

Analysis of Exhaust Manifold using Computational Fluid Dynamics

Marupilla Akhil Teja*, Katari Ayyappa, Sunny Katam and Panga Anusha

SIR C R Reddy College of Engineering, Andhra University, Vizag, India

Abstract

Overall engine performance of an engine can be obtained from the proper design of engine exhaust systems. With regard to stringent emission legislation in the automotive sector, there is a need design and develop suitable combustion chambers, inlet, and outlet manifold. Exhaust manifold is one of the important components which affect the engine performance. Flow through an exhaust manifold is time dependent with respect to crank angle position. In the present research work, numerical study on four-cylinder petrol engine with two exhaust manifold running at constant speed of 2800 rpm was studied. Flow through an exhaust manifold is dependent on the time since crank angle positions vary with respect to time. Unsteady state simulation can predict how an intake manifold work under real conditions. The boundary conditions are no longer constant but vary with time. The main objectives that to be studied in this work is:

- To prepare the cad model in the CATIA software by using the actual parametric dimensions.
- To prepare finite element model in the Computer aided analysis software by specifying the approximate element size for meshing.
- To find and calculate the actual theoretical values for the input boundary conditions.
- To study the flow patterns generated due to the flow of the exhaust gases from the manifold.
- To study the velocity and pressure distribution in ports at maximum flow rate.
- To study the static pressure drop, total pressure drop, and energy loss in the flow pattern generated in the exhaust manifold.

Keywords: Exhaust-manifold; Computational fluid dynamics; Back-pressure

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



Figure 1: Log type 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) Twin Scroll or divided manifold: A twin scroll manifold

*Corresponding author: Teja AM, SIR C R Reddy College of Engineering, Andhra University, Vizag, India, Tel: 5712300489; E-mail: mgsateja@gmail.com

Received July 07, 2016; Accepted July 17, 2016; Published July 28, 2016

Citation: Teja MA, Ayyappa K, Katam S, Anusha P (2016) Analysis of Exhaust Manifold using Computational Fluid Dynamics. Fluid Mech Open Acc 3: 129.

Copyright: © 2016 Teja MA, et al. This is an open-access article distributed under the terms of the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original author and source are credited.



Figure 2: Tubular 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 cylinder's 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-

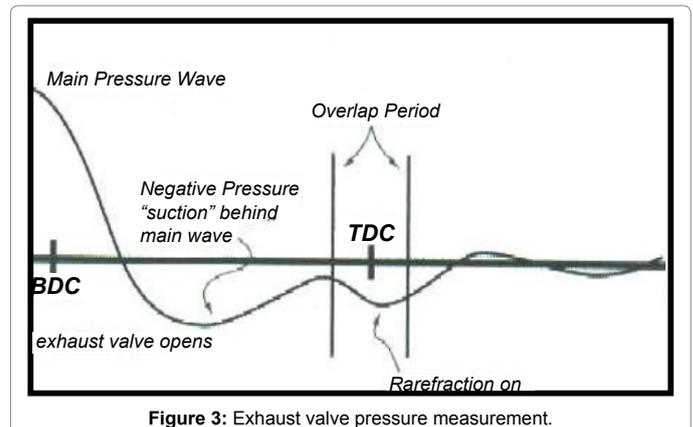


Figure 3: Exhaust valve pressure measurement.

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.

Introduction to CATIA V5R20

Welcome to **CATIA (Computer Aided Three Dimensional Interactive Application)**. As a new user of this software package, you will join hands with thousands of users of this high-end CAD/CAM/CAE tool worldwide. If you are already familiar with the previous releases, you can upgrade your designing skills with the tremendous improvement in this latest release.

CATIA V5, developed by Dassault Systems, France, is a completely re-engineered, Next-generation family of CAD/CAM/CAE software solutions for Product Lifecycle Management. Through its exceptionally easy-to-use and state-of-the-art user interface, CATIA V5 delivers innovative technologies for maximum productivity and creativity, from the inception concept to the final product. CATIA V5 reduces the learning curve, as it allows the flexibility of using feature-based and parametric designs.

CATIA V5 provides three basic platforms: P1, P2, and P3. P1 is for small and medium-sized process-oriented companies that wish to grow toward the large scale digitized product definition.

P2 is for the advanced design engineering companies that require product, process, and resource modeling. P3 is for the high-end design applications and is basically for Automotive and Aerospace Industry, where high quality surfacing or Class-A surfacing is used. The subject of interpretability offered by CATIA V5 includes receiving legacy data from the other CAD systems and even between its own product data management modules. The real benefit is that the links remain associative. As a result, any change made to this external data gets notified and the model can be updated quickly.

CATIA V5 workbenches: CATIA V5 serves the basic design tasks by providing different workbenches. A workbench is defined as a specified environment consisting of a set of tools that allows the user to perform specific design tasks. The basic workbenches in CATIA V5 are **Part Design, Wireframe and Surface Design, Assembly Design, Drafting**.

Part design workbench: The **Part design** workbench is a parametric and feature-based environment in which you can create solid models. The basic requirement for creating a solid model in this workbench is a sketch. The sketch for the features is drawn in the **Sketcher** workbench that can be invoked within the **Part Design** workbench. You can draw the sketch using the tools in this workbench. While drawing a sketch, some constraints are automatically applied to it. You can also apply additional constraints and dimensions manually. After drawing the sketch, exit the **Sketcher** workbench and convert it into a feature. The tools in the **Part Design** workbench can be used to convert the sketch into a sketch-based feature. This workbench also provides other tools to apply the placed features, such as fillets, chamfers, and so on. These features are called the dress-up features. You can also assign materials to the model in this workbench.

Wireframe and surface design workbench: The **Wireframe and Surface Design** workbench is also a parametric and feature-based environment, and is used to create wireframe or surface models. The tools in this workbench are similar to those in the **Part Design**

workbench with the only difference that the tools in this environment are used to create basic and advanced surfaces.

Assembly design workbench: The **Assembly Design** workbench is used to assemble the components using the assembly constraints available in this workbench. There are two types of assembly design approaches:

1. Bottom-up
 2. Top-down
- In the bottom-up approach of the assembly design, the components are assembled together to maintain their design intent.
 - In the top-down approach, components are created inside the assembly in the **Assembly Design** workbench. You can also assemble an existing assembly to the current assembly.
 - The **Space Analysis** toolbar provides the **Clash** analysis tool that helps in detecting clash, clearance, and contact between components and sub-assemblies.

Drafting workbench: The **Drafting** workbench is used for the documentation of the parts or assemblies created earlier in the form of drawing views and their detailing. There are two types of drafting techniques:

- Generative drafting
- Interactive drafting

The generative drafting technique is used to automatically generate the drawing views of the parts and assemblies. The parametric dimensions added to the component in the **Part Design** workbench during its creation can be generated and displayed automatically in the drawing views. The generative drafting is bidirectional associative in nature. You can also generate the Bill of Material (BOM) and balloons in the drawing views.

In interactive drafting, you need to create the drawing views by sketching them using the normal sketching tools and then adding the dimensions.

Generative sheet metal design workbench: The **Generative Sheet metal Design** workbench is used for the designing the sheet metal components. Generally, the solid models of the sheet metal components are created to generate the flat pattern of the sheet, study the design of the dies and punches, study the process plan for designing, and the tools needed for manufacturing the sheet metal components.

Introduction to ANSYS

ANSYS is a large-scale multipurpose finite element program developed and maintained by ANSYS Inc. to analyze a wide spectrum of problems encountered in engineering mechanics.

Program organization: The ANSYS program is organized into two basic levels:

- Begin level
- Processor (or Routine) level

The Begin level acts as a gateway into and out of the ANSYS program. It is also used for certain global program controls such as changing the job name, clearing (zeroing out) the database, and copying binary files. When you first enter the program, you are at the Begin level.

- At the Processor level, several processors are available. Each processor is a set of functions that perform a specific analysis task. For example, the general pre-processor (PREP7) is where you build the model, the solution processor (SOLUTION) is where you apply loads and obtain the solution, and the general postprocessor (POST1) is where you evaluate the results of a solution. An additional postprocessor, POST26, enables you to evaluate solution results at specific points in the model as a function of time.

Material models: ANSYS allows several different material models like:

- Linear elastic material models (isotropic, orthotropic, and anisotropic).
- Non-linear material models (hyper elastic, multi linear elastic, inelastic and Visco elastic).
- Heat transfer material models (isotropic and orthotropic).
- Temperature dependent material properties and Creep material models.

Loads: The word loads in ANSYS terminology includes boundary conditions and externally or internally applied forcing functions, as illustrated in Loads. Examples of loads in different disciplines are:

Structural: displacements, forces, pressures, temperatures (for thermal strain), Gravity.

Thermal: temperatures, heat flow rates, convections, internal heat generation, Infinite surface.

Magnetic: magnetic potentials, magnetic flux, magnetic current segments, source current density, and infinite surface.

Electric: electric potentials (voltage), electric current, electric charges, charge densities, infinite surface.

Fluid: velocities, pressures Loads are divided into six categories: DOF constraints, forces (concentrated loads), surface loads, body loads, inertia loads, and coupled field loads.

- A DOF constraint fixes a degree of freedom (DOF) to a known value. Examples of constraints are specified displacements and symmetry boundary conditions in a structural analysis, prescribed temperatures in a thermal analysis, and flux-parallel boundary conditions.
- A force is a concentrated load applied at a node in the model. Examples are forces and moments in a structural analysis, heat flow rates in a thermal analysis, and current segments in a magnetic field analysis.
- A surface load is a distributed load applied over a surface. Examples are pressures in a structural analysis and convections and heat fluxes in a thermal analysis.
- A body load is a volumetric or field load. Examples are temperatures and fluencies in a structural analysis, heat generation rates in a thermal analysis, and current densities in a magnetic field analysis.
- Inertia loads are those attributable to the inertia (mass matrix) of a body, such as gravitational acceleration, angular velocity, and angular acceleration. You use them mainly in a structural analysis.

- Coupled-field loads are simply a special case of one of the above loads, where results from one analysis are used as loads in another analysis. For example, you can apply magnetic forces calculated in a magnetic field analysis as force loads in a structural analysis.

Analysis types: The following types of analysis are possible using ANSYS

- Structural Analysis: Static Analysis, Modal Analysis, Harmonic Analysis, Transient Dynamic Analysis, Spectrum Analysis, Buckling Analysis, Explicit Dynamic Analysis, Fracture mechanics, and Beam Analysis.
- Thermal Analysis: Steady-state thermal analysis, transient thermal analysis.
- CFD (Computational Fluid Dynamics) Analysis: Laminar or turbulent, Thermal or adiabatic, Free surface, Compressible or incompressible, Newtonian or Non-Newtonian, Multiple species transport.
- Several types of Electromagnetic field analysis and Coupled field analysis.

Post processing: Post processing means reviewing the results of an analysis. It is probably the most important step in the analysis, because you are trying to understand how the applied loads effect your design, how good your finite element mesh is, and so on Two postprocessors are available to review your results: POST1, the general Postprocessor, and POST26, the time-history postprocessor. POST1 allows you to review the results over the entire model at specific load steps and sub steps (or at specific time points or frequencies). POST26 allows you to review the variation of a particular result item at specific points in the model with respect to time, frequency, or some other result item. In a transient magnetic analysis, for instance, you can graph the eddy current in a particular element versus time. Or, in a nonlinear structural analysis, you can graph the force at a particular node versus its deflection.

Literature Review

Specifications of Kawasaki Z 750

Displacement: 748.00 cc

Engine type: In-line four,four stroke

Power: 104.20hp (76.1kw) @10500 rpm

Torque: 78 Nm @ 8000 rpm

Compression ratio: 11.3:1

Bore*stroke: 68.4*50.9 mm

Valves per cylinder: 4

Fuel system: Injection

Ignition: digital

Cooling system: liquid

Global journal of researches in engineering

The numerical formulation of the problem is incomplete without prescribing boundary conditions, which correspond to the specific physical model. The specification of mathematically correct boundary conditions that ensure the uniqueness of the solution, while

being compatible with the physics at the boundaries, is not always straightforward. Before arriving at the boundary conditions at various boundaries, we have to first identify the solution/computational domain of the problem [1]. The physical domain and computational domain usually differ. However, the computational domain largely depends on the geometry of physical domain. The computational domain boundary (truncated from the real boundary) along with appropriate boundary conditions should be chosen in such a way that there is negligible change in the results with further increase in its size.

International journal of automobile engineering

Exhaust manifold is one of the most critical components of an IC Engine. The designing of exhaust manifold is a complex procedure and is dependent on many parameters viz. back pressure, exhaust velocity, mechanical efficiency etc. Preference for any of this parameter varies as per designer's needs. Usually fuel economy, emissions and power requirement are three different streams or thought regarding exhaust manifold design. This work comprehensively analyzes **eight different models** of exhaust manifold and concludes the best possible design for least emissions and complete combustion of fuel to ensure least pollution [2].

Fluent 6.0 provides three mesh motion strategies to accommodate the volume deformation associated with in-cylinder motion: spring smoothing, local re-meshing, and dynamic layering. A common feature of each of these modes of deformation is that the cell topology and node movement of interior cells and nodes is determined automatically by the solver. The user only needs to specify motion at the boundaries, the internal structure of the mesh is determined by the solver at each timestep in response to the prescribed boundary motion [3].

Among the cyclic plasticity models, overlay model have been considered as physically motivated, as many real materials or mechanical systems can be thought of as having a similar structure. For example, most of the engineering materials may have a crystalline structure that is made of a distribution of slip-planes or dislocations of different slip strengths [4]. The overlay model has definite advantages over the classical models. The varieties of effects which cannot be described in the classical models can be done in the overlay model, whereby the description remains fairly simple and qualitative. The basic assumption of overlay model is that material is composed of a series of so-called Jenkin's elements connected in parallel, each of which consists of a linear spring with stiffness [5].

The disadvantage is back pressure can be created in the exhaust system. All engines have a maximum allowable engine back pressure specified by the engine manufacturer [6]. Sometimes it will be high so that may cause some effects on the diesel engine. So this project is aimed to reduce the back pressure in the exhaust silencer by using the software computational fluid dynamics (CFD) the back pressure will be reduced.

Geometric Modeling and Analysis

Drawing details

The four-into-one configured exhaust manifold gets its name from its shape and design, four individual ports exiting the cylinder head which all meet up to a single collector as shown in (Figures 4.1- 4.8). Four-into-one configured exhaust manifolds are generally restricted for high-speed application. The IGES files imported in ICEM, model of all inlet and outlet ports length extended for flow develop (Figures 4.1- 4.8). The thickness of the exhaust manifold deleted to obtain fluid domain. The conversion of 2d drafting to 3d model in CATIA software is known as 3d modeling. The 3d model designed is in STP format (Figures 5 and 6).

ANSYS ICEM CFD

ANSYS ICEM CFD meshing software starts with advanced CAD/ geometry readers and repair tools that allow the user to quickly move to a variety of geometry tolerant meshes and produce high-quality volume or surface meshes with minimal effort. Advanced mesh diagnostics, interactive and automated mesh editing, output to a wide variety of CFD and FEA solvers, and multi-physics post-processing tools make the ANSYS ICEM CFD tool a complete meshing solution. The product vision is to provide a variety of flexible tools that can take the model from any geometry to any solver, in one modern and fully scriptable environment. Features include: Intuitive built-in geometry creation and highly automated.

Functionality for the modification and repair of imported CAD data. Ability to work with both NURBs and faceted geometry types like STL or existing meshes. Robust tools for creating all mesh types: hexahedral, tetrahedral, prism, pyramid, quad, tri or bar elements. Highly tolerant of imperfect/over detailed CAD data containing sliver surfaces, gaps, holes and overlaps (patch independence). Supports Smoothing, coarsening, refinement, and element type conversion, linear or quadratic element. More than 100 CFD/CAE solver interface

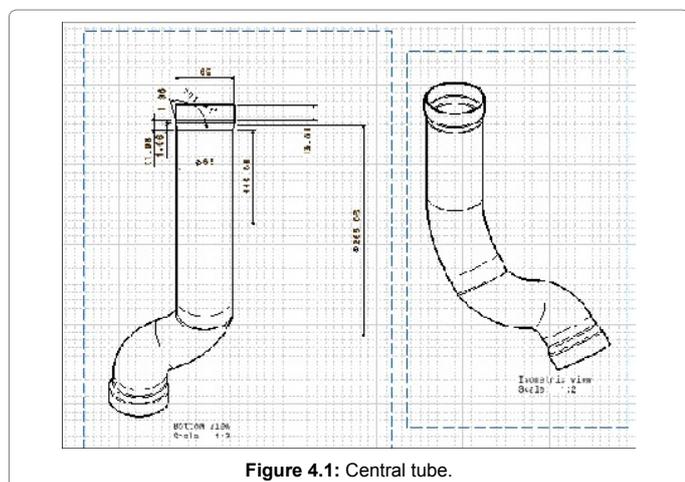


Figure 4.1: Central tube.

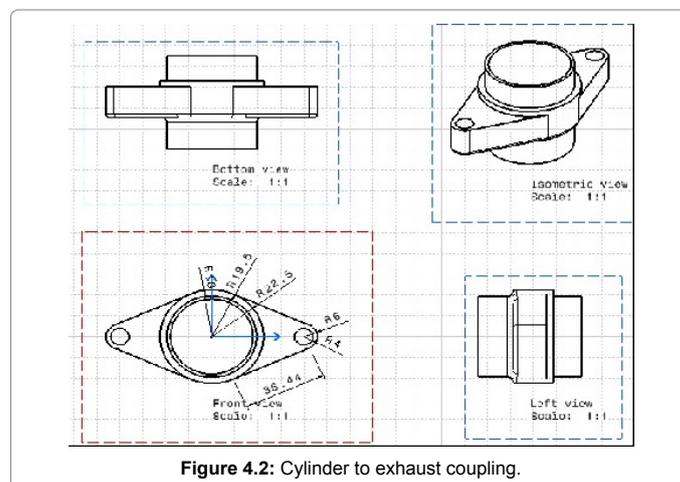


Figure 4.2: Cylinder to exhaust coupling.

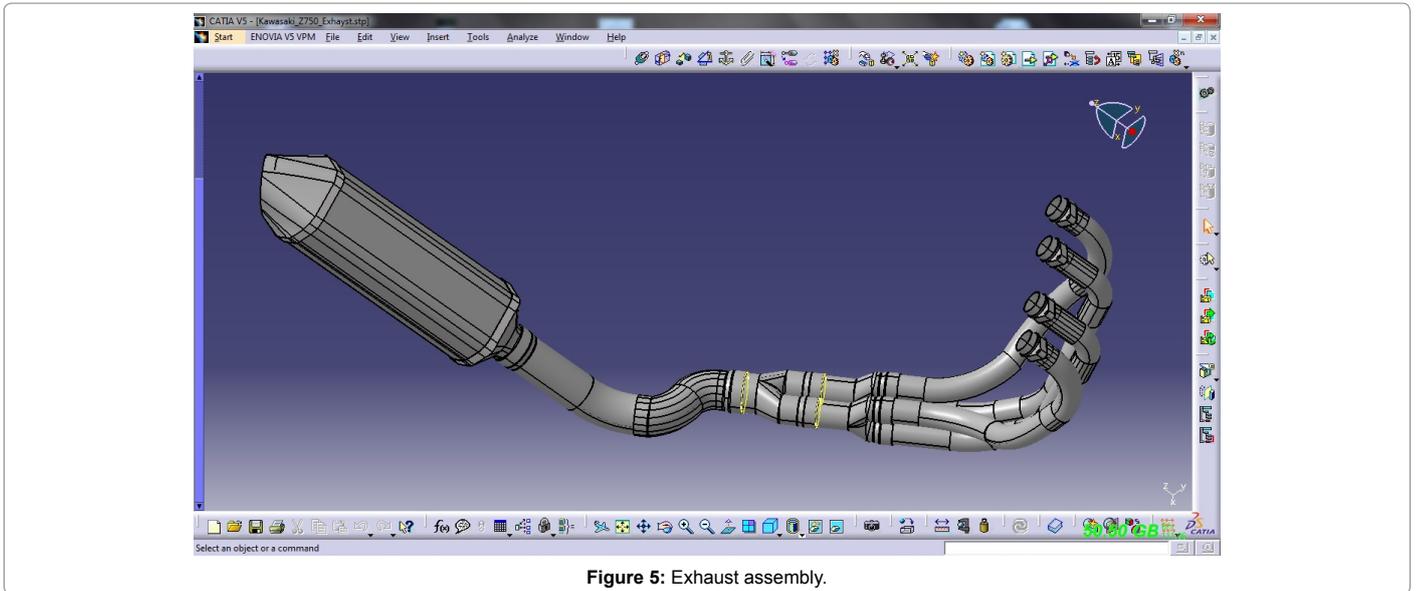


Figure 5: Exhaust assembly.

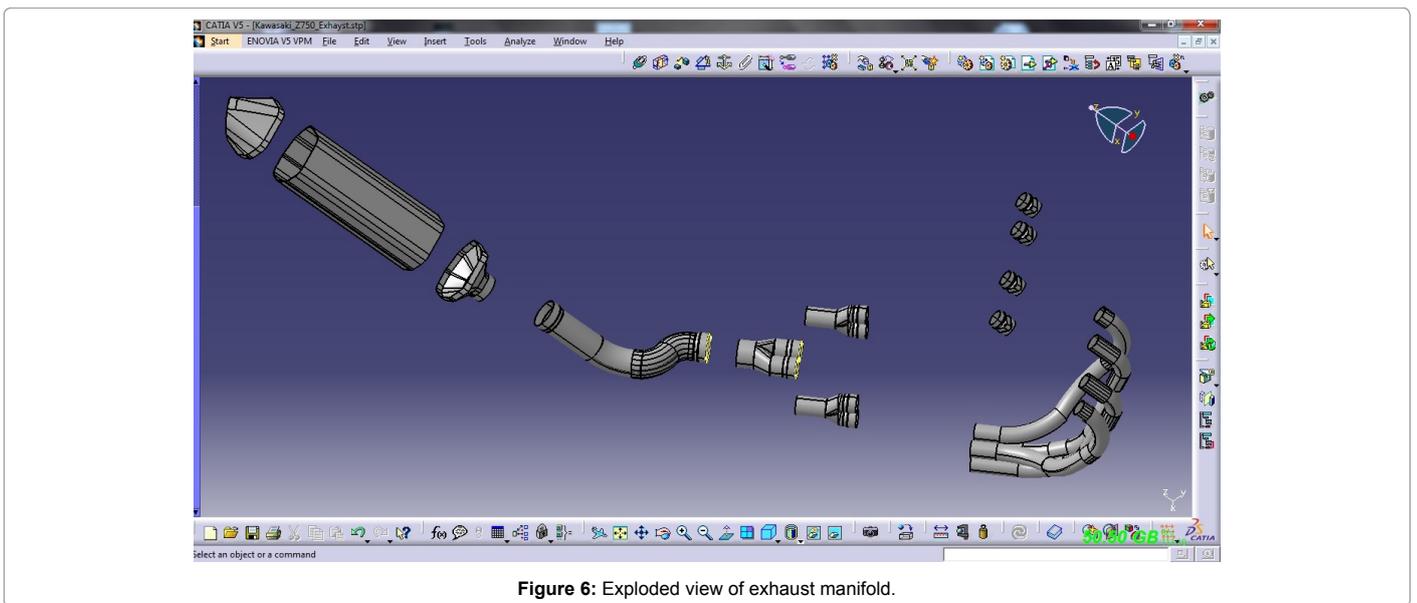


Figure 6: Exploded view of exhaust manifold.

quadrilaterals and triangular cells and for 3D case in hexahedral and tetrahedral cells shown in (Figure 8).

Mesh classification: Mesh can be classified on the bases of connectivity of the elements as follows:

Structured mesh: Structured meshes are widely used in many real CFD applications. In structure mesh all interior vertices are topologically alike. This can be generated algebraically by interpolation from boundaries for example transfinite interpolation. In structured meshes points are laid in the region of interest in a regular pattern; these points form quadrilateral cells in 2D and hexahedral in 3D. A structured mesh generated in 2D region is shown in (Figure 9).

There are some significant advantages associated with structured meshes due to which they are preferred over other grid methods in many applications. These advantages are listed below:

- From the solver point of view, they have simplicity and easy data access.

- From users point of view, they possess high degree of control. For example user can easily control the refinement level in a particular region by placing more dense points on the boundaries of that region.
- Memory requirement to simulate flow in a structured domain is less as compared to other methods.
- Element could easily be flow aligned in structured meshes which offer more accuracy of the results.
- Examination of flow field in post processing is easier with structured mesh.

Unstructured mesh

Unstructured meshes are used in many FEA and CFD applications which have irregular shape of object or region of interest. Unlike structured mesh, they can have elements of arbitrary topology in a confined region. In unstructured meshes points are laid in the region of

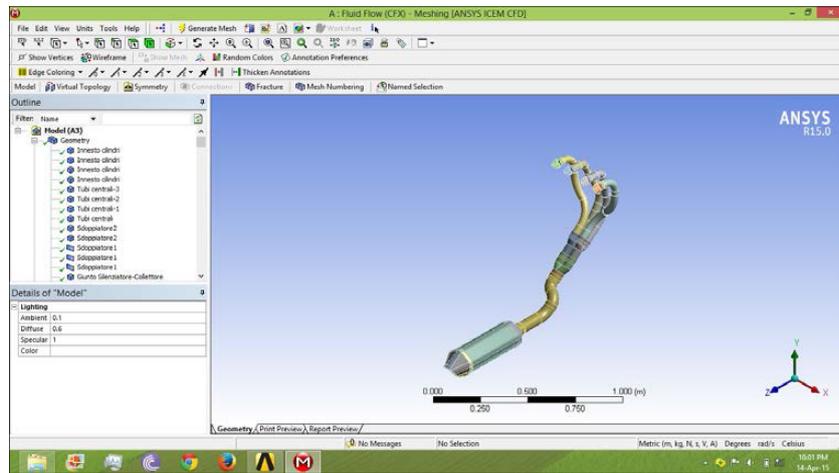


Figure 7: Imported file from CATIA to CFX.

interest in a random order forming triangular subdivision of the region. Unstructured mesh in its most basic form contains triangles in 2D and tetrahedral cells in 3D, but may be made of hexahedral or elements of any shape in general. Unstructured meshes are more easily generated and adapted but require a much more complex solution data structure. In generation of unstructured mesh user control is limited as interior nodes are generated automatically without any interaction of user. An unstructured mesh generated around airfoil in 2D region is shown in (Figure 10).

The advantages of unstructured grid methods are given below:

- They have the ability to adopt better around the complex geometries.
- They can be generated with less efforts and interactions.
- They can be easily adopted by the applications which require automation of the mesh generation process .

Hybrid mesh: General configurations can conceivably be treated with either type of mesh, and hybrid combinations are also possible, using individual structured meshes near boundaries, with the sub-regions being connected by an unstructured mesh. Thus, hybrid mesh is a positive combination of both structured and unstructured mesh and possesses the advantages of both. A hybrid mesh generated around airfoil in 2D region is shown in (Figure 11).

Storage requirement per point for a flow simulation on an unstructured grid is about four times that of structured grid. Even if the unstructured grid is coarser in the far-field the storage of the unstructured grid, with its greater total number of points would typically be 40 times greater than that for a structured grid. Thus, for hybrid grid there is a clear indication that the extent of un-structured grid employed should be as minimum as possible.

Grid generation approach: Grid generation is an important aspect as a good grid can accelerate the convergence of the solution, while a poor grid can lead to divergent iteration history. Grid generation process proceeds from first defining the boundary geometry and distribution of the points on the curves that form the edges of boundary sections. A surface grid is generated on the boundary surface. Finally a volume grid is generated in case of 3D domain. This project utilizes the power of commercial software ANSYS ICEM CFD to automate the process of mesh generation.

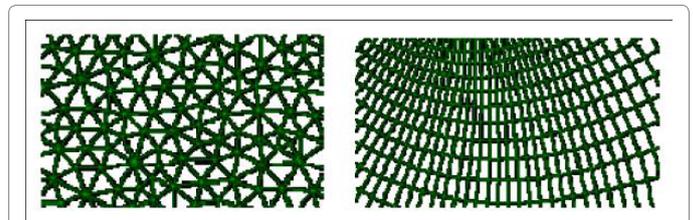


Figure 8: Example of 2D analysis domain.

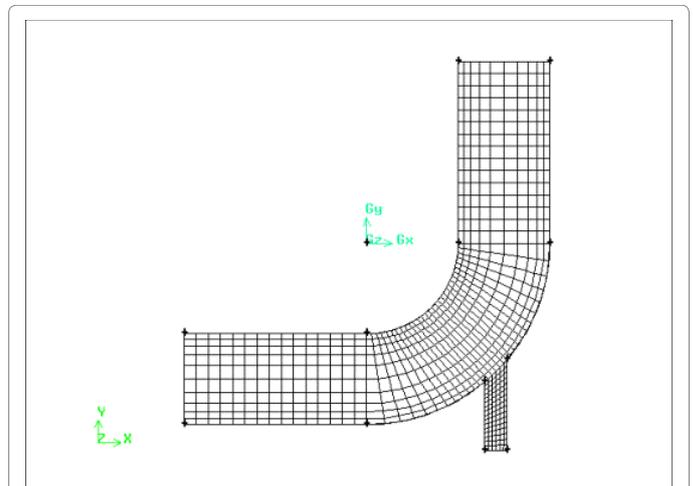
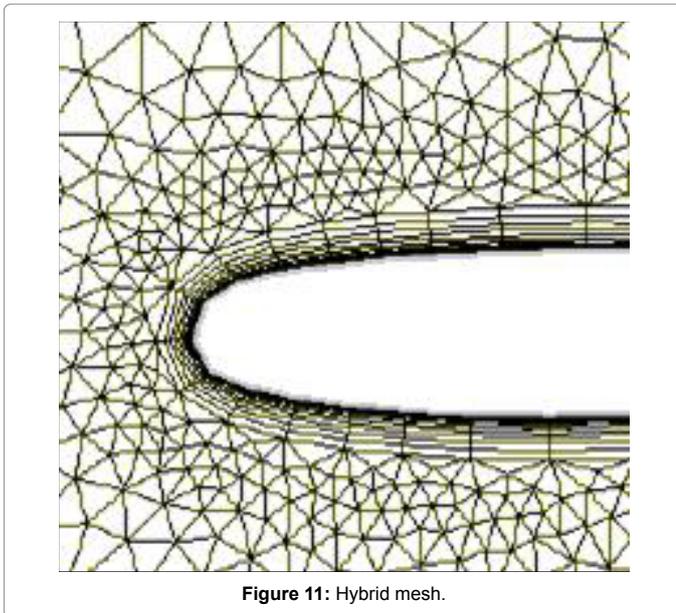
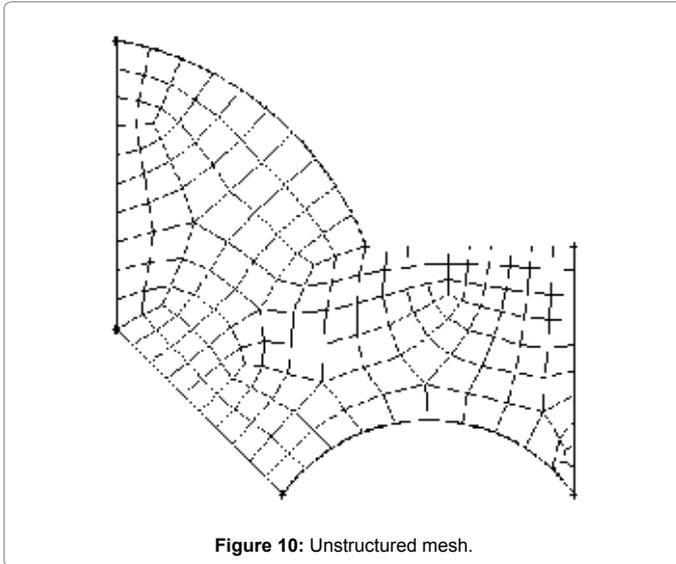


Figure 9: Structured mesh.

Grid generation for the exhaust manifold: The exhaust manifold of IGES files imported in ICEM CFD software, the geometry cleanup has been done, the fluid domain extracted from geometry and different parts were created. Unstructured mesh generated for the Exhaust manifolds. The model-1 contains 417594 nodes and 398080 hexas, 38052 quads cells. The model-2 contains 413618 nodes and 394294 hexas, 37692 quads cells. First layer mesh size of 0.01 mm with spacing factor 1.2. After generating the mesh quality has been checked. It has minimum internal angle is 25 degree and determinant is 0.4. The skewness and nonnegative volume has been checked before importing the mesh in solver, the boundary conditions are defined in the ICEM



CFD itself Fluent as a solver takes mesh file in .msh format. So the generated mesh with declared boundary condition is saved according to the solver input format (Figure 12).

Problem solution using fluent approach

Computational Fluid Dynamics (CFD) is a computational technology that has become an indispensable tool to study the dynamics of fluid and analyzing the fluid flow, pressure distribution, heat and mass transfer etc., in a region of interest. It is a powerful numerical tool used widely to simulate many processes in automobile industries. Using CFD software a 'virtual prototype' of the system or device can be made and applied with the real world physical conditions and then prediction about the performance of that design can be obtained. Recent progression in computing efficacy coupled with reduced costs of CFD software packages has advanced CFD as a viable technique to provide effective and efficient design solutions.

Fluent: Fluent is commercial CFD solver software by ANSYS.

For over twenty years, it has been a leader in the CFD software for simulating fluid flow, heat and mass transfer, and a host of related phenomena involving turbulence, reactions, and multiphase flow. Fluent is state-of-the-art CFD simulation software for modeling fluid flow and heat transfer in complex geometries. Fluent provides complete mesh flexibility, including the ability to solve flow problems using unstructured meshes that can be generated about complex geometries with relative ease. Fluent is ideally suited for incompressible and compressible fluid-flow simulations for complex geometries. Fluent code solves the equation of conservation of mass, momentum, energy and other relevant fluid variables. Fluent supports different mesh/grid types like 2D-triangular/quadrilateral, 3D-tetrahedral/hexahedral/wedge/polyhedral, and mixed (hybrid) meshes. Also FLUENT allows refining (or) coarsening grid based on the flow solution. Fluent can import mesh generated by a number of meshing software including Gambit, T Grid, Gridgen, Ansys, Ideas, ICEM, Hypermesh etc. It supports journaling which can be used in creating script files for automation of solver process. Batch and serial mode solving process is provided. Once the mesh is imported to FLUENT, governing equations (in integral form) for the conservation of mass, momentum, energy and any other relevant variables are applied to each discrete control volume and used to construct a set of non-linear algebraic equations for the discrete dependent variables, Figure 12 shows each iterative pressure solver. In this thesis work all simulations are performed by pressure based.

Computational fluid dynamics-working on CFD: Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using computational methods. Computational Fluid Dynamics (CFD) is a discipline that solves a set of equations governing the fluid flow over any geometrical configuration. The particular form chosen depends on the intended application. The equation can represent steady or unsteady, compressible or incompressible, and inviscid or viscous flows, including non-ideal and reacting fluid behavior. The state of the art is characterized by the complexity of the geometry, the flow physics, and the computer time required for obtaining a solution. The Physical aspects of any fluid flow are governed by three fundamental principles:

Mass is conserved; Momentum and Energy is conserved: These fundamental principles can be expressed in terms of mathematical equations, which in their most general form are usually partial differential equations. The role of CFD in engineering predictions has become so strong that today it may be viewed as a new third dimension of fluid dynamics, the other two dimensions being pure experiment and pure theory.

The standard k- ε model: The simplest "complete models" of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard $\kappa - \epsilon$ model falls within this class of turbulence model and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semiempirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism.

Steps in CFD: CFD is structured around the numerical algorithms that can tackle fluid flow problems. In order to provide easy access to their solving power these include sophisticated user interfaces to input

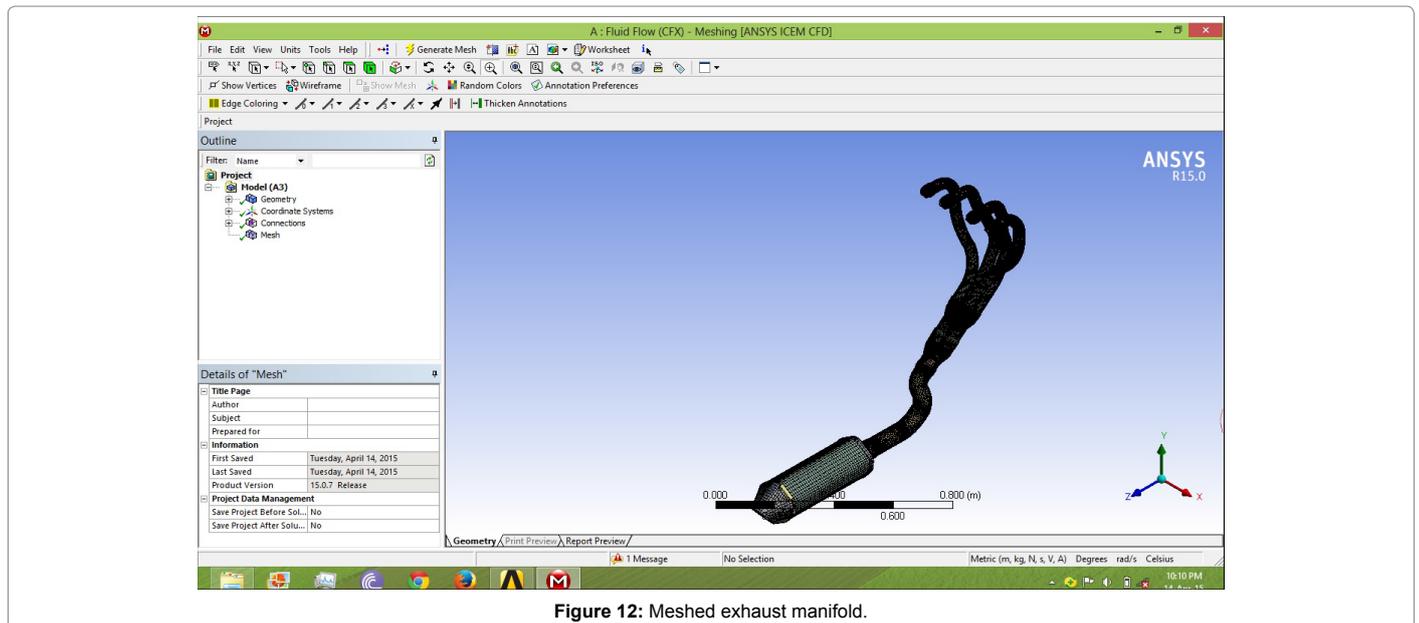


Figure 12: Meshed exhaust manifold.

problem parameters and to examine the results. Hence all contain three main elements.

- a) Pre-processor
- b) Solver and
- c) Post-processor

Pre-processor: Pre-processing consists of the input of a flow problem to a CFD problem by means of an operator-friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The user activities at the pre-processing stage involve:

- Definition of the geometry of the region of interest: the computational domain.
- Grid generation-the sub-division of the domain into a number of smaller subdomains: a grid (or mesh) of cells (or control volumes or elements).
- Selection of the physical and chemical phenomena that need to be modeled.
- Definition of fluid properties.
- Specification of appropriate boundary conditions at cells which coincides with or touch the domain boundary.

The solution to a flow problem (velocity, pressure, temperature etc.) is defined at nodes inside each cell. The accuracy of a CFD solution is governed by the number of cells in the grid. In general, larger the number of cells better the solution accuracy. Both the accuracy of a solution and its cost in terms of necessary computer hardware and calculation time are dependent on the fineness of the grid. Optimal meshes are often nonuniform finer in areas where large variations occur from point to point and coarser in regions with relatively little changes. At present it is still up to the skills of the CFD user to design a grid that is a suitable compromise between desired accuracy and solution cost. Over 70% of the time spent in industry on a CFD project is devoted to the definition of the domain geometry and grid generation.

Input Parameters used in CFD are:

- Temperature: 1500 K
- Pressure: 2 bar
- Mass flow rate: 0.01 kg/s

Output parameters used are:

- Temperature: 300 K
- Pressure: 25 kpa

Considered steady state condition:

- Incompressible ideal gas as a working fluid.
- The flow was considered to be turbulent. 10% turbulent intensity was taken and K-ε model.

Boundary conditions are:

Inlets: At an inlet, fluid enters the domain and, therefore, its fluid velocity or pressure or the mass flow rate may be known. Also, the fluid may have certain characteristics, such as the turbulence characterizes which needs to be specified.

Inlet: mass flow inlet.

Outlet: pressure outlet.

No-slip impermeable wall: Wall boundary conditions are used to bound fluid and solid regions. In viscous flows, the no-slip boundary condition is enforced at walls by default, but you can specify a tangential velocity component in terms of the translational or rotational motion of the wall boundary, or model a "slip" wall by specifying shear.

(You can also model a slip wall with zero shears using the symmetry boundary type, but using a symmetry boundary will apply symmetry conditions for *all* equations.

Solver: There are three distinct streams of numerical solution techniques: finite difference, finite element and spectral methods. In outline the numerical methods that form the basis of the solver perform the following steps:

- Approximation of the unknown flow variables by means of simple functions.
- Discretization by substitution of the approximations into the governing flow equations and subsequent mathematical manipulations.
- Solution of the algebraic equations.

The main differences between the three separate streams are associated with the way in which the flow variables are approximated and with the discretization processes.

Post-processor: Post-processor is versatile data visualization tools. These include:

- Domain geometry and grid display (Figure 13)
- Vector plots (Figure 14)

- Vectors (Figure 15)
- Line and shaded contour plots (Figure 16)
- 2D and 3D surface plots
- Particle tracking
- View manipulation (translation, rotation, scaling etc.)
- Color postscript output

Results and Discussions

Fluent

Analysis is carried out on exhaust manifold at different engine speeds varying with flow rate, so that following results are obtained from (CFD) FLUENT analysis for further discussions.

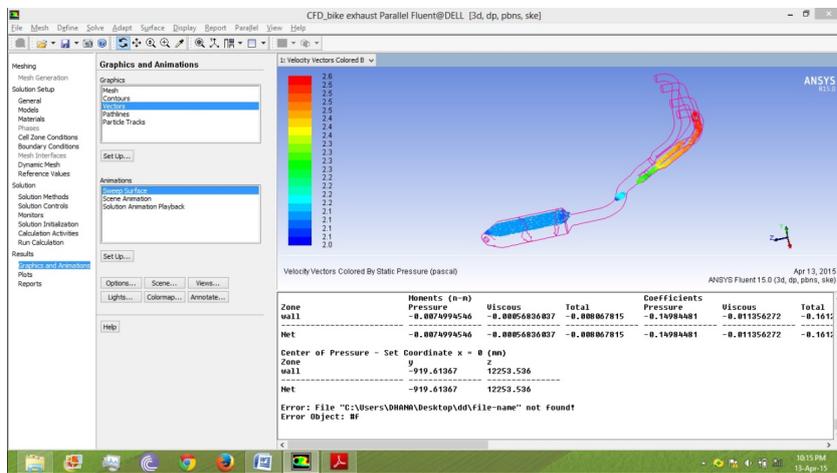


Figure 13: Domain geometry.

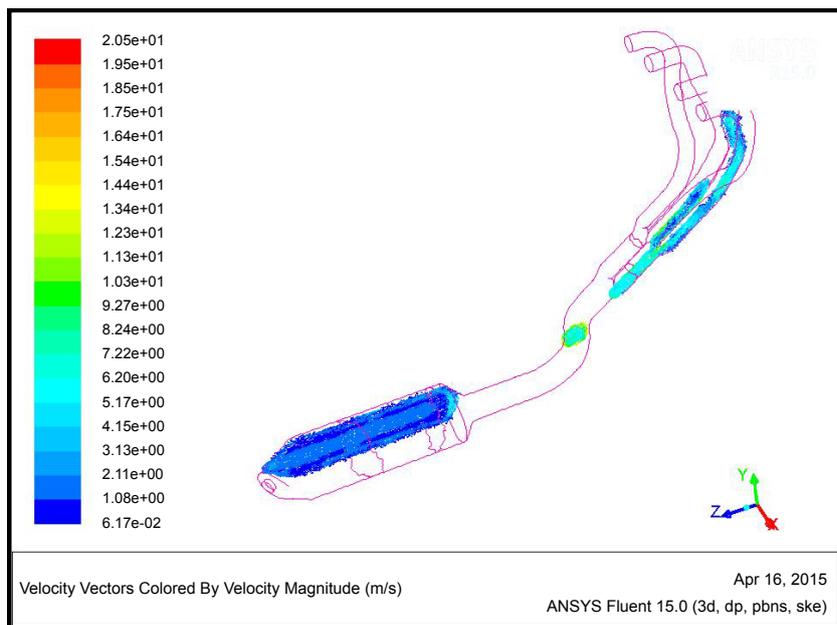


Figure 14: Vector plot.

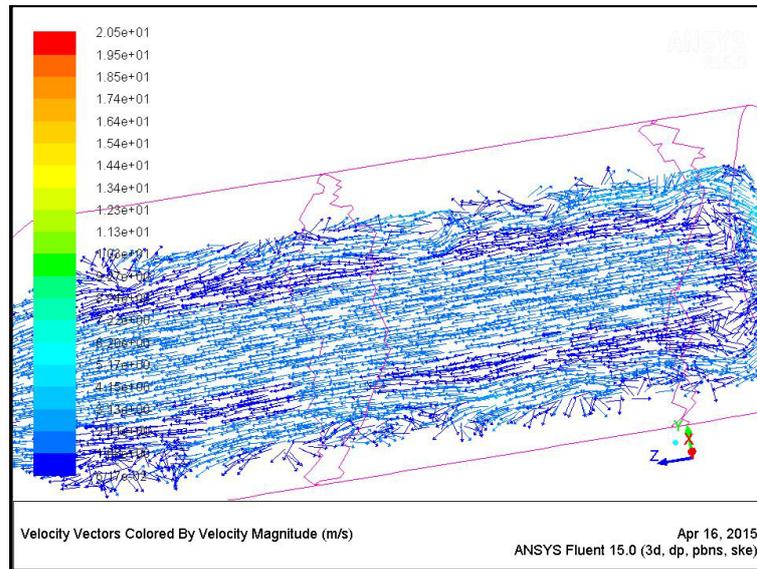


Figure 15: Vectors.

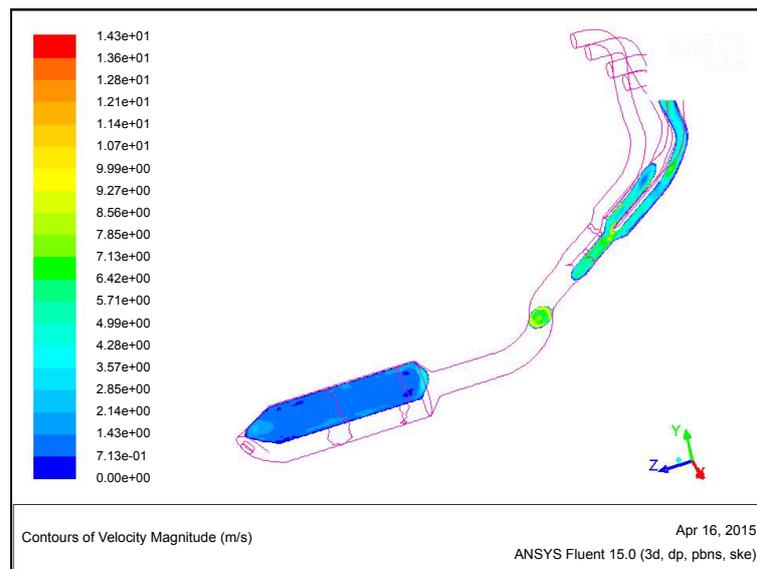


Figure 16: Contour plot.

Speed: 12000 rpm, flow rate 3.143 m³/s

1. Turbulent kinetic energy: The maximum turbulent kinetic energy obtained at mean position viz., $4 \times 10^7 \text{ m}^2/\text{s}^2$ (Figure 17).
2. Static pressure: Static pressure is maximum at 0.01m away from the mean position of exhaust manifold viz., 5×10^6 Pascal (Figure 18).
3. Velocity magnitude: Velocity magnitude is maximum at mean position of the exhaust manifold is $3 \times 10^4 \text{ m/s}$ (Figure 19).
4. Wall shear: The above diagram shows that maximum and minimum wall shear along the walls of exhaust manifold obtained are $2 \times 10^7 \text{ Pa}$ and 3.77 Pa respectively (Figure 20).

Speed: 13000 rpm, flow rate: 4.516 m³/s

1. Turbulent kinetic energy: Turbulent kinetic energy is maximum at distance 0.05m away from the mean position and is $5.5 \times 10^6 \text{ m}^2/\text{s}^2$ (Figure 21).
2. Static pressure: Static pressure distribution of exhaust manifold is maximum at mean position and is $5 \times 10^6 \text{ Pa}$ (Figure 22).
3. Velocity magnitude: Velocity magnitude is maximum at mean position of exhaust manifold viz., $3.5 \times 10^4 \text{ m/s}$ (Figure 23).
4. Wall shear: Wall shear at walls of the exhaust manifold. Wall shear is maximum at tail end of the exhaust manifold viz., $1.72 \times 10^5 \text{ Pa}$.

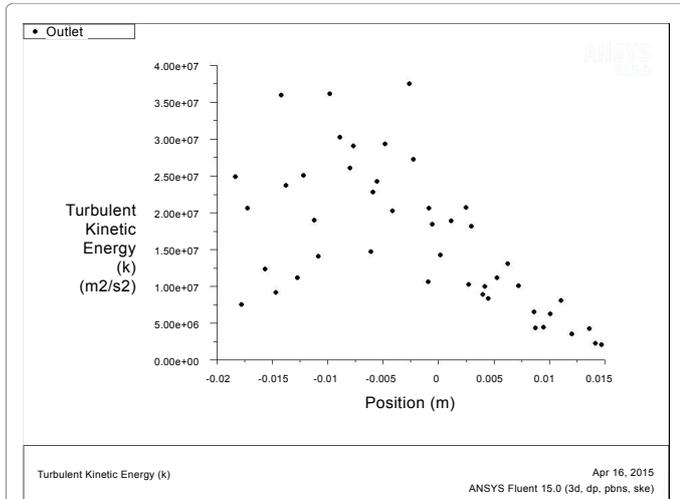


Figure 17: Turbulent kinetic energy at different positions of exhaust manifold.

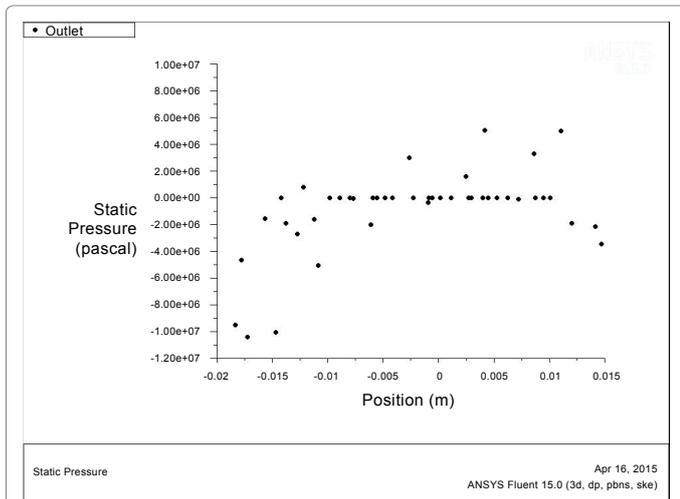
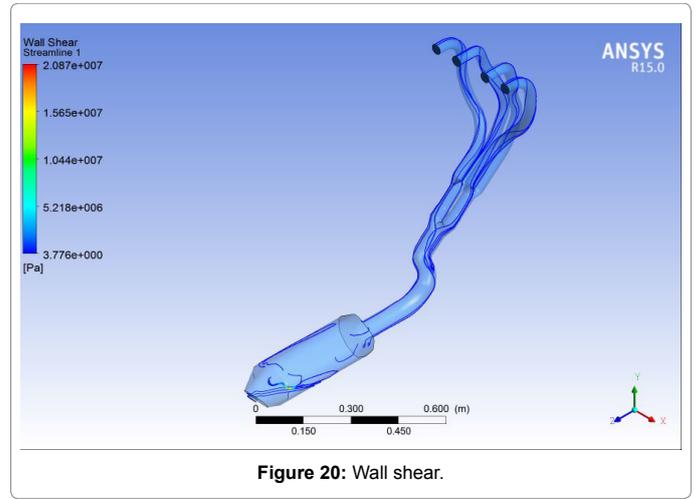


Figure 18: Static pressure at different positions of exhaust manifold.

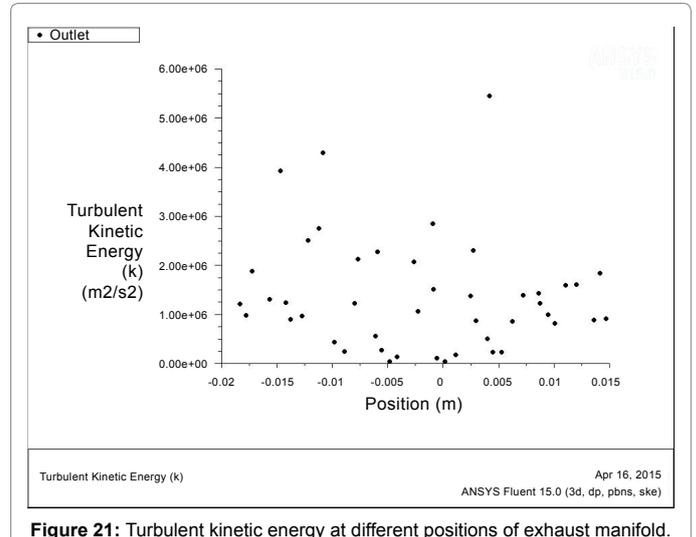


Figure 21: Turbulent kinetic energy at different positions of exhaust manifold.

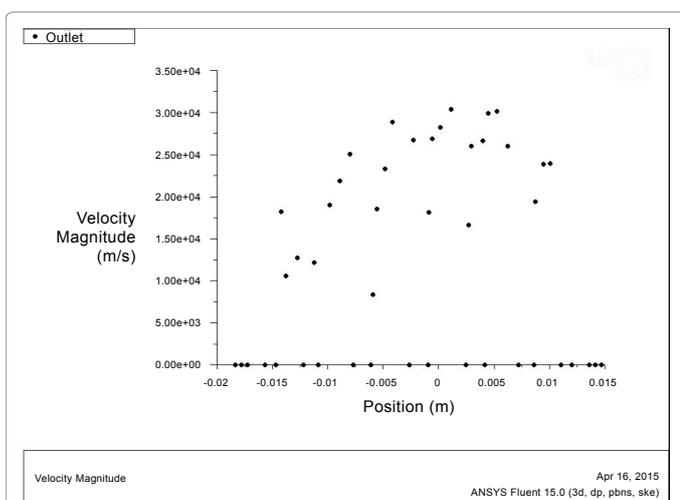


Figure 19: Velocity at different positions of exhaust manifold.

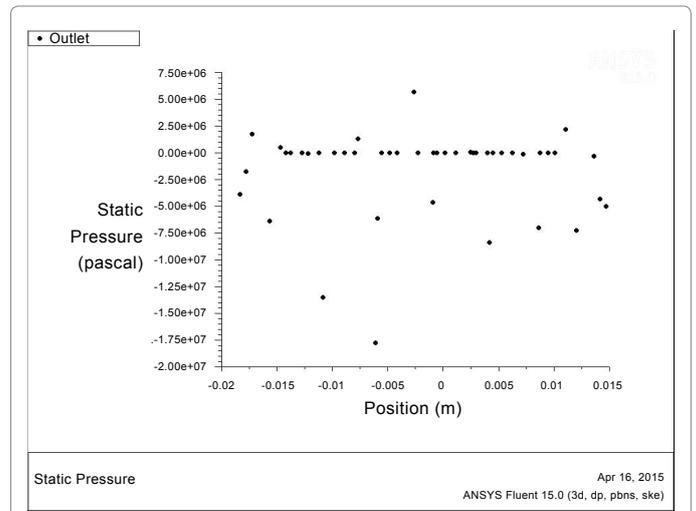


Figure 22: Static pressure at different positions of exhaust manifold.

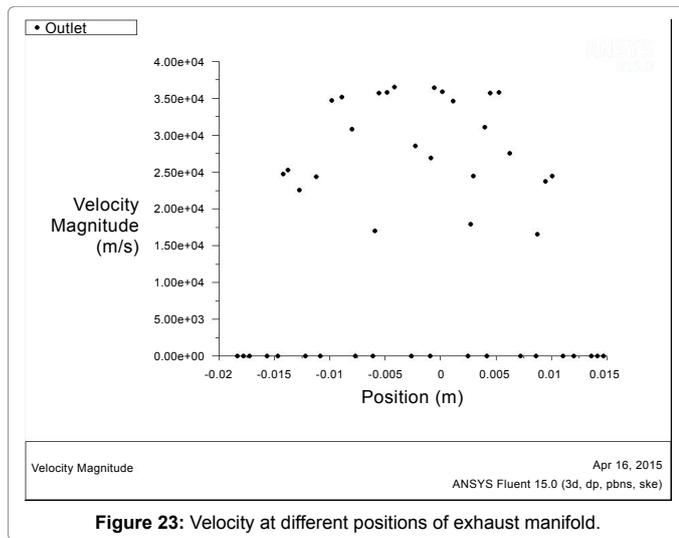


Figure 23: Velocity at different positions of exhaust manifold.

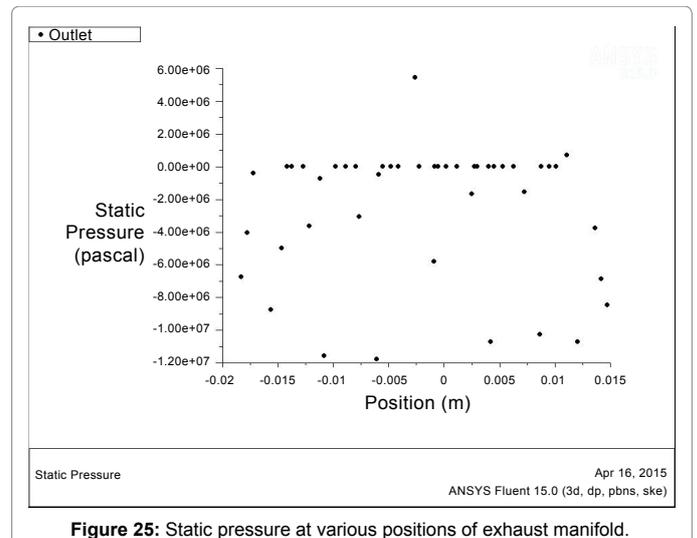


Figure 25: Static pressure at various positions of exhaust manifold.

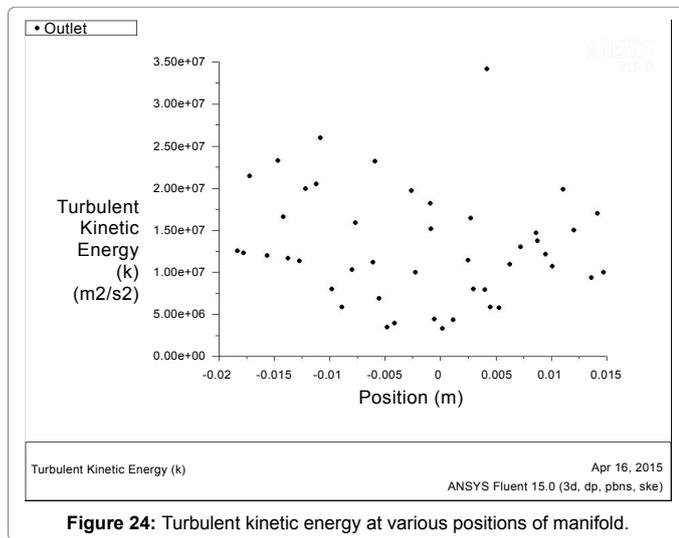


Figure 24: Turbulent kinetic energy at various positions of manifold.

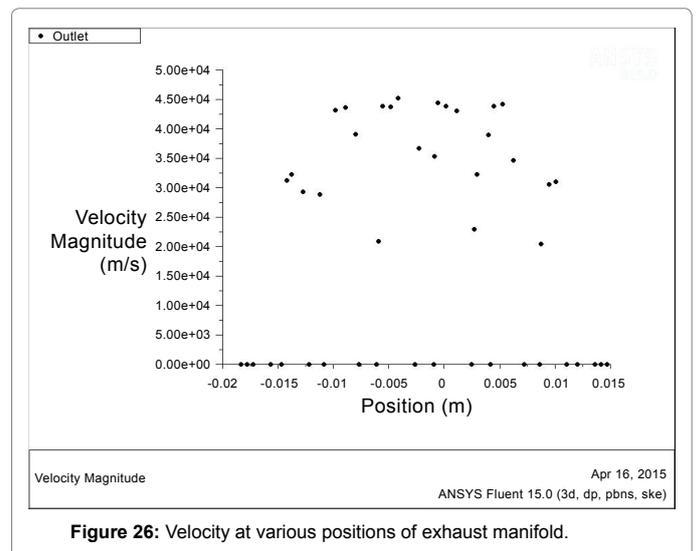


Figure 26: Velocity at various positions of exhaust manifold.

Speed: 1400 rpm, flow rate: 4.18 m³/s

1. Turbulent kinetic energy: Turbulent kinetic energy is maximum at distance of 0.005m away from the mean position of exhaust manifold is $3.5 \times 10^7 \text{ m}^2/\text{s}^2$ (Figure 24).
2. Static pressure: Static pressure distribution is maximum at mean position of exhaust manifold viz., $6 \times 10^6 \text{ Pa}$ (Figure 25).
3. Velocity magnitude: Velocity magnitude is maximum at mean position of exhaust manifold and is $4.5 \times 10^4 \text{ m/s}$ (Figure 26).
4. Wall shear.

Speed: 1500 rpm, flow rate: 5.627 m³/s

1. Turbulent kinetic energy: Turbulent kinetic energy is maximum at distance 0.005m away from the mean position of exhaust manifold and is $4 \times 10^7 \text{ m}^2/\text{s}^2$ (Figure 27).
2. Static pressure: Static pressure distribution is maximum at mean position of exhaust manifold and is $7.5 \times 10^6 \text{ Pa}$ (Figure 28).
3. Velocity magnitude: Magnitude of velocity distribution is

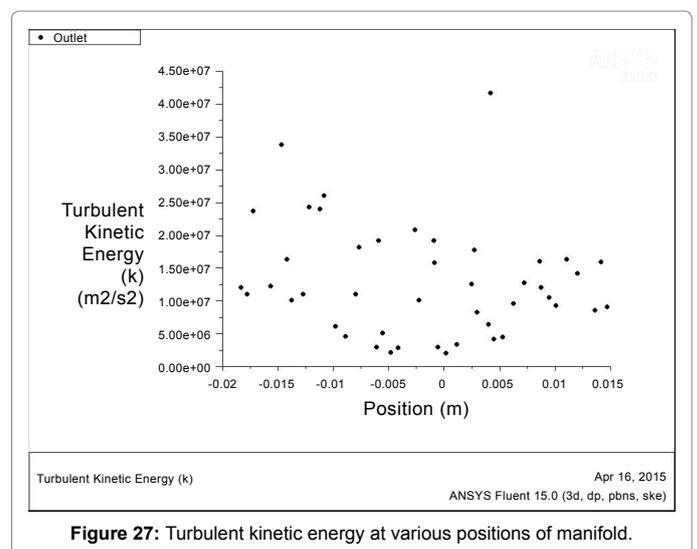


Figure 27: Turbulent kinetic energy at various positions of manifold.

maximum at mean position of exhaust manifold and is 5×10^4 m/s (Figure 29).

4. Wall shear (Table 1 and Figure 30).

The Figure 30 shows that as the speed of the engine increases Turbulent kinetic energy varies, at speed 1300 rpm its reaches its threshold limit then the value of turbulence KE increases further increase of engine speed.

The Figure 31 shows that the static pressure of exhaust gases coming from the cylinder is varies with engine speed.

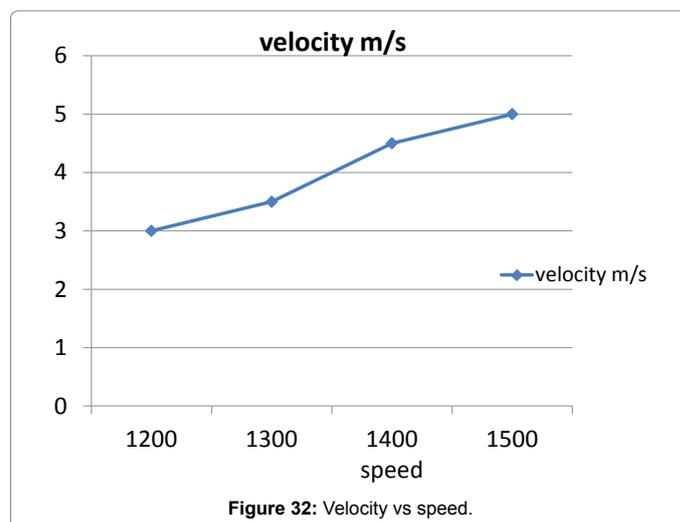
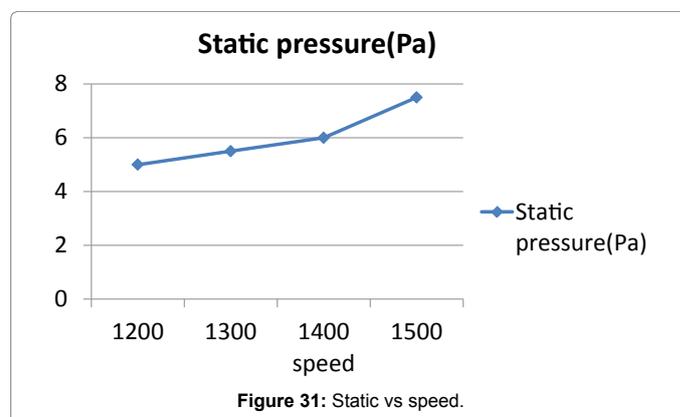
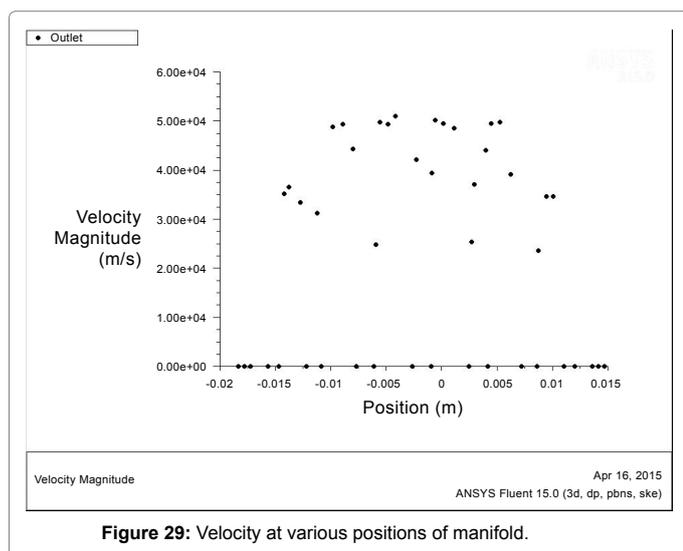
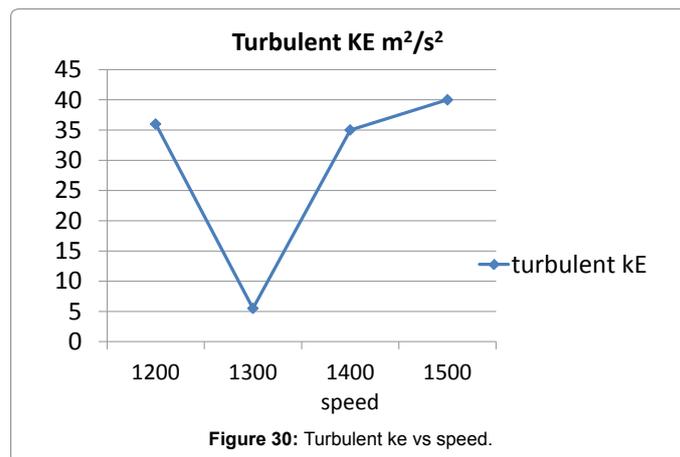
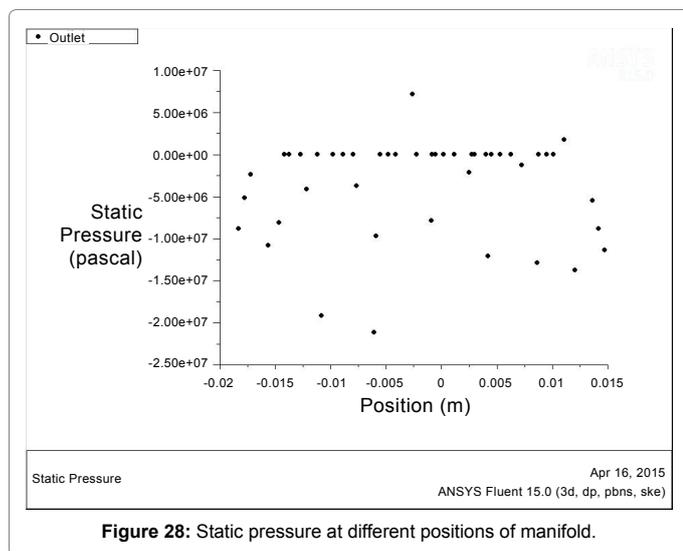
The Figure 32 shows that velocity of the exhaust emission gases increases with the increase of engine speeds.

The Figure 33 shows that the wall shears in the exhaust manifold fluctuates with the increase of speeds of engine.

Conclusions

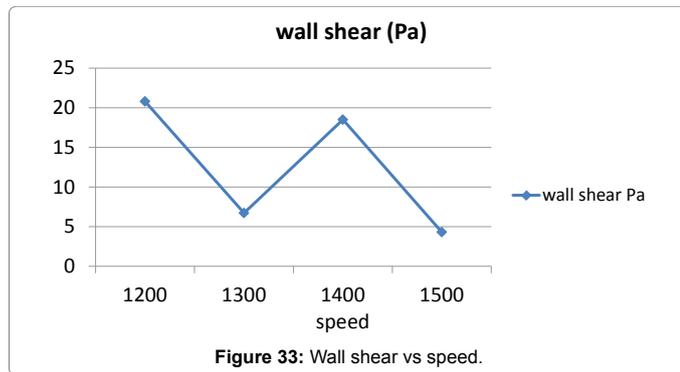
It has been presented that the research methodology depending on three-dimensional model of airflow in the outlet systems with the application of standard numeric methods.

The model may be the basis for performing changes in geometry of outlet system concerning minimizing the flow losses and shaping the



| Speed (rpm) | Flow rate (m³/s) | Static pressure (Pa)*10 ⁶ | Turbulent KE (m²/s²)*10 ⁷ | Velocity (m/s)*10 ⁴ | Wall shear (Pa)*10 ⁶ |
|-------------|------------------|--------------------------------------|--------------------------------------|--------------------------------|---------------------------------|
| 1200 | 3.143 | 5 | 3.6 | 3 | 20.8 |
| 1300 | 4.516 | 5.5 | 5.5 | 3.5 | 6.72 |
| 1400 | 4.981 | 6 | 3.5 | 4 | 18.5 |
| 1500 | 5.627 | 7.5 | 4 | 5 | 4.3 |

Table 1: Wall shear.



field of velocity. The model enables of calculations of outlet system of internal combustion engines on the stage of its construction.

The conclusions from the analysis of velocity and air distribution:

- The dominant resistance has been affirmed in the space of direction change where the highest pressure differences appear.
- This has been affirmed patchy distribution of pressure and velocity of fumes in the cross-section area of outlet ducting, which indicates the deficiencies in development of manifold.

- The increase of flow velocity appears in the field of manifold, which is the result of decreasing the field of section space.

The conducted considerations suggest the necessity of changes in geometry of manifold.

References

1. Pandey KM, Bidesh R (2012) CFD Analysis of Intake Valve for Port Petrol Injection SI Engine. *Global Journal of Researches in Engineering* 12: 1-9
2. Sumesh VK, Pravin K, Rajagopal K (2013) CFD Analysis of Exhaust Manifold of Multi-Cylinder Si Engine Todetermine Optimal Geometry for Reducing Emissions. *International Journal of Automobile Engineering Research and Development* 3: 45-56.
3. Colucci, P.J, Lee D, Lim CK, Goldin G (2002) In-cylinder Engine Modeling Developments at Fluent. *Twelfth International Multidimensional, Lebanon, NH.*
4. Rathnaraj DJ (2012) Thermomechanical Fatigue Analysis of Stainless Steel Exhaust Manifolds. *IRACST Engineering Science and Technology: An International Journal* 2: 2.
5. Rathnaraj JD, Bose RJ, Kumar MN (2006) Simulation and Experimental Investigation on Variation of Swirl with Valve Lift in DI Diesel Engine using CFD. *Journal on Future Engineering and Technology* 1: 75-87
6. Tamizharasan T, Barnabas JK, Pakkirisamy V (2012) Multi-Response Optimization of Hard Milling Process: RSM Coupled With Grey Relational Analysis. *International Journal of Engineering and Technology (IJET)* 226: 1159-1173.