frac{\partial}{\partial t}(\rho_m \epsilon) + \nabla \cdot (\rho_m \vec{v}_m \epsilon) = \nabla \cdot \left(\frac{\mu_{t,m}}{\sigma_\epsilon} \nabla \epsilon \right) + \frac{\epsilon}{k}(C_{1\epsilon} G_{k,m} - C_{2\epsilon} \rho_m \epsilon)$$

The properties of the fluid [10] used through the pipe described below:
Density = 1.0685 (kg/m^3)
Viscosity= 3.0927 x 10^{-5} (Pa-s)
Specific heat = 1056.6434 (J/Kg-K)
Thermal conductivity = 0.0250 (W/m-K)

The energy equation used here is the k-e (k-epsilon) equation for describing the nature of flow, which is turbulent. As per Teja MA *et al* [1], the turbulence percentage was taken to be 10% at the inlet. The boundary conditions as discussed in the calculations were taken to be:

At inlet:
Velocity: 41.83 m/s (from eq. 6)
Pressure: 2bar
Temperature:1500K
Turbulence: 10%

At outlet:
Pressure: 1.01325bar
Temperature: 303K

The properties of pressure and temperature at inlet and outlet are referred from Teja M A *et al* [1]. Convergence was set to 10^{-6} for all the variables involving in the equation. As the computation power was limited, the highest achievable continuity was 10^{-5} with the number of iterations around 250-300. With extremely precise mesh qualities, the accumulated error tends to a negligible value.

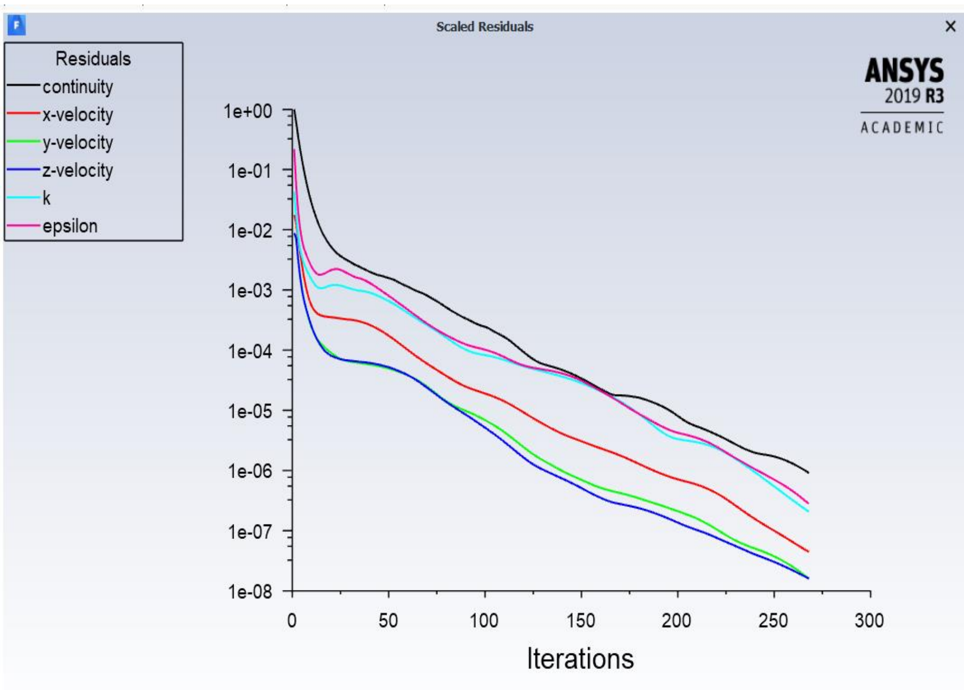

Figure 11. Continuity levels with respect to number of iterations

6. Results and Discussion

Backpressure is directly looked through as a localized change in the direction of flow of gases through the pipe. Also, various contours containing contrasting color with their measures indicated on the scale aids us to understand the change of a particular characteristic as it progresses through the length of the pipe. Let us look at all the four results and compare their velocity streamlines and reverse pressure gradients.

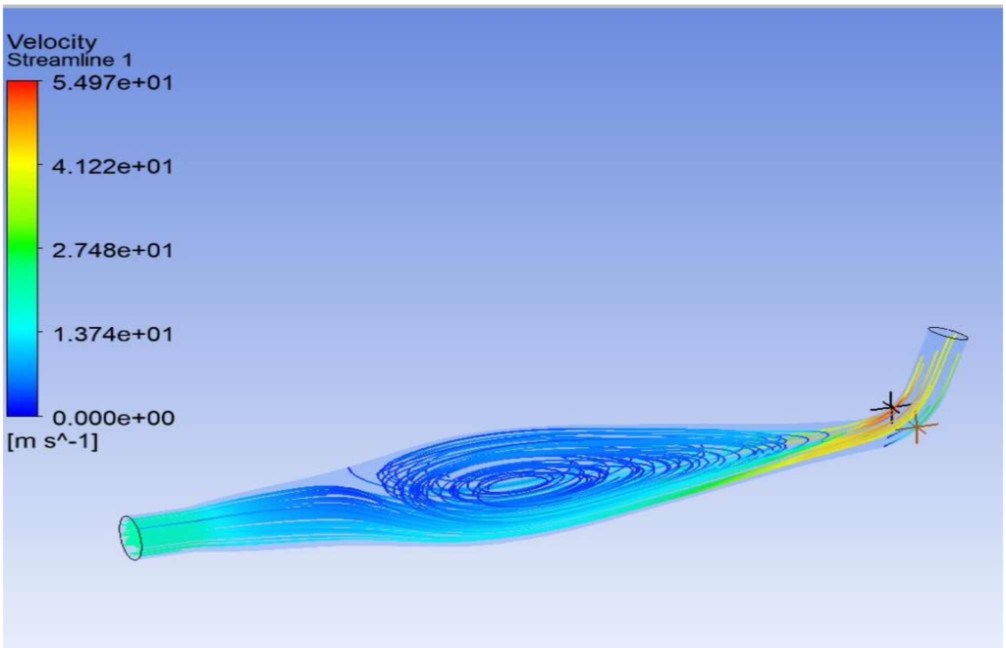

Figure 12. Velocity streamlines from inlet of iteration-1

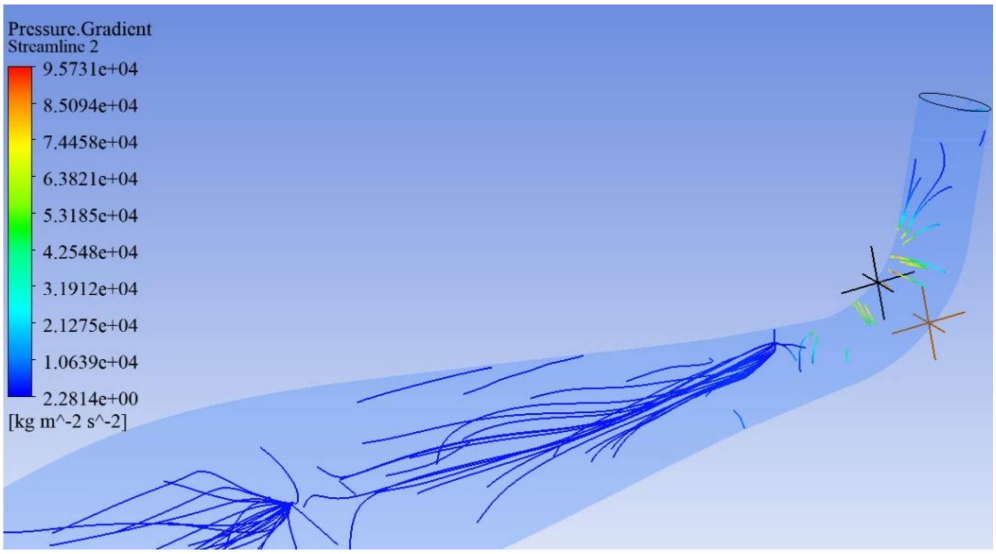

Figure 13. Reverse pressure gradient-1

For the iteration 1, Figure 13 shows the reverse pressure gradient with maximum pressure points. This when tested in the total pressure region shows that there is a change in the nature of streamlined flow in the convergent divergent area region. Thus, a swirl occurs from the expanding region back to the inlet. This creates a low-pressure region. This draws the streamlines to form a vortex in the expanded region of the exhaust. Thereby extreme back-pressure is visible. This design was taken as a base reference and improvised in the following iterations.

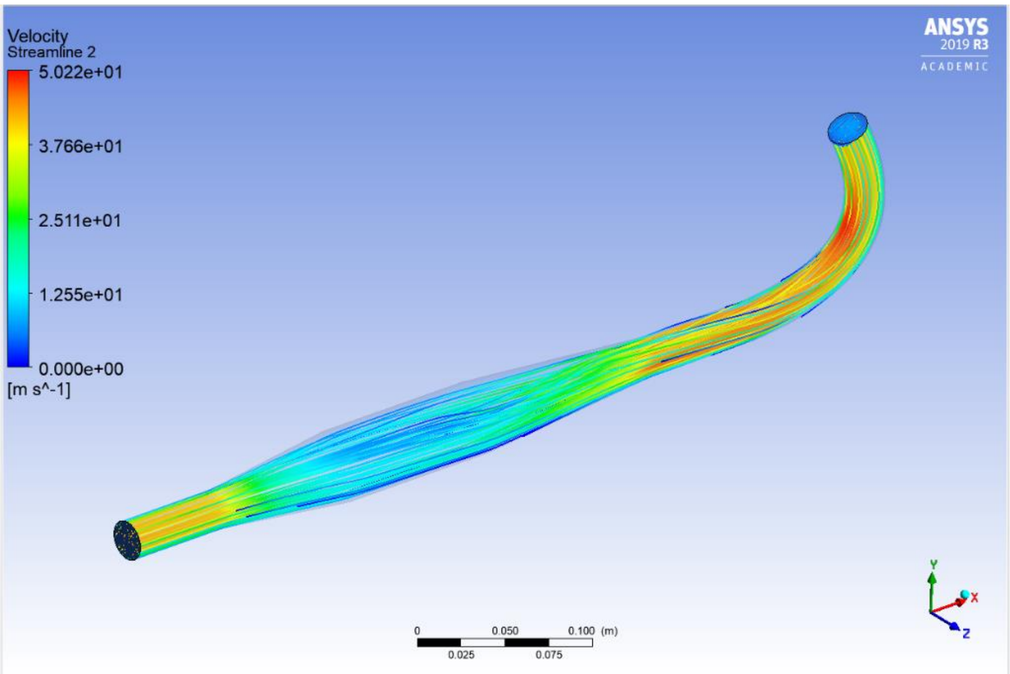

Figure 14. Velocity streamlines from inlet – iteration 2

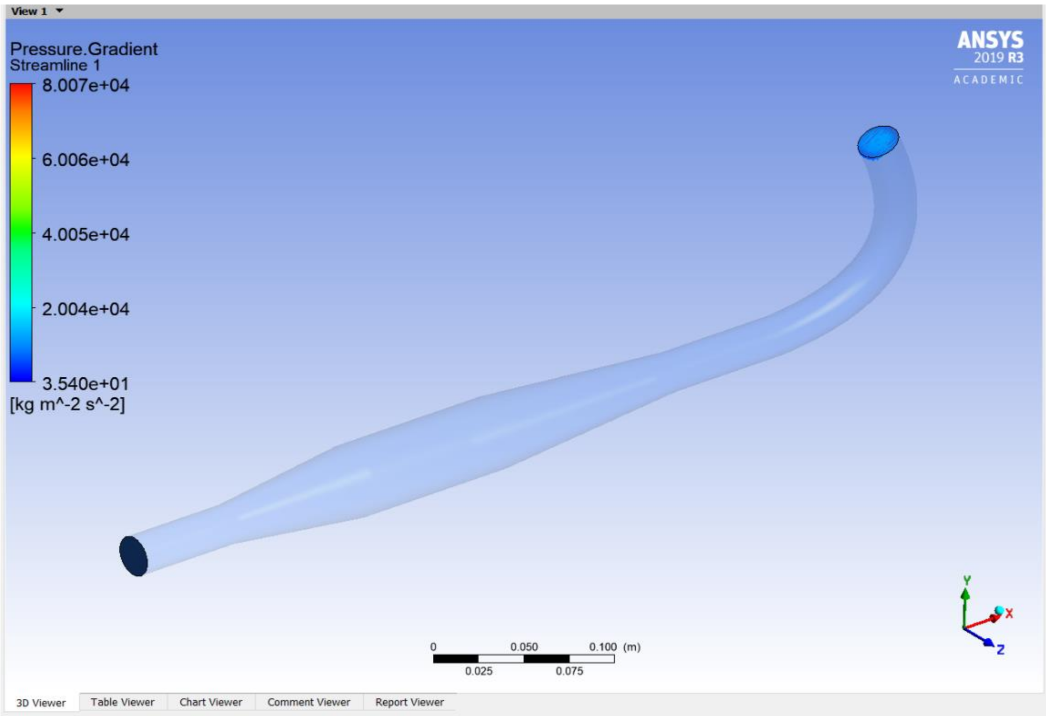

Figure 15. Reverse pressure gradient –iteration 2

In the iteration 2, we find that the reverse flow of gases with respect to the pressure gradient is almost zero for the same boundary conditions. Also referring to the figure 14 the velocity of the exhaust gas at the exit is around 37m/s which is relatively faster while compared to the iteration 1. The gradual change in the bend angle contributes to the least amount of accumulated reverse pressure. The dimensions of the convergent-divergent region also do not change drastically which benefits the flow nature.

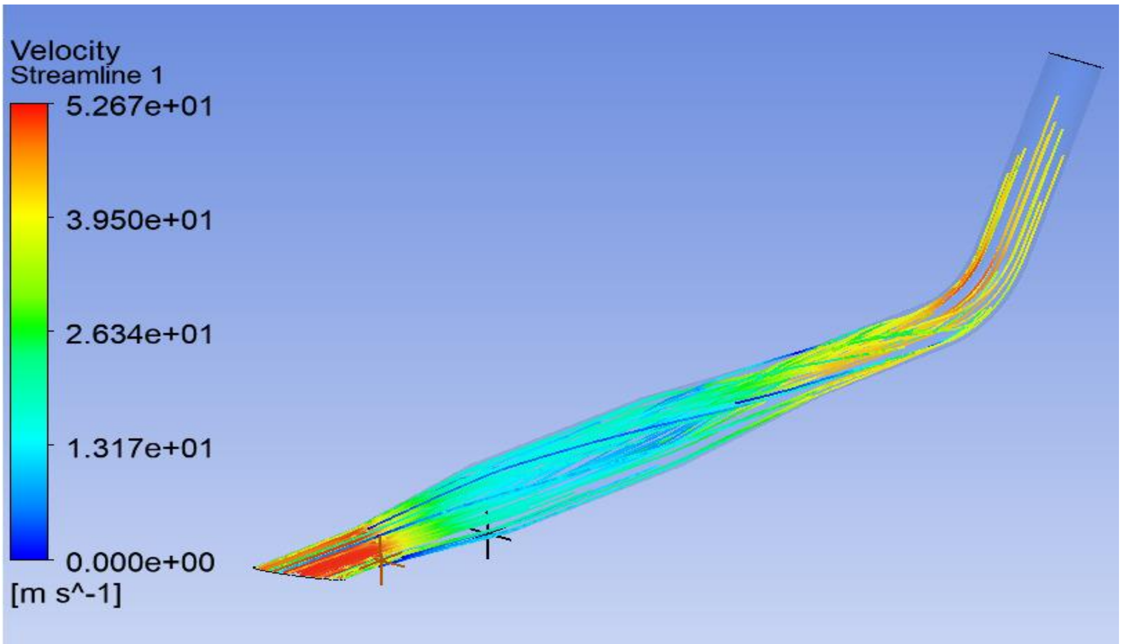

Figure 16. Velocity streamlines from inlet – iteration 3

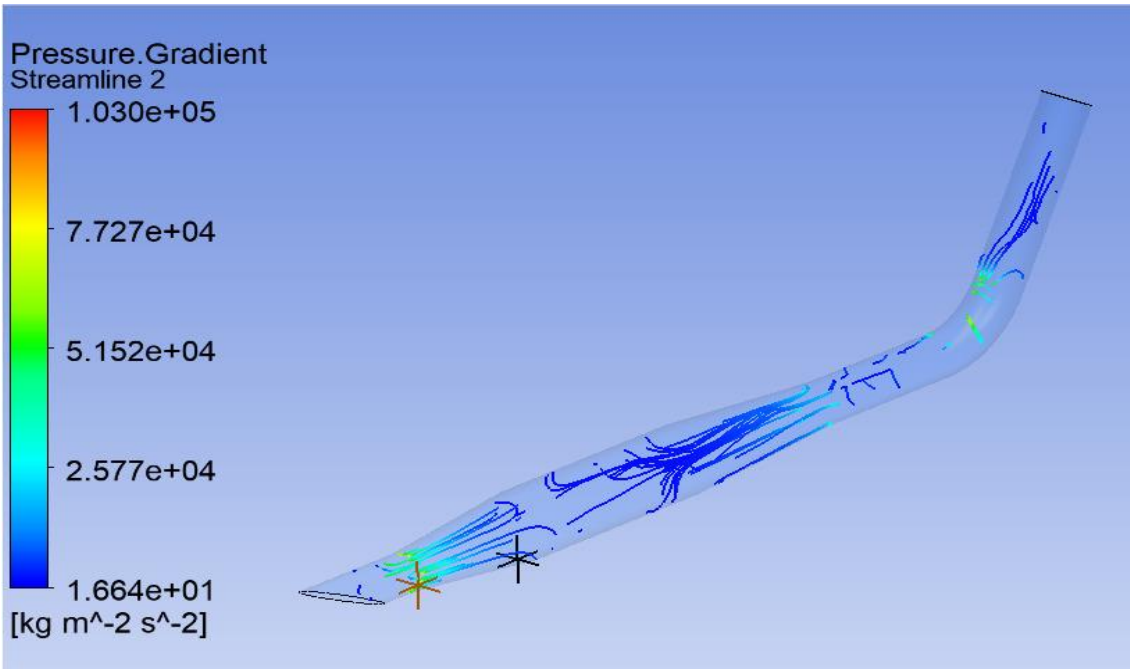

Figure 17. Reverse pressure gradient –iteration 3

In the third iteration we find that the magnitude of the velocity is quite high at the exit of the pipe. This increase is due to the cut angle provided at the outlet. This reduces the boundary resistance of the gases thus flushing the gases out faster. But the reverse pressure gradient is higher than iteration 3 with maximum pressure points near the exit region. This can counteract the increase in exit velocity due to the built-up pressure at various points through the flow.

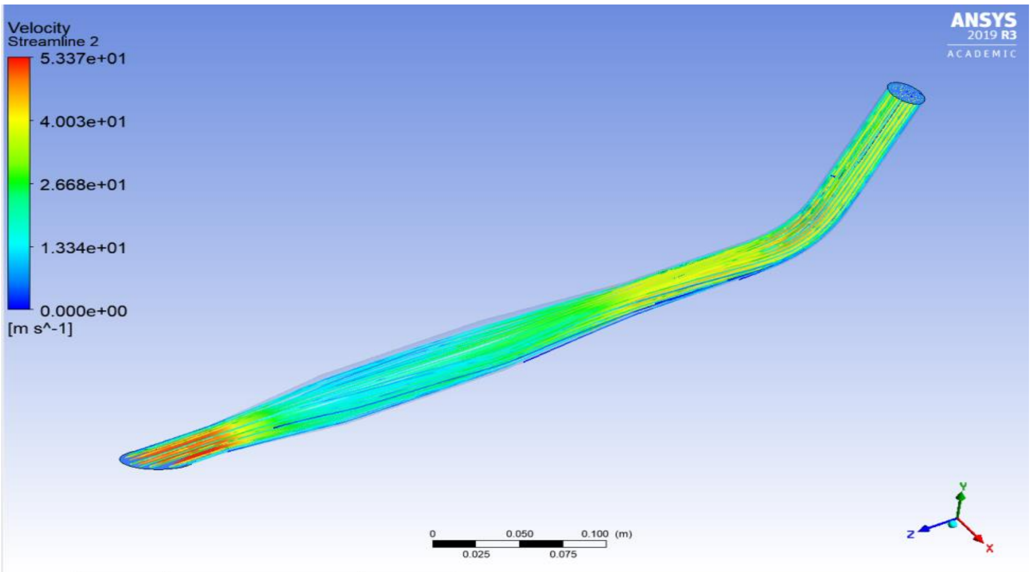

Figure 18. Velocity streamlines from inlet – iteration 4

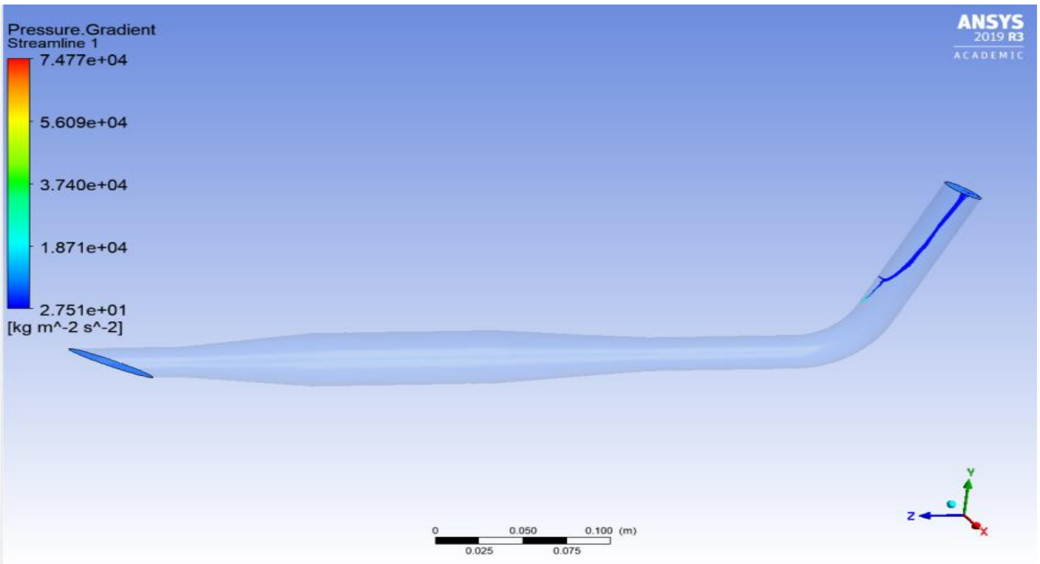

Figure 19. Reverse pressure gradient –iteration 4

The difference between the iteration 3 and 4 is that the path length is 600mm and the bend angle is 115 degrees. Just by a small variation, a large amount of reverse pressure gradient is excluded. The velocity is higher at the exit than iteration 2 which had nearly zero reverse pressure gradient. The reverse pressure gradient in the iteration 4 is significantly lesser than iterations 1 and 3.

7. Conclusion

From the analysis of four iterations, it is evident that iteration-2 has the least reverse pressure gradient through its flow. It has the following dimensions:

Table 4. Dimensions of iteration -2

Inlet diameter	24mm
Outlet diameter	24mm
Path Length	620mm
Convergent -divergent length	313.8 +/- 0.5mm
Bend angle	135°
Exit cone angle	180°

Exhaust gases tend to maintain a favourable velocity profile in iteration-2, through the bend region. This region is prone to incur a loss in velocity of gases, which is exhibited in other iterations. Reverse pressure gradient is almost zero at any point in iteration-2 which offsets the difference in a minuscule drop in velocity at the exit compared to other iterations.

Another differential advantage is that, the nature of flow in the region of lower velocity tends to be streamlined and does not show a vortex characteristic. Also, the inlet and outlet diameter are the same which simplifies the range of material dimensions required to manufacture. Hence it can be concluded that the iteration-2 model would be the perfect exhaust model for the input conditions and output requirements of this study.

8. Future Scope

Subsequent work involves analyzing noise attenuation on the desirable exhaust system. Parametric optimization for all the dimensional parameters be performed and the best iteration is to be selected. Experimental analysis has to be performed on the exhaust for the suitable engine. Cross verification with the experimental results is to be carried out to complement the theoretical simulation results. To facilitate this, theoretically obtained ideal exhaust design has to be fabricated with the provision to physically measure the flow parameters. Usage of MAF sensor and pressure measurement devices have to be employed in the verification process.

9. References

[1] Teja M A, Ayyappa K, Katam S and Anusha P 2016 *Fluid Mech Open Access* **3** 129
[2] Onorati A, Ferrari G, Errico G D and Montenegro G 2002 *SAE technical paper series* **01** 0901
[3] Gopaal,Varma and Dr L Suresh Kumar 2014 *International Journal of Engineering Trends and Technology* **17** 295
[4] Venkatesan S P, Ganesan S, Devaraj R and Hemanandh J 2018 *International Journal of Ambient Energy* **41** 659-664
[5] Navadagi Vand Sangamad S 2014 *International Journal of Engineering Research and Technology* **3** 92-97
[6] Aradhye O and Bari S 2017 *Proc. of the ASME 2017 Int. Mechanical Engineering Congress and Exposition*, *ICME* vol 6 **(**Florida: ASME)
[7] Bajpai K, Chandrakar A, Akshay A and Shekhar S 2017 *IOSR Journal of Mechanical and Civil Engineering* **14** 23-29
[8] Manikandan P, Durai A S, Kumar S S, Kumar R S and Krishnan M N 2019 *South Asian Journal of Engineering and Technology* **8** 257-261
[9] Kennedy, Woods *et al* 2011 *Proc. of the ITRN* (Ireland: University College Cork)
[10] Umesh K S , Pravin V K and Rajagopal K 2013 *Journal of Automobile Engineering Research and development* **3** 11-22