ADAMA SCIENCE AND TECHNOLOGY UNIVERSITY
SCHOOL OF MECHANICAL, CHEMICAL AND MATERIALS ENGINEERING
MECHANICAL SYSTEMS AND VEHICLE ENGINEERING PROGRAM

Computational Fluid Dynamics (CFD) Analysis of Three Wheeler Vehicle Muffler

A thesis submitted in partial fulfillment of the requirements for the award of the degree of Master of Science in Automotive Engineering

By

Hunde Solomon Rufi

Advisor: Dr. Addisu Bekele

June 21, 2016

Adama, Ethiopia

CANDIDATE'S DECLARATION

I hereby declare that the work which is being presented in the thesis entitled "**Computational Fluid Dynamics (CFD) Analysis of Three Wheeler Vehicle Muffler**" in partial fulfillment of the requirements for the award of the degree of Master of Science in Automotive Engineering is an authentic record of my own work carried out from November, 2015 to June, 2016 under the supervision of Dr. Addisu Bekele, Assistant professor of mechanical engineering, Adama Science and Technology University, Ethiopia.

The matter embodied in this thesis has not been submitted by me for the award of any other degree or diploma. All relevant resources of information used in this thesis have been duly acknowledged.

Name Signature Date

Hunde Solomon ____________ ____________

Student

This is to certify that the above statement made by the candidate is correct to the best of my knowledge and belief. This thesis has been submitted for examination with my approval.

Name Signature Date

________________________ ______________ ____________

Advisor

ADAMA SCIENCE AND TECHNOLOGY UNIVERSITY

SCHOOL OF MECHANICAL, CHEMICAL AND MATERIALS ENGINEERING

MECHANICAL SYSTEMS AND VEHICLE ENGINEERING PROGRAM

CFD Analysis of Three Wheeler Vehicle Muffler

By

Hunde Solomon

Approved by board of Examiners

__________________________________ ____________

Chairman, Department graduate committee

__________________________________ ____________

Advisor

__________________________________ ____________

Internal Examiner

__________________________________ ____________

External Examiner

ACKNOWLEDGEMENT

All glory to the ALMIGHTY GOD for His grace on me, who guided and strengthened me, even at times when I felt; I were not capable of doing it myself.

I would like to thank my advisor Dr. Addisu Bekele, Assistant professor of mechanical engineering, for allowing me the opportunity to perform this research, and for his guidance, emphasis on critical thinking, and intellectual input.

It is my great pleasure to acknowledge Mr. Getaneh Abdisa, for his invaluable support and contribution in enhancing the quality of my work.

I would like to thank my fiancé Simbo and my family for their usual support.

Finally, I would like to thank all my friends and classmates who have either directly or indirectly supported me during the hard time of the thesis work.

ABSTRACT

The exhaust noise pollution from Automobiles has become one of the major problems of environment. Muffler is being employed to reduce such noise coming from vehicles exhaust system. Day to day the noise pollution from vehicles is increasing due to ever increasing number of vehicles on the roads. So there exists a need to improve the performance of mufflers in attenuation of the noise emitted from vehicles exhaust system. In this thesis the functionality of muffler being used in three wheelers vehicles (Bajaj) has been analyzed using Computational Fluid Dynamics (CFD) method. Similarly the pressure distribution in the muffler is analyzed. Further modification has been done on the existing muffler and detail analysis is carried out using ANSYS software in order to improve the efficiency of the muffler. The simulation results also verify that the assembly performance of the muffler modified is better than the base muffler.

Keywords: *CFD, Muffler, Three Wheelers, Exhaust Noise*

Table of Contents **Page**

LIST OF FIGURES PAGE

LIST OF TABLES **PAGE**

ACRONYMS AND ABBREVIATIONS

AF	Air Fuel ratio
ASA	Acoustical Society of America
BDC	Bottom Dead Center
CAD	Computer Aided Design
CFD	Computational Fluid Dynamics
EDS	Exhaust Diffuser System
EPFM	Exterior penalty function method
FEM	Finite Element Method
FSAE	Formula Society Automotive Engineer
FDM	Feasible Direction Method
FCB	Functional Cargo Block
IPFM	Interior Penalty Function Method
LFN	Low frequency noise
LHV	Lower Heating Value
MMM	Mode Matching Method
REF	Reflection Coefficient
TDC	Top Dead Center
TMM	Transfer Matrix Method
u, v and w	velocity components in the x, y and z directions
ρ	density
μ	dynamic viscosity.
a_1	area of valve port
v_1	velocity of gas through the port
C_i	average flow coefficient
a_p	area of the piston
v_{mp}	mean piston velocity
a_{ip}	area of muffler inlet pipe

v_{ip}	velocity of gas at the muffler inlet pipe
C_p	specific heat at constant pressure
Φ	dissipation function
T	temperature
P	pressure
Cv	specific heat at constant volume
γ	ratio of specific heats
R	universal gas constant
V_s	swept volume,
m_a	mass of air
m_f	mass of fuel
m_{ex}	mass of exhaust
dB	decibel

CHAPTER ONE
INTRODUCTION

1.1 Background and Justification

Noise exists wherever people live specially in industrial cities. Because the life of human has knotted with machines and annoying noise has been produced while the engine of machine is working. The noise comes from the exhausters of jet engines, automobiles, funnels of power houses and so on. Four types of vehicle noise sources threaten the human hearing when they are inside. They are engine noise, wind noise, road noise and exhaust noise. Generally, noise control safety standards come into play when the generation of noise cannot be avoided and must be addressed in some manner. Common solutions include the use of silencers or enclosures/cabins. Noise control is vital to protect the safety of human/workers, as well as the comfort levels of those outside the workplace but still close enough to be affected. As it is known three wheelers are major transport servicing vehicles in Adama city which are producing maximum amount of noise. These vehicles are mostly single cylinder and two cylinders that their exhaust systems should be looked carefully to design optimized one. So this study analyzes the mufflers of these three wheelers. The exhaust from the automobiles is at a high pressure and leads to generation of noise while rejection to the atmosphere. To reduce the exhaust pressure and subsequently to reduce the noise a muffler should be improved in the exhaust system of the automobiles. Due to increased environmental concerns requiring less noise emissions combined with reduced emission of harmful gases, it is becoming very crucial to carefully design the exhaust system mufflers for road transport applications.

Internal combustion engines are the major power sources for automobiles and also the work developers for the small scale power generation units. They operate by taking in a mixture of air and fuel (petrol engines) or by taking only air (diesel engines) and compressing it followed by combustion. At this stage power is produced while combustion of the fuel and ultimately exhaust gases are produced and have to be pushed out of engine cylinder so that there is ample volume left for the fresh charge to be induced. The exhaust from the automobiles is at a high pressure and leads to generation of noise while rejection to the atmosphere. To reduce the exhaust pressure and subsequently to reduce the noise a muffler is improved in the exhaust system of the automobiles. A muffler is a device that is used for reducing the amount of noise produced by the engines. The basic constructions of muffler usually consist of the tubular metal jacket, perforated tubes and the expansion chamber. The arrangement of these components will guide the exhaust gas to flow from the inlet pipe of the muffler to the outlet (tail pipe). Inside the muffler, the noise from the exhaust gas will be cancelled out by the basic physics principle on noise cancellation before the gas flows out to the atmosphere. The noise cancellation will reduce the noise that radiated by the vehicle to the surroundings.

A noise safety standard refers to many causes engaged in averting injuries or suffers resulting from the effects of sound. By designing devices in a way as to decrease noise product, controlling that noise before it touches people, or through the use of personal protective equipment such as ear muffs, noise safety takes many forms, each created to meet the specific kinds of noise exist in an environment.

Noise safety standards look for reducing the occurrence of negative effects of noise, coming from temporary distraction to short form of hearing loss, all the way through to continuous hearing loss or deafness. Furthermore, these standards are not only using in determining where noise should be attenuated but also in guiding the process, from the initial measurement of noise , to the choices available to reduce it, their productivity, execution, and overall outcome of noise safety plans. One of the main organizations of noise safety standards is Acoustical Society of America (ASA).

Nowadays, reduction of various existed noises is the significant issue for scholars in the area of Acoustics and vibrations to overcome. This thesis focuses on the effective parameters on maximization of the pressure loss to reduce the noise in the mufflers. Geometric modeling of RE 4-STROKE PETROL ENGINE (Bajaj) muffler was carried out using Solid Works package and further geometric model was imported into ANSYS Workbench to carry out simulation analysis.

1.2. Basic Definitions

a. Noise and Sound

The fluctuations in the air pressure in any media (like air) are appeared as an issue which is called **noise** and **vibration** and it is sensed by ear drum of the human that is called **sound**. The frequency ranges for healthy human hearing is from 20Hz to 20,000Hz. The term of "noise" is used to show the unwanted sound.

b. Sound Wave

Variation of pressure which contains energy when it travels through different medium, it is called sound wave. These waves travel in the air by vibrating the molecules of air from source of sound.

c. Silencers

There is no technical difference between a muffler and silencer, and these two terms are utilized interchangeably. Although the terms might not be unknown for many people, mufflers make everyday life much more amiable. The demand of mufflers is mainly directed to the machine components or areas where there is a large amount of emitted sound such as exhaust tubes which has high pressure, gas turbines, and rotary pumps. Muffler consists of four main components.

- Inlet pipe
- Outlet pipe
- Shell/Resonator chamber
- Perforated pipe

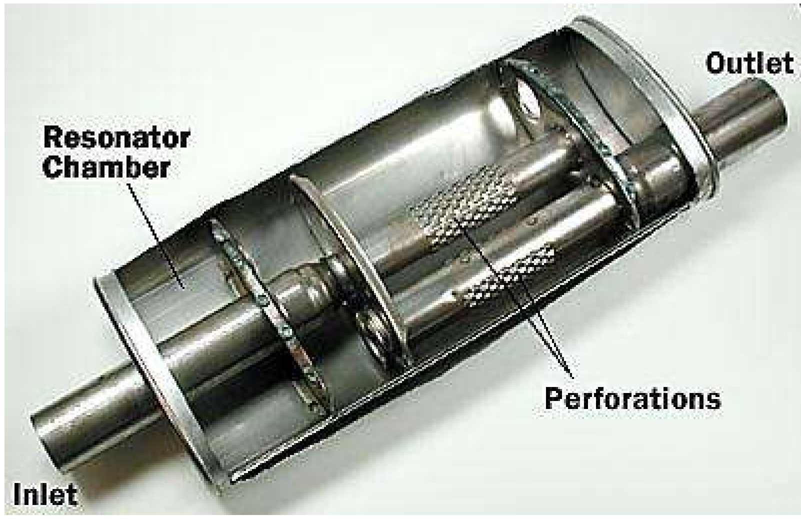


Figure 1. Cut way View of a Conventional Muffler

When the exhaust gases from inlet pipe pass through the perforations inside the shell, the gases get scattered in different directions. After reflection from the inside surface of the shell, the sound cancellation of waves occurs. The gases pass through the perforations multiple times and even get reflected from the shell surface. Due to the combined effect of these, the level of sound at the muffler outlet is reduced significantly. The flow through the muffler and variation of various parameters such as velocity and pressure along the length of the model can be accurately demonstrated with the help of CFD analysis which display accurate results within a short span of time.

Although there are several demands for mufflers, there are really only two chief types which are used. They are absorptive and reactive silencers. Absorptive mufflers formed into corporation of sound absorbing materials to wrap the emitted energy in gas flow. Reactive silencers utilize a series of complicated paths to maximize sound attenuation while encountering set specifications such as dropping the pressure, volume flow, etc. Several of complicated silencers today combine both methods to optimize sound attenuation and prepare practical specifications.

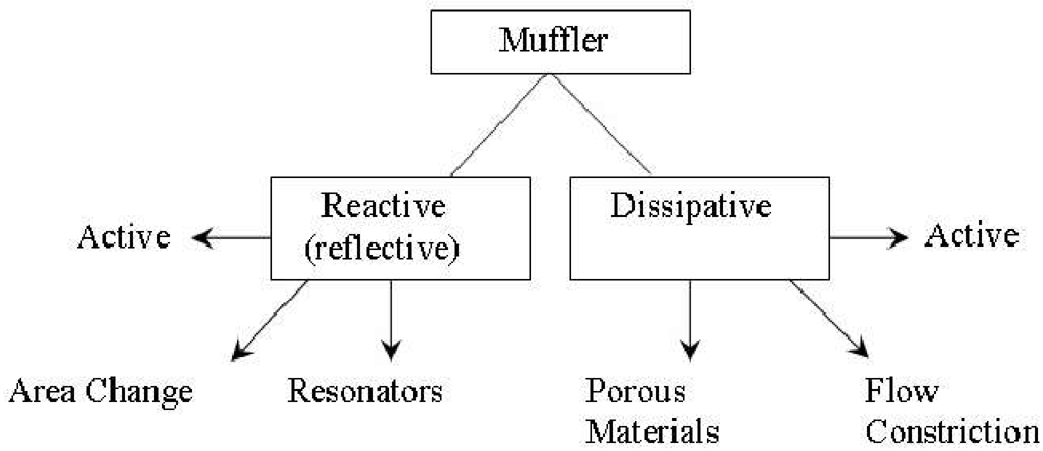


Figure 2. Types of Muffler.

i. Absorptive Silencer

An acoustic material is materials that decrease vibration and noise. There are four various types of material which are usually used to control, automotive noise and vibration. They are barriers and absorptions for noisy area and Isolation and damping in the vibrant area. Some materials control airborne noise and some of them control structure borne noise. In this area, each type of mentioned material has an impressible parameter on the sound transmission loss (TL). This type of muffler is combined with the absorbing materials to transmute energy of acoustic into heat. Absorptive mufflers are mostly straight pipes which several layers of absorptive materials like fiberglass to decrease the emitted sound power. The attenuation property of the absorptive materials is constant and it is the significant feature of this type of the muffler.

A dissipative muffler utilizes materials which absorb sound to remove the energy of the acoustic motion in the wave, as it emits through muffler. More energy breaking and lower emitted sound power can be leaded by higher attenuation constants. One of the renowned applications of this type of muffler is in racecars where the engine performance is requested, and great back pressure is not produced to muffle the sound as well. This leads to advance the muffler performance. This type is good in the applications when the source produces noise in a broad frequency band and is particularly effective at high frequencies, but special precautions must be taken if the gas stream has a high speed and temperature and if it contains particles or is corrosive. Some mufflers are a combination of reactive and dissipative types. Selection of these mufflers will depend upon the noise source and several environmental factors.

ii. Reactive Silencer

Reactive mufflers are usually composed of several chambers of different volumes and shapes connected together with pipes. They are essentially sound filters and are mostly useful when the noise source to be reduced contains pure tones at fixed frequencies or when there is a hot, dirty, high-speed gas flow. Reactive muffler for such purpose can be made quite inexpensively and require little maintenance.

To decrease the amount of acoustic energy transmuted, reactive mufflers are used with a number of complicated paths. This is done by alteration in impedance at the junctions and makes ascent to reflected waves. Degradation the engines' performance is happened by minimizing the amount of energy transmuted, so the energy which back to the source is really high. This type of muffler is really well-known to use for combustion engines, and mufflers in rough environments. The performance of the muffler is efficient in low frequencies while the absorptive ones uses in high frequencies.

There is a difference between these two types of silencers; absorptive muffler breaks the energy of acoustic, while reactive one retains the energy and ruinous interference to reduce the emitted sound power. Reactive silencers, which are usually used in automotive applications, reflect the sound waves return towards the source and hinder sound from being transmitted along the pipe.

1.3. Statement of the problem

The worst situations in the rate of exhaust noise calls for hand to hand efforts of all concerned bodies and the society in general. It is a well known fact that the socio economic, physical and psychological crisis caused by vehicle exhaust noise need to be dealt with seriously. Various research and studies should be conducted to overcome these problems. The ever growing of interest of problem finding and solving will bring many changes in the future that we think it would be possible to improve the present technologies we are using every day in our life. In automotive industries also many improvements have been done so far and will in the future. This research is attempting a different approach to this problem (exhaust noise) particularly on the mufflers of three wheelers by modifying internal structures. The system intended to be innovative in this research will decrease pressure during exhaust. This research is dedicated to find solution for this known problem by modifying and analyzing internal Structure of muffler by using original muffler as a bench mark. Models are created in Solid Works and ANSYS 15.0 is used for analysis.

1.4. Objectives of the Study

1.4.1. General objective

General objective of this thesis is to analyze muffler of a three wheeler vehicles using Computational Fluid Dynamics (CFD) and modification to reduce noise.

1.4.2. Specific objectives:

1. To collect the geometrical details of muffler used in Three Wheelers.
2. To develop model of muffler using SOLID WORKS 2014.
3. To simulate gas flow through muffler using ANSYS FLUENT.
4. To analyze the muffler for its efficiency in attenuation of noise using ANSYS FLUENT.
5. Based on the analysis modification of three wheelers muffler.

1.5. Significance of the study

In Adama transport system that is serving the public are mostly three wheeler vehicles. The study has significance to optimize sound emitted from exhaust system of three wheelers. As urbanization increases, the numbers of these vehicles in the city increases proportionally to give transport service for the public while increasing the noise emission. In addition to this, in the working principles of muffler if the steady flow is not significantly impeded the so-called 'back pressure' will be very low and the engine will function more efficiently. Because of the back pressure the volumetric efficiency decreases and specific fuel consumption increases. Hence there is a need for an optimum muffler design.

1.6. Scope of the study

The scope of this study is generally design of models and analysis of the three wheelers muffler using software; SOLID WORKS 2014 and ANSYS FLUENT 15.0. Model is generated to FLUENT ANSYS work bench. After the analysis is done, CFD results will be drawn and review the airflow streamlines. The exhaust gas velocity and pressure plots will be studied across the internal tubes and perforated holes. Based on the analysis the result will be drawn and discussed.

1.7. Limitation of the study

The limitations of this thesis work are acoustic extension package is not available in the ANSYS. Analysis of sound wave transmission is not done. There is no sufficient literature for CFD analysis of flow of gas inside the muffler especially that of three wheeler vehicle is not studied using ANSYS software. Thus it is not possible to compare this result with literatures reviewed.

CHAPTER TWO

LITERATURE REVIEW

Ozdemir et al (2013) modeled a muffler shown in the figure 3 the muffler consists of perforated inlet and outlet pipes and two perforated baffles. Furthermore, the muffler is divided into three main parts; front, middle and expansion chamber. Perforated parts of inlet and outlet pipes create a cross flow inside the muffler. Front and back baffles are also perforated.

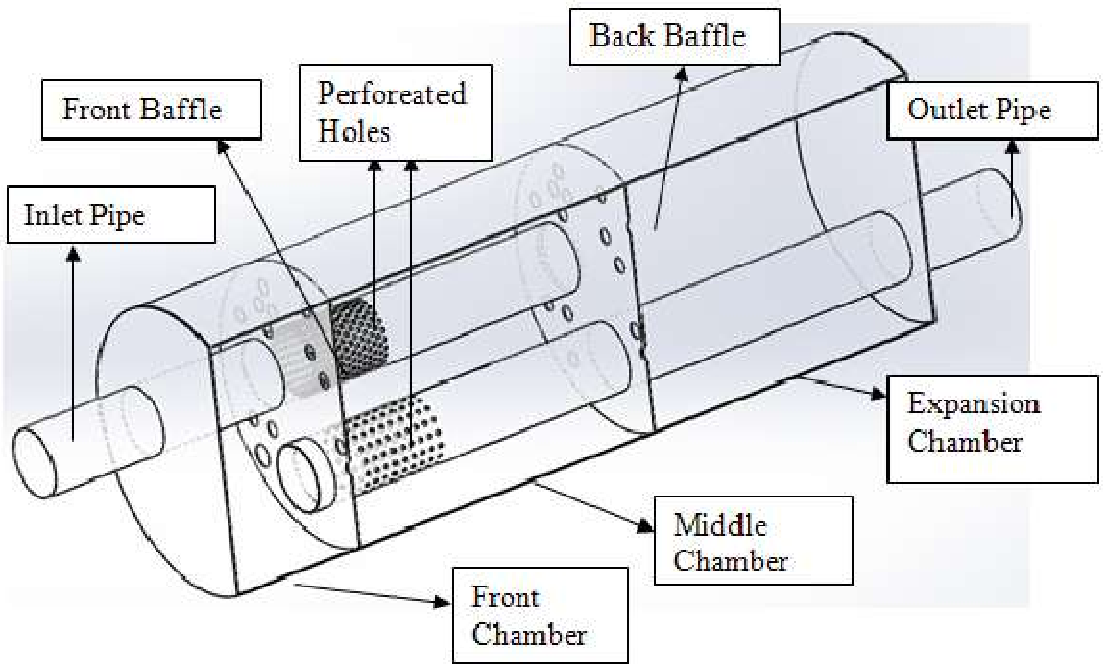


Figure 3. Schematics of the Inner Structure of a Base Muffler [1]

The design changes for the purpose of reducing the total volume of the muffler. Expansion chamber has an important role for the muffler. In this chamber, jet flow or cross flow occur and acoustic attenuation is obtained by reflecting sound wave to source. In this view, front chamber in the present muffler can be thought as expansion chamber. Increasing the number of expansion chamber in the mufflers which have more than one expansion chambers enhance the transmission loss of the mufflers; however, the influences on back pressure changes depending on the rate of perforation**.** For this purpose, in this study, three alternative models are attained by decreasing front chamber length, middle chamber length and expansion chamber length of the base muffler with certain percentage. Obtained alternative muffler model can be seen in Figure 4. [1]

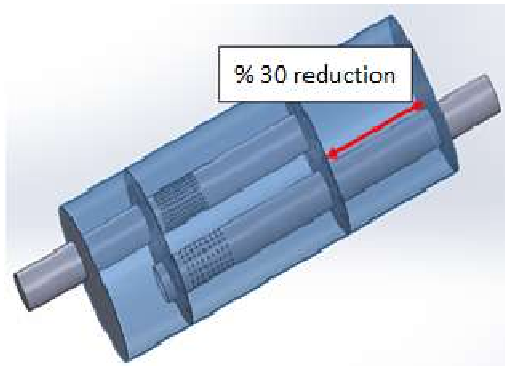


Figure 4. A New Designed Model Muffler with 30% Reduction of the Main Expansion Chamber Length [1]

The model comprised of a perforated inlet and had four pipes between the expansion chambers. The analysis was performed by changing the lengths of the front, middle and expansion chambers as it was found that the noise attenuation can be done by increasing the length of the middle chamber. The air inlet was considered at 473 K and the air was considered to be incompressible ideal gas for the analysis. It is observed that 30% reduction on length of rear chamber did not make any difference on acoustic characteristics with base muffler model. To generate cross flow, inlet and outlet pipe's perforated part stand in the middle chamber. A decrease in the length of middle chamber prevents the cross flow. Thus, It can be concluded that a greater pressure loss occur at this model. [1]

Tutunea et al [2] conducted research on resistance muffler with the fields of acoustics, fluid dynamics, heat transfer and mechanism design. The author simulated the field by numerical method with Cosmos Flow and analyzed the effect of the internal flow field on the performance of the muffler, which may be a credible guidance of the muffler structural design. With this method the pressure distribution in the muffler is simulated and the pressure loss is predicted for the structure modification. The experimental results verify that the assembly performance of the muffler modified is better than the original muffler.

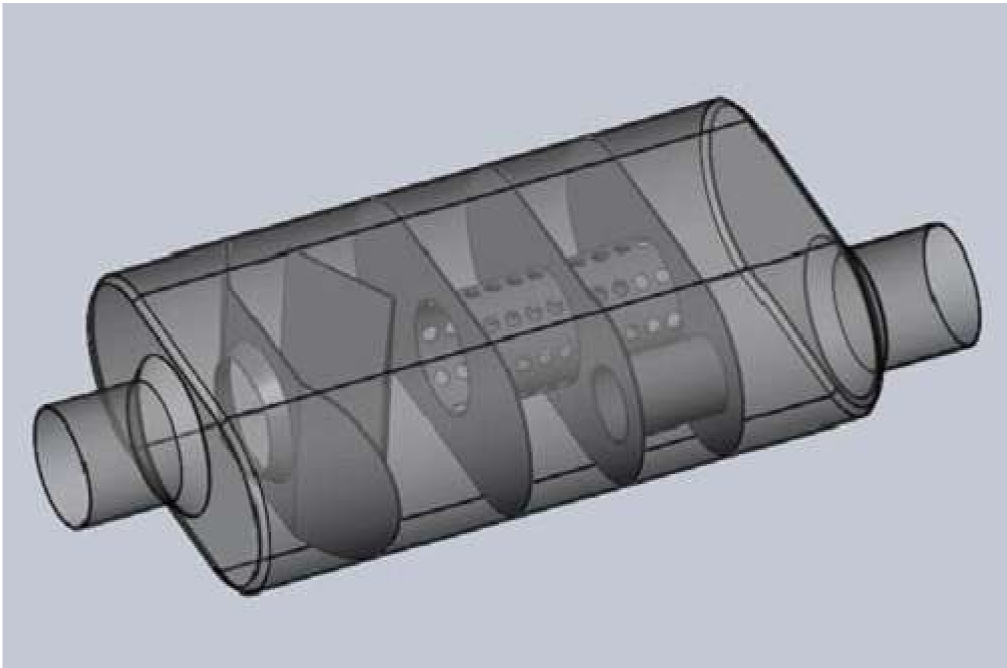


Figure 5. A 3D Model of a New Designed Muffler [2]

The model is analyzed by considering the flow of burning gases at a speed of 25 m/s and a temperature of 280°C. For the environment condition is selected an ambient temperature of 20°C and a normal atmospheric pressure. Due to the high subsonic compressible internal flow the flow simulation was computed on an I7 Intel Core System with 6 Gb. of RAM. The solution has converged after 2745 iteration.[2]

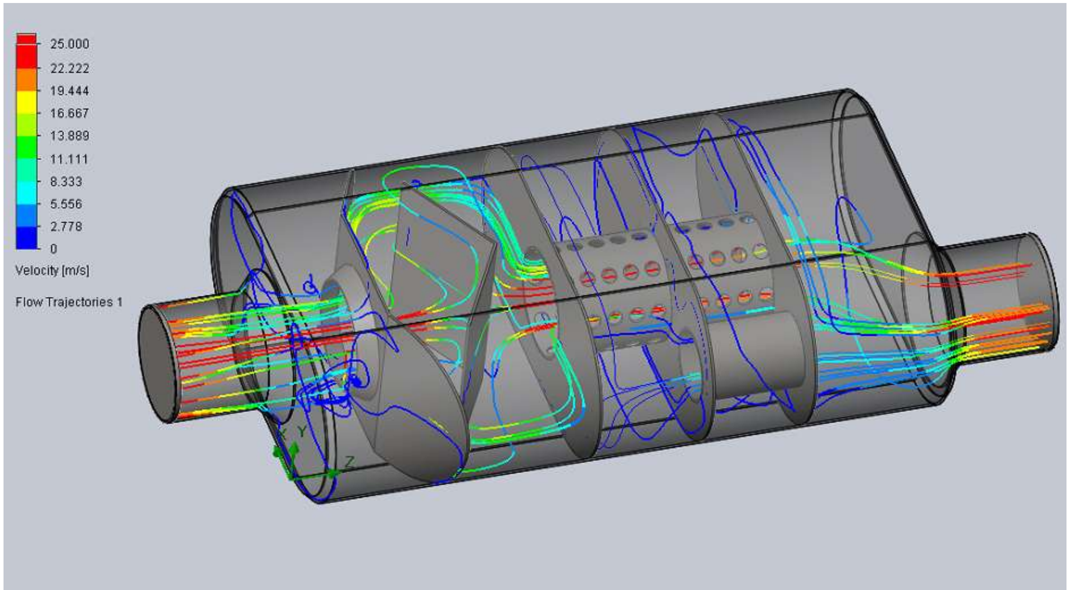


Figure 6. A Gas Flow Pattern in the Model Muffler [2]

Sileshi Kore et al (2011) analyzed a muffler model which the muffler was designed for the simulation of the Alfa Romeo 1995 model. The analysis was performed in ANSYS FLUENT and the meshing of the geometry was done using GAMBIT. The solver implemented was a 2-D, segregated implicit solver with 2nd order implicit time stepping. Second order upwind discretization was used for the density, momentum, energy, turbulent kinetic energy and the turbulent dissipation rate equations. The inlet conditions were considered as pressure inlet of air (ideal gas) at a pressure of 101325 Pa and 300 K temperature. In this case the inlet velocity is normally 80 m/s and the outlet boundary condition has been set at atmospheric pressure. From the above analysis it was observed that the maximum transmission loss is around 14db with an optimal amount of backpressure for the developed model.

Tutunea et al (2013) conducted a research on a muffler which had three main chambers and the middle chamber had a perforated tube. Here the model was generated using SOLIDWORKS 3D modeling package and the analysis and simulation of the internal flow was carried out using ANSYS FLUENT 14. The boundary conditions were incorporated as velocity inlet, where the velocity was considered as 25m/s and at 280 ^{0}C of temperature. It was observed that there was a considerable decrease in the pressure at the outlet of the muffler and the results were in good agreement. The flow of the exhaust was assessed by generating a CAD model and simulating them in the virtual environment of ANSYS fluent.

Denia et al (2001) mentioned that elliptical geometry is widely used in automotive mufflers. However, there are relatively few reported studies of its acoustic attenuation performance. They also studied the problem of wave propagation in elliptical chamber mufflers from several points of view. The solution of the wave equation in elliptic co-ordinates is expressed in terms of the Mathieu functions, obtaining the natural frequencies and mode shapes. The modal superposition technique and the point source method have been applied to a number of elliptical mufflers (expansion chambers and reversing chambers) to obtain their transmission loss. The results are compared with those obtained from the finite element method and from experiments, showing a good agreement. Also the boundary element method and the finite element method (FEM) are the most interesting ones. One of the major advantages of the FEM is the possibility to analyze problems with arbitrary geometry and general boundary conditions, but at the cost of long computation times if sufficient accuracy is desired at high frequencies, due to the large number of elements required.

Further study on elliptical muffler was carried out by Denia et al (2003) and presented a detailed analysis of the acoustic attenuation performance of elliptical expansion chambers with single-inlet and double-outlet.

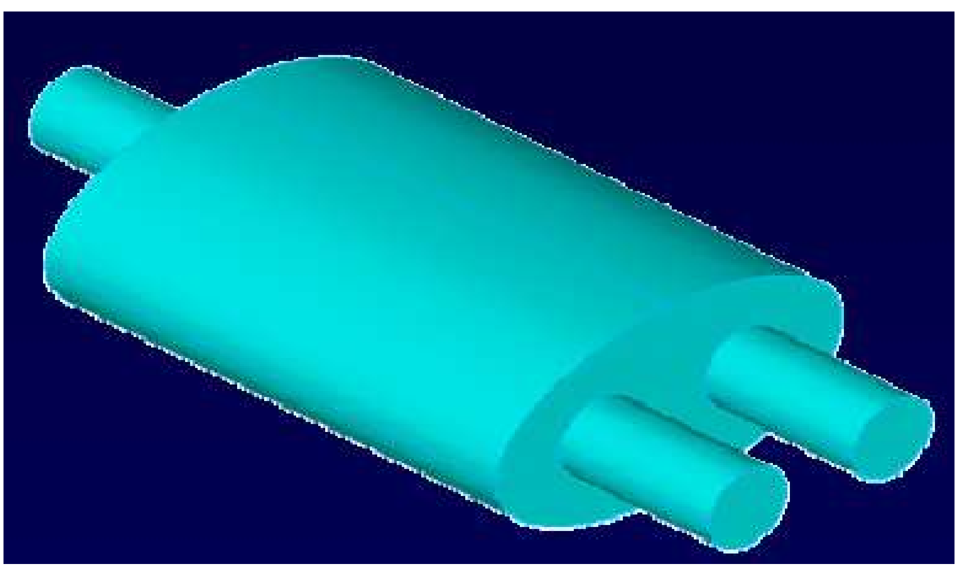


Figure 7. An Elliptical Mufflers with Single-inlet and Double-outlet [6]

First, the Finite Element method (FEM) is considered in order to obtain a reference solution. Then, the Mode matching method (MMM) is applied at the area discontinuities of the muffler, considering a circular inlet pipe, a central elliptical chamber and two circular outlet Pipes. This method reduces the computational requirements and enables to couple the Acoustic fields within each region, described by means of Mathieu functions in the Elliptical chamber and Bessel functions in the circular pipes. The solution given by this analytical approach is compared with finite element results and experimental**.** Measurements for a selected configuration show a good agreement. The acoustic behavior is then analyzed in detail as a function of the chamber length, the Eccentricity of the elliptical cross-section and the position of the inlet and outlet pipes. Some potential means to improve the acoustic performance are proposed.

Choi and Lee (2013) formulated an acoustical shape optimization problem for optimal design of a perforated reactive muffler with offset inlet/outlet. The mean transmission loss value in a target frequency range is maximized for an allowed pressure drop value between an inlet and an outlet. Partitions in the chamber are divided into several sub-partitions, whose lengths are selected as design variables. Each sub-partition has the same number of holes, whose sizes are equal. A finite element model is employed for acoustical and flow analyses. A gradient-based optimization algorithm is used to obtain an optimal muffler. The acoustical and fluidic characteristics of the optimal muffler are compared with those of a reference muffler. Validation experiment is carried out to support the effectiveness of our suggested method.

Abdullah et al (2011) implemented three-dimensional finite element analysis to predict the transmission loss of an elliptic expansion chamber muffler for a given frequency range. Finite element models have been established by ANSYS 12.0.1. Six muffler configurations of different major/minor axis ratios of elliptic expansion chamber (a/b), with equal chamber volume have been modeled. Results demonstrated the ability of FEM to represent the acoustic attenuation performance of acoustical muffler. It also showed the effect of (a/b) ratio on the cut-off frequency due to multidimensional modes propagation. The FEM results were interpreted in various muffler configurations to determine the relationship between the number of domes in transmission loss characteristic curve and (a/b) ratio. In this study each model was meshed using tetrahedral FLUID30 element. The resulting mesh size produces 10 elements per acoustic wave length at the upper band of the frequency range being analyzed.

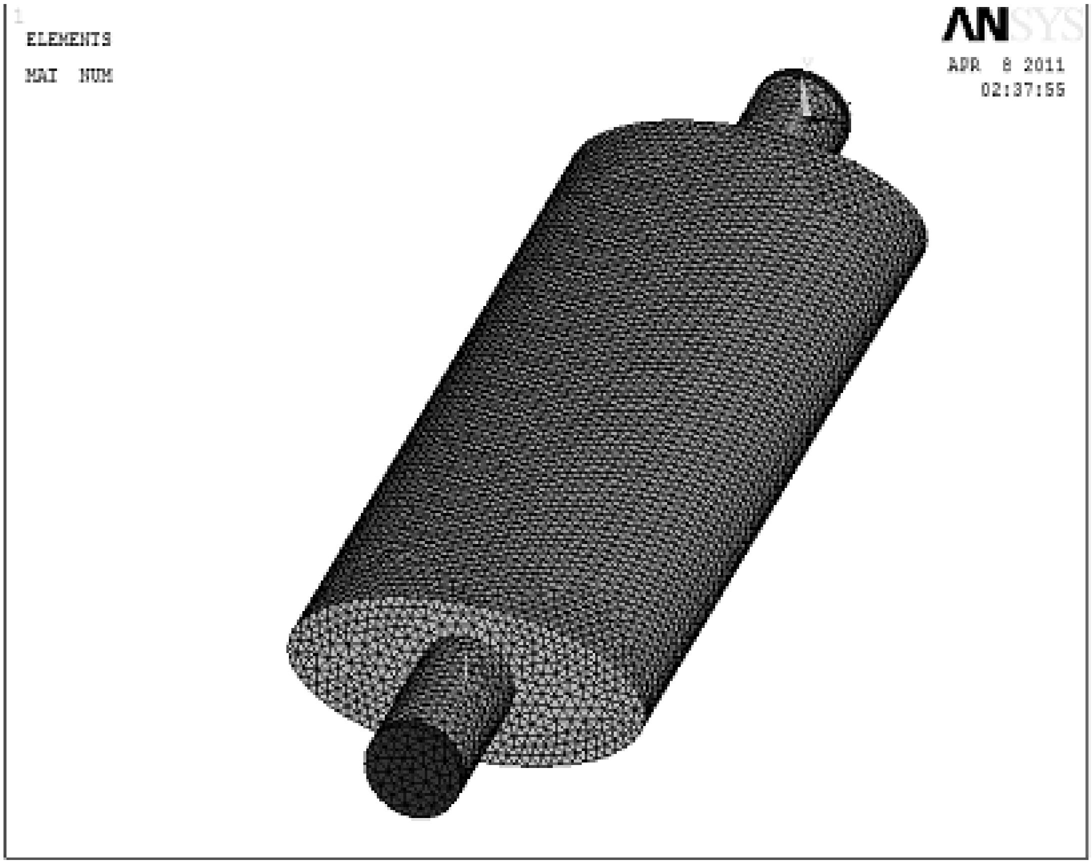


Figure 8. Typical FE Model of an Elliptical Expansion Chamber [8]

A good FEA model should be able not only to mathematically represent the physical system as accurate as possible but also to avoid unnecessary computational cost as much as possible.

Patil et al (2014) had studied the exhaust system designed and through CFD (Fluent) analysis, a compromise between two parameters namely, more maximization of brake thermal efficiency with limited back pressure was aimed at. In CFD analysis, three exhaust diffuser system (EDS) models with different angels are simulated using the appropriate boundary conditions and fluid properties specified to the system with suitable assumptions. The back pressure variations in three models are discussed in. Finally, the model with limited backpressure was fabricated and Experiments are carried out on single cylinder four stroke diesel engine test rig with rope brake dynamometer. CFD analysis of exhaust diffuser systems and the performance of the engine are discussed.

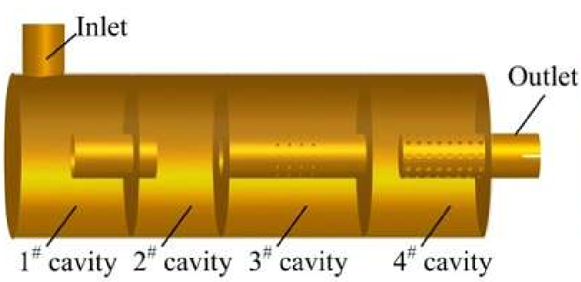
Özdemir et al (2013) in this study, a reactive, perforated and cross-flow muffler which damped the noise of a spark ignition engine is investigated. Some geometrical dimensions of the muffler are decreased and new models are redesigned. Models are analyzed to obtain flow characteristic by using computational fluid dynamics.

Different configurations of simple expansion chamber mufflers, including extended inlet/outlet pipes and baffles, have been modeled numerically using Computational Fluid Dynamics (CFD) in order to determine their acoustic response. The CFD results are compared with published experimental results. The CFD model consists of an axisymetric grid with a single period sinusoid of suitable amplitude and duration imposed at the inlet boundary. The time history of the acoustic pressure and particle velocity is recorded at two points, one point in the inlet pipe and one point in the outlet pipe. These time histories are Fourier Transformed and the transmission loss of the muffler is calculated. Calculated results show excellent agreement with the published data. The mean flow performance has also been considered. The mean flow model of the muffler uses the same geometry, but has a finer mesh and has a suitable inlet velocity applied at the inlet boundary and the pressure drop across the muffler is found [11].

This Study comprehensively analyzes four different models of exhaust muffler and concludes the best possible design for least pressure drop. Back pressure was obtained based on the flow field analysis and was also compared with all muffler design. Virtual simulation for back-pressure testing is performed by Computational Fluid Dynamic (CFD) analysis using Acusolve CFD. Finite Element (FE) model generation of the muffler structure is performed using Hyper Mesh as the preprocessor. The structural mesh is modeled using 2D shell elements, wherein the internal tubes with fine perforated holes are considered. The CFD fluid meshing is done with tetra elements using AcuConsole as the CFD preprocessor. The back pressure generated across the muffler is determined by measuring and inputting the mass flow rate of the exhaust gases entering the muffler inlet pipe. Field View is used for post processing the CFD results and reviewing the airflow streamlines. The exhaust gas velocity and pressure plots are studied across the internal tubes and perforated holes, and the back pressure is measured [12].

In this paper the effect of change in dimensions of perforation diameter and change in porosity of internal tube is investigated using CFD analysis and the simulated data is compared with experimental results. It is found that the porosity of the muffler has pronounced effect on the Backpressure. The backpressure reduced almost by 75% if the porosity is doubled. Also, if the diameter of the hole increases the backpressure decreases sharply by 40%. The change in diameter of holes has remarkable effect on back pressure. CFD and Experimental values are in good agreement with each other [13].

In order to evaluate performance of reactive muffler and its effect on power loss of engine, flow field of muffler was discussed by CFD comparing with experimental test and the structure of reactive muffler was optimized. Based on results of simulation and optimization, the reactive muffler used in vibratory rollers with weight of 13t was fabricated and its field test was carried on. The structure and critical parameters were defined according to the spectral characteristics of the exhaust noise of a roller. The 3D models of the 1# muffler without optimization and the 2# muffler with optimization were conducted using Pro-E software, of which were transformed into Step file, as shown in Figure below.[14]



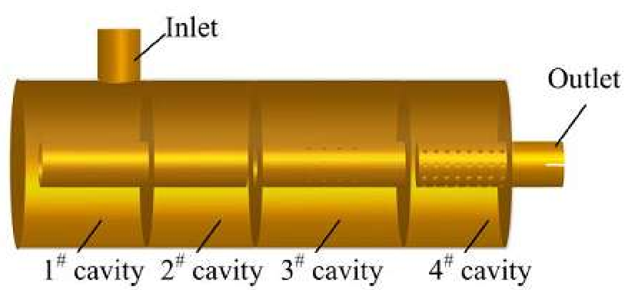


Figure 9. The 3D Models of the Muffler (a) Muffler-1, (b) Muffler-2 [14]

The simulate results showed that velocity field coincided with the pressure field basically, which indicates that the optimized muffler -2# has excellent aerodynamic characteristics and rational design of damping units. The results of field tests showed that 2# muffler had better acoustic insertion loss with little pressure loss. Acoustic insertion loss was 17~18.4 dB (A) at engine speed of 2450 rpm, which meets the designing goal [14].

Higgs Benjamin and Rupke Ryan (2007) designed the muffler system to meet the demanding needs of a Formula SAE prototype race car. This development must adhere to the FSAE standards as they relate to noise control. In order to fully understand the problem definition, a comprehensive study of sound properties is completed. This research elaborates on sound waves and how they may be reflected, diffracted or absorbed or actively cancelled using constructive interference. Reflection and diffraction properties represent destructive interference and depend on the angles at which they hit an object. Absorption, also a form of destructive interference, relates to the material that the sound wave comes in contact with. Constructive interference occurs when a 35second sound source is placed 180° out of phase from the first source. The source of noise that a muffler must attenuate is the result of combustion from the engine.

Deal with the design and flow simulation through the muffler. Two muffler designs have been modeled and CFD gas flow simulation has been carried in both of them. Based upon the gas flow through them, on comparison the optimum model was selected.

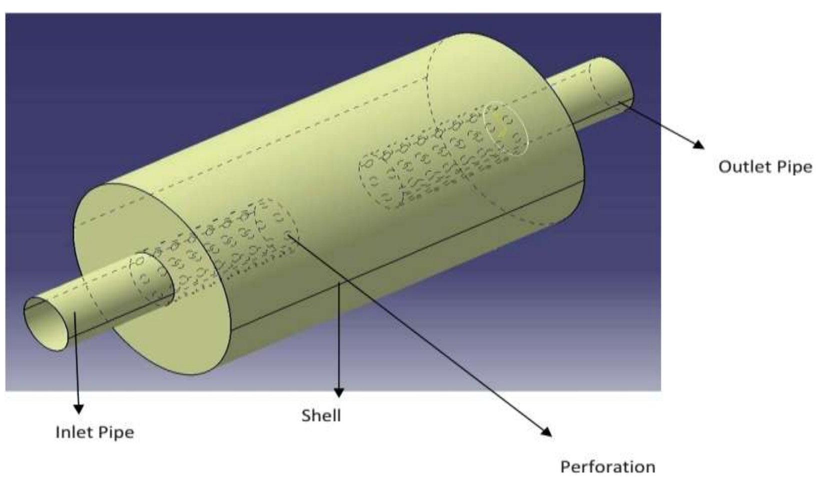


Figure 10. A Solid Model of the Muffler [15]

When the exhaust gases from inlet pipe pass through the perforations inside the shell, the gases get scattered in different directions. After reflection from the inside surface of the shell, the sound cancellation of waves occurs. The gases pass through the perforations multiple times and even get reflected from the shell surface. Due to the combined effect of these, the level of sound at the muffler outlet is reduced significantly. The flow through the muffler and variation of various parameters such as velocity and pressure along the length of the model can be accurately demonstrated with the help of CFD analysis which display accurate results within a short span of time.

Patil et al (2014) The exhaust system designed and through CFD (Fluent) analysis, a compromise between two parameters namely, more maximization of brake thermal efficiency with limited back pressure was aimed at. In CFD analysis, three exhaust diffuser system (EDS) models with different angels are simulated using the appropriate boundary conditions and fluid properties specified to the system with suitable assumptions. The back pressure variations in three models are discussed in. Finally, the model with limited backpressure was fabricated and Experiments are carried out on single cylinder four stroke diesel engine test rig with rope brake dynamometer. CFD analysis of exhaust diffuser systems and the performance of the engine are discussed [16].

Finite element methods are used to obtain flow characteristics and back pressure values of mufflers. Having used of this method, effect of different parameters can be examined without prototyping and best suitable muffler can be determined in the design process. Furthermore time and money can be saved. In this study, a reactive, perforated and cross-flow muffler which damped the noise of a spark ignition engine is investigated. Some geometrical dimensions of the muffler are decreased and new models are redesigned. Models are analyzed to obtained flow characteristic by using computational fluid dynamics [17].

Puneetha et al studied comprehensively four different models of exhaust muffler and concludes the best possible design for least pressure drop. Back pressure was obtained based on the flow field analysis and was also compared with all muffler design. Virtual simulation for back-pressure testing is performed by Computational Fluid Dynamic (CFD) analysis using Acu solve CFD. Finite Element (FE) model generation of the muffler structure is performed using Hyper Mesh as the preprocessor. The structural mesh is modeled using 2D shell elements, wherein the internal tubes with fine perforated holes are considered. The CFD fluid meshing is done with tetra elements using Acu Console as the CFD preprocessor. The back pressure generated across the muffler is determined by measuring and inputting the mass flow rate of the exhaust gases entering the muffler inlet pipe. Field View is used for post processing the CFD results and reviewing the airflow streamlines. The exhaust gas velocity and pressure plots are studied across the internal tubes and perforated holes, and the back pressure is measured [18].

Selamet et. Al (1997) has investigated deeply on the effects of various amounts of length on the performances of acoustic diminution of expansion chambers which have common axis. In this study, three methods are put to use to conclude the transmission loss [19].

ŁUSZCZYŃSKA et. Al (2005) have studied the influence of low frequency noise(LFN) on the execution of the mental performance. In this case study, they have examined and exposed that the LFN which is occurring at normal levels in the industrial control rooms have the influences on the human mental function and personal contentment [20].

A.Pandhare et al [21] conducted research on design and flow simulation through the muffler. Two muffler designs have been modeled and CFD gas flow simulation has been carried in both of them. Based upon the gas flow through them, on comparison we have selected the optimum model. When the exhaust gases from inlet pipe pass through the perforations inside the shell, the gases get scattered in different directions. After reflection from the inside surface of the shell, the sound cancellation of waves occurs. The gases pass through the perforations multiple times and even get reflected from the shell surface. Due to the combined effect of these, the level of sound at the muffler outlet is reduced significantly. The flow through the muffler and variation of various parameters such as velocity and pressure along the length of the model can be accurately demonstrated with the help of CFD analysis which display accurate results within a short span of time.

A reactive muffler has been built by Bhattacharya et. Al (2010) and the outcomes which were estimated at 1200 rpm have been compared with the three frequent types of muffler. They were brake thermal efficiency, brake specific fuel, and drip of pressure [22].

Chen and Shi (2011) made a research on a very basic model which has a perforated inlet pipe and has 4 pipes between expansion chambers using Fluent [23]. In 2010 Hou et.al modeled an exhaust system which consists of pre and after muffler and he analyzed it by using Fluent program. It was shown that the main pressure drop happens in the after muffler and declares that back pressure is 20,8kPa. He emphasized that when he looked at velocity range in the after muffler according to sudden direction changes of velocity vectors the back pressure increases. It is understood that back pressure can be optimized by size of baffle holes [24].

Fang et al (2009) studied flow characteristics of exhaust muffler of a steam shovel by finite element methods. Also he verified analysis results with the experiment. Analysis results showed that inlet total pressure is 256,566 Pa. It is seen that back pressure value reduces when the inlet velocity is decreased. Afterwards Fang gets real pressure loss between muffler inlet and outlet at 20m/s and he compared these values with analysis results [25]. In 2010, Avcu et al analyzed the flow characteristics of a dry and wet type exhaust muffler of a diesel-driven ship with Ansys CFX. Analysis results of the dry type muffler show that the back pressure is approximately 63.7 mbar. With some geometrical changes, the back pressure of the system was decreased to 23 mbar. The same analyses were repeated for wet type muffler and back pressure of it was obtained 22mbar [26].

Lee and Velh (2008) analyzed flow and acoustic behavior of the different type mufflers which has the same number of perforated hole; but different hole distribution. It was emphasized that the predicted value of the back pressure is less than the measured one and this will not change the performance of the engine so much. They asserted that the error likely arise from the faults during measurement of the velocities [27].

Gupta and Suman[28] studied in this paper the details of measurement of the Acoustic Transmission Loss of various Expansion Chamber is shown by using Finite element Analysis (FEM) Acoustic module is explained. For this purpose evaluation of transmission loss of different cross-sections by keeping constant volume of expansion chamber is explained. Also the design validation has been done with existing result to show the validity of results. After the validation process results are compared to observe the effect of changing in the cross-section of muffler.

a. Transmission Loss Measurement by FEA Acoustic Module

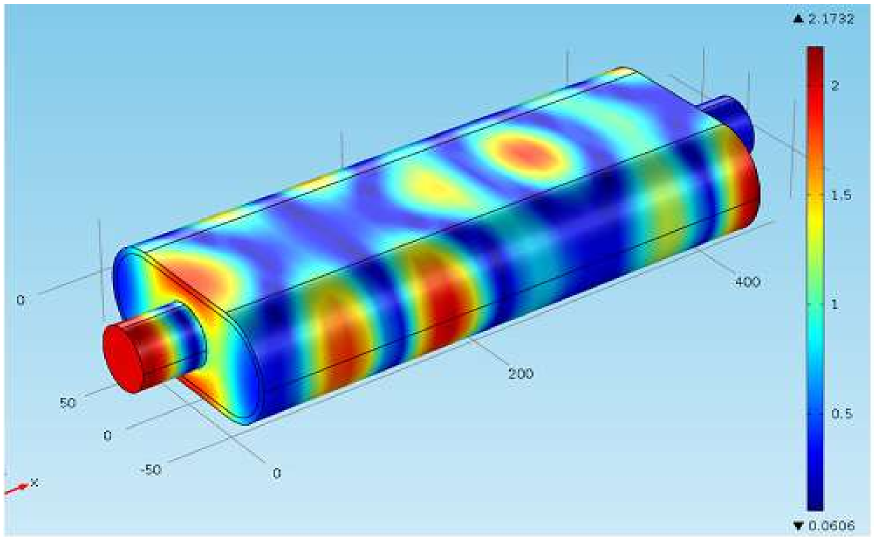


Figure 11. Existing Results Transmission Loss of a Muffler [28]

Table 1.Modeling of Types of Expansion Chamber [28]

S.No.	Types Of Expansion Chamber Having Length 211mm	Cross Section Dimensions With constant volume 6636500 mm^3
1	Circular	Radius = 100 mm.
2	Elliptical	Major & Minor Radius: 166mm X 60 mm
3	Rectangular	307.17 mm X 102.39 mm
4	Square	177 X 177 mm

a. Circular Expansion Chamber with radius 100 mm (Cross section)

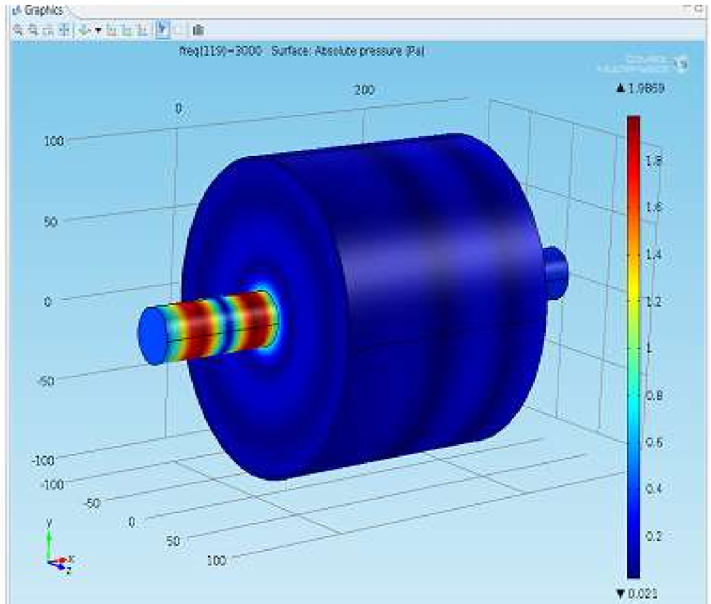


Figure 12. Absolute Pressure Analysis of a Circular Expansion Chamber [28]

b. Elliptical Expansion Chamber with Major & Minor Radius : 166 mm X 60mm (Cross section)

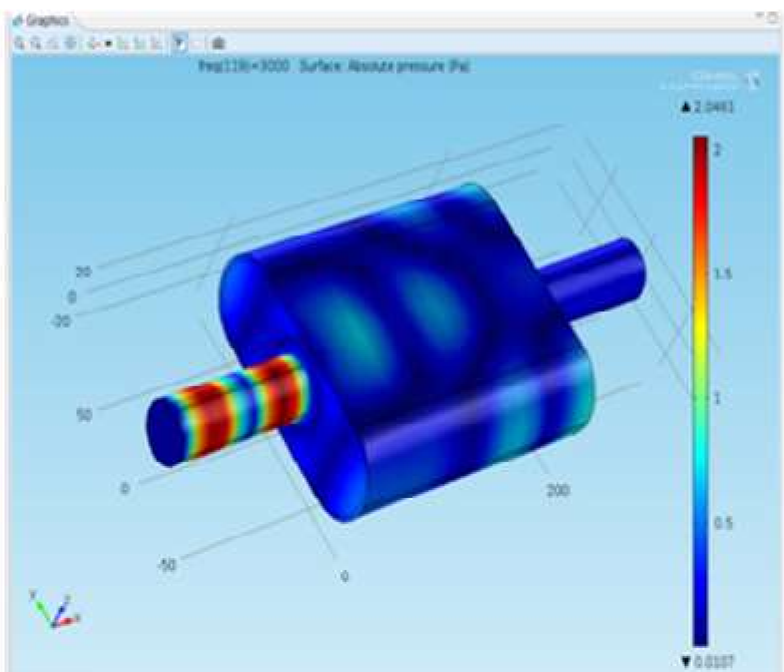


Figure 13. Absolute Pressure Analysis of Elliptical Expansion Chamber [28]

c. Rectangular Expansion Chamber with 307.17 mm X 102.39 mm (Cross section)

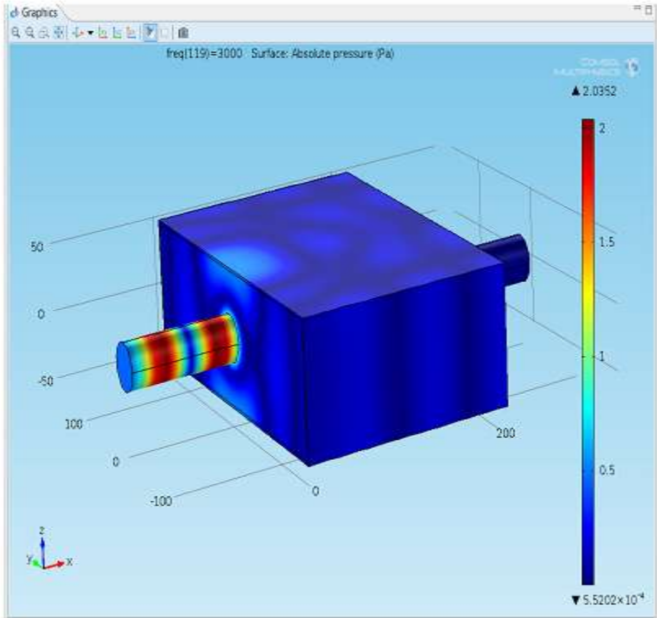


Figure 14. Absolute pressure analysis of rectangular expansion chamber [28]

d. Square Expansion Chamber with 177 mm X 177 mm (Cross section)

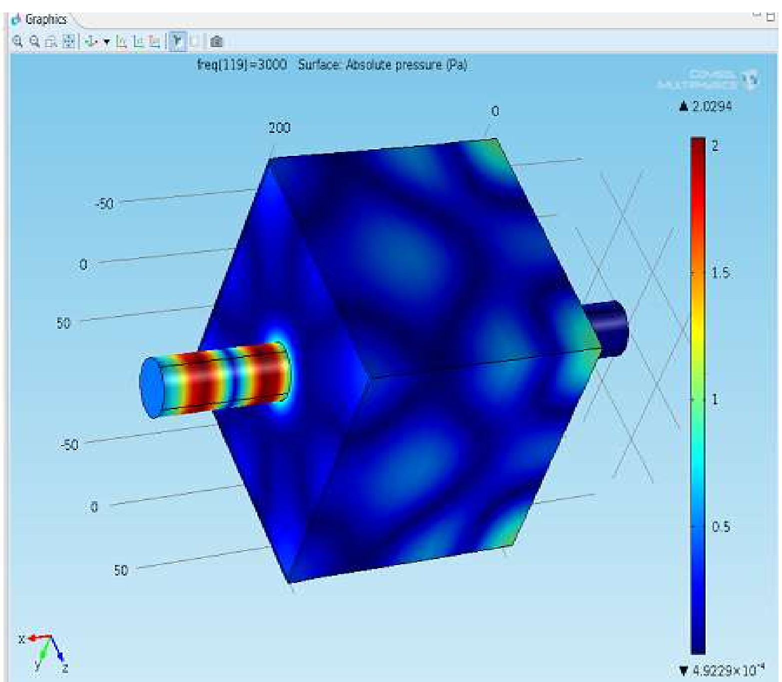


Figure 15. Absolute Pressure Analysis of square Expansion Chamber [28]

Saripalli and Sankaranarayana (2015) used Computational Fluid Dynamics (CFD) method to explore the aerodynamic performance of the muffler. Resistance muffler research relates with the fields of acoustics, fluid dynamics, heat transfer and mechanism design. The project report simulates the field by numerical method with Cosmos Flow and analyses the effect which the internal flow field has on the performance of the muffler. With this method the pressure distribution in the muffler is simulated and the pressure loss is predicted for the structure modification. The experiment results verify that the assembly performance of the muffler modified is better than the original muffler. The muffler being simulated is being designed as per the engine output of Maruti Ciaz.

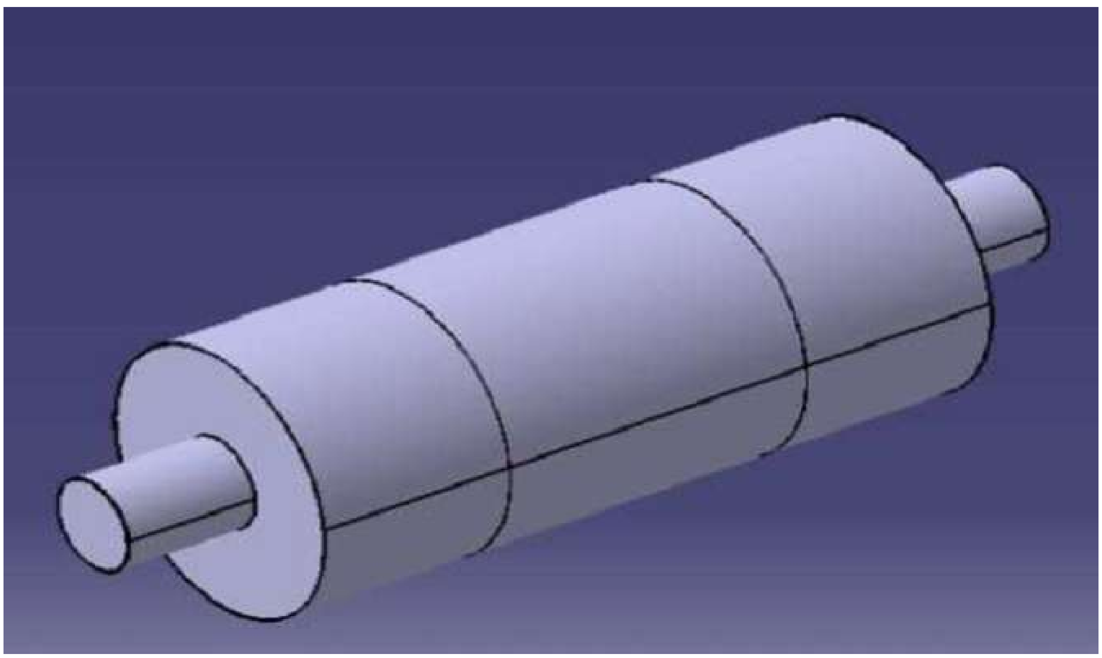


Figure 16. CATIA Model of the Designed Muffler [29]

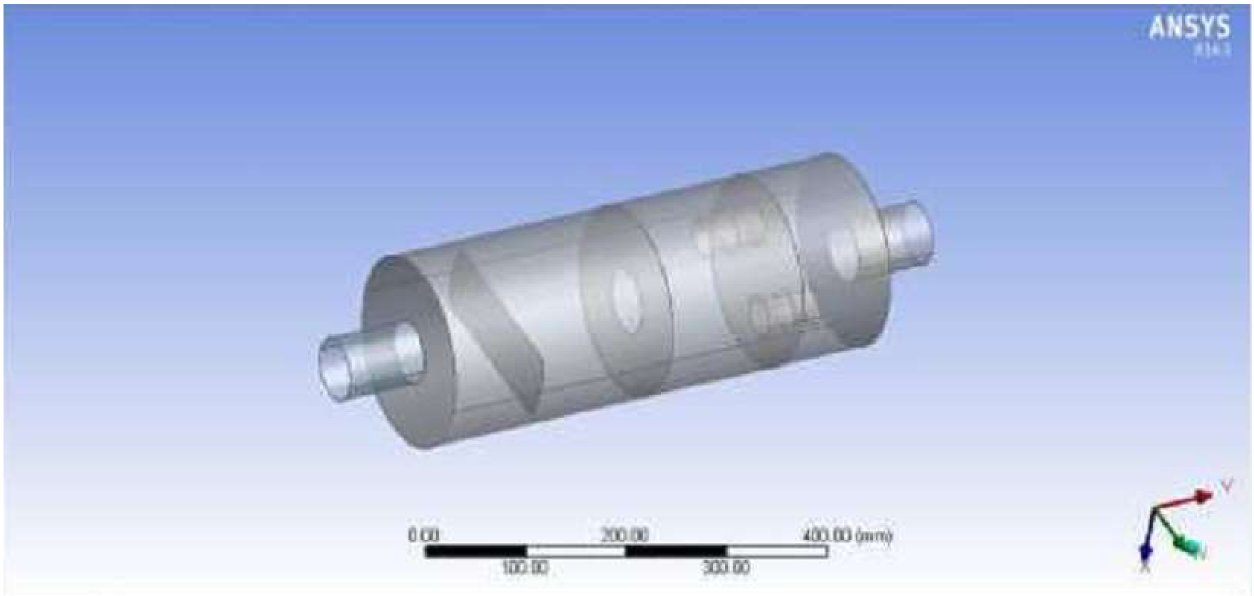


Figure 17. Internal View of Designed Muffler (model 1) [29]

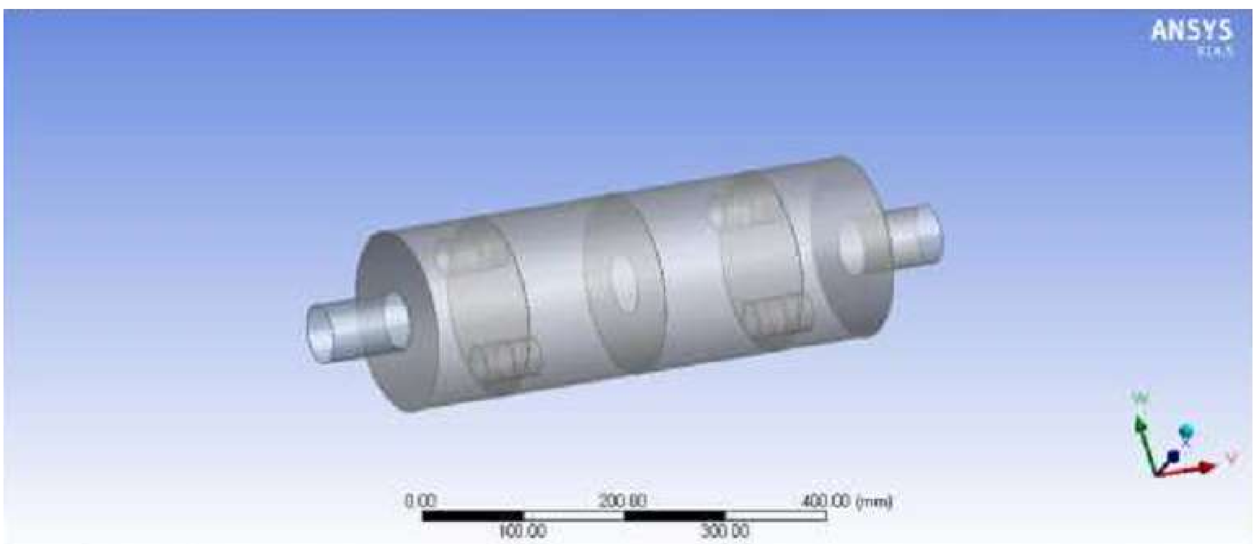


Figure 18. Internal View of Designed Muffler (model 2) [29]

After the construction of exhaust muffler model in CATIA it is analyzed in ANSYS FLUENT 14.5. The FLUENT analysis is carried out by considering two types of inlet boundary conditions i.e pressure inlet and velocity inlet. The mean flow performance of the muffler considered in the flow analysis has been assessed. The results of the simulated muffler models obtained with the use of CFD modeling are very encouraging. Even though the second model displayed a similar behavior, there was a more significant drop in the pressure of the exhaust gases in the first case than the second. The drop in the pressure of the exhaust gases in the first model was about 57% whereas the drop in the second model was nearly 51%.

The above literature review gave an idea to take a problem on noise. The kind of present work is not done before. The aim is to analyze the muffler of petrol engine using CFD FLUENT with different models i.e. with mufflers of elliptical shape with different internal sections to check the performance of the mufflers by measurements of gas speed and pressure distribution inside the muffler shell.

CHAPTER THREE

RESEARCH METHODOLOGY

3.1 Introduction

According to related researches increasing the number of expansion chamber in the mufflers which have more than one expansion chambers enhance the transmission loss of the mufflers. For flow analysis of the muffler, reflective base muffler shown figure below in cut condition is drawn with SOLID WORKS 2014. Additional model(s) are also drawn to compare the results. ANSYS Workbench is used to get mesh the muffler models. Tetragonal elements will be taken due to the thin thickness, small dimensions and especially perforated holes, hexagonal mesh is not preferable while meshing the muffler. Face sizing will be applied to perforated holes in baffles, pipes and inlet and outlet pipes of the muffler.

Figure 19. A Cut way View of a Reflective Base Muffler

The procedure of this thesis work includes the following steps;

- Collection of the necessary data related to muffler design parameters.
- Modeling the mufflers using SOLID WORKS 2014.
- Analyzing CFD results and reviewing the airflow streamlines.
- The exhaust gas velocity and pressure plots are studied across the internal tubes and perforated holes.

3.2 Software's used

3.2.1 ANSYS Fluent 15.0

The currently used CFD software for flow calculations in the air induction system group is the commercial package Fluent. It contains a variety of physical modeling capabilities to model most kinds of flow situations encountered in engineering. Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of

- mathematical modeling (partial differential equations)
- numerical methods (discretization and solution techniques)
- software tools (solvers, pre- and post processing utilities)

The solver used is 3-D, dp, ske. Second order upwind spatial discretization is used for the pressure, momentum, energy, turbulent kinetic energy and the turbulent dissipation rate equations.

3.2.2. Solid Works 2014

The Solid Works® CAD software is a mechanical design automation application that lets designers quickly sketch out ideas, experiment with features and dimensions, and produce models and detailed drawings.

Case Study – RE 4-STROKE PETROL ENGINE (Bajaj)

Engine Data:

Bore (D) = 63.5 mm

Stroke (L) = 62.8 mm

No. Cylinders (n) = single

Engine Displacement = 198.88 cm^3

Maximum Engine power (P) = 8.10 kW at 5000±250 RPM

Maximum net Torque (T) = 18.00 Nm at 3500±250 RPM

Muffler dimension:

Table 2. Dimensions of the Actual Muffle Model

Entity	Dimensions (mm)
Shell length	460
Shell Major dia	130
Shell minor dia	65
Inlet pipe dia	26
Inlet pipe length	50
Outlet pipe dia	26
Outlet pipe length	50

3.3. Engine Parameters

Average piston speed is:

$$V_{mp} = 2SN = 10.5 \text{ m/s at } 5000 \text{ RPM}$$

N is engine speed generally given in RPM (revolutions per minute), V_{mp} in m/sec (ft/sec), and B and S in m or cm (ft or in.).

Average piston speed for all engines will normally be in the range of 5 to 15 m/sec (15 to 50 ft/sec), with large diesel engines on the low end and high-performance automobile engines on the high end. Neglecting losses and assuming the fluid (gas) is incompressible, the volume flow rate of the fluid passing through the valve port is equal to the fluid (gas) passing through the cylinder bore. Hence, from ‘continuity equation’ (fluid mechanics) [30]

Mass flow rate conservation:

$$\dot{m}_{I} = \dot{m}_{o}$$

$$a_1 v_1 \rho_1 = a_p \; v_{mp} \; \rho_p = a_{ip} v_{ip} \rho_i = \quad \Rightarrow \quad a_1 = \frac{a_p \; v_{mp}}{C_i \, v_1} \quad \rightarrow \quad \rho = costant$$

V_{ip}= 68m/s

Where, C_i (0.15 to 0.35)

Table 3. Material Properties of the Muffler Model (Steel Stainless 302) [2]

Property	Units	value
Density	kg/m3	7900
Specific heat	J/(kg*K)	500
Thermal conductivity	W/(m*K)	16.2999
Conductivity type	isotropic	isotropic

3.4 Governing Equation

CFD provides a way to model and simulate real fluid flows and heat transfer by numerical solving of a set of governing equations. The fluid space that is to be modeled, known as the computational domain, is discretized into a grid of cells that is called mesh. The core of CFD is the Navier-Stokes (N-S) equations which describe the motion of fluids (Sayma, 2009) and are solved for each cell in the mesh. The N-S equations primarily consist of the conservation equations for mass and momentum, which are solved for all flow problems in FLUENT.

Conservation of mass:

$$-\frac{\partial \rho}{\partial t} = \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} \qquad (1)$$

Conservation of momentum:

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = -\frac{\partial P}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) + \mathrm{F}x \qquad (2)$$

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = -\frac{\partial P}{\partial x} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) + \mathrm{F}y \qquad (3)$$

$$\rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = -\frac{\partial P}{\partial x} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) + Fz \qquad (4)$$

In these equations *u*, *v* and *w* are the velocity components in the *x*, *y* and *z* directions. ρ is the density, *p* is the pressure and μ is the dynamic viscosity.

Temperature distribution inside the muffler can be studied using the following energy equation. Conservation of energy:

$$\rho\, c_p\left(\frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z}\right) = \Phi + \frac{\partial}{\partial x}\left[k\frac{\partial T}{\partial x}\right] + \frac{\partial}{\partial y}\left[k\frac{\partial T}{\partial y}\right] + \frac{\partial}{\partial z}\left[k\frac{\partial T}{\partial z}\right] + \left(u\frac{\partial p}{\partial x} + v\frac{\partial p}{\partial y} + w\frac{\partial p}{\partial z}\right) \qquad (5)$$

Where C_p is the specific heat at constant pressure, T is the temperature and Φ is the dissipation function. As the N-S equations are discretized and solved for each cell, flow variables, such as velocity and temperature, will be obtained at the discrete locations.

Besides sound pressure the flow velocity and the gas temperature are known from the wave calculation. Thus the flow generated noise can be calculated with the empirical formula from Green and Smith [30]. According to this formula the sound power level (L_w) depends on the flow velocity (V), gas temperature (T), diameter of the inlet tube (D) and the efficiency factor of the rear muffler (E_w).

$$L_w = E_w - 17.5*\log(T) + 20*\log(D) + 45*\log(V) + 1.87$$

3-5 Otto Cycle

The cycle of a four-stroke, SI, naturally aspirated engine is approximated by the air-standard cycle shown in fig.20 for analysis. This ideal air-standard cycle is called an Otto cycle**,** named after one of the early developers of this type of engine. The intake stroke of the Otto cycle starts with the piston at TDC and is a constant-pressure process at an inlet pressure of one atmosphere (process 6-1). This is a good approximation to the inlet process of a real engine at WOT, which will actually be at a pressure slightly less than atmospheric due to pressure losses in the inlet air flow. The temperature of the air during the inlet stroke is increased as the air passes through the hot intake manifold. The temperature at point 1 will generally be on the order of 25° to 35°C hotter than the surrounding air temperature. The second stroke of the cycle is the compression stroke, which in the Otto cycle is an isentropic compression from BDC to TDC (process 1-2). This is a good approximation to compression in a real engine, except for the very beginning and the very end of the stroke. In a real engine, the beginning of the stroke is affected by the intake valve not being fully closed until slightly after BDC. The end of compression is affected by the firing of the spark plug before TDC. Not only is there an increase in pressure during the compression stroke, but the temperature within the cylinder is increased substantially due to compressive heating. The compression stroke is followed by a constant-volume heat input process 2-3 at TDC. This replaces the combustion process of the real engine cycle, which occurs at close to constant-volume conditions. In a real engine combustion is started slightly bTDC, reaches its maximum speed near TDC.

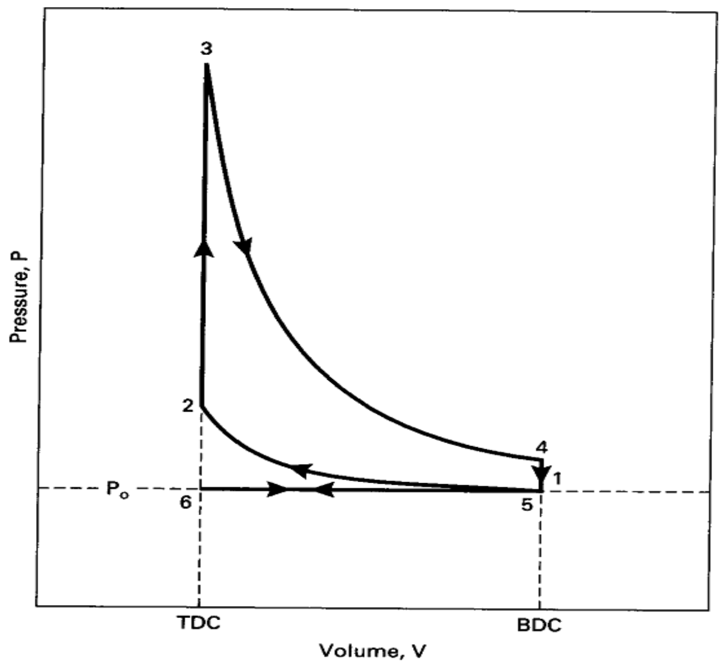


Figure 20. P-V diagram of an Ideal Air-standard Otto Cycle [31]

3.6. Exhaust Process

The exhaust process consists of two steps: blow down and exhaust stroke. When the exhaust valve opens near the end of the expansion stroke the high temperature gases are suddenly subjected to a pressure decrease as the resulting blow down occurs. A large percentage of the gases leave the combustion chamber during this blow down process, driven by the pressure differential across the open exhaust valve. When the pressure across the exhaust valve is finally equalized, the cylinder is still filled with exhaust gases at the exhaust manifold pressure of about one atmosphere. These gases are then pushed out of the cylinder through the still open exhaust valve by the piston as it travels from BDC to TDC during the exhaust stroke. Temperature of the gases in the exhaust system of a typical SI engine will average 400^{o}c to 600^{o}c. This drops to about 300^{o}c to 400^{o}c at idle conditions and goes up to about 900^{o}c at maximum power. This is about 200^{o}c to 300^{o}c cooler than the exhaust gases in the cylinder when the exhaust valve opens. The difference is because of expansion cooling. All temperatures will be affected by the equivalence ratio of the original combustion mixture.

The exhaust system is defined as the hardware necessary to vent the exhaust from the vehicle beginning at the exhaust plane defined by the engine manufacturer and necessary to isolate the exhaust thermally from vehicle structures. The virtual design of the exhaust muffler, as a minimum, include an accurate estimate of space required for the exhaust, backpressure to the engine, system weight, gas species distributions, gas temperature distributions, the interaction of the plume with external surfaces both on the vehicle and the ground, and the thermal interaction of the exhaust system with external surfaces through internal convection, conduction, and radiation **[**31**]**.

3.7. Air/Fuel Cycle Simulation of a Gasoline Engine

The following data are taken from SI automobile engine operating at WOT on four stroke air standard Otto cycle [31].

Assumptions:

- Air-standard Otto cycle
- Compression ratio, r =8 for gasoline engine
- Combustion efficiency, η_c = 100%
- Fuel, Isooctane: C_8H_{18}
- AF = 15
- LHV =44300KJ/Kg
- Ambient Condition:
  P1 = 100Kpa = 1bar
  T1 = 60^0c+273 = 333K
- Exhaust residual, $E_{Residual}$ = 4%
- Cp =1.108KJ/Kg.K
- Cv =0.821 KJ/Kg.K
- γ = Cp/Cv =1.35
- R = Cp-Cv =0.287KJ/Kg.K

Solution:

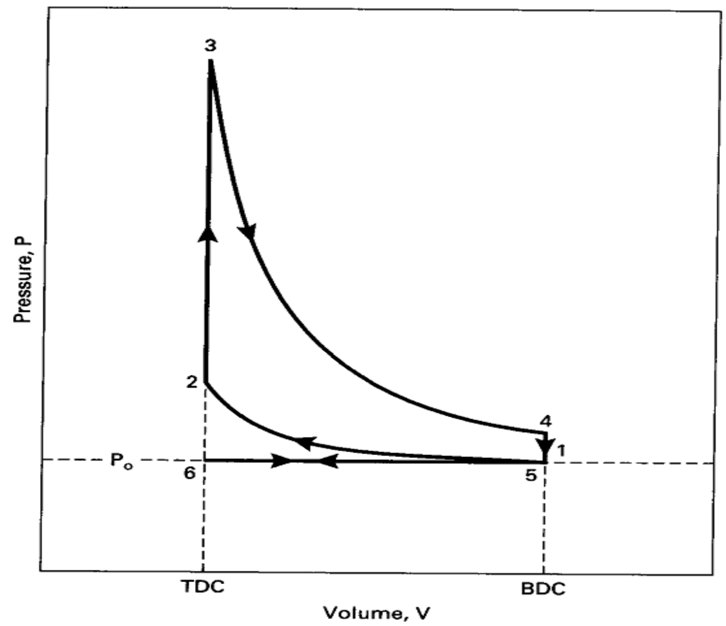


Figure 21. P-V Diagram of the Otto Cycles Process

Swept volume,

$V_s = V_d = 198.88 cm^3$

$r = \frac{Vs + v_c}{v_c}$

$8 = \frac{vs+vc}{vc}$ => $Vc = V_2 = 0.0000284m3$

$V_1 = V_d + V_c = 0.00022728\ m^3$

i. Pressures and temperatures at all points :

Adiabatic compression process 1-2:

Mass of gas mixture in cylinder can be calculated at state 1. The mass within the cylinder will then remain the same for the entire cycle.

$m_m = P_1 * V_1 / R * T_1 = 0.00024Kg$

$\frac{T_2}{T_1} = (\frac{V_1}{V_2})\gamma - 1$

$T_2 = 689.5K$

Also,

$P_1{V_1}^{\gamma} = P_2{V_2}^{\gamma}$

$P_2 = 16.564$ bar

Constant volume process 2-3:

Heat added during the process,

The mass of gas mixture m_m in the cylinder is made up of air m_a, fuel m_f, and exhaust residual m_{ex}:

Mass of air	$m_a = (15/16)*(0.96)*(0\ .00024)$	$= 0.000216$ kg
Mass of fuel	$m_f = (1/16)*(0.96)*(0.00024)$	$= 0.0000144$ kg
Mass of exhaust	$m_{ex}= (0.04)*(0.00024)$	$= 0.0000096$ kg
Total		$= 0.00024$ kg

$Q_{in}= m_f* Q_{LHV} *\eta_c = m_m *C_v*(T_3-T_2)$

$(0.0000144)*(44300)*(1.00) = (0.00024)*(0.821)*(T_3 - 689.5K)$

$T_3 = 3927$ K $= 3654^0$c $= T_{max}$

$$\frac{P_2}{T_2} = \frac{P3}{T3}$$

$P_3 = 94.34$ bar $= P_{max}$

Adiabatic Expansion process 3-4:

$$\frac{T_3}{T_4} = (\frac{V_4}{V_3})^{\gamma-1} = r^{\gamma-1}$$

$T_4 = 1896.6$K

$P_3{V_3}^{\gamma} = P_4{V_4}^{\gamma}$

$P_4 = 5.695$bar

Table 4. Initial Values for Velocity Inlet as the Inlet Boundary Condition.

Area(m2)	0.18
Temperature(K)	1173
Viscosity(kg/ms)	2.7e-05
Enthalpy(J/kg)	749575.3
Density(kg/m3)	0.696
Length(mm)	460
Velocity(m/s)	68
Ratio of specific heats	1.35

3.8. Geometry of Modeled Muffler

The models are designed on Solid Work and imported to ANSYS work bench for the result analysis and simulation. The models designed are original and modified model. After the construction of exhaust muffler model in Solid Works it is analyzed in ANSYS FLUENT 15.0. The FLUENT analysis is carried out by considering inlet boundary condition i.e velocity inlet.

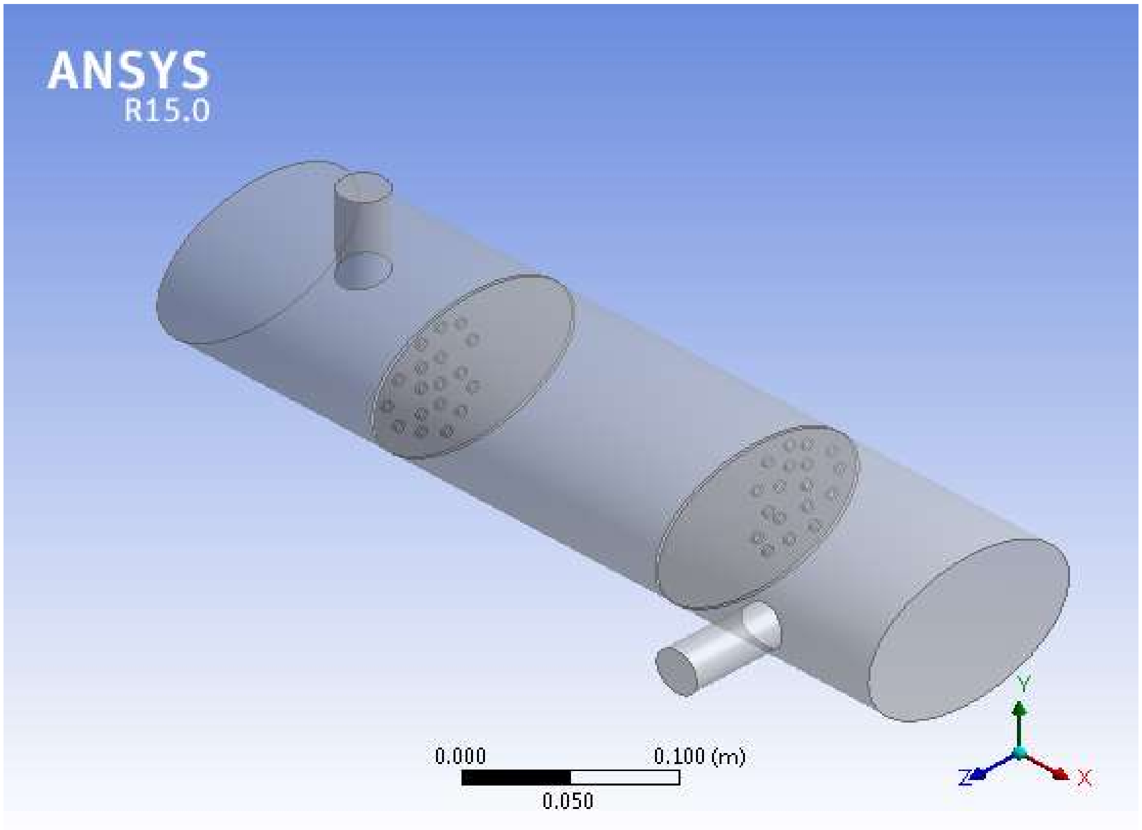


Figure 22. 3D Base Model of the Reflective/Reactive Muffler

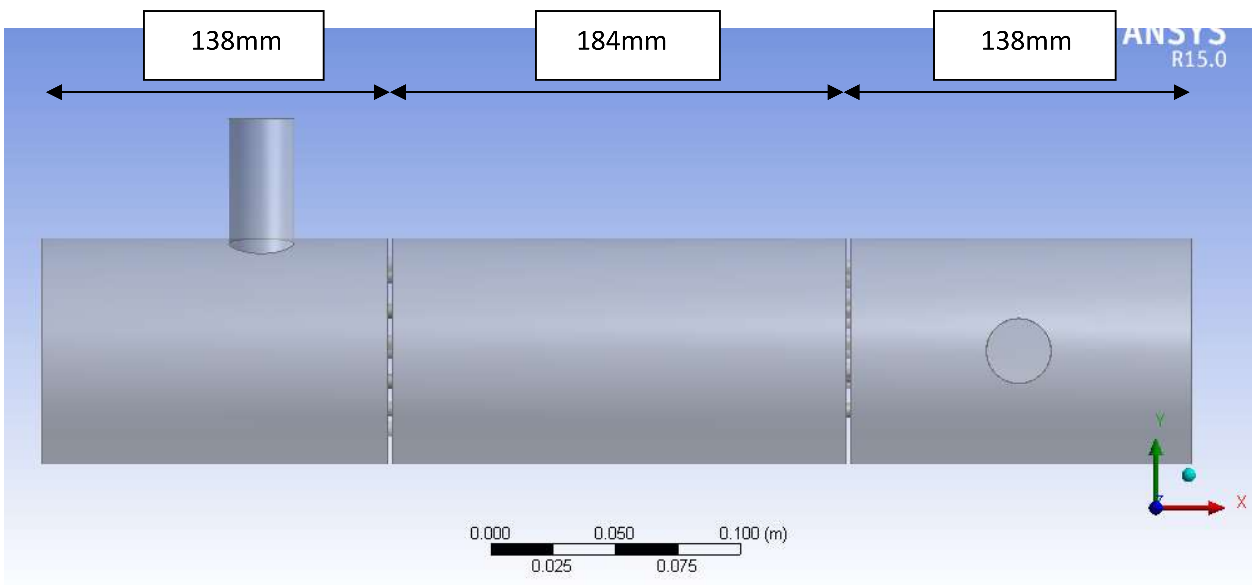


Figure 23. 2-Dimensional View of the Base Model Muffler

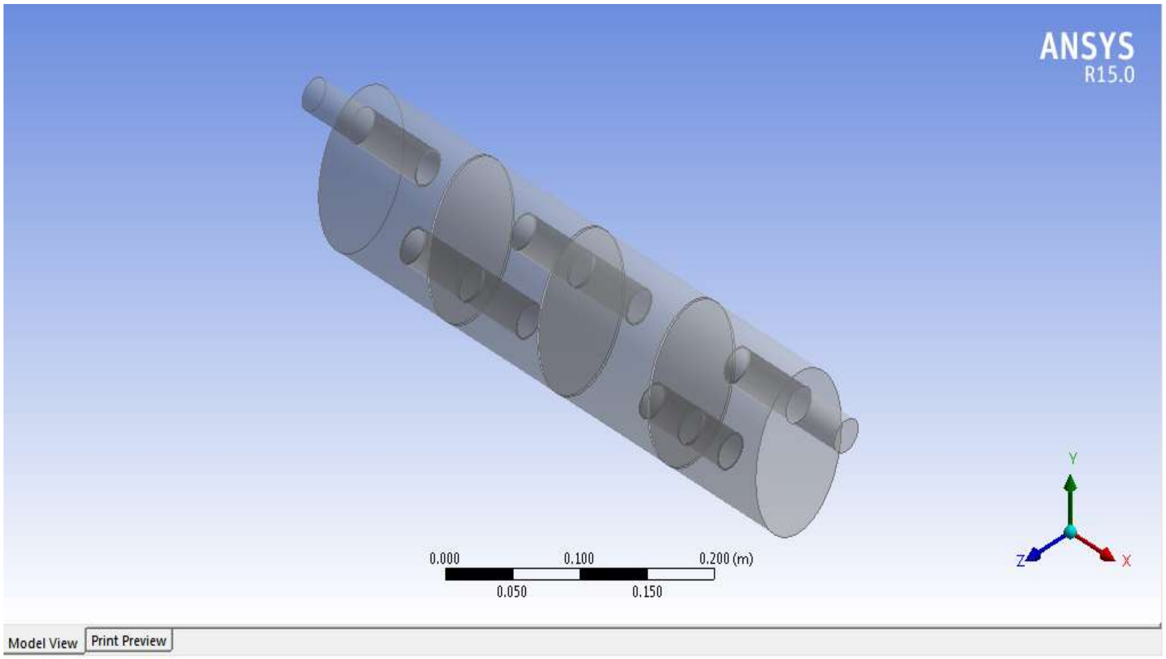


Figure 24. Internal View of Designed Muffler (3D model)

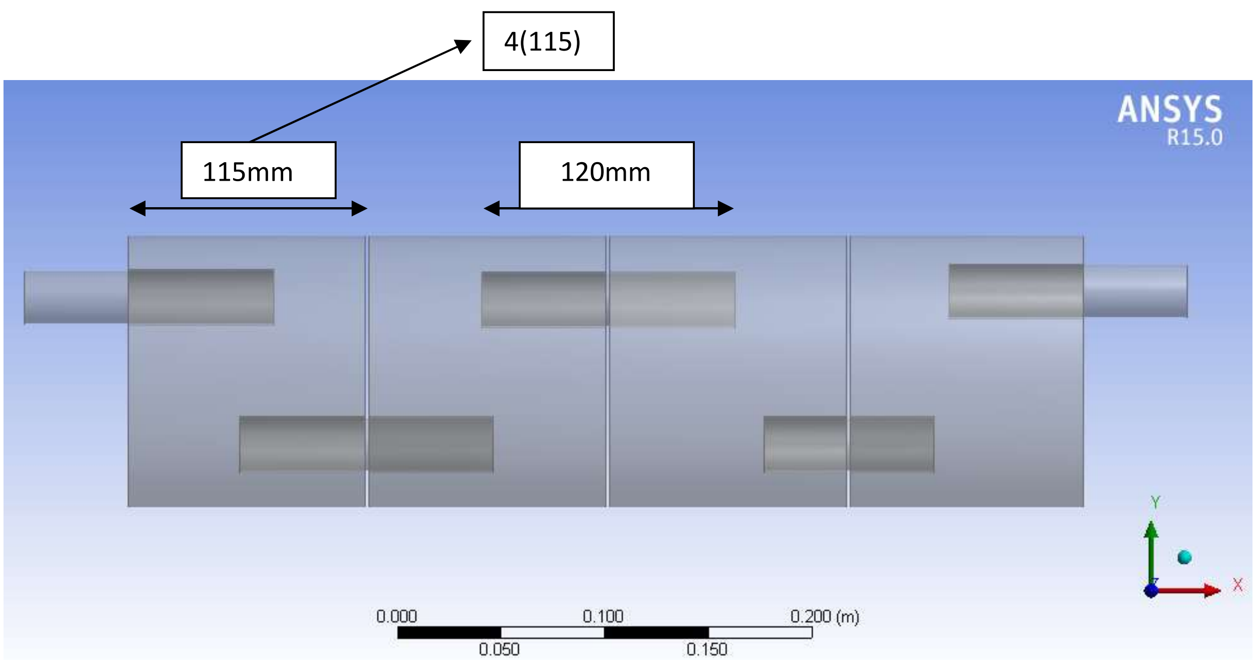


Figure 25. **2-Dimensional View of the Modified Model Muffler**

3.9. Grid generation

It is important to note that the solution is the numerical solution to the problem that you posed by defining the mesh and boundary conditions. The more accurate your mesh and boundary conditions, the more accurate your "converged" solution will be. Mesh generation is an essential step in computational simulations requiring high quality meshes to accurately capture the complex physical phenomena. High quality and high density meshes are required to accurately capture the complex physical phenomena but it is computationally expensive. For grid generation ANSYS meshing is used. The selection of element is based on the grid independence test therefore to make assure it. Sizing of the models are show in the table below.

Table 5. Mesh Sizing of Base Model Muffler

Sizing	
Use Advanced Size Function	On: Proximity and Curvature
Relevance Center	Fine
Initial Size Seed	Active Assembly
Smoothing	High
Transition	Slow
Span Angle Center	Fine
Curvature Normal Angle	12.0 °
Num Cells Across Gap	1
Min Size	7.5e-004 m
Proximity Min Size	7.5e-004 m
Max Face Size	2.5e-003 m
Max Size	7.5e-003 m
Growth Rate	1.20
Minimum Edge Length	1.885e-002 m

Table 6. Mesh Statistics of the Base Model Muffler

Statistics	
Nodes	229435
Elements	1226322
Mesh Metric	Skewness
Min	4.28935313687706E-05
Max	0.796006207158784
Average	0.231441112212359
Standard Deviation	0.120273547827075

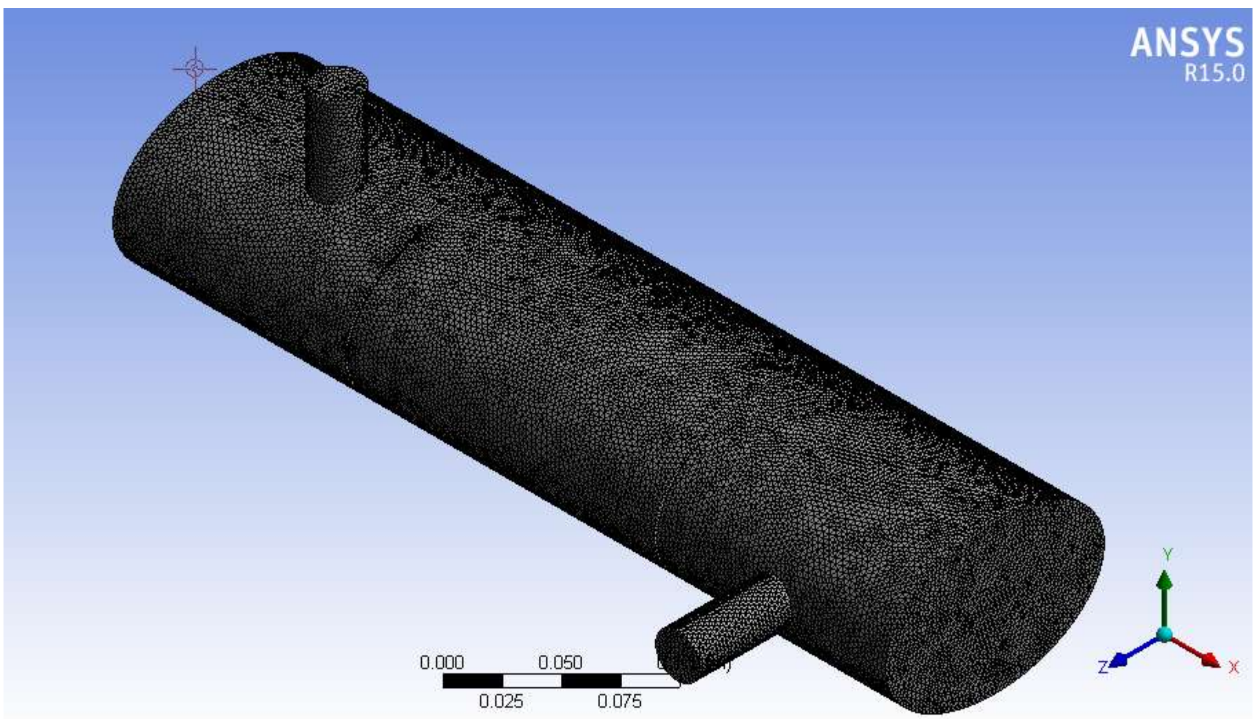


Figure 26. 3D Mesh of the Base Model Muffler

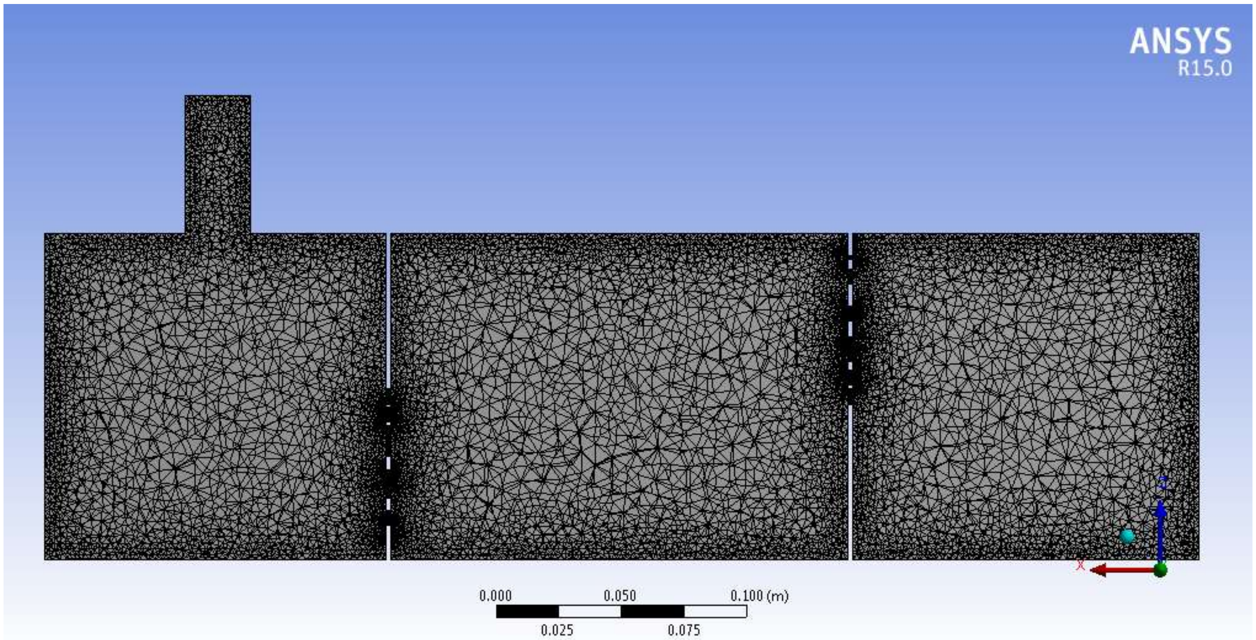


Figure 27. Sectioned Mesh of the Base Model Muffler

Table 7. Mesh Sizing of the Modified Model

Use Advanced Size Function	On: Proximity and Curvature
Relevance Center	Medium
Initial Size Seed	Active Assembly
Smoothing	High
Transition	Slow
Span Angle Center	Fine
Curvature Normal Angle	Default (18.0 °)
Num Cells Across Gap	Default (3)
Min Size	Default (1.4501e-004 m)
Proximity Min Size	Default (1.4501e-004 m)
Max Face Size	Default (1.4501e-002 m)
Max Size	Default (2.9002e-002 m)
Growth Rate	Default (1.20)
Minimum Edge Length	7.854e-002 m

Table 8. Mesh Statistics of the Modified Model Muffler

Statistics	
Nodes	266919
Elements	1483190
Mesh Metric	Skewness
Min	2.55670073093794E-05
Max	0.799780611041817
Average	0.23728144687665
Standard Deviation	0.122770662062964

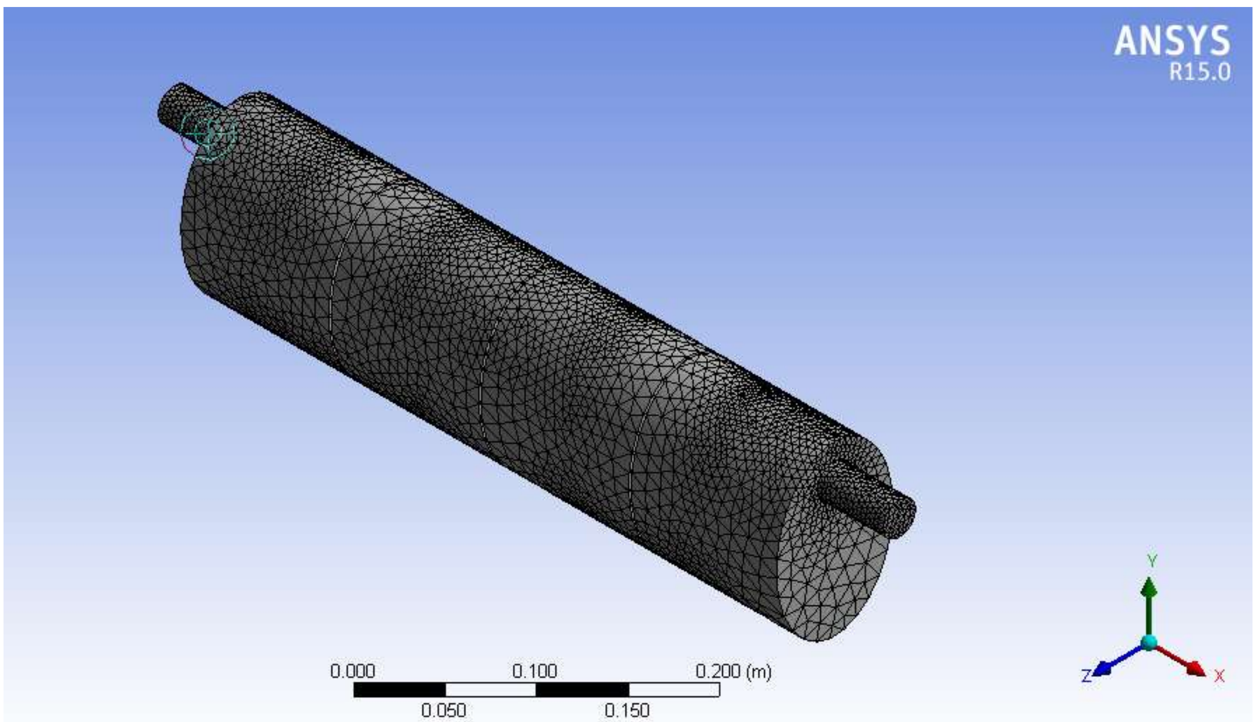


Figure 28. 3D Mesh of the Modified Model Muffler

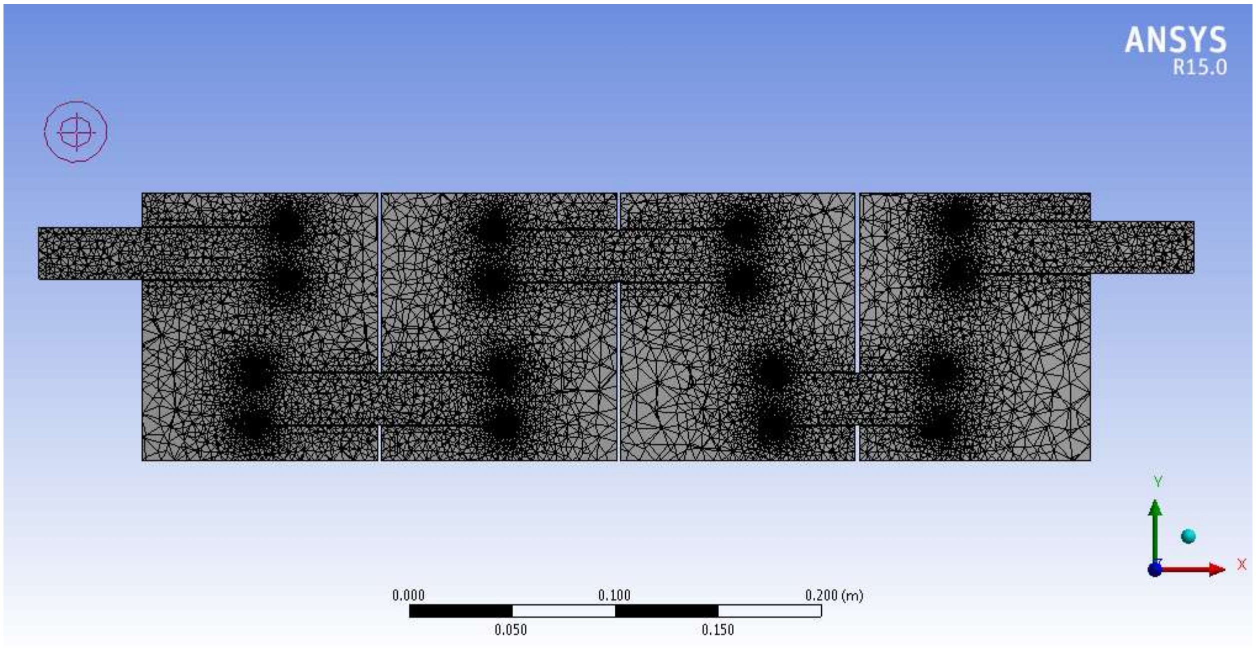


Figure 29. Internal Mesh of the Modified Model

3.10. Grid independence test

Grid convergence is the term used to describe the improvement of results by using successively smaller cell sizes for the calculations. A calculation will approach the correct result as the mesh becomes finer, hence the term grid convergence. As the grid is refined (grid size become smaller and the number of cells in the flow domain increase) and the time step is refined (reduced). The spatial and temporal discretization errors, respectively, should asymptotically approach zero, excluding computer round-off error.

3.11. Grid Convergence Index (GCI)

The GCI is a measure of the percentage the computed value is away from the value of the asymptotic numerical value. It indicates an error band on how far the solution is from the asymptotic value. It indicates how much the solution would change with a further refinement of the grid. A small value of GCI indicates that the computation is within the asymptotic range. Roaches (1994) suggests a grid convergence index GCI to provide a consistent manner in reporting the results of grid convergence studies and perhaps provide an error band on the grid convergence of the solution.

The GCI can be computed using two levels of grid; however, three levels are recommended in order to accurately estimate the order of convergence and to check that the solutions are within the asymptotic range of convergence. One must recognize the distinction between a numerical result which approaches an asymptotic numerical value and one which approaches the true solution. It is hoped that as the grid is refined and resolution improves that the computed solution will not change much and approach an asymptotic value (i.e. the true numerical solution). There still may be error between this asymptotic value and the true physical solution to the equations.

Roaches has provided a methodology for the uniform reporting of grid refinement studies. The basic idea is to approximately relate the results from any grid refinement test to the expected results from a grid doubling using a second-order method. The GCI is based upon a grid refinement error estimator derived from the theory of generalized Richardson Extrapolation. It is recommended for use whether or not Richardson Extrapolation is actually used to improve the accuracy, and in some cases even if the conditions for the theory do not strictly hold. The object is to provide a measure of uncertainty of the grid convergence. A grid independence test was performed on the unmodified and modified geometry and the total pressure outlet and convergence time was selected as the criteria.

Table 9. Grid Independence Test for Base Model Muffler

Model	Base1	Base2	Base3
Cells No.	336195	780054	1226322
$Ptot_{max}$ (pascal)	4926.518	5391.47	5453.35
Convergence time	4hours	9 hours	16hours

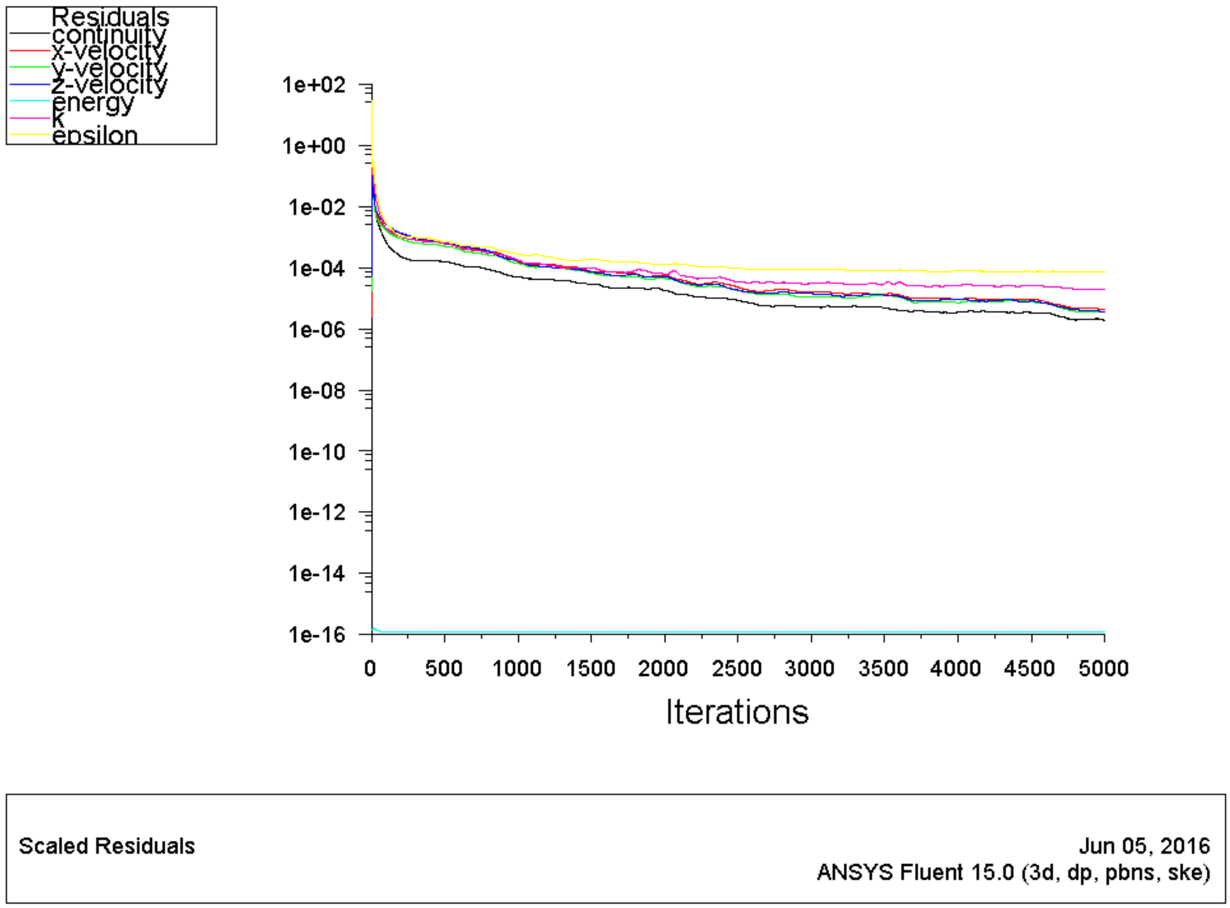


Figure 30. Scaled Residual Convergence History of Base Model Muffler

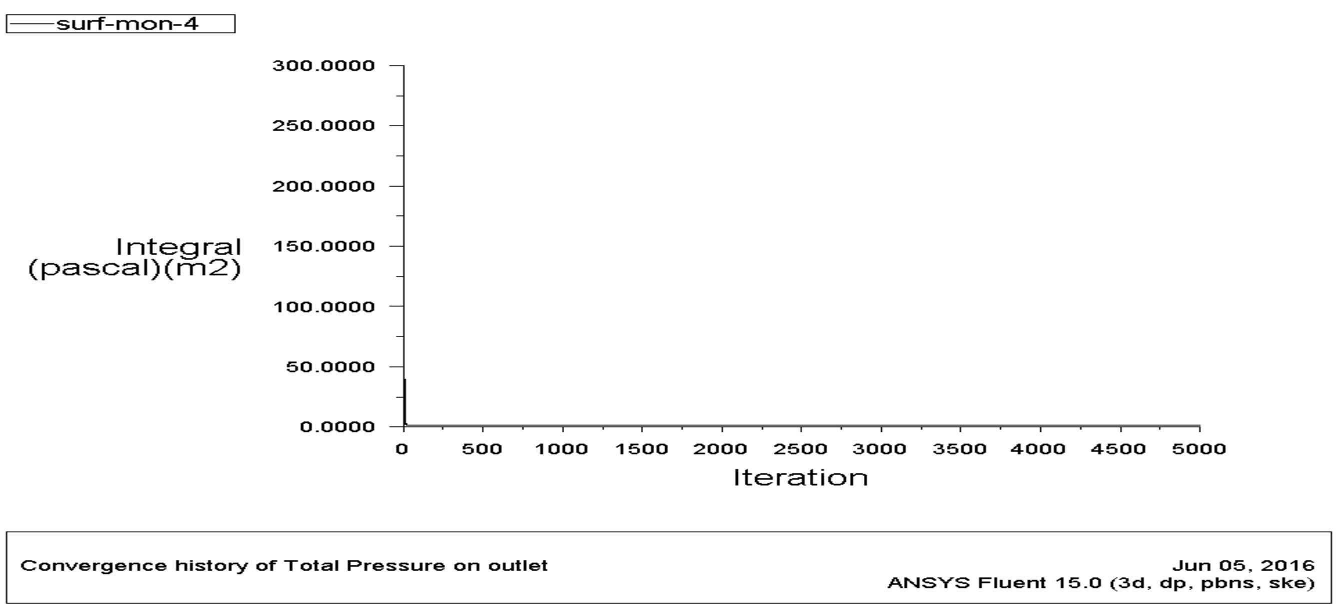


Figure 31. Total Pressure on Outlet Convergence History of Base Model

Table 10. Grid independence test for Modified Model

Model	Model1	Model2	Model3
Cells No.	1043829	1118268	1483190
$Ptot_{outlet}$ (pascal)	3511.958	3462.056	3419.204
Convergence time	6hours	8 hours	12hours

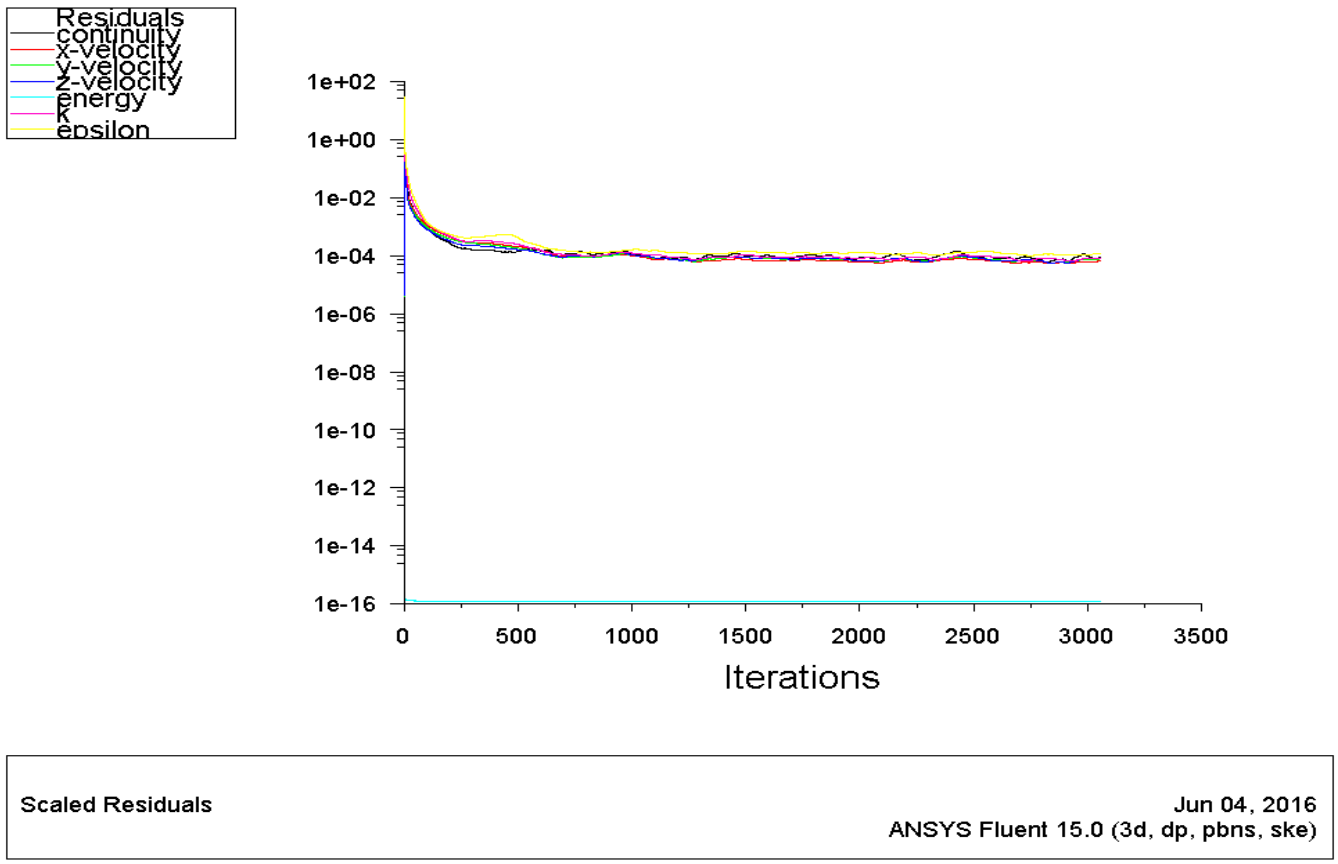


Figure 32. Scaled Residual Convergence History of Modified Model Muffler

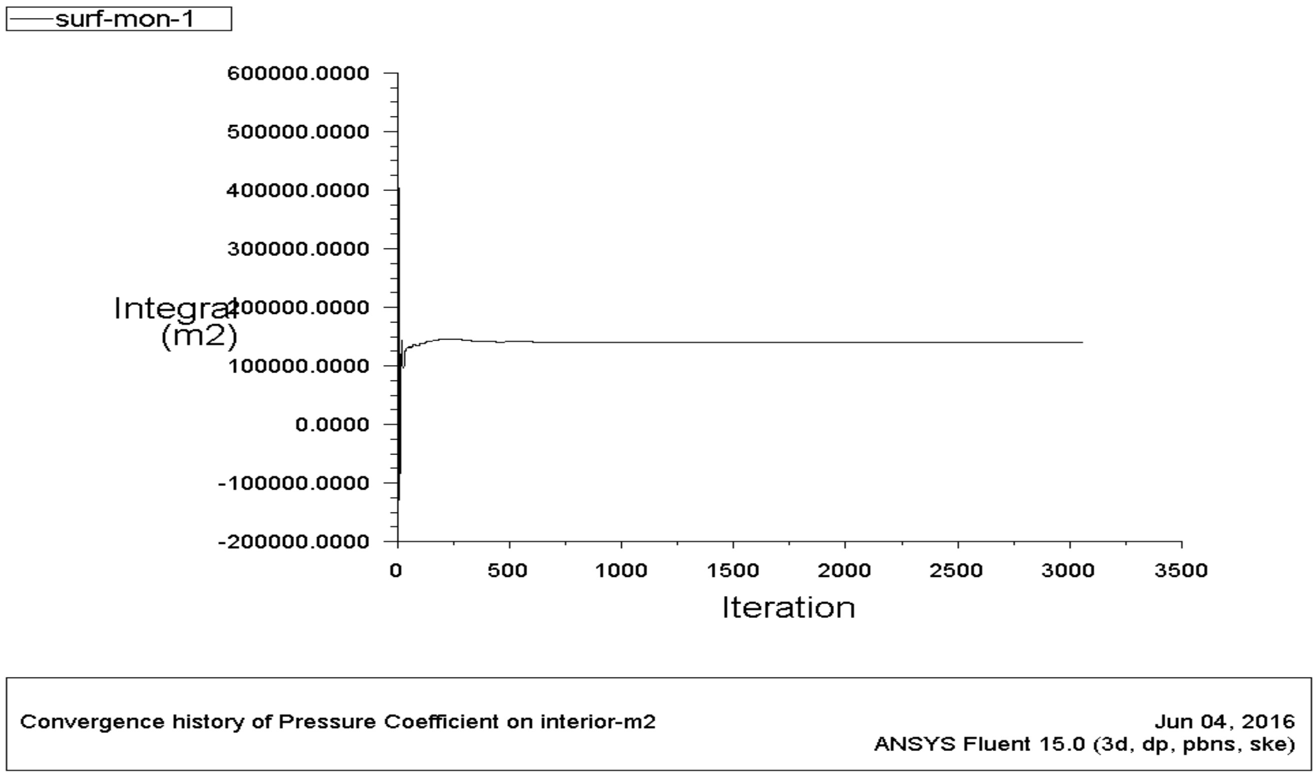


Figure 33. Pressure Coefficient Convergence History of Modified Model

Referring to the results in tables shown above, as the mesh became finer; the total pressure at the outlet reached an asymptotic value. Balance between calculation, time and the accuracy order of the simulation has been made and the setting for the fine grid is considered to be sufficiently reliable. Therefore, the finest models are taken from both models i.e Base model (Base3) and Modified model (Model3) which they are having longest convergence time and largest number of cells.

3.12. The k-epsilon model

One of the most prominent turbulence models, the *k-ε* (*k*-epsilon) model, has been implemented in most general purpose CFD codes and is considered the industry standard model. It has proven to be stable and numerically robust and has a well established regime of predictive capability. For general purpose simulations, the *k-ε* model offers a good compromise in terms of accuracy and robustness.

CHAPTER FOUR

RESULTS AND DISCUSSION

From figure 33 it has been observed that for a velocity inlet boundary condition in base model, the exhaust from the engine enters the muffler at a velocity of 68m/s and increases to a magnitude of about 101 m/s in the first expansion chamber once it entering the second chamber. This is observed as a result of decrease in the flow area the pressure increases and subsequently the velocity increases. After this the gases get spread in the chamber and they enter the next chamber. During entry its velocity again become maximum of about 101m/s and then on hitting the wall they expand and the gases enter the outlet pipe and come out of the exhaust at an increased speed of about 91m/s. Also from figure 35 we observe that, the gases are having maximum pressure of 21133.59Pascal at inlet and expand. After it is expanded it enters the second chamber at a decreased pressure of about 13700Pascal and more decreased to about 12500bar. In the second chamber gases get spread and enters third chamber again at a pressure of about 13700Pascal. As it is expanded it spread inside the third chamber by pressure of almost about 7570Pascal and leaves exhaust pipe at an outlet average pressure of around 5453.347 Pascal.

For a velocity inlet boundary condition in modified model as evident from figure 40 it has been observed that the gases at the inlet pipe is about a pressure of 24057.14Pascal and on hitting the wall at the front of the first chamber by the pressure intensity coming from the inlet pipe and spread in this chamber by the pressure of about 22000Pascal. The flow of the gases is more or less similar as that in base model but due to the increased numbers of chambers uniform distribution of pressure was observed in the each chamber. The gas flows to the second chamber at pressure of about 19800Pascal and this pressure is uniformly distributed in this chamber. The pressure is decreased to about 11100Pascal inside the third chamber and it is more decreased to about 3890Pascal in the fourth chamber. The gases come out of the outlet pipe at a speed of 53.8 m/s and a pressure of 3419.204Pascal. Even though base model displayed a similar behavior, there was a more significant drop in the pressure of the exhaust gases in the modified model than base model. The drop in the pressure of the exhaust gases in the base model was about 43.2%

whereas the drop in the modified model was 56.8%. The reduction of pressure from inlet to outlet of base and modified muffler is shown in the table below.

Table 11.Total Pressure Reduction in Each Model

Models	Inlet P_{Total}(Pascal)	Outlet P_{Total}(Pascal)	P_{Total} Reduction
Base	21133.59	5453.347	15680.243
Modified	24057.14	3419.204	20637.936

Table 12. Acoustic Power Level in Each Model

Models	Inlet L_w(dB)	Outlet L_w(dB)
Base	63.3	114
Modified	68.2	98.5

4.1. Analysis of base model

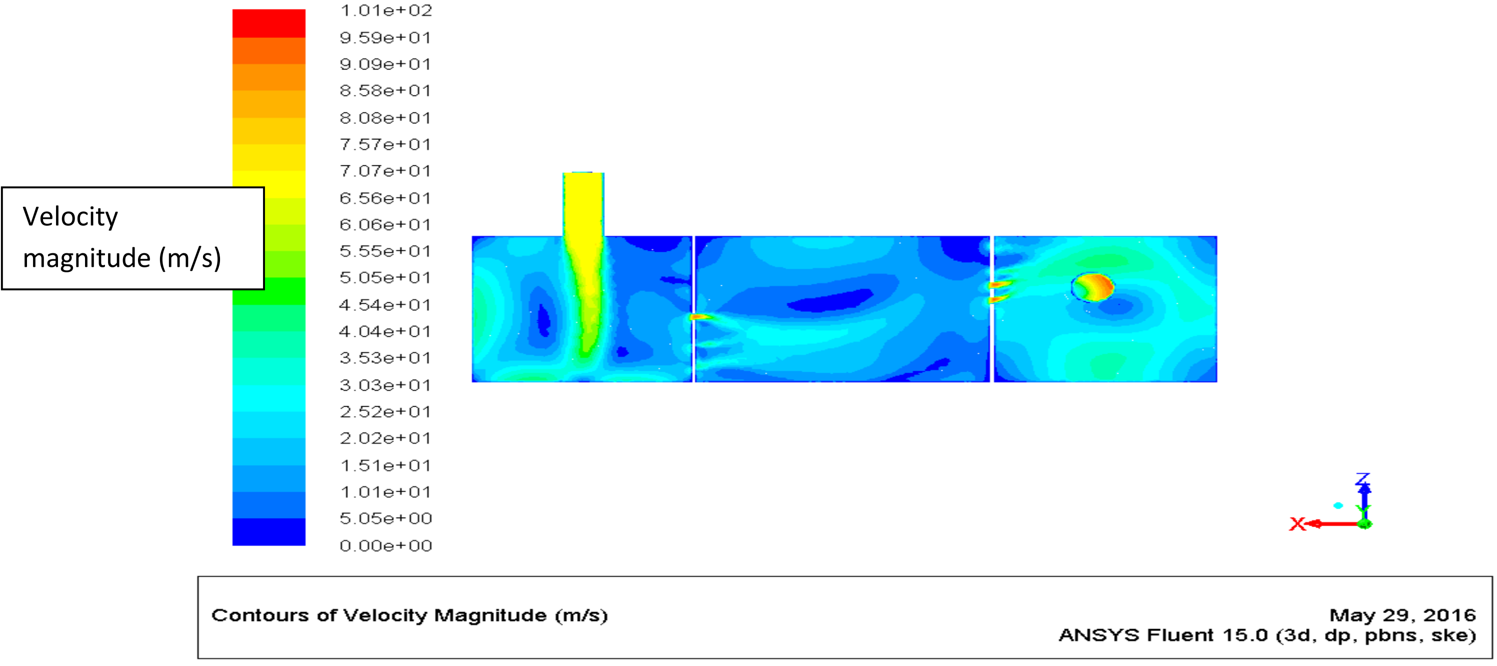


Figure 34. Velocity Distribution Cut Plot in the Base Model Muffler

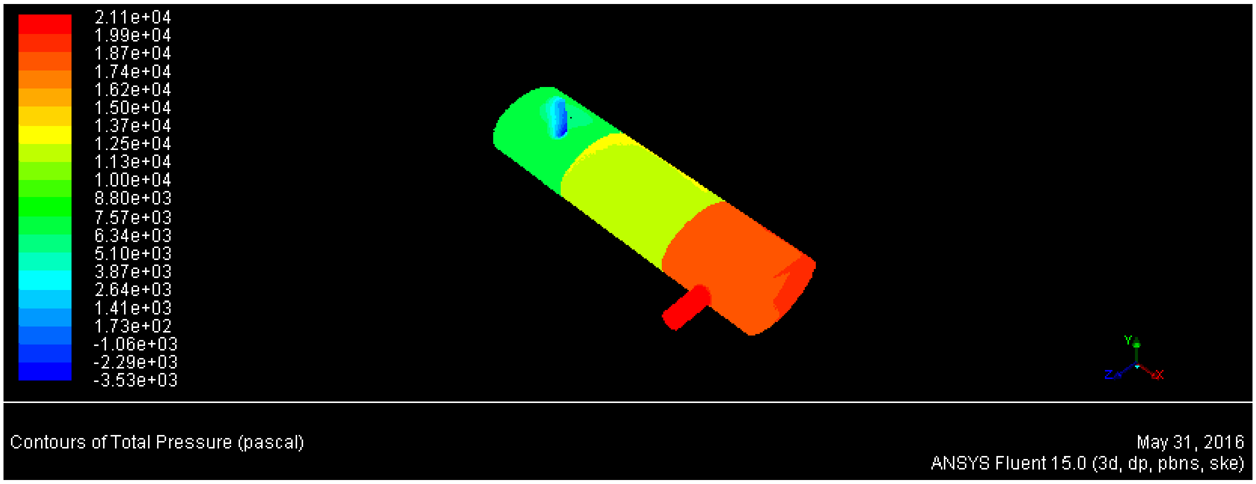


Figure 35. Total Pressure Distribution interior the Base Model Muffler

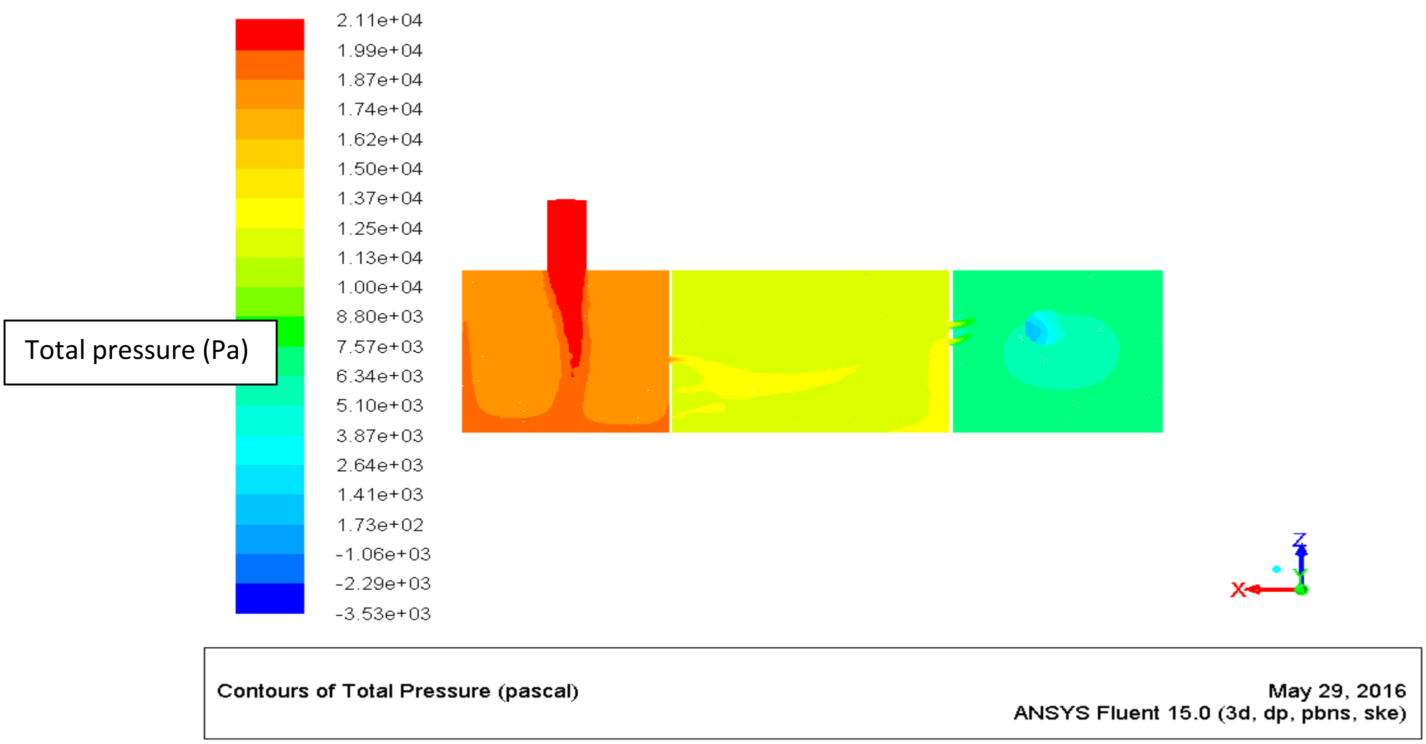


Figure 36. The Total Pressure Distribution in the Meridian Section of the Base Model Muffler

Velocity vectors

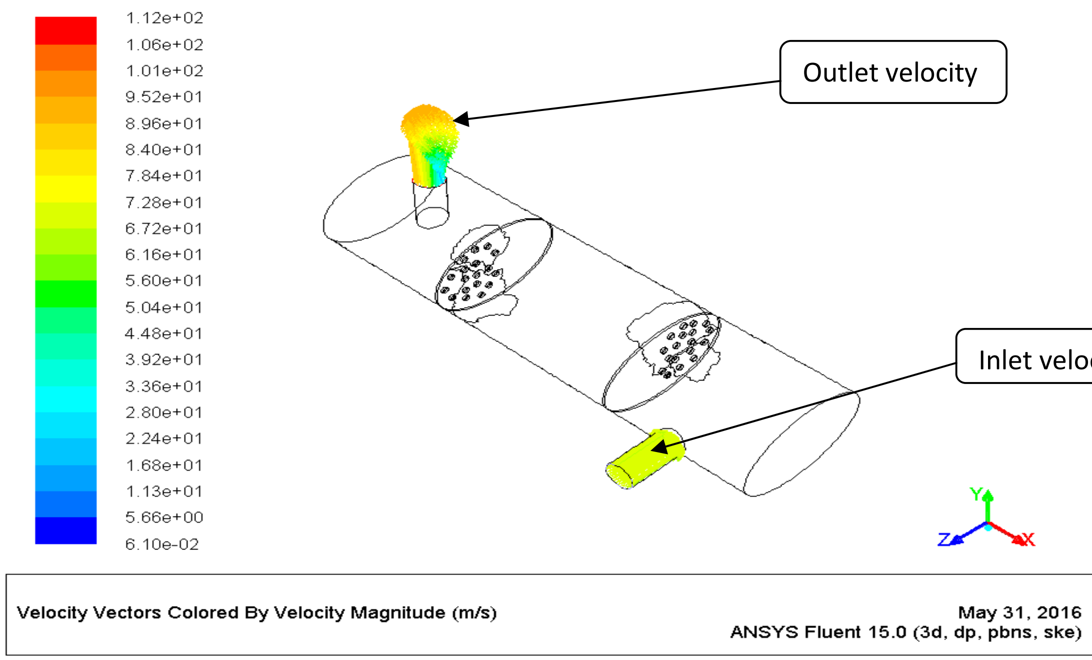


Figure 37. Base Model Muffler Inlet-Outlet Velocity Magnitude

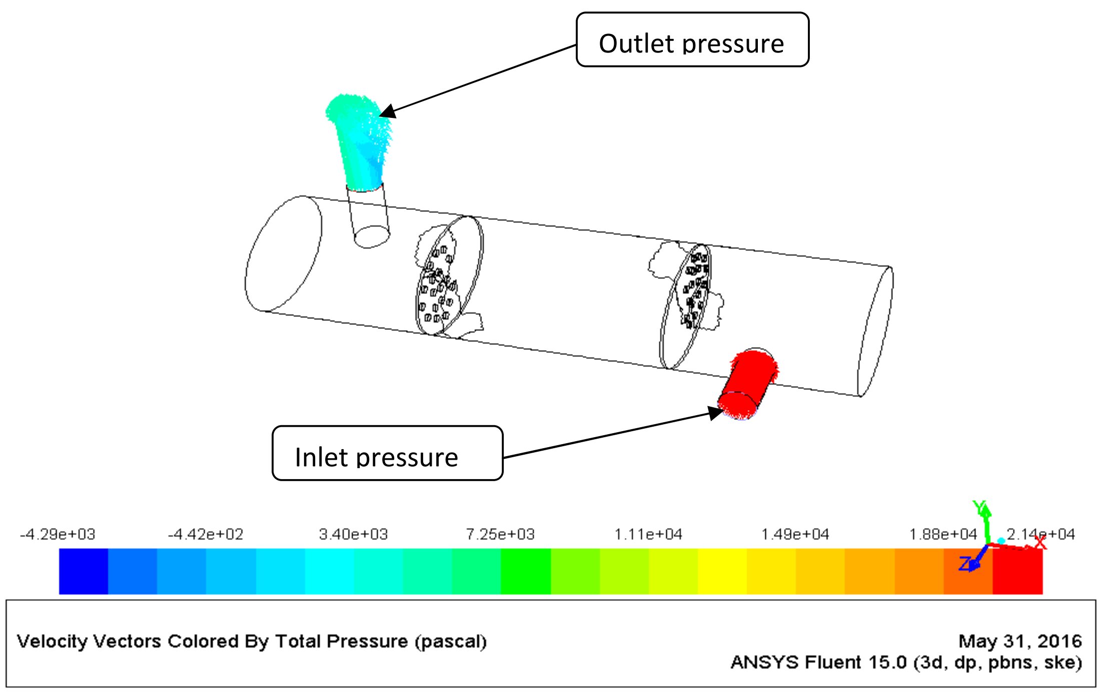


Figure 38. Base Model Muffler Inlet-Outlet Total Pressure

4.2. Analysis of modified model

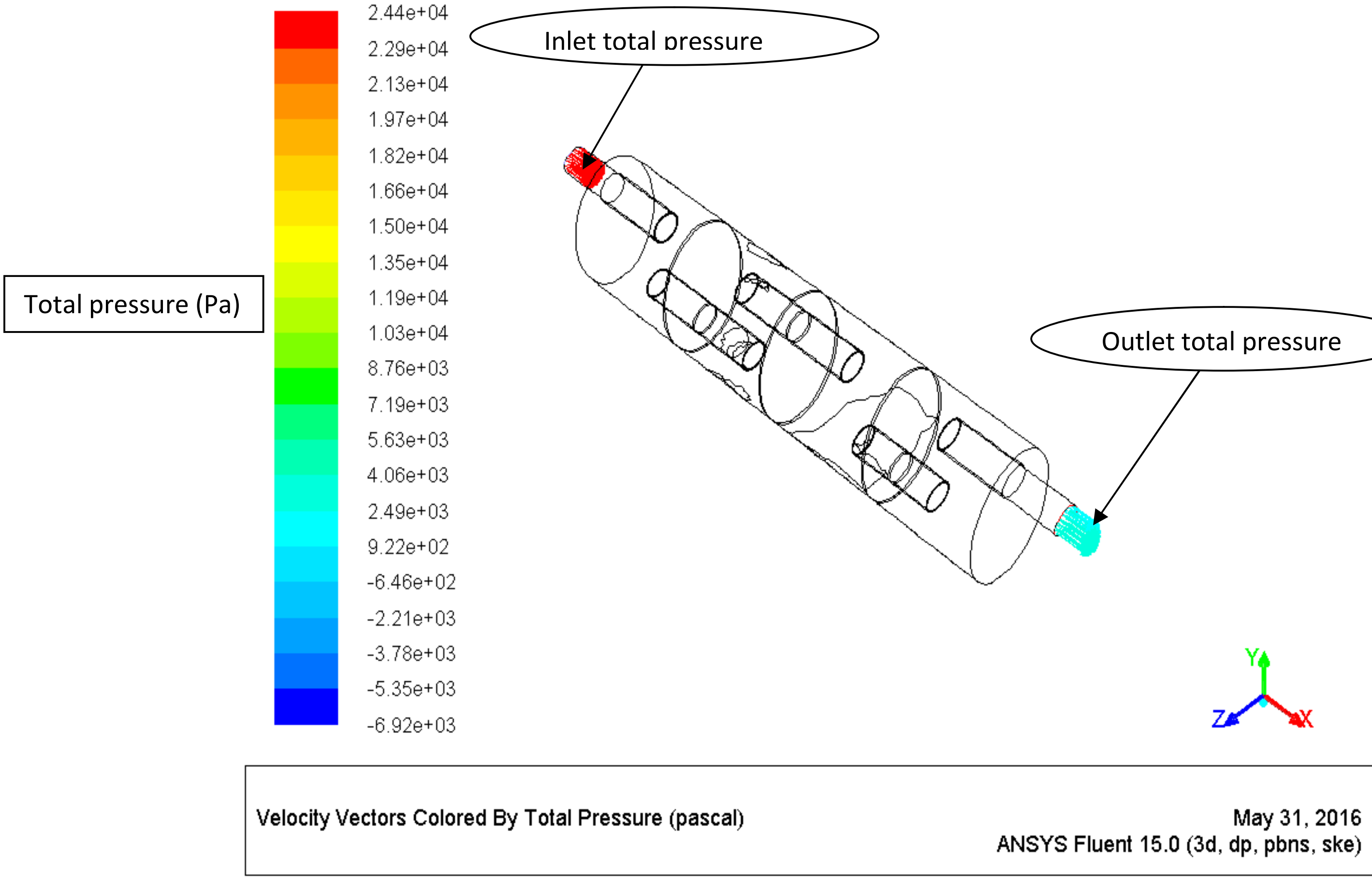


Figure 39. Inlet-Outlet Total Pressure of Modified Model Muffler

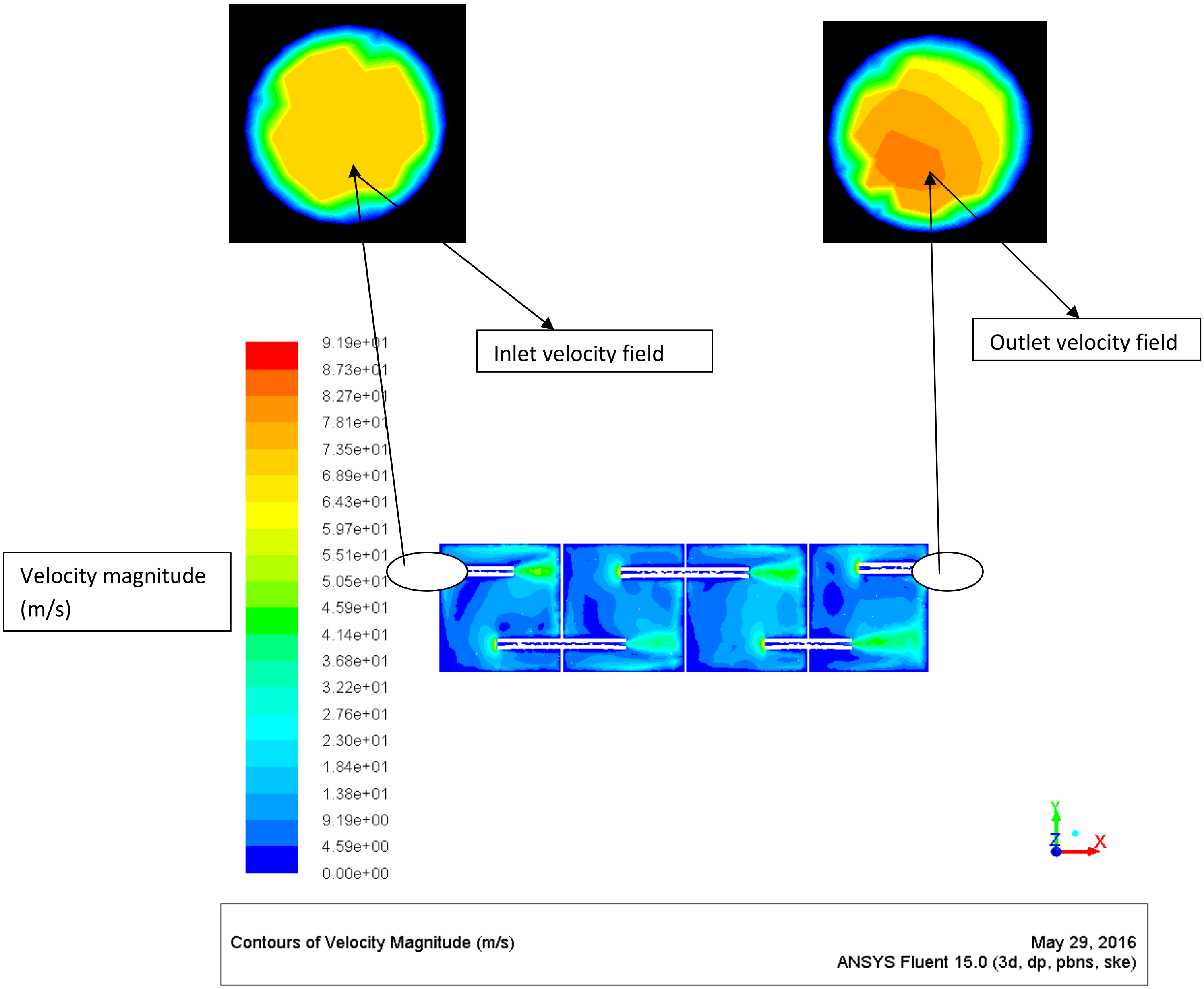


Figure 40. Velocity Distribution Cut Plot in the Modified Model Muffler

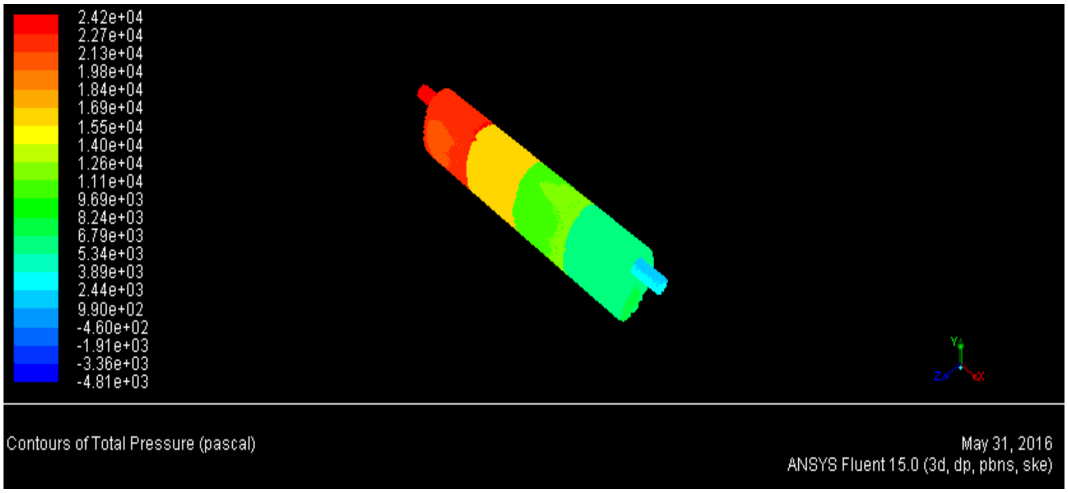


Figure 41. Total Pressure Distribution interior the Modified Model Muffler

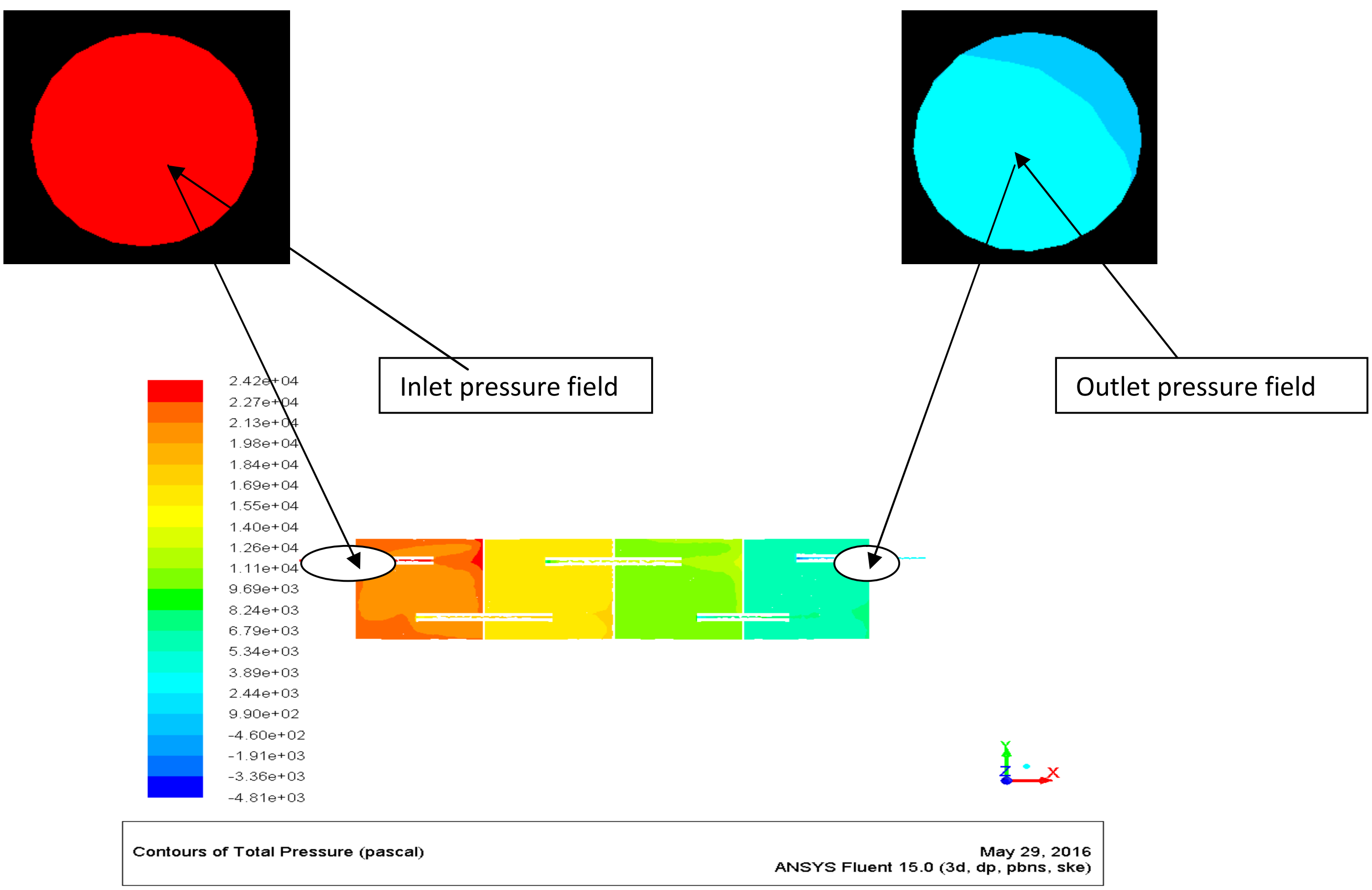
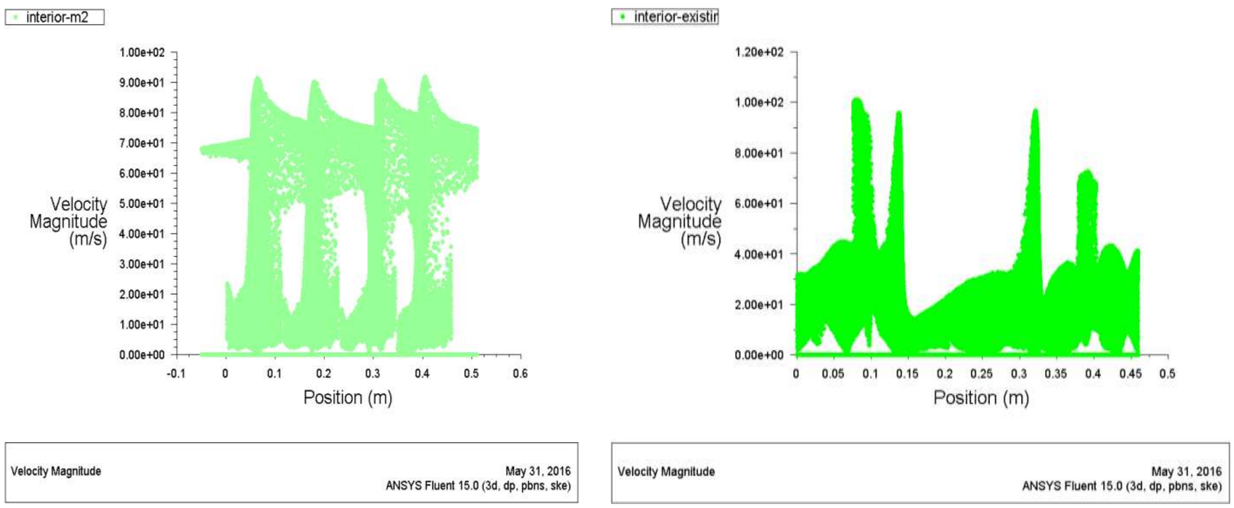


Figure 42. Velocity Distribution Cut Plot in the Modified Model Muffler

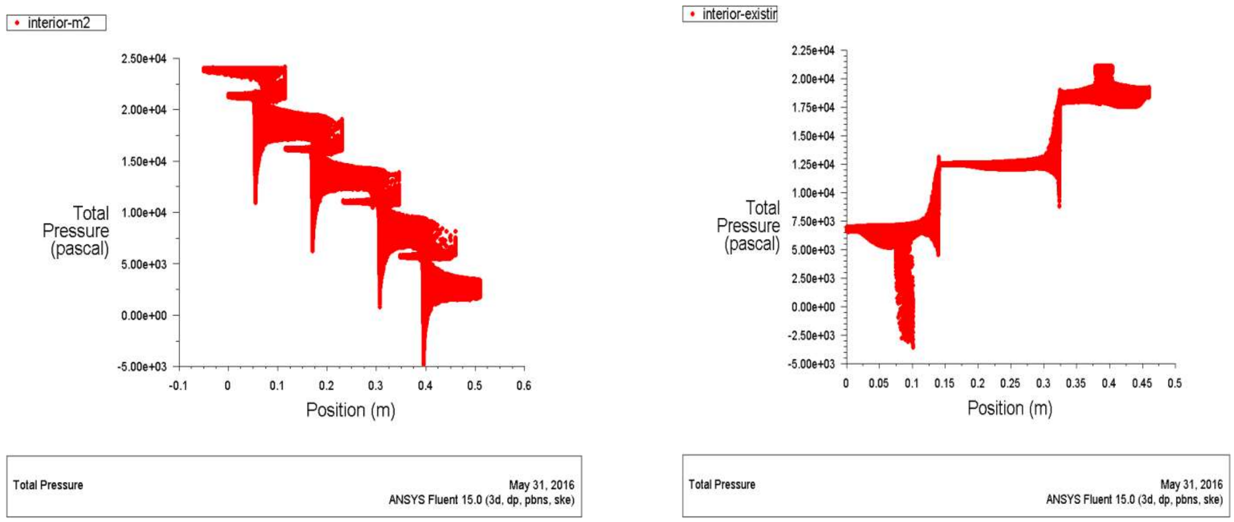
XY plot



(a) (b)

Figure 43. Variation of gas Velocity with the Location of (a) the Modified Model, (b) the Base Model

The velocity magnitude for modified model at the outlet pipe is about 73m/s whereas for base model the velocity increased to about 100m/s as shown in the figure above. Due to the reduction in the flow area the velocity of the gases increases and reaches a maximum intensity of 92 m/s at the transition region in modified model. In base model velocity reaches 98m/s at the transition position.



(a) (b)

Figure 44. Variation of gas Pressure with the Location of (a) the Modified Model, (b) the Base Model

Figure 43 shows total pressure distribution interior the muffler shell verses position for modified (a) and base model (b). For each model pressure drop occurred when the gas passes from one chamber to another chamber when it goes from inlet to outlet pipe.

CHAPTER FIVE

CONCLUSION AND RECOMMENDATION

5.1. Conclusion

In this work two different models of muffler have been designed for the engine output of an RE 4-stroke petrol engine (Bajaj) and the flow has been simulated using ANSYS FLUENT. The flow characteristics in the domain obtained through the simulation were reasonable. From the numerical study, we observe that though both the models have same similar design parameters, the modified model was more effective in reducing the exhaust pressure than the base model. The numbers of chambers section are increased without increasing the shell length. Increased number of chambers increased the pressure reduction inside the modified model. Velocity inside the modified muffler is also reduced due to increased number of chambers sections.

1. Maximum velocity in base model is 101 m/s.

2. Maximum velocity in modified model is 92 m/s.

3. Exhaust pressure reduction in base model is 43.2%

4. Exhaust pressure reduction in modified model is 56.8%

The reduction in pressure of exhaust in the base model is 43.2% whereas the reduction in exhaust pressure in the modified model is 56.8%. Hence we are certain that the modified muffler model has lower acoustic noise level than the base muffler model.

5.2 Recommendation for future works

I recommend department of mechanical engineering or automotive industries and any interested individual to take further research on noise reduction of exhaust system especially on three wheeler vehicles since they are increasing in the cities particularly in Adama town. Hybrid type silencer can be used for more reduction of exhaust noise.

One of the major limitation of the current work is that, analysis of noise reduction using acoustic extension is not done because of unavailability of the software, therefore, in the future analysis of noise reduction using acoustic extension package can be done and further in noise reduction is possible by means of using hybrid type muffler, shape optimization and cost analysis can be done.

REFERENCES

[1] Erdem Özdemir, Rifat Yılmaz, Zeynep Parlar, Şengül Ar, AN ANALYSIS OF GEOMETRIC PARAMETERS' EFFECTS ON FLOW CHARACTERISTIC OF A REACTIVE MUFFLER, 17th International Research/Expert Conference "Trends in the Development of Machinery and Associated Technology" TMT 2013, Istanbul, Turkey, 10-11 September 2013.

[2] D. Tutunea, M.X. Calbureanu and M. Lungu. "The computational fluid dynamics (CFD) study of fluid dynamics performances of a resistance muffler". INTERNATIONAL JOURNAL OF MECHANICS

[3] Sileshi Kore, Abudlkadir Aman and Eddesa Direbsa, PERFORMANCE EVALUATION OF A REACTIVE MUFFLER USING CFD, Journal of EEA, Vol. 28, 2011.

[4] D. Tutunea, M.X. Calbureanu and M. Lungu, The computational fluid dynamics (CFD) study of fluid dynamics performances of a resistance muffler, INTERNATIONAL JOURNAL OF MECHANICS Issue 4, Volume 7, 2013.

[5] F.D. Denia, J. Albelda, and F.J. Fuenmayor, "Acoustic Behaviour of Elliptical Chamber Mufflers," Journal of Sound and Vibration, vol. 241, 2001, pp. 401-421.

[6] Francisco D. Denia, "acoustic behavior of elliptical mufflers With single-inlet and double-outlet", July 2003.

[7] Dong Wook Choi and Jin Woo Lee. "Optimal Design of a Muffler with Perforated PlatesConsidering Pressure Drop", Feb 15,2013.

[8] Abdullah a. dhaiban, m-emad s. soliman, and m.g. el-sebaie. "finite element simulation of acoustic attenuationperformance of elliptical muffler chambers", november 2011.

[9] Atul A. Patil,L.G. Navale,V.S. Patil. "Experimental Verification And CFD Analysis Of Single Cylinder Four Strokes C.I. Engine Exhaust System", Dec 2014.

[10] Erdem Özdemir, RifatYılmaz, ZeynepParlar, Şengül Arı. "An Analysis of Geometric Parameters' Effects On Flow Characteristic Of A Reactive Muffler", September 2013.

[11] Middelberg, J.M., Barber, T.J., Leong, S. S., Byrne, K.P and Leonardi, E. "Computational Fluid Dynamics Analysis Of The Acoustic Performance Of Various Simple Expansion Chamber Mufflers",Nov 2004.

[12] Puneetha C G. "Backpressure Study in Exhaust Muffler of Single Cylinder Diesel Engine using CFD Analysis", 2015.

[13] SudarshanDilipPangavhane et al. “Experimental and CFD Analysis of a Perforated Inner Pipe Muffler for the Prediction of Backpressure”, Nov 2013.

[14] Yunshi Yao. “Experiment and CFD Analysis of Reactive Muffler”, Sep 20, 2013.

[15] Higgs Benjamin and Rupke Ryan, “Noise reduction and automotive muffler design”, MME-419, April (2007).

[16] Atul A. Patil,L.G. Navale,V.S. Patil. “Experimental Verification and CFD Analysis of Single Cylinder Four Strokes C.I. Engine Exhaust System”.Pratibha: International Journal Of Science, Spirituality, Business And Technology (Ijssbt), Vol. 3, No. 1, Dec 2014.

[17] ErdemÖzdemir, RifatYılmaz, ZeynepParlar. “AN ANALYSIS OF GEOMETRIC PARAMETERS’ EFFECTS ON FLOW CHARACTERISTIC OF A REACTIVE MUFFLER.17th International Research/Expert Conference.

[18] Puneetha C G. Manjunath H. Shashidhar M.R. “Backpressure Study in Exhaust Muffler of Single Cylinder Diesel Engine using CFD Analysis”. Altair Technology Conference.

[19] Selamet, A., &Radavich, P. M. (1997).The Effect of Length on the Acoustic Attenuation Performance of Concentric Expansion Chambers: an Analytical, Computational and Experimental Investigation, Journal of Sound and Vibration, 201(4), 396-315.

[20] Łuszczyńska, M. P., Dudarewicz, A., Waszkowska, M., Szymczak, W., &Kowalska, M. Ś. (2005). The Impact of Low Frequency Noise on Human Mental Performance, International Journal of Occupational Medicine and Environmental Health, 18(2):185-198.

[21] Prof. Amar Pandhare. “CFD Analysis of Flow through Muffler to Select Optimum Muffler Model for Ci Engine”.International Journal of Latest Trends in Engineering and Technology (IJLTET).

[22] Bhattacharya, P., Ghosh, B., & Bose, P. K.(2010). Transmission Loss And Performance Test of A Two Cylinder Four Stroke Diesel Engine, Journal of Engineering Science and Technology Vol. 5, No. 3, 284 – 292.

[23] Chen, J. ve Shi, X.: CFD Numerical Simulation of Exhaust Muffler, Seventh International Conference on Computational Intelligence and Security, Wuhan, China.,2008.

[24] Hou, X., Guo, X., Liu, Z., Yan, F. vePeng, F.: Flow Field Analysis and Improvement of Automobile Exhaust System Cold End,Computational Intelligence and Software Engineering (CiSE), 2010 International Conference on, Wuhan, P.R. China,2010.

[25] Fang, J., Zhou, Y., Jiao, P., ve Ling, Z.: Study on Pressure Loss for a Muffler Based on CFD and Experiment, International Conference on Measuring Technology and Mechatronics Automation, China, 2009.

[26] Avcu, M., Kopuz, Ş., Liu, Z. veTeke, M.,: MTU 16V 4000 M90 Brand/Model Diesel Engine Exhaust System Design,Journal of Naval Science and Engineering, Vol. 6, pp. 39-58,201, 2010.

[27] Lee, S.-H. veIh, J.-G.,: Effect of non-uniform perforation in the long concentric resonator on transmission loss and back pressure, Journal of Sound and Vibration,311, 280–296, 2008.

[28] Amit Kumar Gupta, Rahul Suman. “An Approach for Transmission Loss Validation and Measurement of Various Reactive Mufflers with FEA Acoustic Module”. International Journal for Research in Applied Science & Engineering Technology (IJRASET).

[29] Pradyumna Saripalli, K. Sankaranarayana. “CFD Analysis on Flow Through a Resistance Muffler of LCV Diesel Engine”. International Journal of Science, Technology and Society. Vol. 3, No. 4, 2015, pp. 132-145. doi: 10.11648/j.ijsts.20150304.16

[30] A.J. Green and P.N. Smith, IMechE 1988, C17/88, 47

[31] Willard W. Pulkrabek. “Engineering Fundamentals of the Internal Combustion Engine.

[32] Roache, P.J., "Perspective: A Method for Uniform Reporting of Grid Refinement Studies", ASME Journal of Fluids Engineering, Vol. 116, September 1994.