CFD Analysis of Exhaust Manifold of Multi-Cylinder Petrol Engine for Optimal Geometry to Reduce Back Pressure.

Chowdari Vikram[1], V. Pradeep Kumar[2] & P. Ram Mohan[3]
M.Tech[1], Associate Professor[2]
Gokul Group Of Institutions,AP

Abstract— Exhaust manifold is one of the critical components of IC engine for improving the volumetric efficiency. The volumetric efficiency of the engine can be increased by reducing the backpressure in the exhaust manifold. This work analyzes the flow through two different models of exhaust manifold using CFD. The design of exhaust manifold is modified to get optimal geometry. CFD analysis was conducted for different rotational speeds of engine and peak load of engine as well.

Introduction

Exhaust manifold

The exhaust manifold is a pipe, receives the exhaust gases from the combustion Chamber and leaves it to the atmosphere. Exhaust manifolds are mounted to the cylinder head. V- Type engines have two exhaust manifolds, and an in-line engine usually has one. When intake and exhaust manifolds are on opposite sides of an in-line engine, the head is called a cross-flow head. This design improves breathing capacity of an engine. Exhaust manifolds are typically made of cast iron or steel, although some latest-model cars use stainless steel manifolds. Cast iron is a good material for exhaust manifolds. Like the frying pan on a stove, it can tolerate fast and severe temperature changes. Exhaust gas temperature is related to the amount of load on the engine.

Types of turbo manifolds

Log manifold:

The log type manifold is the most economical manifold on the market. They are almost bullet proof when it comes to reliability and they are the most compact manifolds available. Most manifolds you find on factory turbocharged cars are log type manifolds. This is understandable when you (Figure 1) manufacturer is all about reliability. When it comes to performance, a log manifold will leave you wanting. All of the exhaust gases flow into a common way where they often collide with each other causing a lot of turbulence before the turbo. This hurts the turbo’s performance and hurts the motors performance as well being that a log manifolds common plenum is so close to the other cylinders, it causes a lot of back flow of exhaust.

Tubular manifold:

A tubular manifold is one that is custom made for that particular user’s goal for the car. Most road racing cars will build an average runner, equal length manifold that will support great flow characteristics throughout the entire rev range. A car being set-up for

Tubular manifold

drag racing will use a long runner tubular manifold, which will favor flow in the upper rev range. Tubular manifolds are very efficient when compared to a log manifold. They offer superior flow characteristics, and offer less back flow of exhaust gases back into the Motor. Now for the negative tubular manifolds are often prone to cracking causing them to lose points in the reliability column. For those focused on getting the most performance they can, a tubular manifold is a must (Figure 2).

(a) Single scroll manifold: A single scroll manifold is one where all the exhaust runners come to a common collector where they enter the turbocharger. Also all the runners share a common waste gate to expel exhaust gases. This is the most common type of tubular manifold.

(b) (b) Twin Scroll or divided manifold: A twin scroll manifold is designed for the most ideal exhaust flow a motor can offer. These manifolds are built so that runners that come together are paired to be 180 degrees apart from each other on their firing order. This is done so that there is little chance of exhaust flow interference from another cylinders exhaust gases. This not only helps the motor to better expel its exhaust, but also helps to keep the exhaust velocity as high as possible, aiding in the turbochargers spool and the motors torque output. A twin scroll manifold must be used with a turbo that is equipped with a twin scroll housing that keeps the exhausts divided all the way to the turbo's exhaust wheel.

Factors to be considered during the design and development of exhaust manifold

Runner length:

This is arguably one of the most important factors. First would be to make sure that the runners are as equal length as possible. The idea being that the exhaust pulses will be spaced out evenly and arriving at the turbine wheel on the turbo at their own time in the firing order. If they arrive sooner or later, they may interfere with the exhaust pulses from the next firing cylinder. Next, a longer runner manifold will have better flow up top, while a shorter manifold can yield a faster spool, with also less transient lag.

Runner volume:

Runner volume needs to be considered when building a turbo manifold. While a larger runner diameter does facilitate lower exhaust backpressure for better flow on the top-end, it does cause a lower exhaust velocity. A lower exhaust velocity will cause longer spool times, and less transient response out of the turbo.

Collectors:

A collector's job is to tie all of the cylinder's pipes together in one common place and send them into a single exit pipe. A collector is generally a conglomeration of pipes all merged together, allowing for a smooth transition from the primaries or secondaries into the rest of the exhaust. When properly constructed, a good collector will take the low pressure waves created earlier and send them back up the primaries, thus quickening the entire evacuation process. There are two scenarios in which exiting exhaust gases will encounter once they move past the valve: low-pressure or high-pressure. Low-pressure situations within the exhaust pipes help promote better flow by allowing for increased velocity through the exhaust ports while high-pressure situations do the opposite.

Back pressure:

Back pressure can be produced at two places, i.e., when the exhaust valve opens and cam overlap taking place shown in figure 2. Pressure measurements at the exhaust valve during the start of the exhaust stroke at bottom dead centre (BDC) to cam overlap at the end of the exhaust stroke/beginning of the intake stroke at top dead centre (TDC). Notice the positive (backpressure) spike at the far left as the exhaust valve just opens at BDC. The exhaust gases must now push against this POSITIVE (back) pressure before it can leave the combustion chamber. The pressure tracing is upwards and positive. Energy must be used up in order to overcome the initial positive (back) pressure in the exhaust system before the exhaust gas is pushed out of the combustion chamber. Further we must be able to overcome the positive backpressure. It well known that, the exhaust gas begins to travel faster and creates a NEGATIVE pressure (Figure 3).

In the figure 1 the pressure tracing is downward or has a negative value. When it is with a more negative pressure, then it means that, there is creation of more suction or a vacuum in the system. The system is literally sucking or pulling out exhaust gas from the combustion chamber or cylinder. This sucking or "SCAVENGING" affect not only helps to remove more exhaust gas from the cylinder. It also helps to suck more intake air and fuel mixture. Faster the exhaust gas travel more will be the vacuum creation. However, it is necessary to have much as negative pressure creation before cam overlaps.

Exhaust pulses and energy pulses

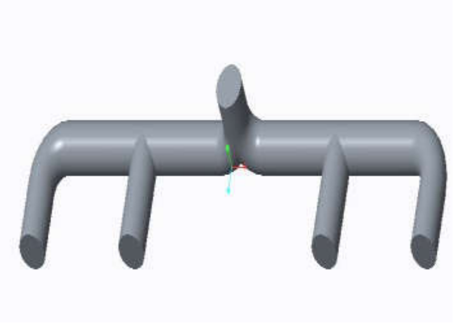
To understand how a typical exhaust system works, you first have to understand what exhaust pulses are. As for the exhaust manifold, its job is simple: to contain the gases that leave the cylinder head’s exhaust ports just after each power stroke. These gases exit the ports in the form of pulses, high-speed, high-pressure pulses created by the piston moving upward thus squishing everything out. The piston creates the pressure, but said pressure disappears once the exhaust valves close. Surrounding each exhaust pulse on a well-designed header is a large pressure differential with more pressure in front of the pulse than behind it – this is due to the valve closing. This pressure differential actually assists in evacuating the exhaust gases most efficiently. The process is as follows: once the high-speed, high-pressure exhaust gases exit past the exhaust valves, a low-pressure zone is left behind; in the best scenarios it not only leaves low pressure, but also leaves the negative pressure in the form of a vacuum. If you can control when this negative pressure referred to as the vacuum reaches the port then you can expect significant power gains. The process relies on the inertia created by the fast-moving gases. When strong enough, the inertia can actually help coerce gases through the open valves during overlap periods. A side from the exhaust gas pulses, there’s another kind of pulse going on here. Sound waves are also emitted into the manifold each time an exhaust valve opens up.

Traveling at the speed of sound, these waves move at a quicker rate than the exhaust pulses do. Also in contrast to exhaust pulses, pressure waves bounce around within the pipes, and depending upon diameter changes and a number of other factors, they may often revert direction. The results are exhaust gas pulses traveling in one direction and low pressure energy waves bouncing around all over the place. If the manifold can allow for these low-pressure energy waves to arrive back at the exhaust valves just as they once again open, the whole process can be used to the engine’s advantage. But this requires everything to be timed correctly and this is nothing short of difficult to figure out sometimes

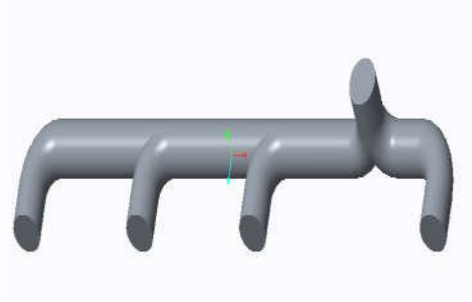
CONSTRUCTION

The exhaust manifold system considered in the present case has 4 inlets connected to the exhaust port of the engines and a single outlet from where the flow is passed on to the exhaust system before ejection into the ambient. Due to lack of experimental data, an industrially available

manifold geometry has been considered for the present analysis. It consists of the pipe diameter of 42 mm and total span of the manifold to be 0.6 m. The base model has the outlet placed besides the first port having smaller length of the curved pipe from the individual engine exhausts. The modified geometry has the outlet port placed in the middle with longer curved pipes from the individual engine exhausts



modified



Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reaction (e.g., combustion), and related phenomena by solving the mathematical equations that govern these processes using a numerical algorithm on a computer. The technique is very powerful and spans a wide range of industrial and non-industrial application areas.

GOVERNING EQUATIONS OF FLUID FLOW:

The governing equations of fluid flow represent mathematical statements of the conservation laws of physics. Each individual governing equation represents a conservation principle. The fundamental equations of fluid dynamics are based on the following universal laws of conservation. They are,

- Conservation of mass
- Conservation of momentum
- Conservation of energy

Continuity Equation:

The equation based on the principle of conservation of mass is called continuity equation. The conservation of mass law applied to a fluid passing through an infinitesimal, fixed control volume yields the following equation of continuity,

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \qquad (4.1)$$

$$\frac{\partial \rho}{\partial t}+div(\rho V)=0 \qquad (4.2)$$

Where 'ρ' is the fluid density, u, v, and w is the fluid velocity vectors. For an incompressible flow, the density of each fluid element remains constant.

Energy Equation:

This equation is based on the principle of conservation of energy; the energy equation is derived from first law of thermodynamics which states that the rate change of energy of a fluid particle is equal to the rate of heat addition to the fluid particle plus the rate of work done on particle,which is

$\rho \frac{DE}{Dt}$ = The rate of change energy of a fluid particle

E = Internal energy + kinetic energy + gravitational energy

$$\mathrm{E}=i+½\left(u^2+v^2+w^2\right)+\mathrm{g} \qquad (4.11)$$

$-\frac{\partial q_x}{\partial x}-\frac{\partial q_y}{\partial y}-\frac{\partial q_z}{\partial z}=-div\,\mathrm{q}$ = The rate of heat addition to the fluid particle. (4.12)

$q_{x,}q_{y,}q_z=-k\frac{\partial T}{\partial x},-k\frac{\partial T}{\partial y},-k\frac{\partial T}{\partial z}$ = Heat flux components (4.13)

$\left[\frac{\partial[u(-p+\tau_{xx})]}{\partial x}+\frac{\partial(u\tau_{yx})}{\partial y}+\frac{\partial(u\tau_{zx})}{\partial z}\right]\delta x\,\delta y\,\delta z$ = Net rate of work done by force in ‘x’ direction (4.14)

Energy equation in conservative form:

$$\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right]+\nabla\cdot\left[\rho\left(e+\frac{V^2}{2}\right)V\right]=\rho\dot{q}+\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}-\frac{\partial(wp)}{\partial z}$$

$$+\frac{\partial(u\tau_{xx})}{\partial x}+\frac{\partial(u\tau_{yx})}{\partial y}+\frac{\partial(u\tau_{zx})}{\partial z}+\frac{\partial(v\tau_{xy})}{\partial x}+\frac{\partial(v\tau_{yy})}{\partial y}+\frac{\partial(v\tau_{zy})}{\partial z}+\frac{\partial(w\tau_{xz})}{\partial x}+\frac{\partial(w\tau_{yz})}{\partial y}+\frac{\partial(w\tau_{zz})}{\partial z}+pf.V \tag{4.15}$$

COMPUTATIONAL SOLUTION FOR TURBULENT FLOWS:

There are three main classes of techniques for dealing with turbulent flows:

- Direct numerical simulation (DNS)
- Large eddy simulation (LES)
- Reynolds averaged Navier-Stokes equations (RANS)

To resolve a turbulent flow by direct numerical simulation (DNS) requires that all relevant length scales be resolved from the smallest eddies to scales on order of the physical dimensions of problem domain. To get statistical quantities of interest, which cannot be measured, experimentally, can be evaluated from the simulations. DNS is used for developing an understanding of the physics of the flow and deployed in developing turbulence models for simple flows. However, from an engineering point of view, it provides far more information than an engineer needs, and it is simply too expensive to be employed on a regular basis.

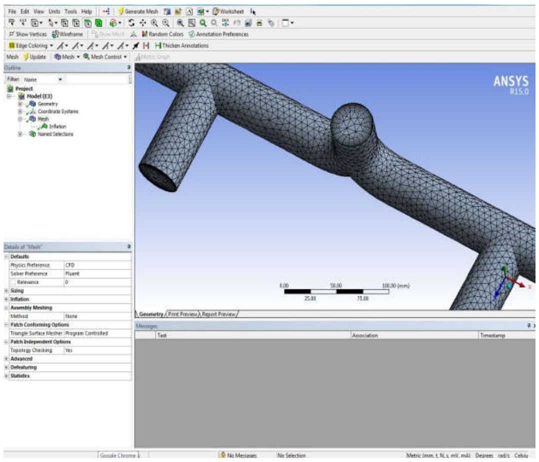
Most engineering flows of interest are in the turbulent regime, which contains a wide range of length and time scales. In large eddy simulation(LES) the large eddies are computed and the smallest eddies are modeled, as the large-scale motions are more energetic than the small scale ones and are responsible for most of the transport. The small-scale turbulence serves mainly to drain energy from the large scales through the cascade process, and is more universal and nearly isotropic, which makes it more suitable to be modeled The computational effort required for LES is less than that of DNS by approximately a factor of 10 using present day methods.

The main thrust of present-day research in computational fluid dynamics in turbulent flows is through the time-averaged Navier Stokes equations. These equations are referred to as the Reynolds equations of motion or the Reynolds averaged Navier-Stokes(RANS) equations. Time averaging the equations of motion gives rise to new terms, which can be interpreted as "apparent" stress gradients associated with the turbulent motion. These new quantities must be related to the mean flow variables through turbulence models.

The Reynolds equations are derived by decomposing the dependent variables in the conservation equations into time-mean (obtained over an appropriate time interval) and fluctuating components and then time averaging the entire equation.

MESHING:

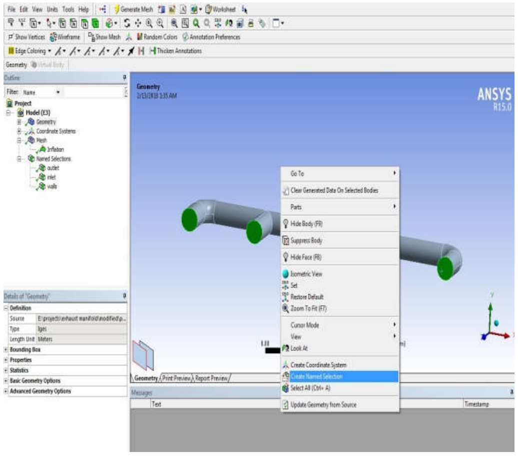
Initially mesh was generated with global mesh settings and was refined using inflation taking 10 layers around the the boundary walls.



Details of "Mesh"

Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Sizing	
Inflation	
Assembly Meshing	
Method	None
Patch Conforming Options	
Triangle Surface Mesher	Program Controlled
Patch Independent Options	
Topology Checking	Yes
Advanced	
Defeaturing	
Statistics	

Selection of inlet, outlet and walls boundaries has been done using named selection



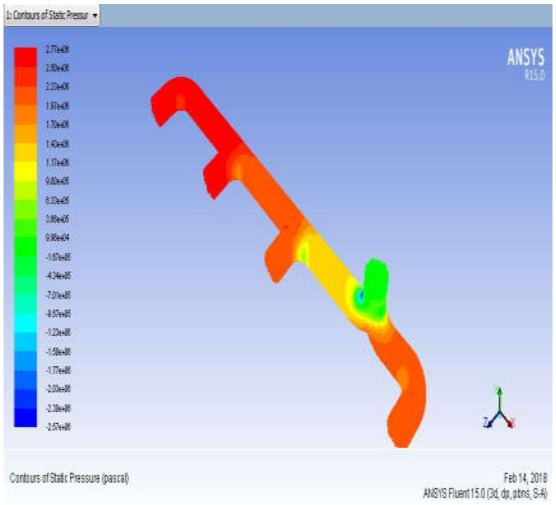
BOUNDARY CONDITIONS The two models will be tested for 4 different engine loads corresponding to different mass-flow rates at each inlet:

Cases Engine speed(rpm) Total exhaust gas	flow rate (m3/s)
1300	3.143
1700	4.516
1800	4.981
5500rpm	12.4525

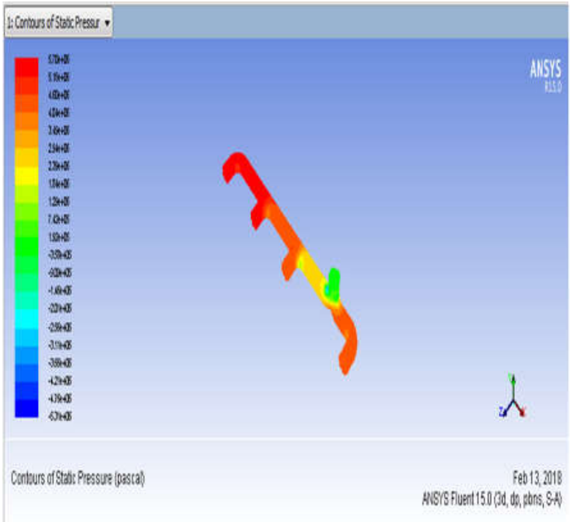
Results:

Pressure contours for existing model:

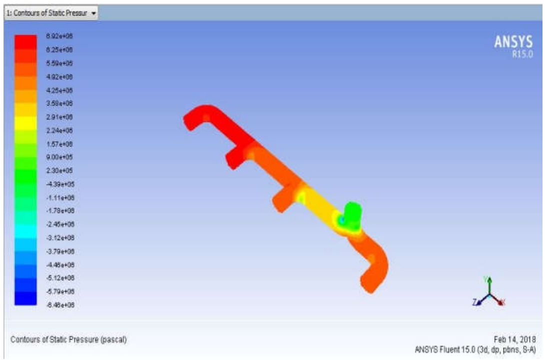
1300rpm



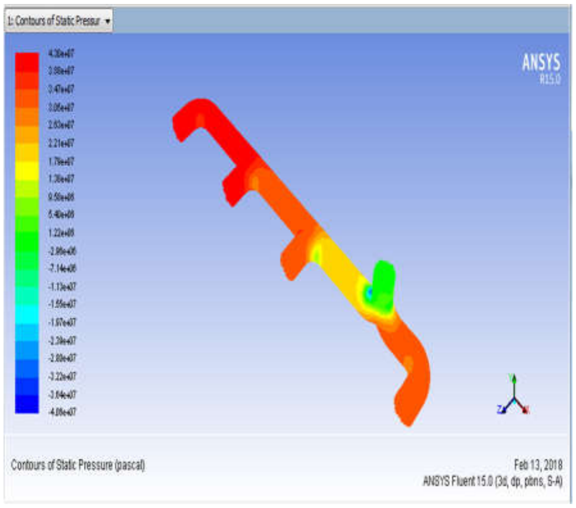
1700RPM



1800RPM

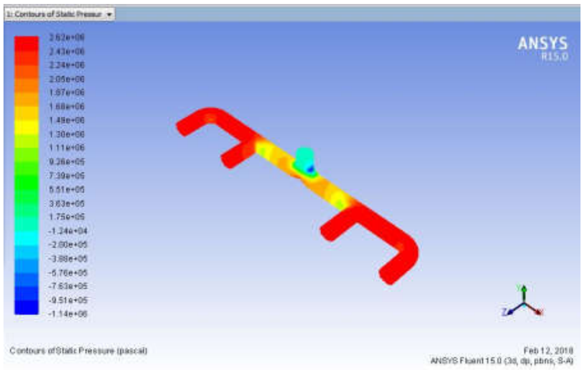


5500RPM

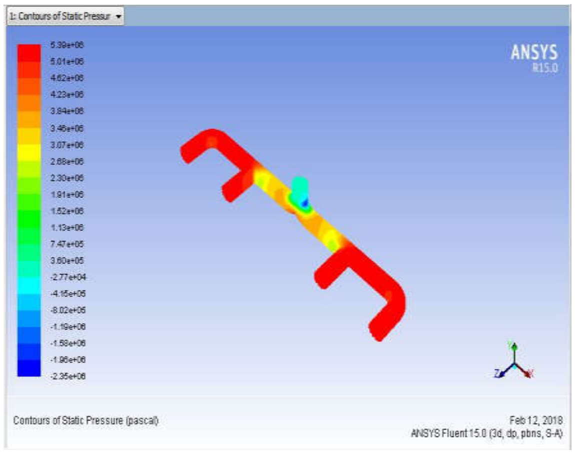


Pressure contours for modified model:

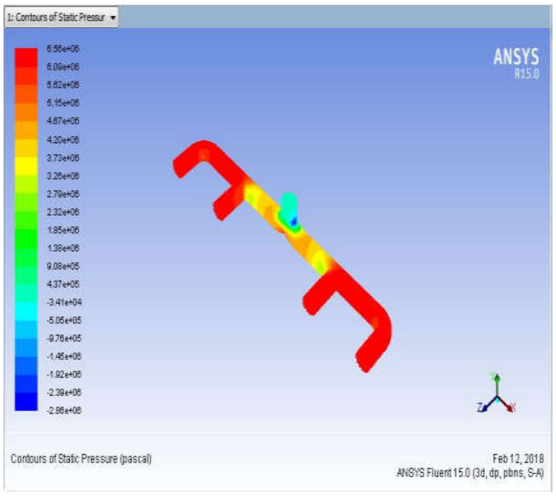
1300 rpm



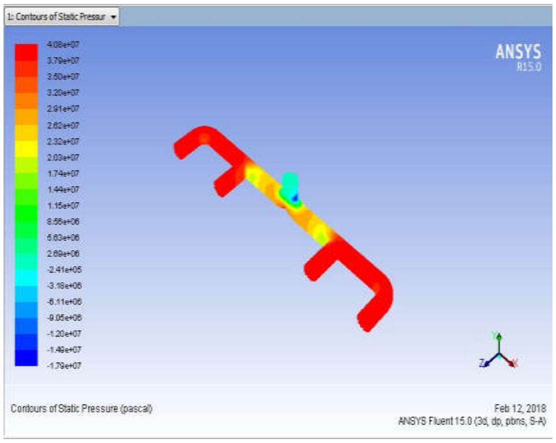
1700 rpm



1800rpm



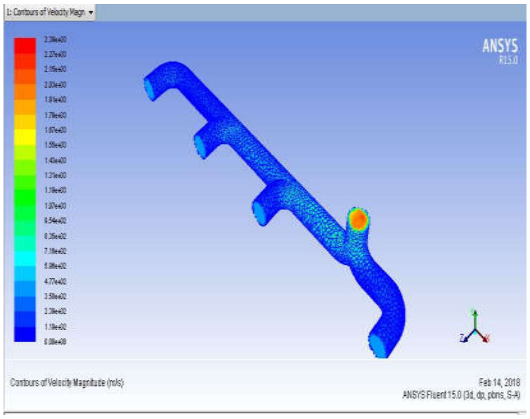
5500rpm



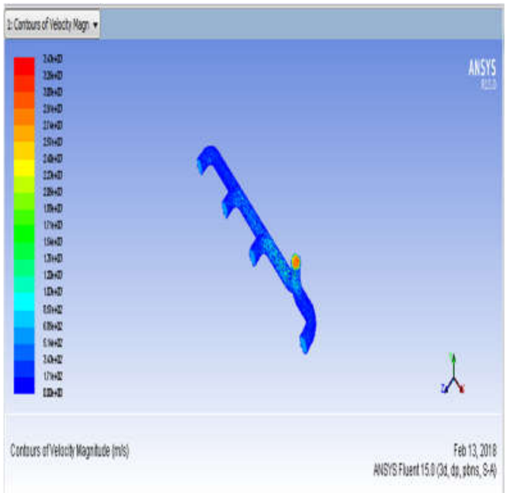
Velocity contours:

Existing model

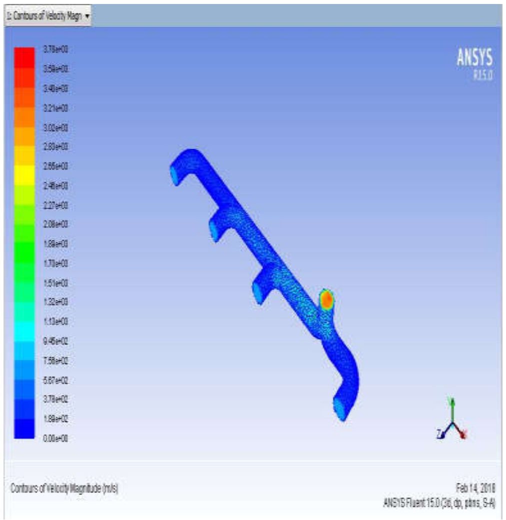
1300rpm



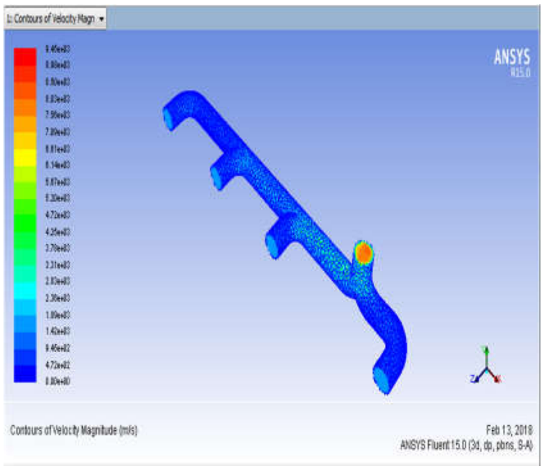
1700rpm



1800rpm

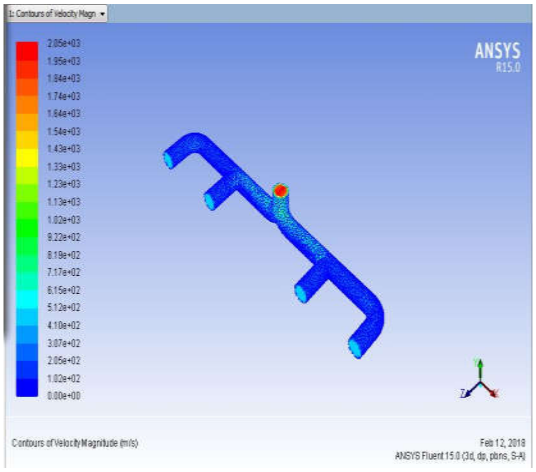


5500rpm

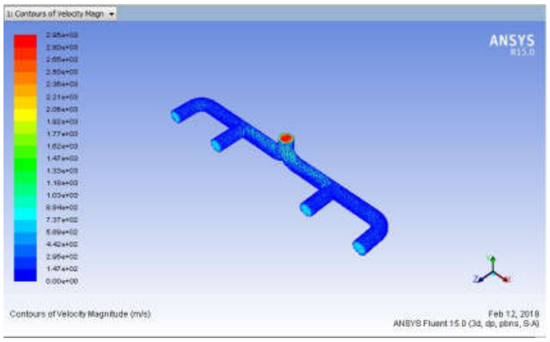


Modified model

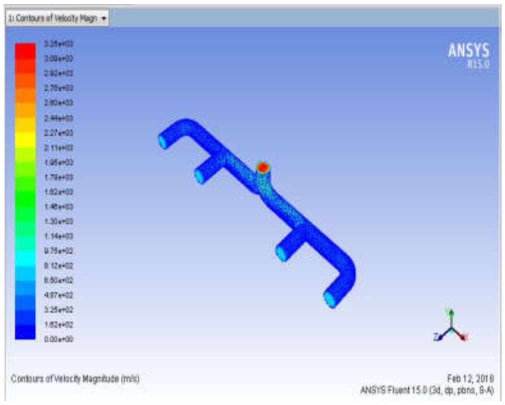
1300 rpm



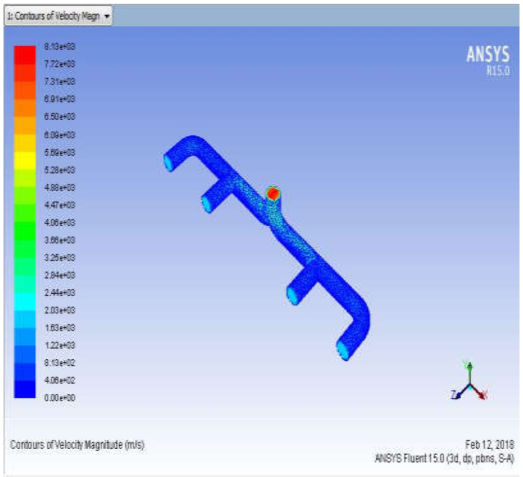
1700 rpm



1800rpm



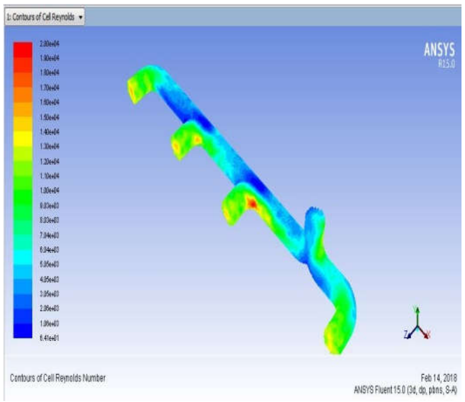
5500 rpm



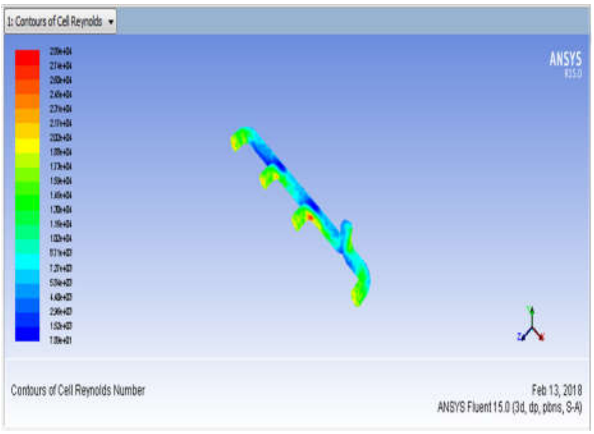
Reynolds number variation:

Existing model:

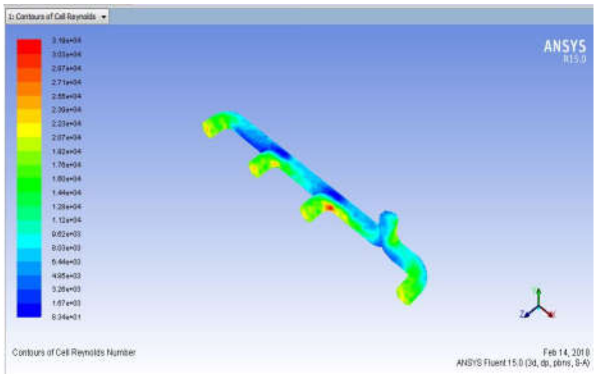
1300rpm



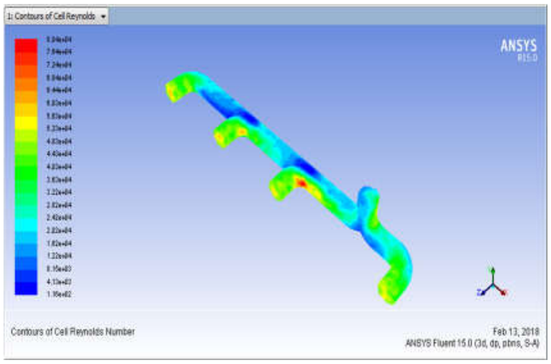
1700rpm



1800rpm

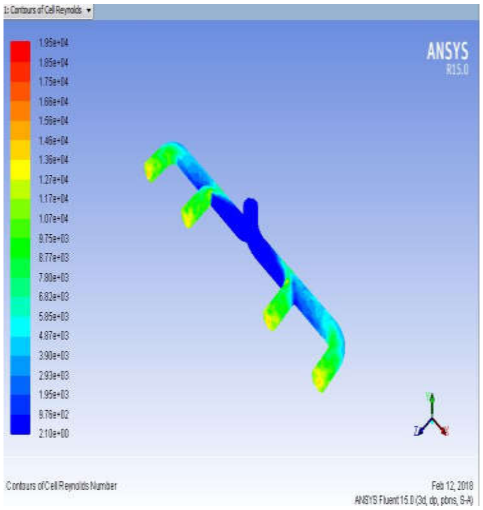


5500rpm

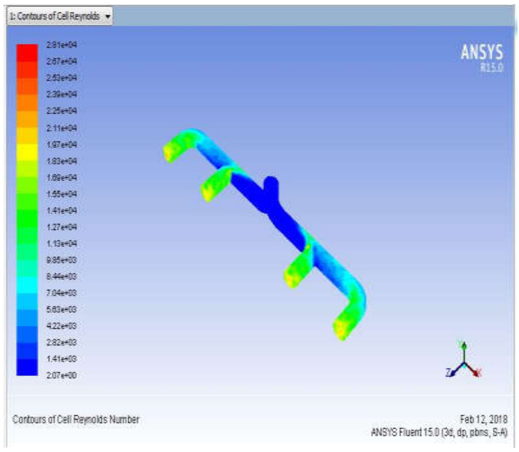


Modified model:

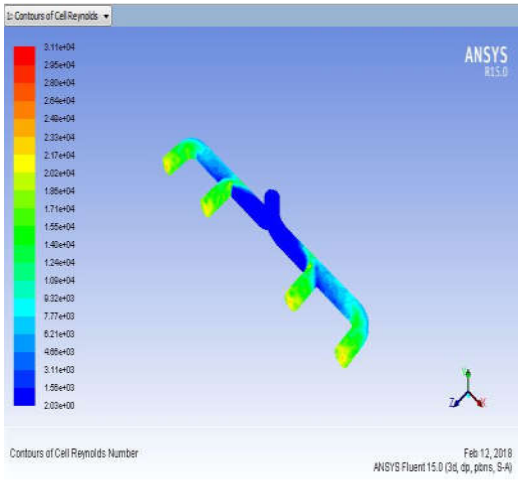
1300 rpm



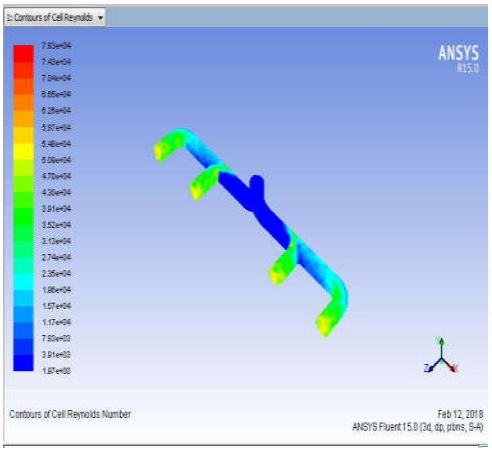
1700rpm



1800 rpm



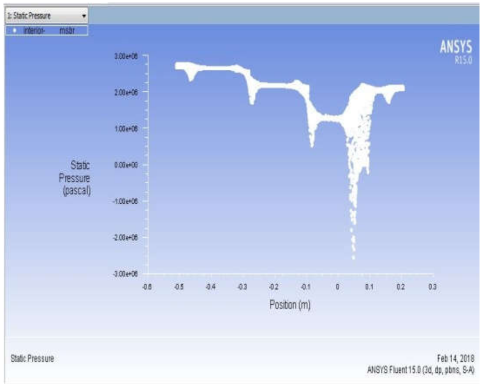
5500rpm



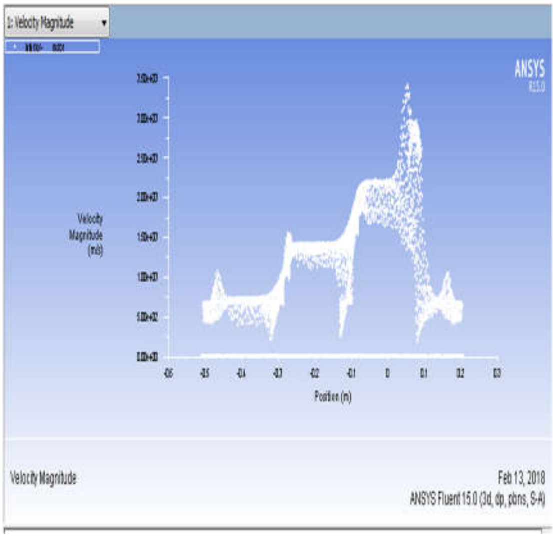
Pressure vs X-length

Existing model:

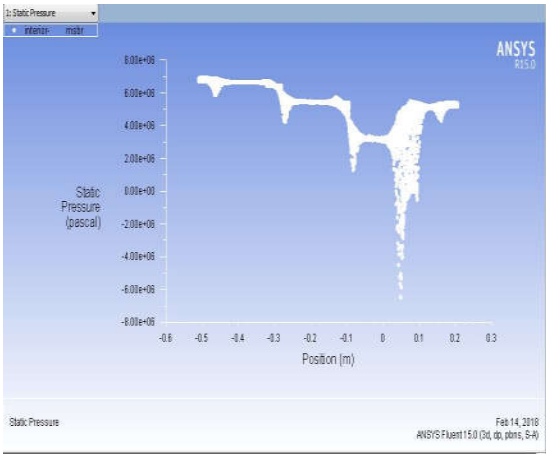
1300rpm



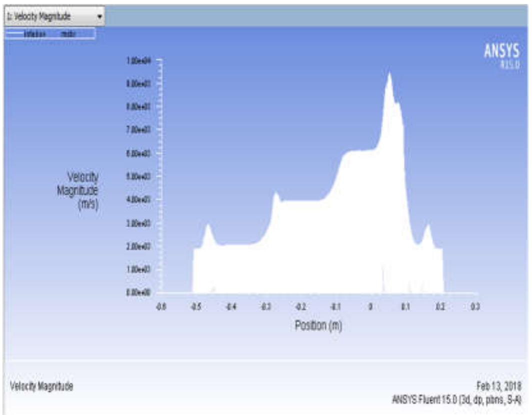
1700rpm



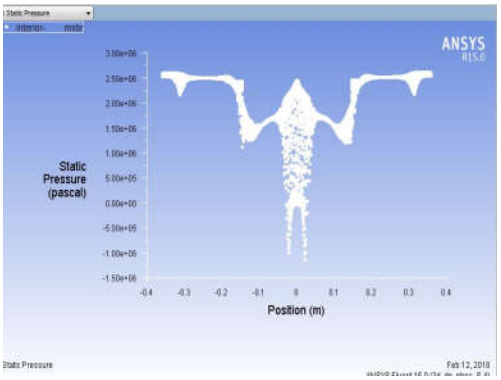
1800rpm



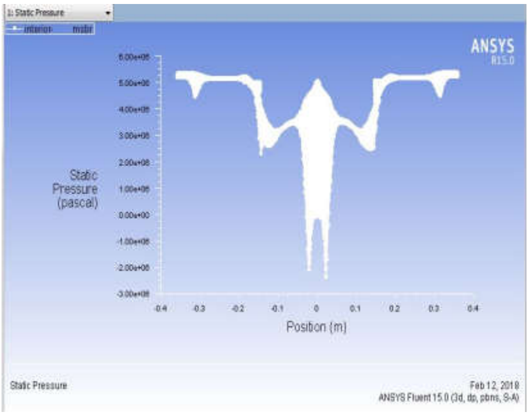
5500rpm



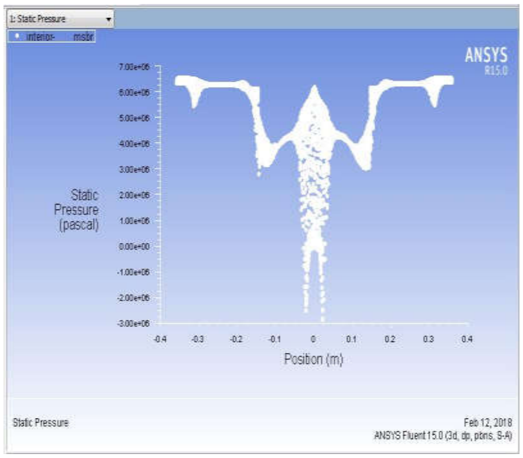
1300rpm



1700rpm

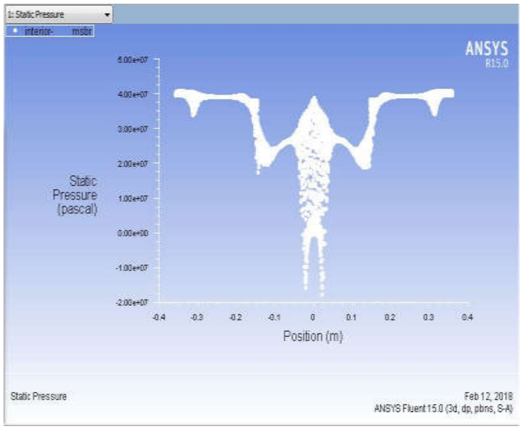


1800rpm



5500 rpm

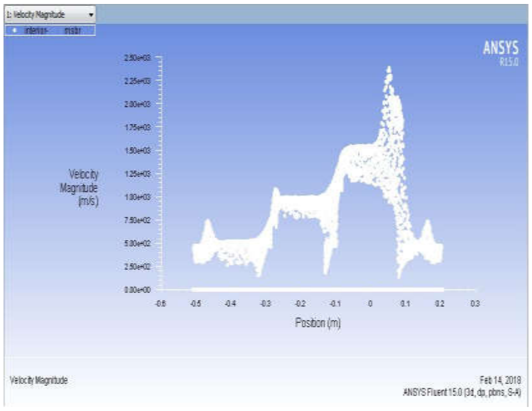
Modified model:



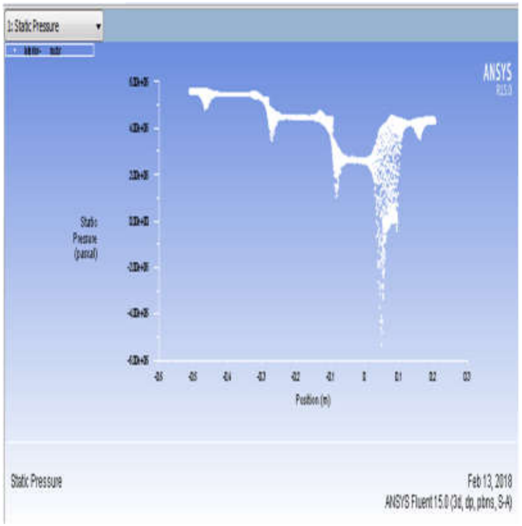
Velocity vs X-length plot

Existing model:

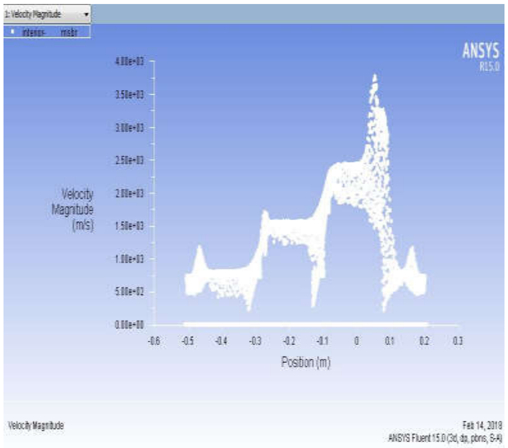
1300rpm



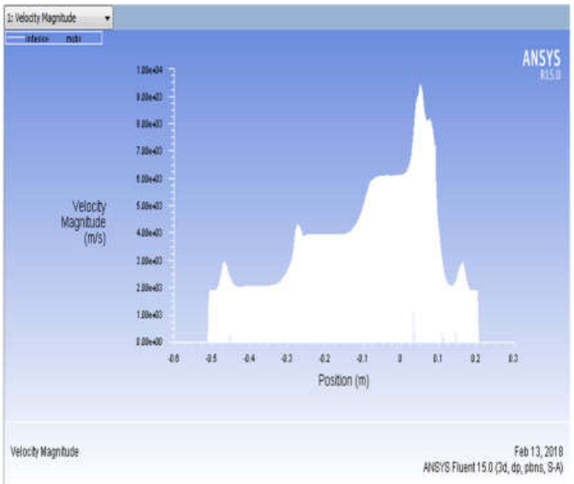
1700rpm



1800rpm

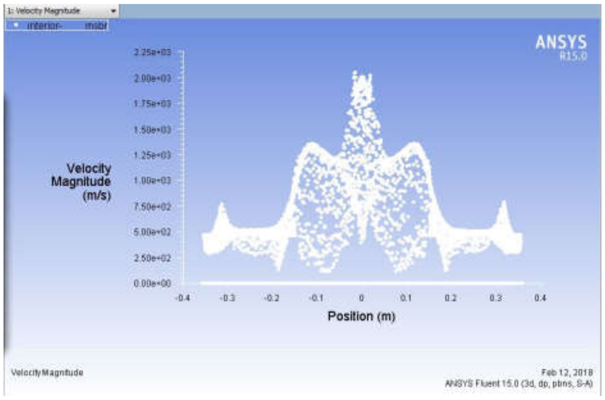


5500rpm

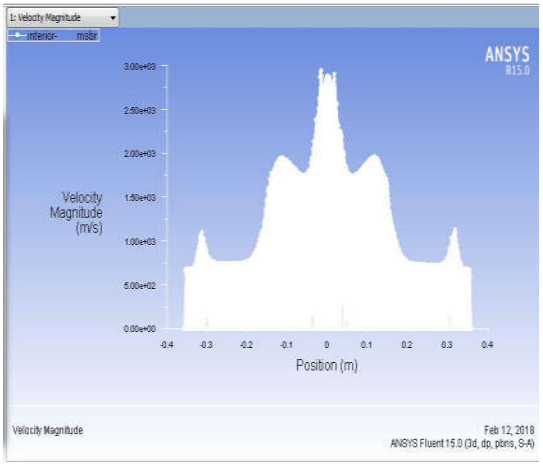


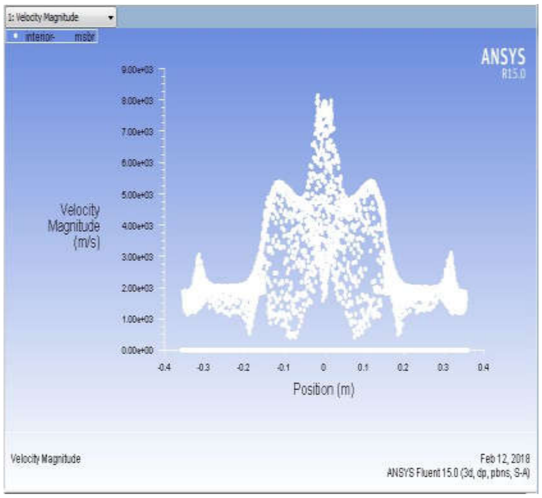
Modified model:

1300rpm

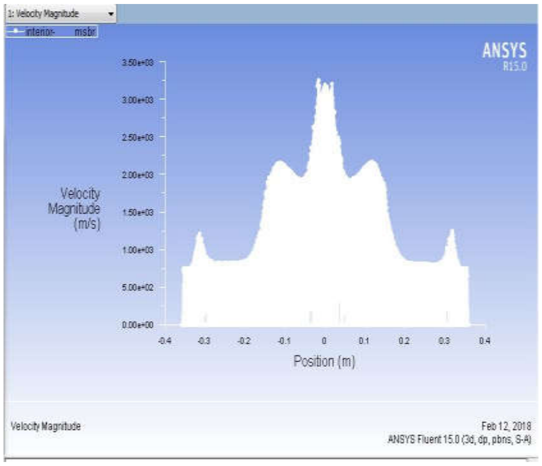


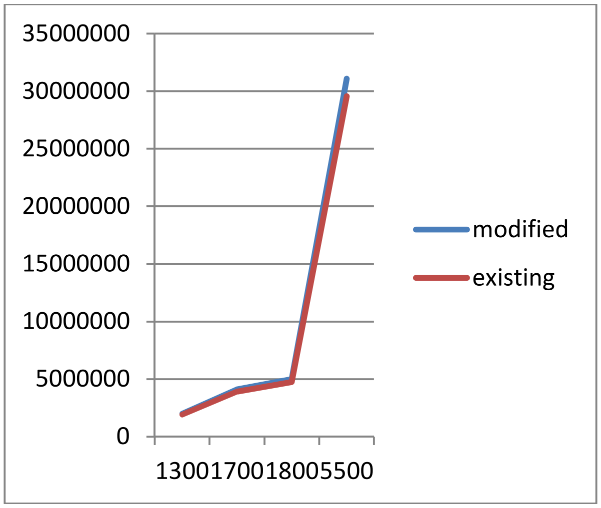
1700rpm





1800rpm





5500 rpm

COMPARISON OF PRESSURE DROP

Speed(Rpm)	existing	modified
1300	1900083	1991116.4
1700	3913070	4105002.5
1800	4757497	4992287.5
5500	29574228	31094092

CONCLUSION

The flow analysis of exhaust manifold was performed. The existing manifold is modified by changing its geometry to get optimal geometry. Both old and new models are analyzed under same boundary conditions. The results of new model are compared with the existing model. Pressure and velocity graphs are drawn for the new model and are compared with existing model. The decrease in back pressure is shown by using contour and vector diagrams. The flow is made efficient by decreasing the exhaust gas back pressure in the newly modified model thus increasing the volumetric efficiency of the engine.

REFERENCES

1. Seenikannan P., Periyasamy V.M. and Nagaraj P. "An experimental analysis of a Y section exhaust manifold system with improved engine performance".

2. Yasar Deger, Burkhard Simperl, and Luis P. Jimenez "Coupled CFDFE- Analysis for the Exhaust Manifold of a Diesel Engine" Sulzer Innotec, Sulzer Markets and Technology Ltd, Winterthur, Switzerland.

3. Dr. Kutaiba J.M. AL-Khishali, Dr. Mahmoud A. Mashkour & Ehsan Shamil Omaraa "Analysis of Flow Characteristics In Inlet And Exhaust Manifolds of Engine" Experimental Gasoline Combustion In A VCR Engine".

4. Scheeringa K**.** **"**Analysis of Liquid cooled exhaust manifold using CFD".

5. Gopal P., Senthil Kumar T. and Kumaragurubaran B., "Analysis of Flow Through The Exhaust Manifold of a Multi Cylinder Petrol Engine for Improved Volumetric Efficiency" International Journal of Dynamics of Fluids ISSN 0973-1784 Volume 5 Number 1 (2009)

6. K.S. Umesh, V.K. Pravin and K. Rajagopal "CFD Analysis and Experimental Verification of Effect of Manifold Geometry on Volumetric efficiency and Back Pressure for Multi-cylinder SI