Advanced International Journal for Research (AIJFR)

E-ISSN: 3048-7641 ● Website: www.aijfr.com ● Email: editor@aijfr.com

Design and Thermal Analysis of Hyundai I20 Muffler Using Fem (Case Study)

Bujji Babu Domathoti [1], Durga Rao Vemulapalli [2]

Assistant Professor, Mechanical Engineering, Sir Cr Reddy College of Engineering Eluru

Abstract

A LCV Petrol engine Hyundai i20 muffler is a part of a vehicle's exhaust system that muffles, or reduces, engine noise through soundproofing technique. Exhaust systems are parts of an engine's ventilation that transfer exhaust gases from within an engine's cylinder to the outside atmosphere. This project develops into the design and analysis of a commercial automotive muffler, a crucial component of modern vehicles. The muffler's initial design is generated using commercial CATIA V5 software, and then imported into commercial FEM software ANSYS 19.2 through a .IGS file format. The modeling of these mufflers is heavily influenced by the thermal performance of the material, impedance, and perforations. Due to the passage of hot gases through the muffler, its properties are subjected to change. Therefore, a thorough analysis of the muffler's thermal properties is conducted. Perform the various materials Stainless steel, Titanium alloy, Aluminium alloy, The heat transfer mechanisms within the muffler are identified and characterized. The resulting heat flow is found to induce variations in the muffler's properties.

Keywords: CAD – Computer-Aided Design,**CAE** – Computer-Aided Engineering,**FEA** – Finite Element Analysis,**IC Engine** – Internal Combustion Engine,**HP** – Horsepower,**RPM** – Revolutions Per Minute, **SS** – Stainless Steel,**Ti** – Titanium,Tmax – Maximum Temperature**,Q** – Heat Transfer Rate,**k** – Thermal Conductivity,**Cp** – Specific Heat at Constant Pressure**,HTR** – Heat Transfer Rate,CAD/CFD Coupling – Integration of CAD Design and CFD Analysis,3D – Three- Dimensional,**ANSYS** – (Name of Software) originally Analysis System,**CATIA** – Computer- Aided Three-dimensional Interactive Application

1. Introduction

1.1 Introducion To Muffler

Mufflers are an essential part of any exhaust system, as they help to reduce noise levels and improve engine performance. They work by absorbing and dissipating sound waves, and they can also be used to reduce backpressure in the exhaust system.

The design of a muffler can have a significant impact on its performance. For example, a muffler with a larger volume will be able to absorb more sound waves, while a muffler with a smaller volume will be more compact and lightweight. The shape of the muffler can also affect its performance, with some shapes being more efficient than others.

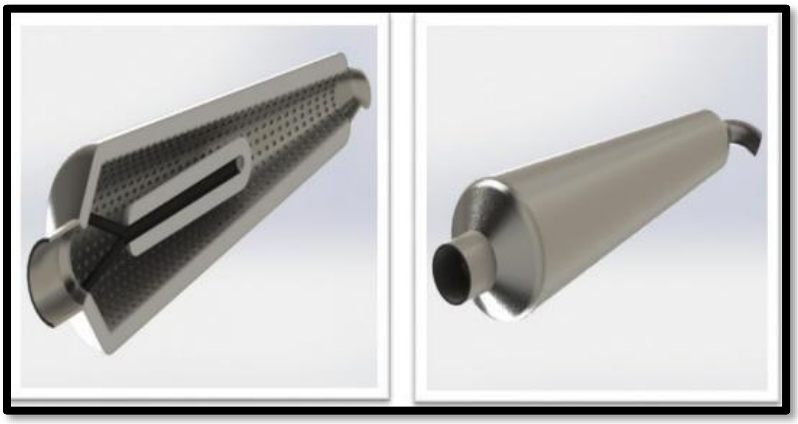


Figure 1.1: Hyundai i20 Muffler.[1]

In addition to the design of the muffler itself, the materials used in its construction can also affect its performance. For example, mufflers made from stainless steel are more durable than those made from other materials, while mufflers made from ceramic are more efficient at absorbing sound.

1.2 HISTORY

The design and modification of mufflers using static thermal analysis using the finite element method (FEM) has a long and rich history. Early mufflers were simple devices that used perforated plates or chambers to absorb sound waves. However, these mufflers were often ineffective and could overheat. In the early 1900s, engineers began to use the FEM to design more efficient mufflers. The FEM allowed engineers to simulate the flow of exhaust gases through the muffler and to identify hot spots. This information could then be used to design mufflers that were both effective and durable. The use of the FEM in muffler design continued to grow in the mid-20th century. As computers became more powerful, engineers were able to develop more complex muffler designs. These designs often used multiple chambers and acoustic resonators to achieve maximum sound absorption. In the late 20th century, the use of the FEM in muffler design became even more widespread. This was due in part to the development of new materials, such as stainless steel and ceramic, which could withstand the high temperatures of exhaust gases.

Today, the FEM is an essential tool for muffler design and modification. It is used by engineers to develop mufflers that are both effective and durable. The FEM has helped to make mufflers a critical component of modern exhaust systems.

1.3 WORKING PRINCIPAL

The design and modification of mufflers using static thermal analysis using the finite element method (FEM) involves a comprehensive process that combines theoretical principles, numerical simulations, and engineering expertise to achieve optimal muffler performance. The theoretical principles underlying muffler design and modification are based on the understanding of sound waves, acoustic propagation, and heat transfer. Sound waves are the primary source of noise generated by an internal combustion engine. Mufflers attenuate sound waves by absorbing and dissipating their energy through various mechanisms. Static thermal analysis focuses on the thermal behaviour of the muffler under the influence of hot exhaust gases, ensuring that the muffler can withstand the high temperatures and thermal stresses without compromising its structural integrity or sound attenuation performance.

The FEM is a powerful numerical tool that enables engineers to simulate the flow of exhaust gases through the muffler and to predict the temperature distribution within the muffler structure. By discretizing the muffler geometry into a mesh of finite elements, the FEM solves a set of governing equations that capture the heat transfer phenomena occurring within the muffler. This allows engineers to identify hot spots, analyze thermal stresses, and evaluate the overall thermal performance of the muffler design. The interpretation and application of results obtained from static thermal analysis require engineering expertise. Engineers must have a thorough understanding of muffler design principles, material properties, and thermal engineering concepts to effectively utilize the FEM and make informed design decisions. They must also consider manufacturing constraints, cost considerations, and noise reduction targets to optimize the muffler design.

1.3.1 An automotive requires a muffler to reduce the amount of noise emitted by a vehicle.

1.3.2 Mufflers are installed along the exhaust pipe as a part of the exhaust system of an I.C. engine to reduce its exhaust noise.

1.3.3 Mufflers use neat technology to cancel out the noise.

1.3.4 The muffler reduces exhaust noise by dampening the pulsations in the exhaust gases and allowing them to expand slowly.

1.3.5 It was usually made of sheet steel, coated with aluminum to reduce corrosion. Some are made of stainless steel.

1.3.6 A muffler contains perforated pipes, baffles and resonance chambers.

1.3.7 Many also contain sound-absorbing material such as fiberglass or wire wool.

1.3.8 The muffler slows down the gases and breaks up the pulsating sound waves, and so reduces the noise.

1.3.9 It must cause as little restriction as possible. Poor design can cause excessive back- pressure that will slow down the escape of the exhaust gases and reduce engine performance.

1.3.10 Some mufflers combine baffles and pipes to change the flow of gases without restricting them. Gases enter through the inlet and must reverse their direction of flow before they exit through the outlet. This is called a reverse-flow muffler.

1.3.11 Some mufflers use double outer-skins to minimize heat and noise transmission.

1.3.12 Some exhaust systems use a resonator as well as a muffler. It looks like a muffler but it usually has a straight-through design and it contains sound absorbing material. It's designed to remove types of sound that mufflers can't remove.

1.3.13 Silencers and mufflers cover a wide range of noise reduction devices and must be considered one of the most powerful weapons available to reduce noise emitted from cars, trucks, motor cycles, boats, vacuum pumps, compressors etc.

1.4 TYPES OF MUFFLERS

There are several different types of muffler design and modification that can be achieved using static thermal analysis using the FEM method. These include:

- **Chambered mufflers:** These mufflers are the most common type of muffler and consist of multiple chambers that reflect and absorb sound waves. The size and shape of the chambers can be optimized using static thermal analysis to maximize sound absorption and minimize heat generation.

Figure 1.2: Chambered mufflers.[2]

- **Resonator mufflers:** These mufflers use Helmholtz resonators to absorb sound waves at specific frequencies. The resonant frequency of each resonator is determined by its size and shape, and static thermal analysis can be used to optimize the design of the resonators for maximum sound absorption.

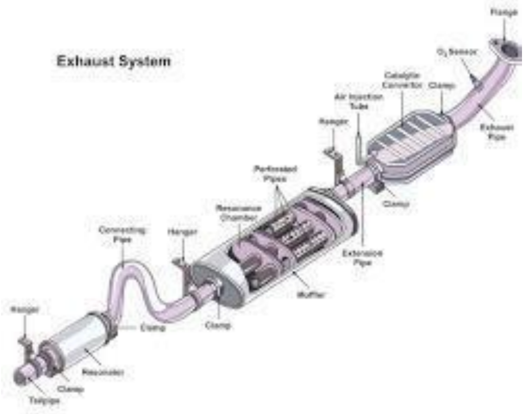


Figure 1.3: Resonator mufflers. [3]

- **Reactive mufflers:** These mufflers use reactive elements, such as perforated tubes or expansion chambers, to generate sound waves that cancel out the incoming sound waves. The design of the reactive elements can be optimized using static thermal analysis to minimize backpressure and improve sound attenuation.

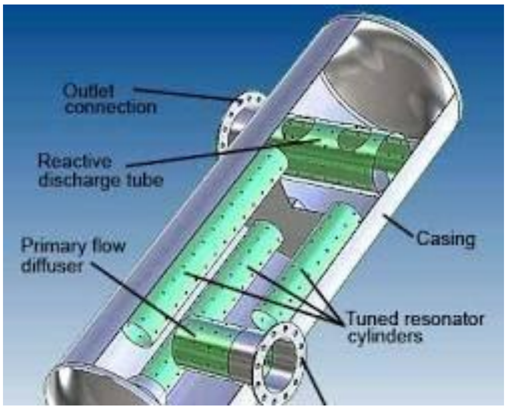


Figure 1.4: Reactive mufflers. [4]

- **Louvered mufflers:** These mufflers use louvered plates to direct the exhaust gas flow through a series of chambers. The size and shape of the louvers can be optimized using static thermal analysis to minimize backpressure and improve sound attenuation.

Figure 1.5: Louvered mufflers. [5]

- **Tubular mufflers:** These mufflers use a series of perforated tubes to absorb sound waves. The size and spacing of the perforations can be optimized using static thermal analysis to maximize sound absorption and minimize backpressure.

Figure 1.6: Tubular mufflers. [6]

- **Absorption mufflers:** These mufflers use porous materials, such as fiberglass or ceramic fiber, to absorb sound waves. The thickness and density of the porous material can be optimized using static thermal analysis to maximize sound absorption.

Figure 1.7: Absorption mufflers. [7]

- **Combination mufflers:** These mufflers use a combination of two or more of the above muffler types to achieve the desired level of sound attenuation.

1.5 APPLICATIONS:

Static thermal analysis using the finite element method (FEM) has a wide range of applications in the design and modification of mufflers for various types of internal combustion engines. Some of the key applications include:

- **Automotive Exhaust Systems:** Static thermal analysis is widely used in the design of automotive mufflers, where it plays a crucial role in optimizing muffler performance, ensuring thermal integrity, and contributing to the overall efficiency of the exhaust system. By analyzing temperature distribution, identifying hot spots, and evaluating thermal stresses, engineers can ensure that the muffler can effectively absorb sound waves, dissipate heat efficiently, and withstand the high temperatures and thermal loads encountered during vehicle operation.

- **Motorcycle Exhaust Systems:** In the design of motorcycle mufflers, static thermal analysis is employed to optimize sound attenuation, reduce backpressure, and ensure the durability of the muffler under the demanding conditions of motorcycle riding. By analyzing the thermal behavior of the muffler under various engine operating conditions, engineers can select appropriate materials, optimize muffler geometry, and prevent potential thermal failures.

- **Marine Exhaust Systems:** Static thermal analysis is essential in the design of marine mufflers, where it helps to address the unique challenges of marine environments. By considering factors such as saltwater corrosion, high humidity, and limited space constraints, engineers can design mufflers that are both effective and durable in these demanding conditions.

- **Industrial Exhaust Systems:**Static thermal analysis plays a critical role in the design of industrial exhaust mufflers, where it helps to reduce noise pollution, protect workers from harmful emissions, and ensure the safe operation of industrial equipment. By analyzing the thermal behavior of mufflers in industrial settings, engineers can optimize heat dissipation, prevent thermal overload, and ensure compliance with environmental regulations.

- **Power Generation Exhaust Systems:** In the design of exhaust systems for power generation equipment, such as generators and turbines, static thermal analysis is crucial for ensuring the long-term performance and reliability of the mufflers. By analyzing the thermal behavior under various operating conditions, engineers can prevent thermal fatigue, optimize material selection, and minimize the risk of muffler failure.

- **Aviation Exhaust Systems:** Static thermal analysis is employed in the design of aviation mufflers to address the unique challenges of aircraft propulsion systems. By considering factors such as high temperatures, high altitudes, and vibration, engineers can optimize muffler performance, prevent thermal failures, and ensure the safety of aircraft operations.

- **Racing Exhaust Systems:** Static thermal analysis plays a vital role in the design of racing exhaust systems, where it helps to maximize performance and minimize weight. By analyzing the thermal behavior of mufflers under extreme operating conditions, engineers can optimize design for maximum power, reduce heat generation, and ensure the durability of the muffler under the stresses of racing.

- **Research and Development:** Static thermal analysis is an invaluable tool in muffler research and development, enabling engineers to investigate new muffler designs, evaluate material properties, and optimize muffler performance under various conditions. By conducting simulations and analyzing results, researchers can gain valuable insights into muffler behavior and contribute to the development of innovative muffler designs.

- **Troubleshooting and Failure Analysis:** Static thermal analysis can be used to troubleshoot muffler failures and identify potential thermal issues. By analyzing the thermal behavior of a failing muffler, engineers can pinpoint the root cause of the problem and develop effective solutions to prevent future failures.

- **Acoustic Performance Optimization:** Static thermal analysis can be used in conjunction with acoustic simulations to optimize muffler performance for both sound attenuation and thermal management. By analyzing the interplay between thermal behaviour and acoustic propagation, engineers can design mufflers that effectively reduce noise while maintaining thermal integrity.

1.6 ADVANTAGES:

Static thermal analysis using the finite element method (FEM) offers several advantages for the design and modification of mufflers:

1. **Improved Muffler Performance:** By analyzing the temperature distribution, hot spots, and thermal stresses within the muffler, engineers can identify and address potential issues that could affect the muffler's ability to absorb sound waves and dissipate heat effectively. This leads to improved overall muffler performance and reduced noise levels.

2. **Enhanced Durability and Reliability:** Static thermal analysis helps ensure that the muffler can withstand the high temperatures and thermal loads encountered during operation. By identifying and mitigating thermal stress concentrations, engineers can prevent material failure and extend the lifespan of the muffler.

3. **Optimized Material Selection:** Static thermal analysis provides valuable information about the thermal behavior of different materials under the expected operating conditions. This allows engineers to select the most suitable materials for the muffler's construction, ensuring optimal performance and durability.

4. **Reduced Development Time and Costs:** Static thermal analysis can help to identify and address

potential thermal issues early in the design process, preventing costly redesigns and manufacturing defects. This can lead to faster development cycles and reduced development costs.

5. **Improved Thermal Management:** Static thermal analysis can be used to optimize the muffler's heat dissipation capabilities, ensuring that the muffler can effectively transfer heat away from the exhaust gases and prevent overheating. This can contribute to the overall efficiency of the exhaust system.

6. **Enhanced Design Flexibility:** Static thermal analysis allows engineers to explore a wider range of design options and evaluate their thermal performance virtually. This flexibility enables the development of innovative muffler designs that meet specific performance requirements.

7. **Accurate Prediction of Thermal Behaviour:** Static thermal analysis provides a detailed and accurate representation of the thermal behaviour of the muffler under various operating conditions. This allows engineers to make informed design decisions based on reliable data.

8. **Insightful Failure Analysis:** Static thermal analysis can be used to analyze muffler failures and identify the root cause of the problem. This information can be used to prevent future failures and improve the overall design of mufflers.

9. **Efficient Design Iterations:** Static thermal analysis facilitates efficient design iterations, allowing engineers to quickly evaluate the impact of design changes on the muffler's thermal performance. This iterative approach leads to improved optimization and design refinement.

10. **Compliance with Thermal Standards:** Static thermal analysis can be used to demonstrate compliance with thermal standards and regulations, ensuring that the muffler meets the required safety and performance requirements.

1.7 LIMITATIONS:

Despite its numerous advantages, static thermal analysis using the finite element method (FEM) also has certain limitations that need to be considered during the design and modification process of mufflers:

1. **Simplified Representation of Physical Phenomena**
2. **Accuracy Dependence on Mesh Resolution**
3. **Interpretation Challenges**
4. **Assumption of Static Conditions**

2. LITERATURE REVIEW

Dr. K. Satyanarayana et.al [2022] " Modelling And Analysis Of Diesel Engine Muffler Using Cfd " Noise pollution is the major drawback of I.C engines. Automotive engineers and researchers have been working consistently on reducing automotive noise as well as pollution. Automobile engineering is one of the fields where in the advancements were witnessing a steep upward trend. On comparing with early design, the recent automobiles were highly efficient and were formed with sophisticated systems. However, the area which still demand improvisation in automobile design exists in the areas like fuel economy, efficiency and exhaust systems. Exhaust systems focus majorly on reducing the emission of pollutants into atmosphere and also on controlling the sound that comes through exhaust. The devise used for reducing the sound is called as Muffler and the devise used for controlling the emissions is called catalytic convertor. Mufflers make use of different techniques and components to reduce the noise. Usually the noise is reduced when the transmission loss increases inside the muffler. Components like perforated tubes, sound absorbing materials, pipe bends, sudden expansions and contractions of pipe etc. In this work focus was laid on altering the design of a muffler to further reduce the noise and increasing the performance of the muffler. This is carried out by designing and using flow simulation through the muffler. Two muffler designs have been modelled and CFD gas flow simulation has been carried in both of them at various boundary conditions. Based upon the gas flow through them, on comparison we have selected the optimum model, for which the performance is best and sound reduction is maximum. The geometry of the model is prepared in CATIA V5 and analysis is carried out in ANSYS 2020 R2 using Computational Fluid Dynamics (CFD).

Apoorva Kaushik Singh,1 Ashutosh Gupta,1 and C. Kannan et.al [2021] " The combustion occurring in an internal combustion engine subsequently emits a high- intensity acoustic pulse along with exhaust gas flow. Formula SAE competition is imposing restrictions on acoustic emissions from an engine, which necessitates the activity of design and optimization of a high-performance muffler. The muffler has to be designed with the objectives of satisfying acoustic emission norms imposed and strategically designed to have less obstruction to exhaust flow and be lightweight to maintain the integrity of a Formula Student car. A novel approach of combining the effect of adsorptive and resonating muffler design has been inculcated in this work after the estimation of muffler size and resonance. The acoustic pressure analysis and exhaust flow efficiency are carried out using Ricardo Wave. These simulation results are validated with the developed prototype. The developed prototype is ascertained to fulfill the requirement of acoustic emission norms (103 dB[C]-110 dB[C]) without much compromise on exhaust flow efficiency. The noise reduction comes with a trade-off to the total power generated by the engine. It directly hampers the exhaust gas expelling efficiency and thus increasing backpressure. T hus, the main objective of this research work is formulated to develop a muffler that reduces noise to meet the norms without/with the least compromise on exhaust gas expelling efficiency and backpressure. The present work is dealt with a Honda CBR 600RR engine with a custom exhaust manifold and 20 mm restricted intake manifold. The work starts with determining the noise emitted without any sound reduction devices while the engine was operated at an idling speed of 2,500 RPM and a maximum speed of 12,000 RPM.

Sahil S Ambavane, Vinayak C Achari, Mrunal R Khandebharad, Pratik A Kulkarni et.al [2021] "The Vehicle noise is one of the components of environmental noise pollution. Exhaust noise is one of the main causes of vehicles and the exhaust system is designed to reduce the noise level to meet the demand level and lower the emission quality according to environmental standards. Modern engines must be powerful and also meet stringent pollution standards. In automobiles, exhaust mufflers play an important role

in reducing vehicle noise as well as the ride comfort itself. The muffler mode needs to be analyzed to maintain the desired noise level and comfortable ride. In this project the design and analysis of single outlet and double outlet exhaust muffler is done the design of muffler is done on solid work software and analysis is done on Ansys 14.1 version while analyzing the parameter like wall y + value, pressure contour, temperature contour velocity contour, it was kept in study and proper comparison of both the exhaust muffler is done.

Rohit Suryawanshi , Ajay Kashikar, Vivek Sunnapawar et.al [2020] "Noise pollution is the major drawback of I.C. engines. Automotive engineers and researchers have been working consistently on reducing automotive noise as well as pollution. While designing the mufflers; care has to be taken not only for noise reduction but also for exhaust emission, back pressure, space constraints, cost incurred, etc. Various methods to design and analyze the mufflers have been devised by researchers across the globe. Many guidelines have been provided by the authors in this regard. In the present paper, an attempt is made to present a comprehensive review of literature in context to design and analysis of automotive mufflers.

Praveen. R et.al [2018] "Design And Analysis Of Automobile Muffler: A Review",The paper deals with the design of a commercial automotive muffler, which is being used in current automobile vehicles like hyundai. Initially the muffler is designed in the basic 3D modelling can be done using commercial CAD software package and can be imported into Commercial FEM software using a neutral file format. The propensity to model these mufflers relies vastly on the thermal performance of the material, impedance and the perforations. In muffler the hot gases passes through, which may affect the properties of the muffler. So we have analyzed the thermal properties of the muffler. And defines the heat transfer occurring inside the muffler. The heat flow causes the variation in the properties of the muffler.

P. Srinivas*, Venkata Ramesh Mamilla, G. Lakshmi Narayana Rao, Sowdager Moin Ahmed et.al [2016]“ Design and Analysis of an Automobile Exhaust Muffler’’This paper deals with Present day engines are required to have more engine power and are also required to meet the strict pollution standards. In an automobile the exhaust muffler plays an integral role in reducing the sound of the automobile, as well as the ride itself. In order to maintain a desired noise and comfortable ride, the modes of a muffler need to be analysed. Here dynamic modal analyses were carried out to determine the mode shapes, stresses and deformations of exhaust muffler using CAE analysis, Mufflers are installed within the exhaust system of most internal combustion engines, although the muffler is not designed to serve any primary exhaust function. The muffler is engineered as an acoustic sound proofing device designed to reduce the loudness of the sound pressure created by the engine by way of acoustic quieting. For the majority of such systems, however, the general rule of “more power, more noise” applies. Several such exhaust systems that utilize various designs and construction methods: Vector muffler - for larger diesel trucks, uses many concentric cones, or for performance automotive applications, using angled baffles to cause exhaust impulses to cancel each other out. Spiral baffle muffler - for regular cars, uses a spiral-shaped baffle system Aero turbine muffler - creates partial vacuums at carefully spaced out time intervals to create negative back pressure.

Ankit J Desai et.al [2016] " Design and Analysis of Automotive Muffler"—Noise pollution is a very crucial problem for today's life, so to reduce noise level sound proofing is necessary. Muffler is a very important part of the vehicle exhaust system to reduce the noise produced by engine combustible products when passing through the exhaust system. To achieve maximum noise reduction with the minimum pressure drop is very difficult. A conventional muffler of Maruti-Suzuki WagonR is taken as reference and depending upon parameters new muffler is designed and modelled in software and analysis will be done numerical codes. Analysis ease the design parameters to be change, so that an appropriate design can be generate and maximum amount of noise reduction and pressure drop takes place with minimum back pressure. Comparison of conventional muffler and proposed designed muffler is based on amount of noise reduction, pressure drop and muffler life. In experimental setup pressure drop calculated by the water manometer tube and sound intensity measured by Sound Level Meter (SLM) device.

Kirill V Horoshenkov et.al [2016] "Design of Acoustically Efficient Automotive Mufflers Using Sustainable Materials" This review emphasizes the importance of using sustainable materials in muffler design to minimize environmental impact and reduce the carbon footprint of automobiles. It discusses advanced materials such as porous polymers, carbon fiber composites, and acoustic meta materials that offer promising noise reduction capabilities. Acoustical sustainable materials, either natural or made from recycled materials, are quite often a valid alternative to traditional synthetic materials. The production of these materials generally has a lower environmental impact than conventional ones, though a proper analysis of their sustainability, through Life Cycle Assessment procedures, has to be carried out. Airborne sound insulation of natural materials such as flax or of recycled cellulose fibers is similar to the one of rock or glass wool.

Sibin Babu et.al [2015] " Acoustic Performance Design of Automotive Muffler '' The present work involved the optimization of the mufflers design by designing a hybrid muffler. The objective of the current work is to design a hybrid muffler which is very good in performance and works well in the wide spectrum of frequency. For this purpose, we are going to perform acoustic analysis which can show how the sound pressure is changing when it is passing through the muffler and what the difference in between inlet sound pressure is and outlet sound pressure. .CAD files of the mufflers were established for developing FEA models in ANSYS. When we do acoustic analysis, we actually carry out transmission loss analysis. Two production mufflers were selected for this study. Both mufflers have complex partitions and one of them was filled with absorbent material. FEA models were validated by experimental measurements using a two-source method. After the models were verified, sensitivity studies of design parameters were performed to optimize the transmission loss (TL) of both mufflers. The sensitivity study includes the perforated hole variations, partition variations and absorbent material insertion. Mufflers are installed within the exhaust system of most internal combustion engines, although the muffler is not designed to serve any primary exhaust function. The muffler is engineered as an acoustic sound proofing device designed to reduce the loudness of the sound pressure created by the engine by way of acoustic quieting. Mufflers are a fundamental part of engine exhaust system and are used to minimize sound transmissions caused by exhaust gases. Design of mufflers is a complex function that affects noise characteristics, emission and fuel efficiency of engine. Mufflers presently used in the automotive industry are either reactive muffler or a dissipative muffler which work at a certain target frequency spectrum. For example, the reactive muffler are good at a low frequency ranges whereas the dissipative mufflers work at high frequencies of 1500 2000Hz.

Raj C N Thiagarajan et.al [2015] "Development of Hybrid Mufflers for Improved Noise Reduction and Fuel Efficiency" This review investigates the concept of hybrid mufflers that combine different muffler technologies, such as Helmholtz resonators, reactive-type mufflers, and active noise control, to achieve enhanced noise reduction while minimizing backpressure and ensuring fuel efficiency. Literature review on acoustic methods and materials of a muffler used in different application like automobile, aerospace industry, compressor and industrial noise has been put forwarded by other co workers have been presented in this manuscript. Materials properties like stress, temperature, thermal conductivity and density have been technical presented in this work.

3. OVERVIEW OF PROJECT

3.1 PROBLEM IDENTIFICATION

Noise pollution is a significant drawback of internal combustion (IC) engines, prompting continuous efforts among automotive design engineers and researchers to reduce IC engine noise and emissions. Silencer design necessitates careful consideration of not only noise reduction but also exhaust emissions, backpressure, space constraints, and associated costs. Scientists worldwide have developed various methods for designing and analyzing silencers, with numerous tutorials provided by authors. However, the issue remains a topic of ongoing research and development.

3.2 OBJECTIVE OF THE PROJECT

- **Define Design Requirements**
  Identify noise reduction targets for different frequency ranges. Consider engine operating conditions and exhaust flow characteristics. Determine space constraints and installation limitations

- **Concept Generation and Selection**
  Explore various muffler designs, including chambered mufflers, reactive mufflers, and hybrid mufflers. Evaluate the acoustic performance and backpressure characteristics of each design concept. Select the most promising concept based on design requirements and feasibility

- **Muffler Modeling and Analysis**
  Develop a detailed 3D model of the selected muffler design using CATIA V5 R20 software. Utilize thermal simulations to analyze exhaust gas flow and pressure. Employ different materials to simulations and predict performance

- **Design Optimization**

Apply optimization techniques to refine muffler geometry, material selection, and perforations. Iterate design modifications based on Thermal simulations and results.

- **Documentation and Reporting**
  Document the design process, including methodology, rationale, and results. Prepare detailed design specifications and performance data. Present findings in technical reports or publications

3.3 METHODOLOGY

Effective muffler design requires careful consideration of several key factors, including acoustical, aerodynamic, mechanical, geometrical, and economic aspects. Aerodynamic factors dictate the maximum acceptable backpressure through the muffler, considering the prevailing temperature and mass flow rate. Mechanical criteria determine the material selection for muffler fabrication, ensuring durability and performance under various operating conditions. Geometrical considerations encompass the shape and dimensions of the muffler, influencing its acoustic performance and compatibility with the vehicle's layout. Economic factors play a crucial role in optimizing muffler design, balancing cost-effectiveness with performance requirements.

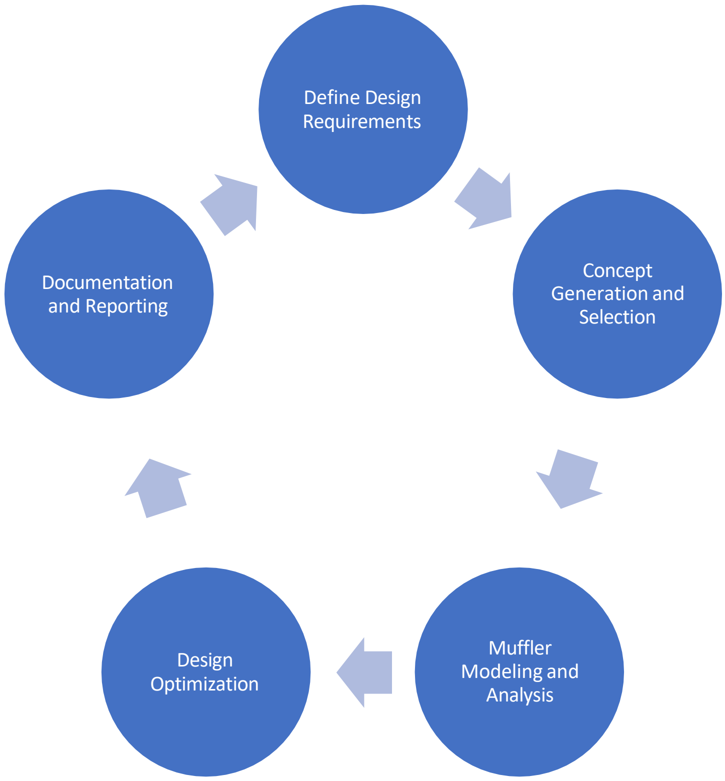


Figure 3.1: Methodology Process. [8]

3.4 MATERIAL USED

3.4.1 STAINLESS STEEL

Stainless steel is a corrosion-resistant alloy of iron containing a minimum of 10.5% chromium, which forms a passive layer of chromium oxide on the surface that prevents rusting. It often contains other alloying elements such as nickel, molybdenum, and manganese to enhance strength, ductility, and corrosion resistance. The most commonly used grade for structural and mechanical applications is **AISI 304**, an austenitic stainless steel known for its excellent forming and welding characteristics, good tensile strength, and superior resistance to oxidation and various chemicals.

3.4.2 ALUMINUM ALLOY

Aluminum alloys are lightweight, corrosion-resistant materials made by combining aluminum with other elements like copper, magnesium, silicon, and zinc. A widely used grade for structural applications is **6061-T6**, which is heat-treated to improve strength while retaining good corrosion resistance and workability. It exhibits excellent mechanical properties, making it suitable for aerospace, automotive, and general-purpose engineering applications. Its combination of low density and decent strength makes it an ideal material where weight saving is crucial.

3.4.3 TITANIUM ALLOY

Titanium alloys are known for their high strength-to-weight ratio, exceptional corrosion resistance, and ability to withstand extreme environments. The most commonly used titanium alloy is **Ti-6Al-4V**, consisting of 6% aluminum and 4% vanadium. This alpha- beta titanium alloy is widely used in aerospace, biomedical implants, and high- performance engineering structures due to its excellent fatigue strength, low density, and high resistance to creep and corrosion. Despite being more expensive and difficult to machine, Ti-6Al-4V provides an unmatched combination of properties where performance is critical.

Properties	Stainless Steel (AISI 304)	Aluminum Alloy (6061- T6)	Titanium Alloy (Ti-6Al- 4V)
Density (kg/m^3)	7930	2700	4430
Young's Modulus (GPa)	200	69	110
Poisson's Ratio	0.29	0.33	0.34
Thermal Conductivity (W/m·K)	16.2	167	6.7
Specific Heat Capacity (J/kg·K)	500	900	560

Table 1: Mechanical Properties.

4 INTRODUCTION TO CATIA V5R20

4.1 INTRODUCTION:

CATIA V5 is a high-end CAD/CAM/CAE software package that is used by thousands of users worldwide. It is a completely re-engineered, next-generation family of software solutions for Product Lifecycle Management (PLM).

CATIA V5 has a state-of-the-art user interface that makes it easy to use. It also offers innovative technologies for maximum productivity and creativity, from the inception concept to the final product.

CATIA V5 provides three basic platforms: P1, P2, and P3. P1 is for small and medium-sized companies that are looking to grow. P2 is for advanced design engineering companies that require product, process, and resource modeling. P3 is for high-end design applications, such as the automotive and aerospace industries.

CATIA V5 is also capable of interpreting legacy data from other CAD systems. This means that you can easily import data from other software packages and use it in CATIA V5.

CATIA V5 provides different workbenches to serve the basic design tasks. A workbench is a specified environment consisting of a set of tools that allows you to perform specific design tasks. The basic workbenches in CATIA V5 are:

- Sketcher: This workbench is used to create 2D sketches.
- Part Design: This workbench is used to create 3D solid models.
- Wireframe and Surface Design: This workbench is used to create 3D wireframe and surface models.
- Assembly Design: This workbench is used to assemble 3D models.
- Drafting: This workbench is used to create 2D drawings from 3D models

4.2 SKETCHER WORKBENCH:

The Sketcher Workbench is a set of tools in CATIA V5 that allows you to create and constrain 2D geometries. These 2D geometries can then be used to create solid features, such as pads, pockets, and shafts.

To access the Sketcher Workbench, you can use the top pull-down menu (Start > Mechanical Design > Sketcher) or the Sketcher icon.

When you enter the Sketcher Workbench, CATIA V5 will prompt you to select a plane to sketch on. You can choose this plane before or after you select the Sketcher icon.

To exit the Sketcher Workbench, select the Exit Workbench icon.

The Sketcher Workbench contains the following standard workbench-specific toolbars:

- Constraints: This toolbar contains tools for applying constraints to your sketches.
- Construction: This toolbar contains tools for creating construction geometry, such as centerlines and extension lines.
- Sketcher: This toolbar contains tools for creating basic sketch entities, such as lines, arcs, and circles.
- Tools: This toolbar contains tools for editing and modifying your sketches.

The Sketcher Workbench is a powerful tool for creating 2D geometries. With a little practice, you can use it to create complex and accurate sketches that can be used to create solid features.

• **Profile toolbar:** You may generate both straightforward geometries (such as rectangles, circles, and lines) and more intricate geometries (such as profiles and splines) with the commands on this toolbar.

• **Operation toolbar:** Using the trim, mirror, chamfer, and other tools found in the Operation toolbar, a profile can be altered after it has been produced.

Figure 4.1:Operation tool bar.[9]

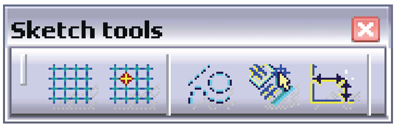


Figure 4.2:Sketch tool bar.[10]

• **Constraint toolbar:** The commands found in the Constraint toolbar can be used to limit profiles with dimensional (distances, angles, etc.) or geometrical (tangent, parallel, etc.) constraints.

• **Sketch tools toolbar:** The commands in this the toolbar allows you to work in different modes which Make sketching easier.

• **User Selection Filter toolbar:** enables the activation of various selection filters.

• **Visualization toolbar:** This feature enables you to select lighting effects, cut the part by the sketch plane, and perform other operations that affect how the part is visualized.

• **Tools toolbar:** This feature enables you to analyze a sketch for issues and create a datum, among other things.

4.3 THE SKETCH TOOLS TOOLBAR

The Sketch Tools toolbar in CATIA V5 contains icons that activate and deactivate different work modes. These work modes assist you in drawing 2D profiles.

The following work modes are available in the Sketch Tools toolbar:

- Grid: This command turns the sketcher grid on and off. The grid can be helpful for aligning your sketches and ensuring that they are accurate.

- Snap to Point: When this icon is active, your cursor will snap to the intersections of the grid lines. This can be helpful for precisely placing your sketch elements.
- Construction / Standard Elements: This icon toggles between the creation of standard elements and construction elements. Standard elements are used to create features in the Part Design workbench, while construction elements are used to help construct your sketch but are not used to create features.
- Geometric Constraints: When this icon is active, geometric constraints will automatically be applied to your sketch. Geometric constraints are relationships between sketch elements, such as tangency, coincidence, and parallelism.
- Dimensional Constraints: When this icon is active, dimensional constraints will automatically be applied to your sketch. Dimensional constraints are relationships between sketch elements that are defined by a numerical value, such as the length of a line or the angle between two lines.

The Sketch Tools toolbar is a powerful tool for creating accurate and precise 2D sketches. By understanding the different work modes available in this toolbar, you can create sketches that are ready to be used to create solid features.

4.4 PART DESIGN WORKBENCH:

The Part Design Workbench in CATIA V5 is a parametric and feature-based environment that allows you to create solid models. A parametric model is a model that is defined by its features, and the features are defined by their dimensions and constraints. This means that if you change the dimensions or constraints of a feature, the entire model will update automatically.

The basic requirement for creating a solid model in the Part Design Workbench is a sketch. A sketch is a 2D drawing that defines the profile of a feature. The sketch can be drawn in the Sketcher Workbench, which can be invoked from within the Part Design Workbench.

When you draw a sketch, some constraints are automatically applied to it. These constraints ensure that the sketch is accurate and that the feature that is created from it is valid. You can also apply additional constraints and dimensions manually. After you have drawn a sketch, you can convert it into a feature in the Part Design Workbench. The tools in this workbench allow you to create a variety of features, such as pads, pockets, and holes.

The Part Design Workbench also provides tools for applying placed features, such as fillets and chamfers. These features are added to the model after the basic features have been created.

4.5 WIREFRAME AND SURFACE DESIGN WORKBENCH:

The Wireframe and Surface Design Workbench in CATIA V5 is a parametric and feature- based environment that allows you to create wireframe or surface models. A parametric model is a model that is defined by its features, and the features are defined by their dimensions and constraints. This means that if you change the dimensions or constraints of a feature, the entire model will update automatically.

4.6 ASSEMBLY DESIGN WORKBENCH:

The Assembly Design Workbench in CATIA V5 is used to assemble components into a complete product. The assembly constraints available in this workbench allow you to define the relationships between the components, such as their position, orientation, and relative motion.

There are two types of assembly design approaches:

- Bottom-up: In the bottom-up approach, the components are assembled one by one, starting with the most basic components and working up to the final product. This approach is typically used when the design of the product is well-defined and the components are already available.
- Top-down: In the top-down approach, the components are created inside the assembly. This approach is typically used when the design of the product is not yet fully defined or when you want to create a product that is made up of a large number of components.

The Assembly Design Workbench also provides tools for analyzing the assembly, such as the Clash Analysis tool. This tool can help you to identify potential clashes between components, which can help you to avoid problems during manufacturing or assembly.

4.7 DRAFTING WORKBENCH:

The Drafting Workbench in CATIA V5 is used to create 2D drawings from 3D models. The drawings can be used for documentation, manufacturing, or assembly purposes.

There are two types of drafting techniques available in the Drafting Workbench:

- Generative drafting: This technique is used to automatically generate the drawing views of the 3D models. The parametric dimensions added to the 3D models during their creation are automatically generated and displayed in the drawing views. This technique is bidirectional associative, which means that any changes made to the 3D models will be reflected in the drawing views automatically.
- Interactive drafting: This technique is used to create the drawing views manually by sketching them using the normal sketching tools. The dimensions are then added to the drawing views manually.

The Drafting Workbench also provides tools for adding annotations to the drawing views, such as dimensions, labels, and notes. It also provides tools for creating the Bill of Material (BOM) and balloons in the drawing views.

4.8 GENERATIVE SHEET METAL DESIGN WORKBENCH:

The sheet metal components are designed using the Generative Sheet Metal Design workbench. To produce the flat pattern of the sheet, examine the design of the dies and punches, examine the process plan for designing, and examine the equipment required for manufacturing the sheet metal components, solid models of the sheet metal components are typically created.

4.9 SYSTEM REQUIREMENTS:

System Requirements for CATIA V5R20

To ensure the smooth running of CATIA V5R20 on your system, you need to meet the following minimum system requirements:

- Operating system: Microsoft Windows 2000 Professional Edition, Windows XP, Windows Vista, or Windows 7.
- Processor: Intel Pentium 4 or Xeon-based workstation.
- Memory: 512 MB of RAM (minimum).
- Disk space: 4 GB of disk space (minimum recommended size).
- Graphics: A graphics adapter with a 3D OpenGL accelerator and a minimum resolution of 1024x768 pixels for Microsoft Windows workstations and 1280x1024 for UNIX workstations.
- Display: A graphic color display compatible with the selected platform-specific graphic adapter. The minimum recommended monitor size is 17 inches.

4.10 GETTING STARTED WITH CATIA V5R20: -

To install CATIA V5R20 on your system, follow these steps:

1. Download the installation file from the CATIA website.
2. Run the installation file and follow the on-screen instructions.
3. Once the installation is complete, you can start CATIA V5R20 by double-clicking on the shortcut icon on your desktop.

You can also start CATIA V5R20 by following these steps:

1. Click on the Start button.
2. Select All Programs.
3. Select CATIA.
4. Select CATIA V5R20.

CATIA V5R20 is a high-end CAD/CAM/CAE software package that is used in a variety of industries, including automotive, aerospace, and shipbuilding. It is known for its ability to create complex and accurate models.

CATIA V5R20 can also import legacy data from other CAD systems. This is a valuable feature, as it allows you to reuse existing data and save time and effort.

The links between CATIA V5R20 and external data are associative. This means that any changes made to the external data will be reflected in CATIA V5R20 automatically. This is a powerful feature that can help you to keep your models up-to-date.

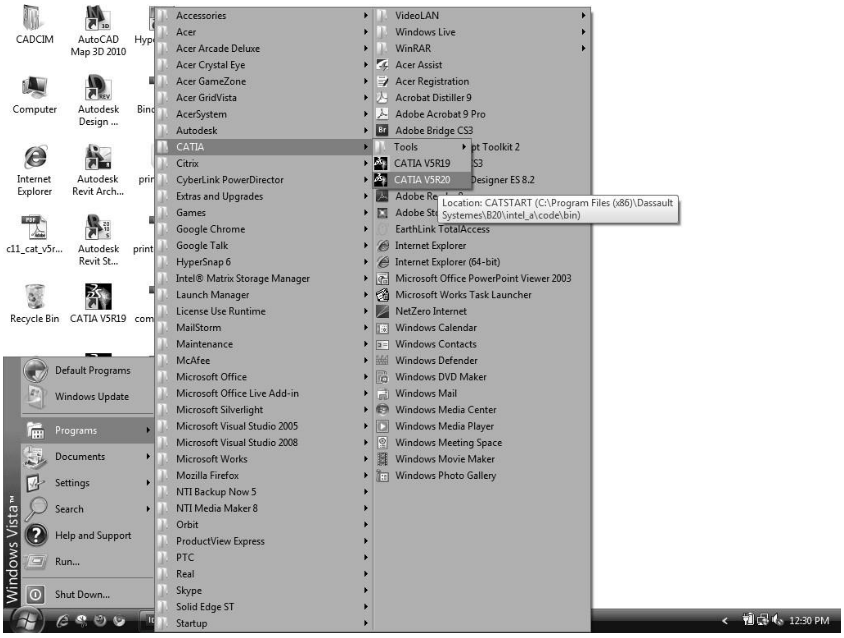


Figure 4.3: Get into CATIA software. [11]

A new product file with the default name Product1 will immediately start once the system has loaded all necessary files to launch CATIA V5R20, as illustrated in Figure.

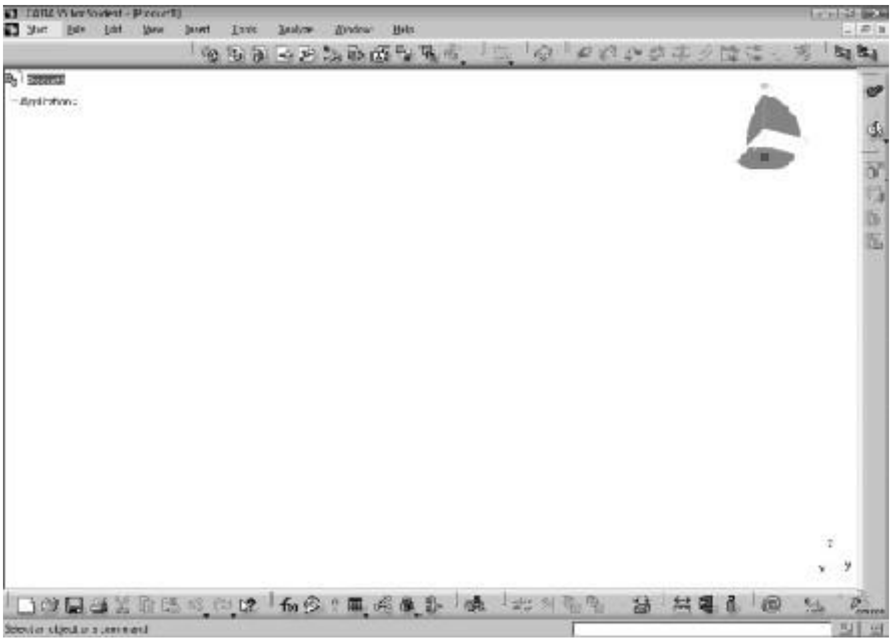


Figure 4.4: User interface of CATIA. [12]

Close this file by choosing **File** > **Close** from the menu bar. Figure 1-3 shows the screen that appears after closing the initial Product file.

4.11 IMPORTANT TERMS AND DEFINITIONS

4.11.1 FEATURE-BASED MODELING

Feature-based modeling is a CAD methodology that creates a 3D model by defining and assembling individual features. A feature is a geometric entity that has a specific function, such as a hole, a slot, or a fillet.

In CATIA V5R20, features are created and modified using the Feature-based Modeling workbench. This workbench provides a variety of tools for creating features, such as the Extrude tool, the Sweep tool, and the Revolution tool.

The Feature-based Modeling workbench also provides tools for modifying features, such as the Chamfer tool and the Fillet tool.

4.11.2 PARAMETRIC MODELING

Parametric modeling is a feature-based modeling methodology that uses parameters to define the shape and size of features. Parameters are variables that can be changed to modify the shape and size of a feature.

In CATIA V5R20, parameters are used to define the dimensions, geometric relationships, and other properties of features.

Parametric modeling allows you to create and modify 3D models quickly and easily. It also allows you to create models that are more accurate and reliable.

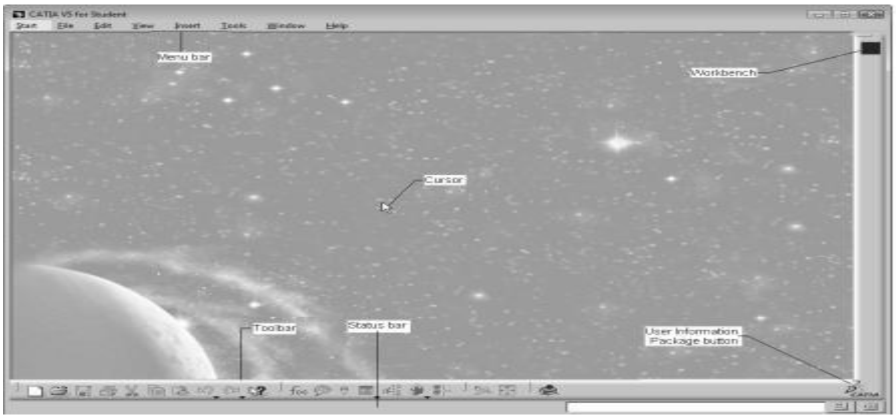


Figure 4.5: Tools in CATIA user interface. [13]

4.11.3 BIDIRECTIONAL ASSOCIATIVITY

Bidirectional associatively is a feature of CATIA V5R20 that ensures that any changes made to a model in one workbench are automatically reflected in other workbenches. This means that you can make changes to a model in the Part Design workbench, and the changes will be reflected in the Assembly Design workbench and the Drawing workbench. Similarly, if you make changes to a model in the Assembly Design workbench, the changes will be reflected in the Part Design workbench and the Drawing workbench. Bidirectional associatively is a powerful feature that can help you to save time and effort when designing and documenting products. It also helps to ensure that your models are accurate and up-to-date.

4.11.4 CAT Part, CAT Product, and CAT Drawing

- CAT Part: A CAT Part file is a file extension associated with all those files that are created in the Sketcher, Part Design, Generative Sheet metal Design, and Wireframe and Surface Design workbenches of CATIA V5. A CAT Part file contains the definition of a 3D solid model.
- CAT Product: A CAT Product file is a file extension associated with all those files that are created in the Assembly Design workbench of CATIA V5. A CAT Product file contains the definition of an assembly, which is a collection of parts.
- CAT Drawing: A CAT Drawing file is a file extension associated with all those files that are created in the Drafting workbench of CATIA V5. A CAT Drawing file contains the definition of a 2D drawing, which is a representation of a 3D model.

4.11.5 SPECIFICATION TREE

The Specification tree is a tree-like structure that shows the hierarchical organization of a 3D model. The Specification tree keeps track of all operations carried out on the part. This means that you can easily see how a part was created and how it is made up of different features.
The Specification tree is a powerful tool that can be used to:

- Inspect the structure of a 3D model.
- Identify the different features that make up a 3D model.
- Understand the relationships between different features.
- Modify the shape and size of a 3D model.

4.11.6 COMPASS

The compass is a tool that is used to manipulate the orientation of parts, assemblies, or sketches. You can also use the compass to orient the view of the parts and assemblies.
The compass is a circular tool that is located in the top right corner of the geometry area. It has three axes: X, Y, and Z. You can use the compass to rotate the model around any of these axes.
The compass is a useful tool that can be used to:

- Rotate the model to get a better view.
- Align the model with other models or sketches.
- Prepare the model for manufacturing or assembly.

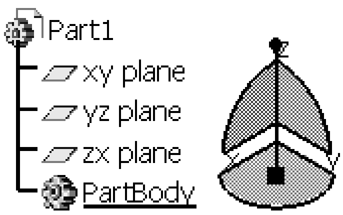


Figure 1.6: The Specification tree Compass. [14]

4.11.7 CONSTRAINTS:

The logical operations known as constraints are applied to the chosen element to specify its size and position in relation to other elements or reference geometries. In CATIA V5, there are two different kinds of limitations. Geometric restrictions are used in the Sketcher workbench to carefully determine the size and placement of the sketch elements in relation to their surrounds. The precise location of the components in the assembly is defined using the assembly constraints accessible in the Assembly Design workbench.

4.12 GEOMETRIC CONSTRAINTS:

Geometric constraints are logical operations that are performed on sketched elements to define their size and position relative to each other. There are two ways to apply geometric constraints: automatically and manually.

Automatic constraints are applied automatically as you draw the sketch. For example, if you draw two parallel lines, CATIA V5 will automatically apply a parallelism constraint between them.

Manual constraints can be applied at any time after the sketch is drawn. To apply a manual constraint, you can use the Constraints Defined in the Dialog Box tool. This tool will open a dialog box that lists all of the available constraints. You can select the appropriate constraint and then click on the check box to apply it.

- Distance: This constraint is used to specify the distance between two points or the length of a line.
- Angle: This constraint is used to specify the angle between two lines or the radius of an arc.
- Symmetry: This constraint is used to make two or more entities symmetrical about a center point or axis.
- Midpoint: This constraint is used to make a point coincide with the midpoint of a line segment.
- Equidistant: This constraint is used to make a point equidistant from two other points.
- Fix: This constraint is used to prevent an entity from moving or being modified.
- Coincident: This constraint is used to make two or more entities coincide.
- Concentric: This constraint is used to make two or more circles or arcs have the same center point.
- Tangent: This constraint is used to make two curves touch each other at a single point.
- Parallel: This constraint is used to make two lines or curves parallel to each other.
- Perpendicular: This constraint is used to make two lines or curves perpendicular to each other.
- Horizontal: This constraint is used to make a line horizontal.
- Vertical: This constraint is used to make a line vertical.

4.13 DESIGNING OF THE MODEL: DESIGN PROCEDURE IN CATIA WORK BENCH:

4.13.1 EXHAUST VALVE DESIGN PROCEDURE IN CATIA:

Create the sectional view of the muffler in Sketcher Workbench by selecting the line and rectangular corner options, and then go to Part Design Workbench and select the shaft option as indicated in the diagram below.

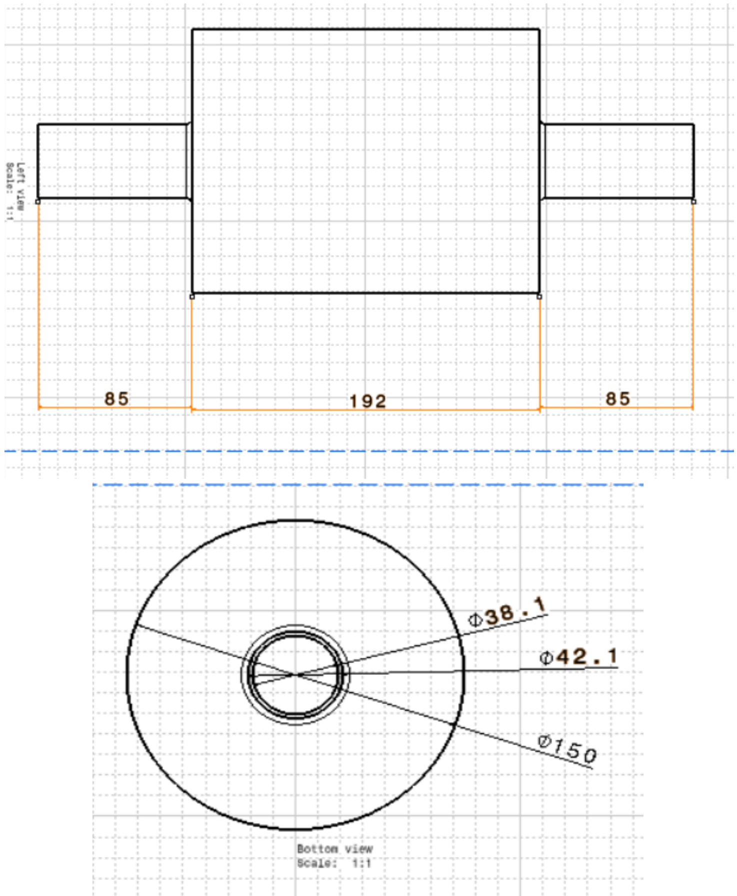


Figure 4.7: 2d Sketched Profile In Sketcher. [15]

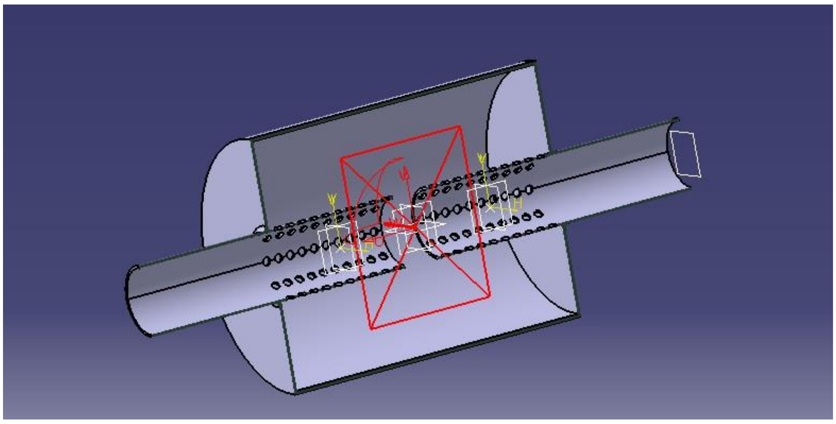


Figure 4.8: Muffler Without Baffle. [16]

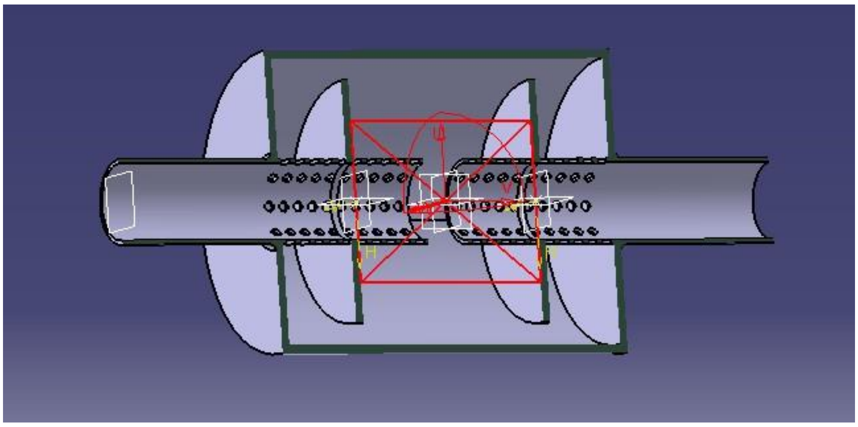


Figure 4.9: Muffler with Baffle. [17]

5 INTRODUCTION TO ANSYS

To analyze a broad range of issues encountered in engineering mechanics, ANSYS is a large-scale multipurpose finite element programmed created and maintained by ANSYS Inc.

5.1 PROGRAM ORGANIZATION:

The ANSYS program is organized into two basic levels:

- Begin level: This is the starting point of the ANSYS program. It is used to control global program settings, such as the job name and the database.
- Processor level: This level contains the different processors that are used to perform specific analysis tasks. For example, the PREP7 processor is used to build the model, the SOLUTION processor is used to apply loads and obtain the solution, and the POST1 processor is used to evaluate the results of the solution.

The Begin level is a gateway to the Processor level. When you first enter the ANSYS program, you are at the Begin level. To access a processor, you must first enter the Begin level and then select the appropriate processor.

The Processor level is divided into two main categories:

- Preprocessing: This category includes processors that are used to create and modify the model.
- Post processing: This category includes processors that are used to analyze and visualize the results of the analysis.

5.2 MATERIAL MODELS:

ANSYS allows for the use of a variety of material models, including:

- Linear elastic material models: These models assume that the material's response is linearly proportional to the applied stress. They are the simplest type of material model and are often used for preliminary analysis or for materials that are known to be linearly elastic, such as steel.
- Nonlinear material models: These models allow for the material's response to be nonlinear, meaning that the stress-strain relationship is not a straight line. They are more complex than linear elastic models but can be used to more accurately simulate the behavior of materials that exhibit nonlinear behavior, such as rubber or concrete.

- Heat transfer material models: These models allow for the simulation of heat transfer within a material. They are used to determine the temperature distribution within a material and the heat flux at its boundaries.
- Temperature-dependent material properties: These models allow for the material properties to vary with temperature. This is important for materials that exhibit significant changes in their properties with temperature, such as plastics.
- Creep material models: These models allow for the simulation of creep, which is the time-dependent deformation of a material under load. Creep is important for materials that are subjected to long-term loads, such as bridges and buildings.

5.3 LOADS:

In ANSYS, the term loads refers to boundary conditions and externally or internally applied force functions. Loads can be divided into six categories:

- DOF constraints: These constraints fix a degree of freedom (DOF) to a known value. For example, you can fix the displacement of a node to zero or specify the temperature of a surface to a certain value.
- Forces (concentrated loads): These loads are applied at a single point in the model. For example, you can apply a force to a node or a moment to a line.
- Surface loads: These loads are applied over a surface in the model. For example, you can apply a pressure to a surface or a heat flux to a surface.
- Body loads: These loads are applied throughout the volume of the model. For example, you can apply a temperature to the entire model or a current density to the entire model.
- Inertia loads: These loads are due to the inertia of the model. For example, you can apply the gravitational force to the entire model.
- Coupled-field loads: These loads are a special case of the other loads, where the results from one analysis are used as loads in another analysis. For example, you can apply the magnetic force calculated in a magnetic field analysis as a force load in a structural analysis.

5.4 ANALYSIS TYPES:

ANSYS can be used to perform a variety of analyses, including:

- Structural analysis: This type of analysis is used to determine the stresses, strains, and deformations of structures under load. It can be used to analyze static structures, dynamic structures, and structures that are subjected to fatigue loading.
- Thermal analysis: This type of analysis is used to determine the temperature distribution within a body or a system. It can be used to analyze steady-state heat transfer problems and transient heat transfer problems.
- CFD (Computational Fluid Dynamics) analysis: This type of analysis is used to simulate the flow of fluids. It can be used to analyze the flow of air around a car or the flow of blood through an artery.
- Electromagnetic field analysis: This type of analysis is used to study the behavior of electromagnetic fields. It can be used to analyze the radiation from a radio antenna or the

magnetic field around a coil.

- Coupled field analysis: This type of analysis is used to study the interaction of two or more physical fields. For example, it can be used to analyze the interaction of the flow of fluid with the heat transfer in a body.

5.4.1 STATIC ANALYSIS:

Static analysis is a type of analysis that determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. In other words, static analysis ignores the effects of time-varying loads, such as vibrations and shocks.

Static analysis can be either linear or nonlinear. Linear static analysis assumes that the materials in the structure behave linearly, which means that the stress is proportional to the strain. Nonlinear static analysis takes into account the nonlinear behavior of materials, such as plasticity and creep.

5.4.2 MODELING OF SOLID ELEMENT:

The SOLID45 element is a higher-order, 3D, 8-node element that can be used to model a wide variety of solids. It has the following capabilities:

- Plasticity: The element can model the plastic behavior of materials, including yielding, strain hardening, and necking.
- Creep: The element can model the creep behavior of materials, including time- dependent deformation and stress relaxation.
- Swelling: The element can model the swelling behavior of materials, such as those that are exposed to high temperatures or pressures.
- Stress stiffening: The element can model the stress stiffening behavior of materials, such as those that become stiffer under load.
- Large deflection: The element can model large deflections of structures, such as those that occur under impact loads.
- Large strain: The element can model large strains of materials, such as those that occur under high pressures.
- Mixed formulation: The element can use a mixed formulation to simulate deformations of nearly incompressible elastic plastic materials, and fully incompressible hyper elastic materials.

The geometry of the SOLID45 element is shown in the figure. The element has 8 nodes, each of which has three degrees of freedom: translations in the x, y, and z directions. The element input data includes the material properties, which can be either isotropic or anisotropic. Orthotropic and anisotropic material directions correspond to the element coordinate directions.

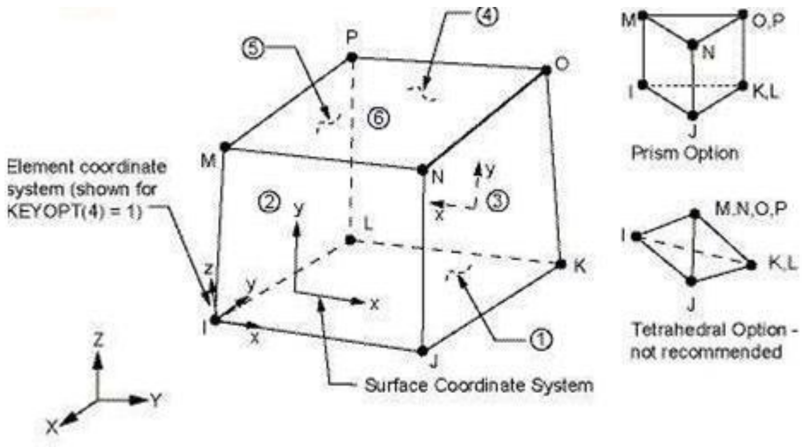


Figure5.1: Modelling of solid element. [18]

5.5 SYSTEM CONFIGURATIONS:

The computational numerical analysis in the current work is carried out using ANSYS version 15.0 on a Pentium IV system with 4GB of RAM and a 160GB hard drive running Windows XP.

6 FINITE ELEMENT METHOD

6.1 INTRODUCTION

- The finite element method (FEM) is a mathematical tool for solving differential equations. It is used to analyze the behavior of structures and materials under different loads and conditions.
- The basic concept of FEM is to divide the body or structure into smaller elements called finite elements. Each element is connected to its neighboring elements at nodes. The displacements and stresses within each element are then approximated by mathematical functions called shape functions.
- The FEM is a powerful tool that can be used to solve a wide variety of problems, including:
- Structural analysis: The FEM can be used to analyze the stresses and deformations of structures under different loads. This is used in the design of bridges, buildings, and other structures.
- Mechanical design: The FEM can be used to design and optimize the performance of mechanical components, such as gears, bearings, and shafts.
- Materials science: The FEM can be used to study the behavior of materials under different conditions, such as high temperatures or stresses. This is used in the development of new materials and the improvement of existing materials.
- Biomedical engineering: The FEM can be used to study the behavior of biological tissues and organs. This is used in the design of medical implants and the development of new medical treatments.
- Aerospace engineering: The FEM is used in the design and analysis of aircraft, spacecraft, and other vehicles.
- The FEM is a versatile tool that can be used to solve a wide variety of problems. It is a valuable

tool for engineers and scientists in many different fields.

- Structural analysis
- Thermal analysis
- Vibrations and Dynamics
- Buckling analysis
- Acoustics
- Fluid flow simulations
- Crash simulations
- Mold flow simulations

Nowadays, even the most simple of products rely on the finite element method for design evaluation. This is because contemporary design problems usually can not be solved as accurately & cheaply using any other method that is currently available. Physical testing was the norm in the years gone by, but now it is simply too expensive and time-consuming also.

6.2 BASIC CONCEPTS:

The Finite Element Method is based on the concepts of breaking down a complex entity into manageable bits or creating a complex object out of simple building blocks. This straightforward concept is used frequently in engineering as well as in daily life. A simple

example, such as calculating the area of a circle, can be used to illustrate the FEA idea. Area of one Triangle: Si = ½ * R2* Sin □I

Area of the Circle: SN = ½ * R2 * N * Sin (2 □ / N) □□ R2 as N □□

Where N = total number of triangles (elements)

Divide the aforementioned area into numerous equal parts if one wishes to calculate the area of the circle without using the standard procedure. The overall area of the circle is calculated by multiplying the area of each triangle by the quantity of such segments.

6.3 A BRIEF HISTORY OF THE FEM:

WHO

The mathematician Courant, the aircraft industry's Turner, the California university's Clough, the aircraft industry's Martin, and the German university's Argyris are all mentioned in the reference. However, it was probably independently founded by a number of explorers.

WHEN

- ✓ The first mathematical formulation of the concept appeared in the 1940s.
- ✓ 1950s application to straightforward technical issues.
- ✓ The 1960s saw the huge computer's implementation.
- ✓ The 1980s saw the development of pre- and post-processors.
- ✓ Analysis of significant structural issues from the 1990s.

WHERE

Implementation and application were concentrated mostly in the aerospace and automotive sectors (because these industries had access to big, quick computers).

WHAT

Large numbers of algebraic equations are arranged into field problems in the form of matrices, and matrices are solved. Algebraic equations are used to represent differential equations. To create the complicated geometry of the engineering challenge, blocks of various geometries are connected.

WHY

The benefit of performing a FEM analysis over building and testing is that it is relatively easy to change the shape, material, and loads for the new product. Almost any problem that can be described as a field problem can be solved using this strategy.

6.4 AVAILABLE COMMERCIAL FEM SOFTWARE PACKAGES

- ANSYS (General purpose, PC, and workstations)
- SDRC/I-DEAS (Complete CAD/CAM/CAE package)
- NASTRAN (General purpose FEA on mainframes)
- LS-DYNA 3D (Crash/impact simulations)
- ABAQUS (Nonlinear dynamic analysis)
- NISA (A General-purpose FEA tool)
- PATRAN (Pre/Post processor)
- HYPER MESH (Pre/post-processor)

MORE ABOUT FEA

The finite element method (FEM) was first developed for use in the aerospace and nuclear industries, where the safety of the structures is critical. Today, the method is used in a wide variety of industries, including automotive, civil, mechanical, and biomedical engineering.

The FEM is a numerical method that approximates the solution of partial differential equations (PDEs) by dividing the domain of the PDE into a finite number of smaller subdomains, called finite elements. The PDE is then solved within each finite element, and the solutions from the individual finite elements are then combined to obtain the solution for the entire domain.

The FEM is a powerful tool that can be used to solve a wide variety of problems, including:

- Structural analysis: The FEM can be used to analyze the stresses and deformations of structures under different loads. This is used in the design of bridges, buildings, and other structures.
- Mechanical design: The FEM can be used to design and optimize the performance of mechanical components, such as gears, bearings, and shafts.
- Materials science: The FEM can be used to study the behavior of materials under different conditions, such as high temperatures or stresses. This is used in the development of new materials and the improvement of existing materials.
- Biomedical engineering: The FEM can be used to study the behavior of biological tissues and organs. This is used in the design of medical implants and the development of new medical

treatments.

- Fluid dynamics: The FEM can be used to simulate the flow of fluids, such as air and water. This is used in the design of aircraft, ships, and other vehicles.
- Heat transfer: The FEM can be used to simulate the flow of heat. This is used in the design of heat exchangers and other devices that transfer heat.

6.5 THE BASIC STEPS INVOLVED IN FEA

The finite element method (FEM) is a numerical method for solving partial differential equations (PDEs). It is used to analyze the behavior of structures and materials under different loads and conditions.

The basic steps of FEM are:

1. Discretization of the domain: The structure or material is divided into a finite number of smaller subdomains, called finite elements. The PDE is then solved within each finite element.
2. Application of boundary conditions: The boundary conditions are applied to the finite elements. Boundary conditions are the constraints that are imposed on the structure or material, such as fixed supports or applied loads.
3. Assembling the system equations: The equations for the finite elements are assembled into a system of equations. The system of equations is typically a matrix equation.
4. Solution for system equations: The system of equations is solved to find the unknown nodal displacements. The nodal displacements are the displacements of the nodes of the finite elements.
5. Post-processing the results: The results of the analysis are post-processed to obtain the desired information, such as stresses, strains, and deformations.

What is an Element?

A study system can be separated into elements, which are separate entities. An element definition can be specified using nodes. The element's shape (area, length, and volume) is determined by the nodes that make up its structure.

What are Nodes?

Nodes are points in space where the degrees of freedom (DOFs) are defined. The DOFs for a node represent the possible movements of that node due to the loading of the structure.Elements are the basic building blocks of a finite element model. They are made up of nodes and are used to approximate the behavior of the structure. The shape of an element is defined by the nodes that make it upThe order of an element refers to the number of DOFs at each node. Linear elements have one DOF per node, quadratic elements have two DOFs per node, and so on.Higher-order elements can represent complex shapes more accurately than lower-order elements. However, they also require more computational resources to solve.

6.6 BRIEF OVERVIEW OF STRUCTURAL STATIC ANALYSIS:

Static analysis is one in which the loads/boundary conditions are not the functions of time and the assumption here is that the load is applied gradually. The most common application of FEA is the solution of stress-related design problems.

Typically in a static analysis the kind of matrix solved is

[K] * [X] = [F]

Where K is called the stiffness matrix, X is the displacement vector and F is the load matrix. This is a force balance equation. Sometimes, the K matrix is the function X. Such systems are called non-linear systems.

Nodal Displacements ui, uj Nodal Forces fi, fj Spring constant k

Spring force displacement relationship F = k □ with □ = uj – ui

K = F/□ (>0) is the force needed to produce a unit stretch Consider the equilibrium forces for the spring. At node i, we have fi = -F = -k(uj – ui) = kui – kuj

And at node j,

fj = F = k(uj – ui) = - kui + kuElement Quality Requirements: Specific factors affect how well the outcomes turn out. The engineer must make sure that these parameters are kept inside the software's allowable bounds in order to get good results. The terms "mesh quality parameters" refer to this.Quad and Hexa elements are the only ones that experience warp ageing. A plane is defined by three points, thus if one of the four nodes is at an angle to another plane, it generates a warp. Zero is the ideal intended warp age value.

6.7 PRE-PROCESSOR:

The preprocessor is the first step in an ANSYS analysis. It is used to prepare the input data for the analysis. The preprocessor has a number of features, including:

Solid modeling mesh generation capabilities

- Data base definition and manipulation of analysis data
- Parametric input, user files, macros, and extensive online documentation
- Extensive graphics capabilities

The preprocessor stage involves the following steps:

1. Specify the title: This is the name of the problem. It is optional, but it can be helpful if you are performing multiple design iterations on the same model.
2. Set the type of analysis: This can be structural, thermal, fluid, or electromagnetic analysis.
3. Create the model: The model can be created in the preprocessor or imported from another CAD software.
4. Define the element type: This is the type of element that will be used to mesh the model.
5. Assign material properties: This includes the Young's modulus, Poisson's ratio, density, and other properties of the material.
6. Apply mesh: This is the process of dividing the model into a mesh of finite elements. The preprocessor stage is an important step in the ANSYS analysis process. It is important to carefully define the input data to ensure that the analysis results are accurate.

6.8 SOLUTION PROCESSOR

The solver is the second step in an ANSYS analysis. It is used to solve the equations that were created in the preprocessor. The solver has three main steps:

1. Pre-solver: The pre-solver reads the model created by the preprocessor and formulates the mathematical representation of the model.
2. Mathematical engine: The mathematical engine solves the equations that were created in the pre-solver.
3. Post-solver: The post-solver calculates the results of the analysis, such as stresses, strains, and deformations.

The solver stage involves the following steps:

1. Select the solution type: This can be static, modal, or transient analysis.
2. Define the loads: This includes point loads, surface loads, thermal loads, and fluid loads.
3. Solve the equations: The solver uses a variety of numerical methods to solve the equations.
4. Calculate the results: The post-solver calculates the results of the analysis, such as stresses, strains, and deformations.

The solver stage is an important step in the ANSYS analysis process. It is important to choose the correct solution type and to define the loads accurately to ensure that the analysis results are accurate.

6.9 POST –PROCESSOR:

Post-processing is the third and final step in an ANSYS analysis. It is used to visualize and interpret the results of the analysis. The post-processor has a number of features, including:

- Geometry distortion: This shows how the geometry of the model has changed due to the applied loads.
- Stress and strain contours: This shows the distribution of stresses and strains in the model.
- Flow fields: This shows the flow of fluids or gasses in the model.
- Safety factor contours: This shows the areas of the model that are most likely to fail.
- Potential field results: This shows the distribution of electrical or magnetic potential in the model.
- Vector field displays: This shows the direction and magnitude of a vector field, such as velocity or force.
- Mode shapes: This shows the natural frequencies and mode shapes of the model.
- Time history graphs: This shows the variation of a quantity over time, such as displacement or stress.

The post-processor can also be used to perform algebraic operations, manipulate databases, and differentiate and integrate calculated results.

The post-processing stage is an important step in the ANSYS analysis process. It is important to visualize and interpret the results of the analysis to ensure that the results are accurate and that the design is safe.

6.10 REVIEW THE RESULTS:

Once the solution has been calculated, the ANSYS postprocessor can be used to review the results. There are two postprocessors available: POST1 and POST26.

- POST1: This is the general postprocessor and can be used to review the results at one sub-step over the entire model or selected portion of the model. It can be used to create contour displays, deform shapes, and tabular listings to review and interpret the results of the analysis. POST1 also offers many other capabilities, such as error estimation, load case combination, calculation among results data, and path operations.
- POST26: This is the time history post-processor and can be used to review results at specific points in the model over all time steps. It can be used to create graph plots of results, data vs. time, and tabular listings. Other POST26 capabilities include arithmetic calculations and complex algebra.

The solution of the analysis results in two types of values:

- Nodal degree of freedom values: These are the primary solution and represent the displacements, stresses, strains, and forces at the nodes of the model.
- Derived values: These are the element solution and represent the results of the analysis at each element, such as the element stiffness matrix and the element stresses.

Structural static analysis is a type of analysis that calculates the effects of loading conditions on a structure while ignoring inertia and damping effects. This means that the analysis assumes that the structure is not moving and that the loads are not changing over time. Static analysis can be used to determine the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects.

The types of loading that can be applied in static analysis include:

- Externally applied forces and pressures
- Steady-state inertial forces
- Imposed displacements
- Temperatures

6.11.1 MESH:

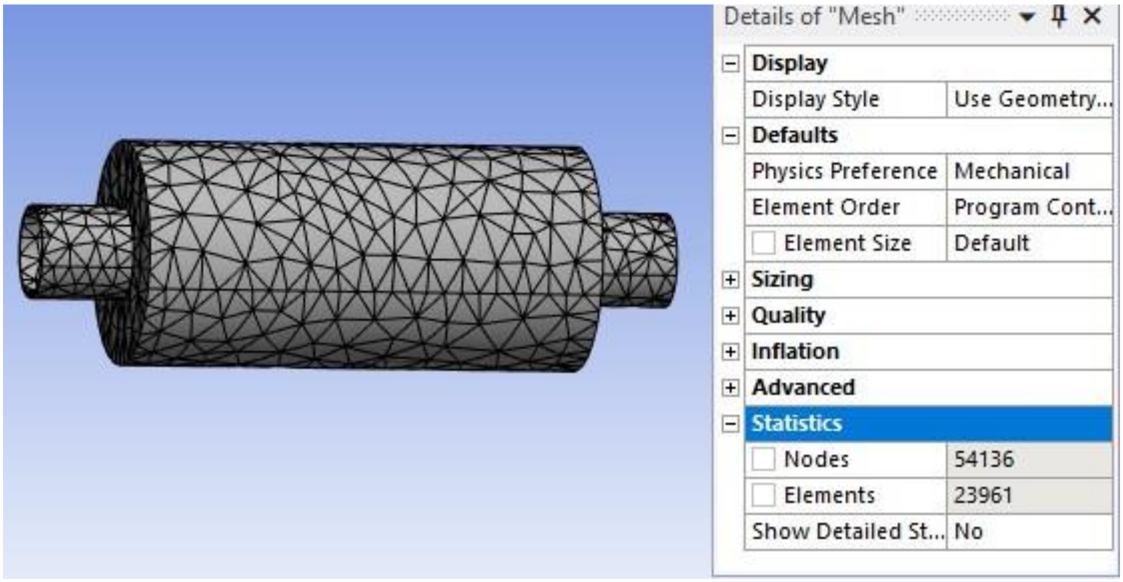


Figure 6.1:Meshing.[19]

6.11.2 BOUNDARY CONDITIONS

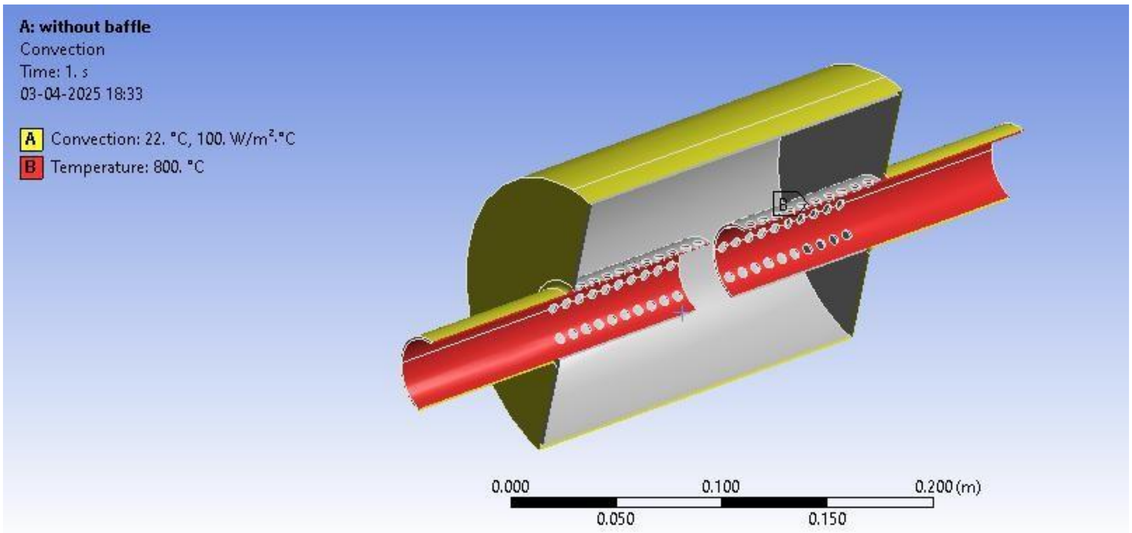


Figure 6.2:Thermal Boundary Conditions For Without Baffle.[20]

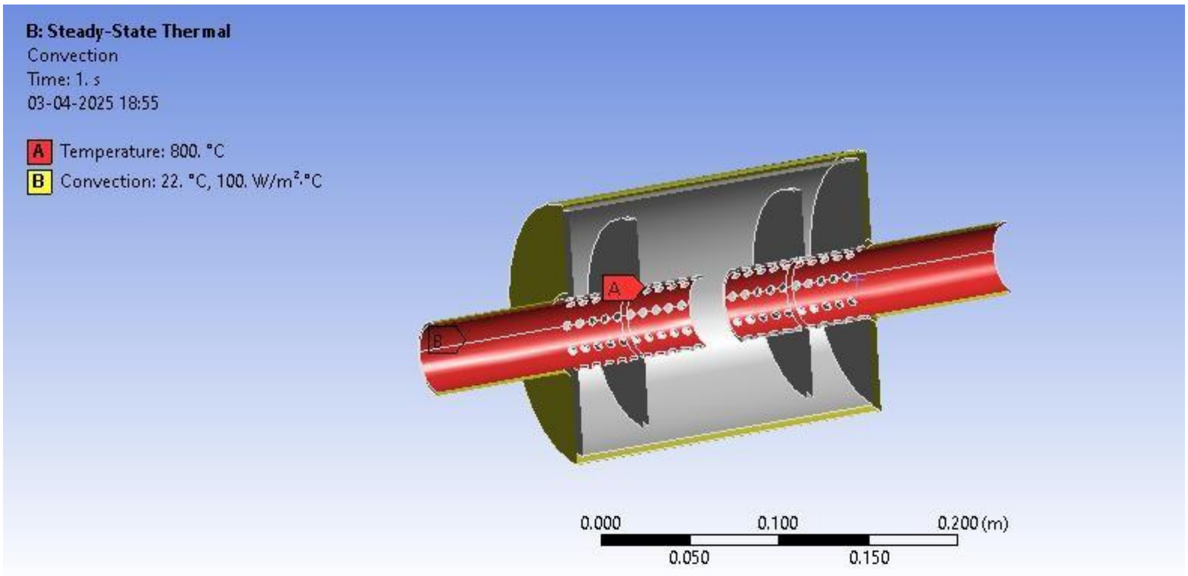


Figure 6.3: Thermal Boundary Conditions For With Baffle.[21]

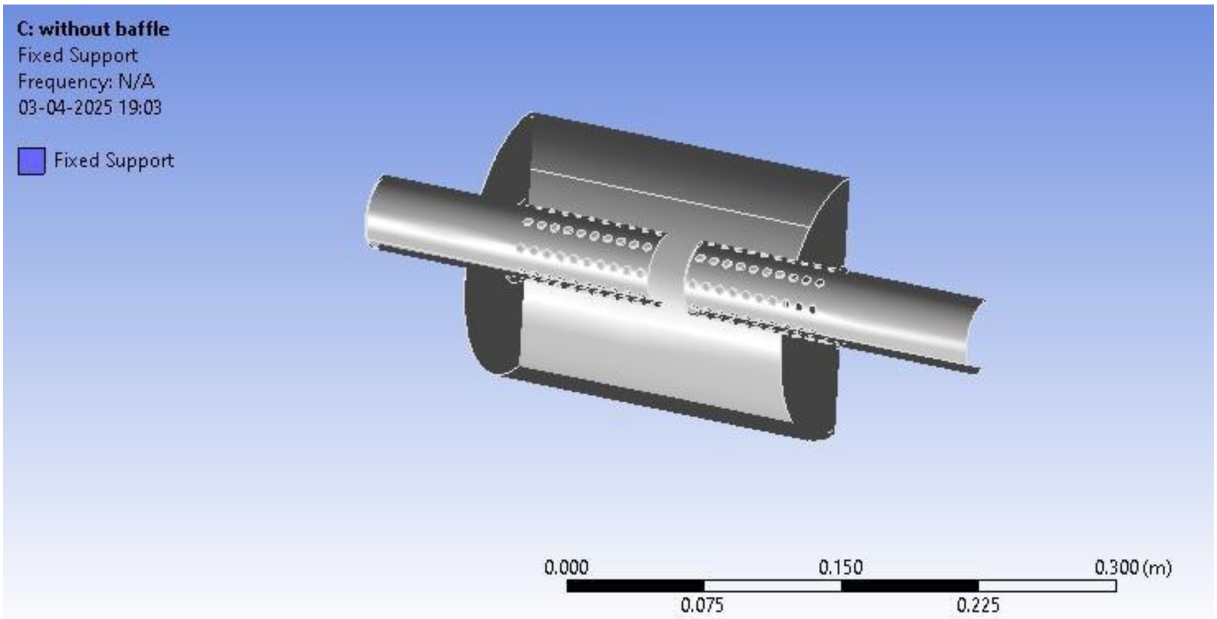


Figure 6.4**:** Modal Analysis boundary conditions.[22]

7 RESULTS AND DISCUSSIONS

The existing muffler having the Frequency of lower values. The new muffler was found to be superior to the existing one in terms of both acoustic performance and engine performance. With the new muffler, thickness of baffles modified 2mm the maximum Frequency obtained was high frequency obtained. The present work has thus experimentally shown that results from Finite Element Analysis can be modified and applied to an alternative design.

Advanced International Journal for Research (AIJFR)
E-ISSN: 3048-7641 ● Website: www.aijfr.com ● Email: editor@aijfr.com

7.1 THERMAL ANALYSIS ON WITHOUT BAFFLE DESIGN

Thermal analysis in ANSYS refers to a simulation process that helps you study how heat is distributed and behaves within a physical system or structure. This is useful for understanding temperature changes, heat flow, and the effects of thermal conditions on materials or mechanical components.

7.1.1 STAINLESS STEEL WITHOUT BAFFLE

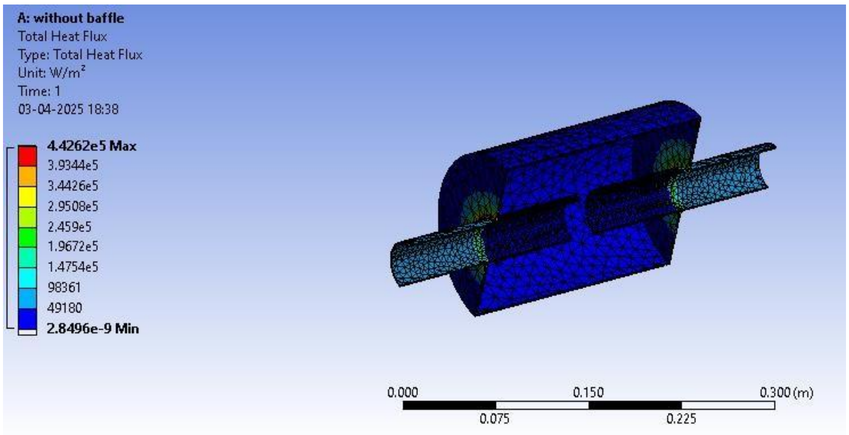


Figure 7.1:Total Heat Flux.[23]

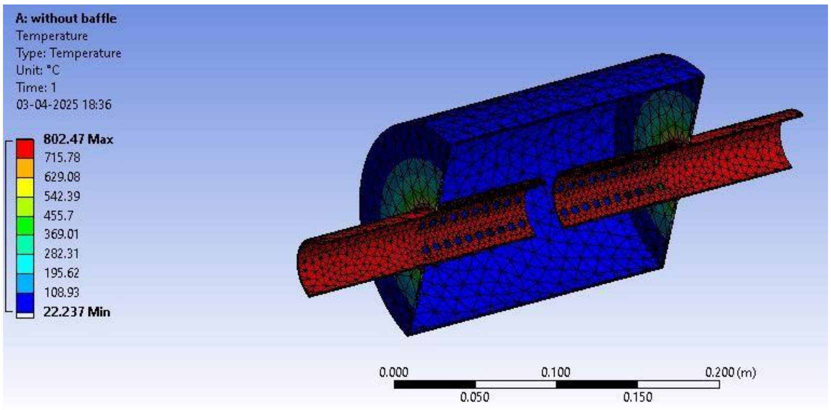


Figure 7.2:Temperature Distribution.[24]

7.1.2 ALUMINIUM ALLOY WITHOUT BAFFLE

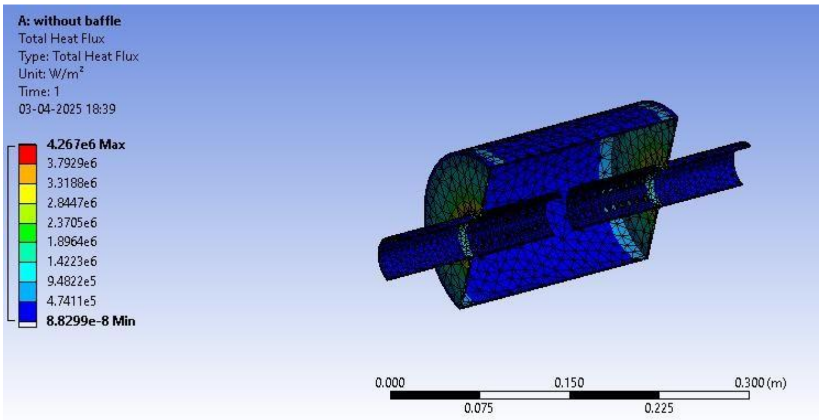


Figure 7.3**:** Total Heat Flux.[25]

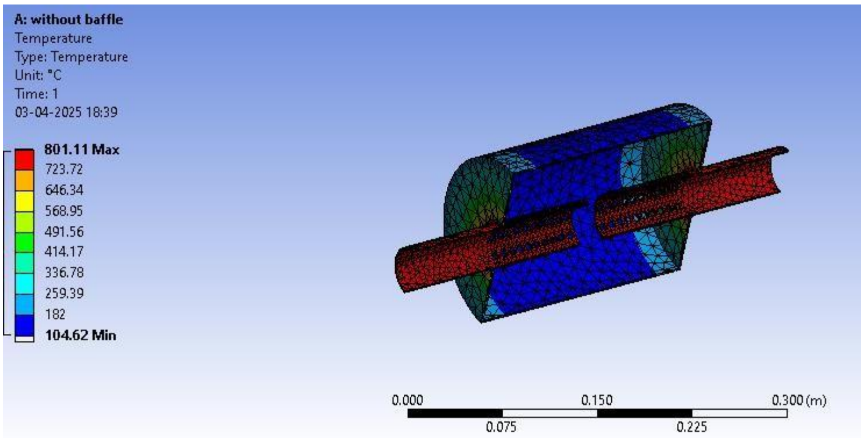


Figure 7.4**:**Temperature Distribution.[26]

7.1.3 TITANIUM ALLOY WITHOUT BAFFLE

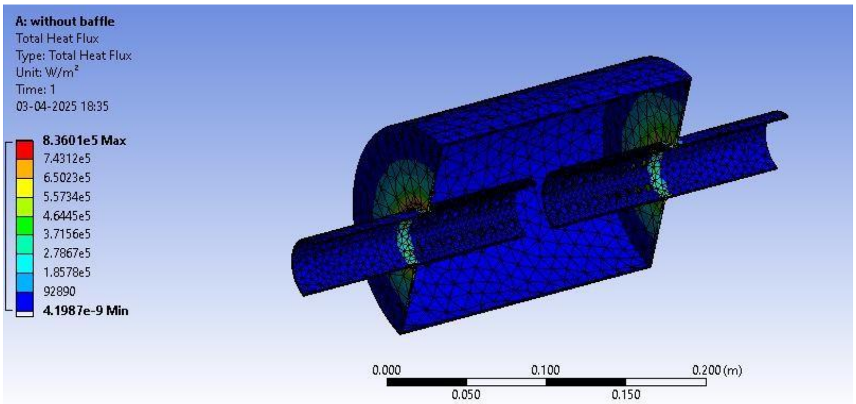


Figure 7.5**:**Total Heat Flux.[27]

Advanced International Journal for Research (AIJFR)
E-ISSN: 3048-7641 ● Website: www.aijfr.com ● Email: editor@aijfr.com

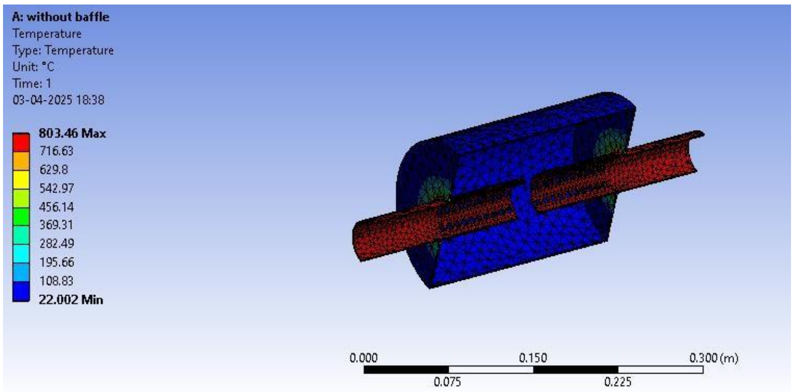


Figure 7.6**:**Temperature Distribution.[28]

7.2 THERMAL ANALYSIS ON WITH BAFFLE DESIGN

7.2.1 STAINLESS STEEL WITH BAFFLE

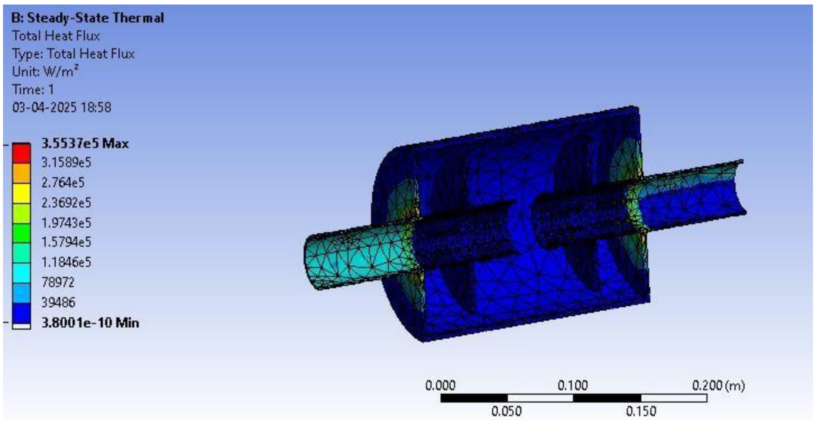


Figure 7.7**:**Total Heat Flux.[29]

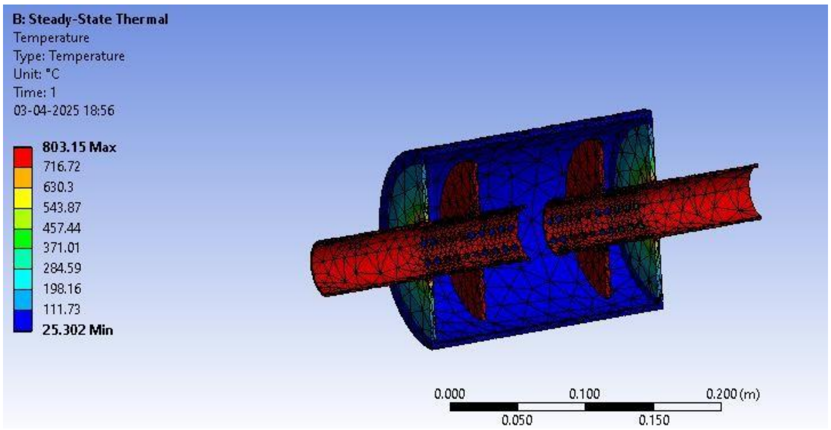


Figure 7.8:Temperature Distribution.[30]

Advanced International Journal for Research (AIJFR)
E-ISSN: 3048-7641 ● Website: www.aijfr.com ● Email: editor@aijfr.com

7.2.2 ALUMINIUM ALLOY WITH BAFFLE

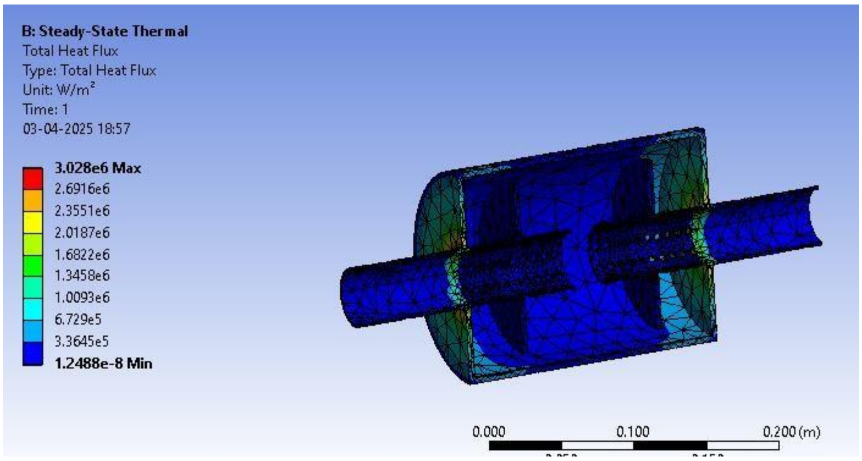


Figure 7.9**:**Total Heat Flux.[31]

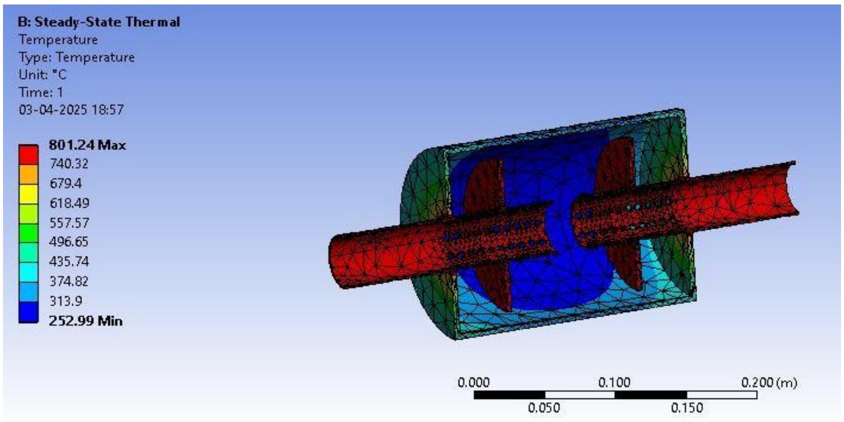


Figure 7.10**:**Temperature Distribution.[32]

7.2.3 TITANIUM ALLOY WITH BAFFLE

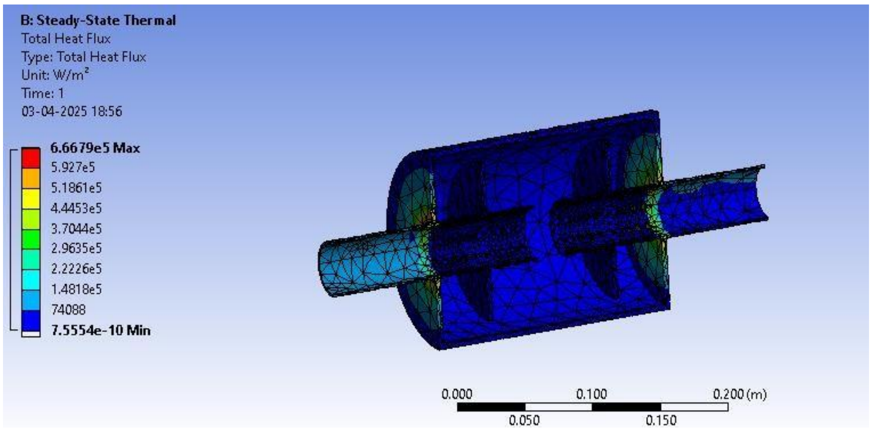


Figure 7.11:Total Heat Flux.[33]

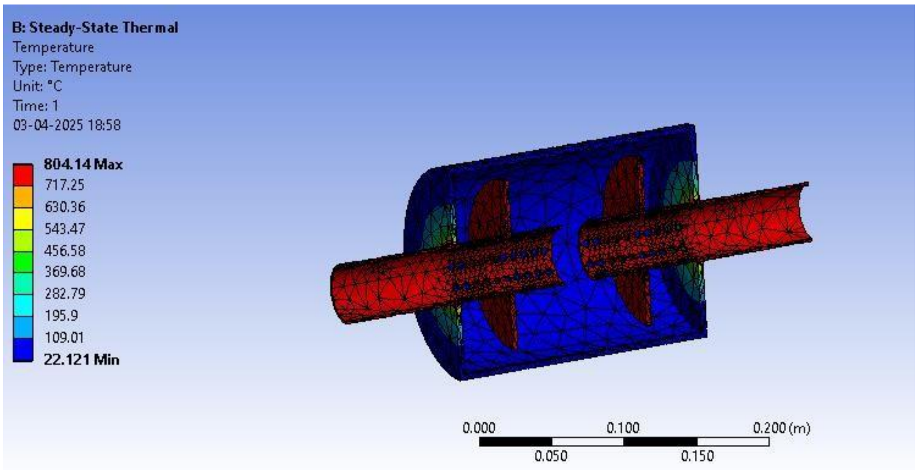


Figure 7.12**:**Temperature Distribution.[34]

7.2.4 TABULAR DATA FOR HEAT FLUX(w/mm2)

MATERIALS	WITHOUT BAFFLE*e^-5	WITH BAFFLE*e^-5
stainless steel	4.4262	3.553
Aluminum alloy	0.4267	0.328
titanium alloy	8.3601	6.667

Table 2: Tabular Data For Heat Flux(w/mm2)

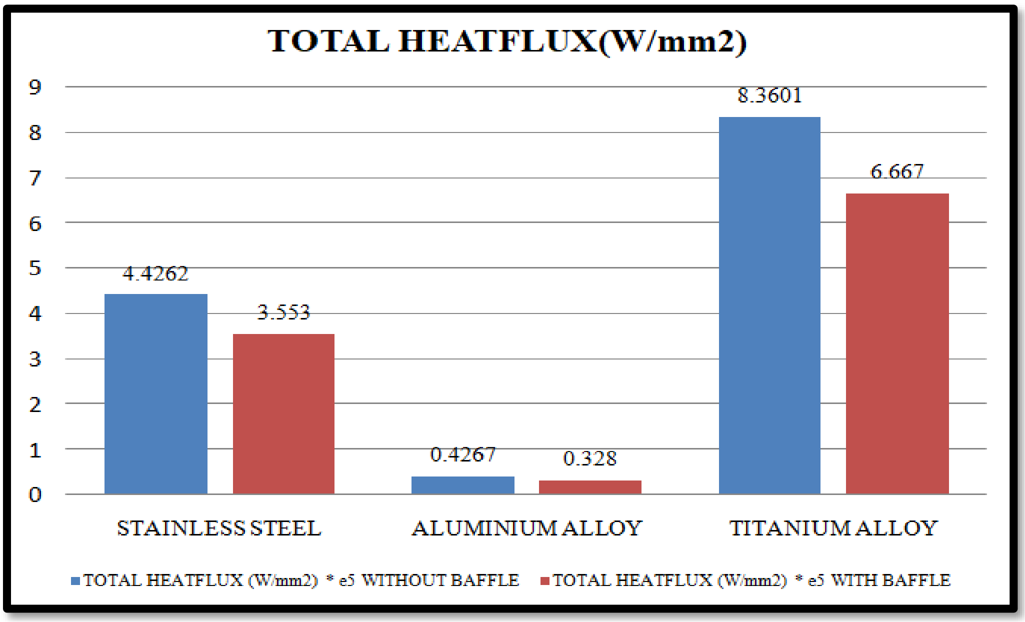
7.2.5 THERMAL TOTAL HEAT FLUX GRAPH

The given bar chart compares the Total Heat Flux (W/mm²) for three materials—Stainless Steel, Aluminum Alloy, and Titanium Alloy—under two conditions: without baffle (blue bars) and with baffle (red bars).

- **Stainless Steel:**
  - Without baffle: **4.4262 × 10^{-5} W/mm²**
  - With baffle**: 3.553 × 10^{-5} W/mm²**
  - A slight reduction in heat flux with the use of a baffle.
- **Aluminum Alloy:**
  - Without baffle: **0.4267 × 10^{-5} W/mm²**
  - With baffle: **0.328 × 10^{-5} W/mm²**
  - The lowest heat flux among the three materials, showing minimal heat transfer.

- **Titanium Alloy:**
  - Without baffle: **8.3601×10^{-5} W/mm²**
  - With baffle**: 6.667×10^{-5} W/mm²**
  - The highest heat flux, indicating significant heat transfer.

Titanium Alloy exhibits the highest total heat flux, making it the most efficient in dissipating heat. Thus, Titanium Alloy is the best material for applications where heat dissipation is crucial, while Aluminum Alloy is preferable for insulation or thermal resistance applications.



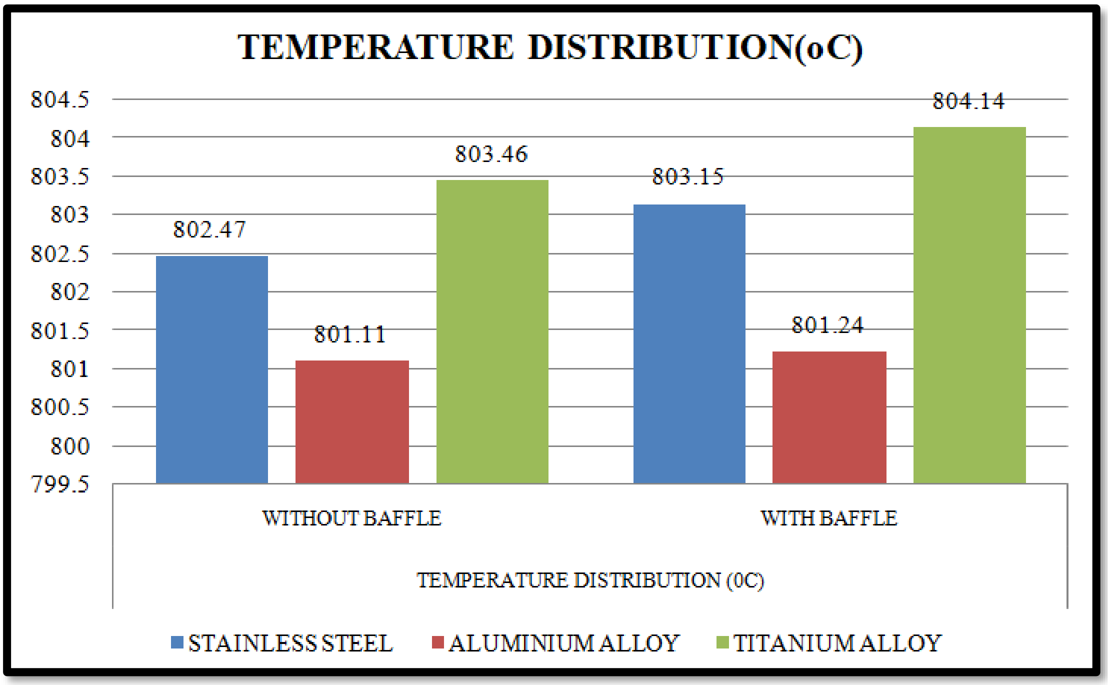
Graph 1:Total heat flux of both models

7.2.7 TABULAR DATA FOR TEMPERATURE DISTRIBUTION (°C)

MATERIALS			
MODELS	**STAINLESS STEEL**	**ALUMINIUM ALLOY**	**TITANIUM ALLOY**
WITHOUT BAFFLE	802.47	801.11	803.46
WITH BAFFLE	803.15	801.24	804.14

Table 3: Tabular Data For Temperature Distribution (°C)

7.2.8 THERMAL TEMPERATURE DISTRIBUTION GRAPH



Graph 2: Temperature Distribution of both models

The given bar chart represents the temperature distribution (°C) for three materials— Stainless Steel, Aluminum Alloy, and Titanium Alloy—under two conditions:

1. Without Baffle
2. With Baffle

- Stainless Steel:
  - Without baffle: 802.47°C
  - With baffle: 803.15°C
  - Slight increase in temperature with the use of a baffle.
- Aluminum Alloy:
  - Without baffle: 801.11°C
  - With baffle: 801.24°C
  - Smallest change in temperature among the three materials.
- Titanium Alloy:
  - Without baffle: 803.46°C
  - With baffle: 804.14°C
  - Highest recorded temperature among the materials, indicating higher heat retention.

Titanium Alloy Shows Highest Temperatures:With or without a baffle, titanium alloy consistently shows the highest temperatures, implying better heat retention or lower thermal conductivity.

7.3 MODAL ANALYSIS ON MUFFLER WITHOUT BAFFLE PLATES

The resonant frequency of muffler without having any baffle plates are found out six modes of total deformations and their maximum boundary loads are obtained in the following results for titanium alloy as a best among the thermal properties.

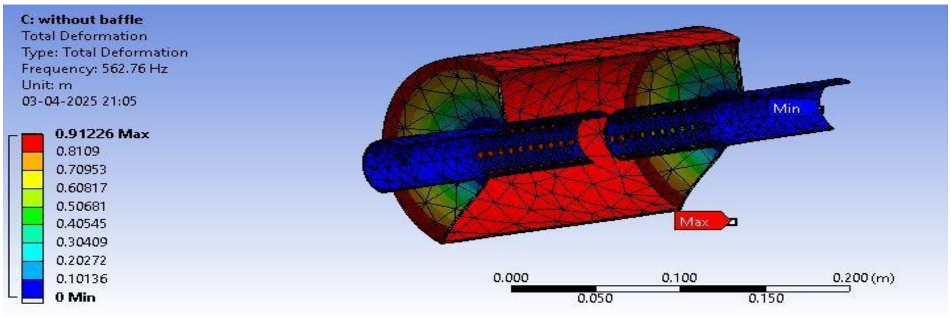


Figure 7.13**:** Mode1.[35]

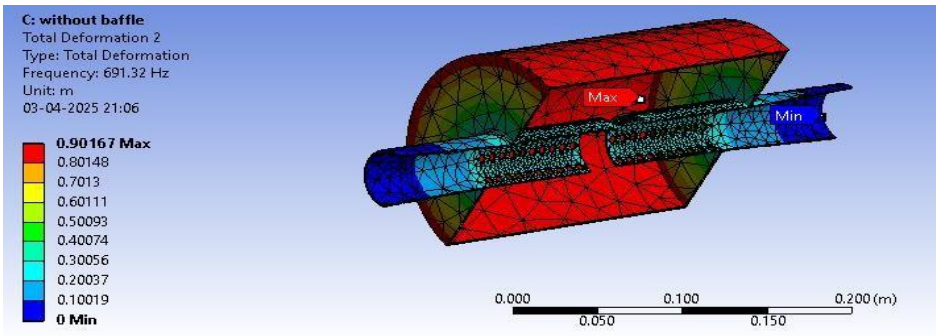


Figure 7.14**:** Mode2.[36]

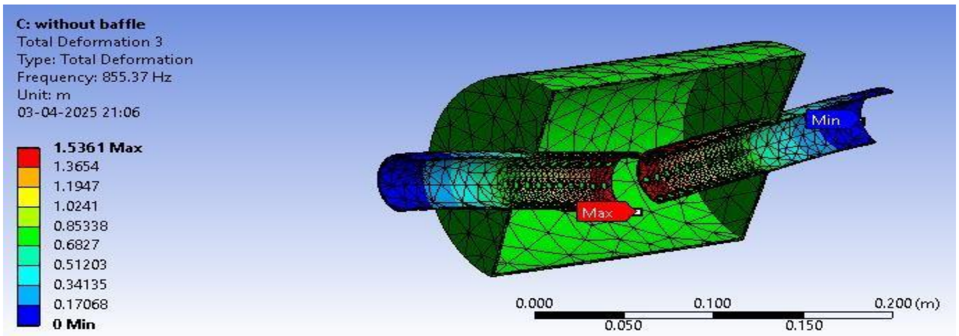


Figure 7.15**:** Mode3.[37]

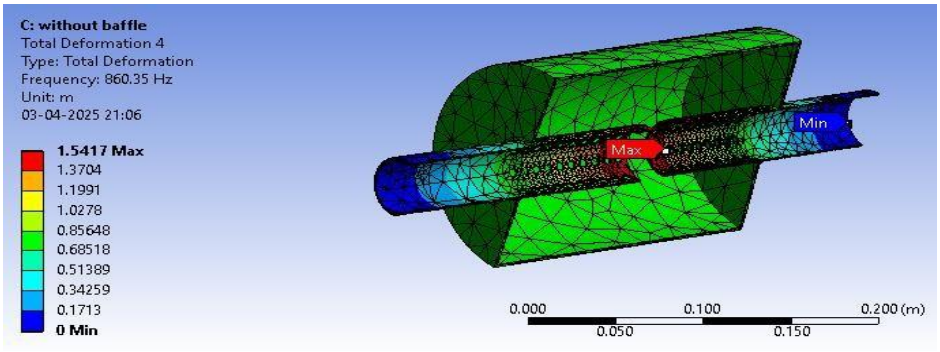


Figure 7.16**:** Mode4.[38]

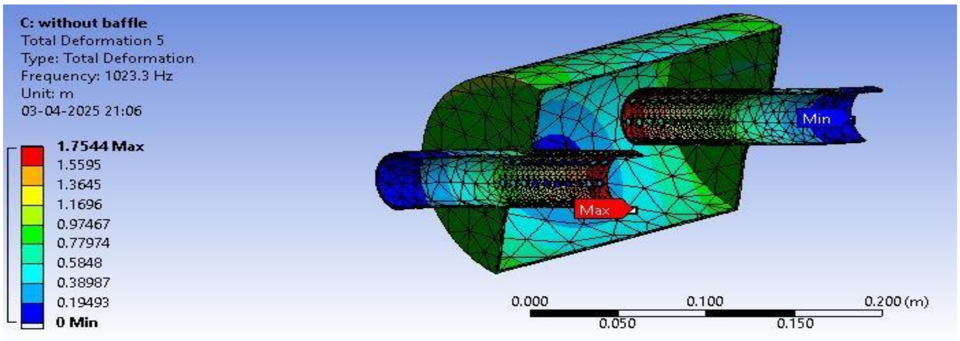


Figure 7.17**:**Mode5.[39]

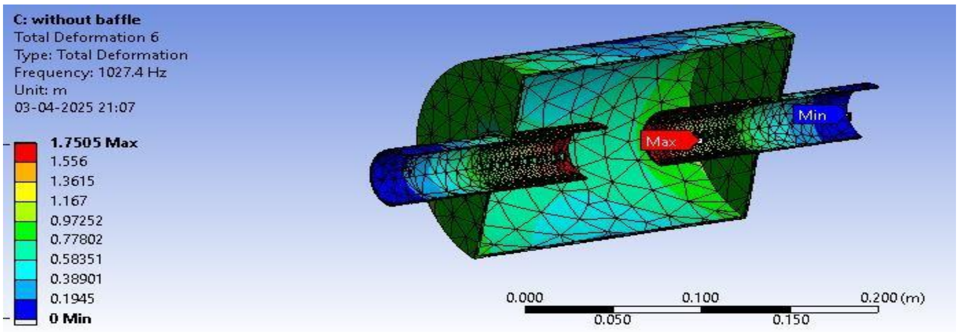


Figure 7.18**:**Mode6.[40]

7.4 MODAL ANALYSIS ON MUFFLER WITH BAFFLE PLATES

The resonant frequency of muffler with 2 mm thickness baffle plates are find out six modes of total deformations and their maximum boundary loads are obtained in the following results.

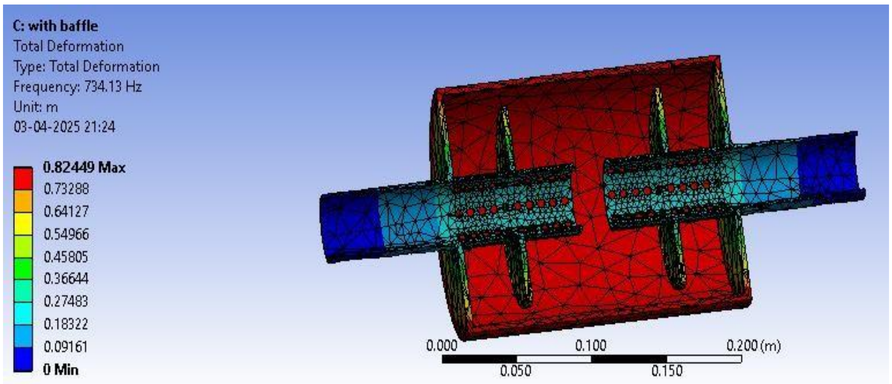


Figure 7.18:Mode1.[41]

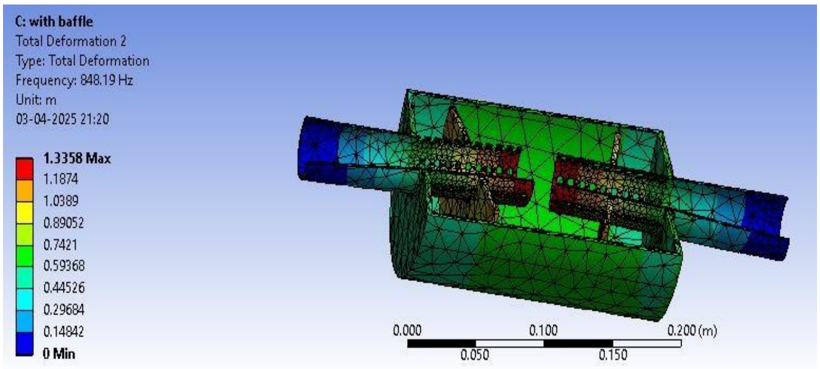


Figure 7.19:Mode2.[42]

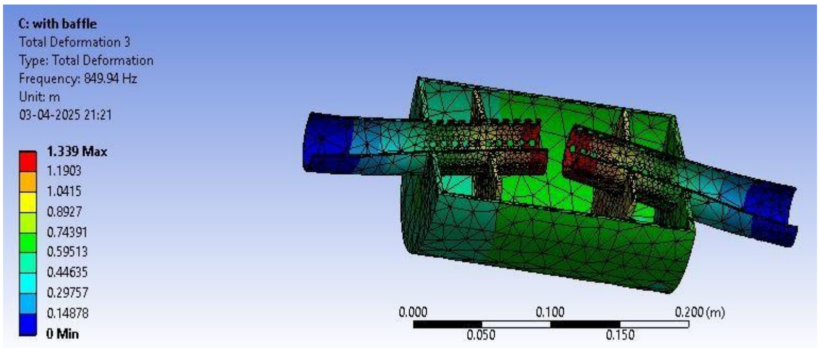


Figure 7.20:Mode3.[43]

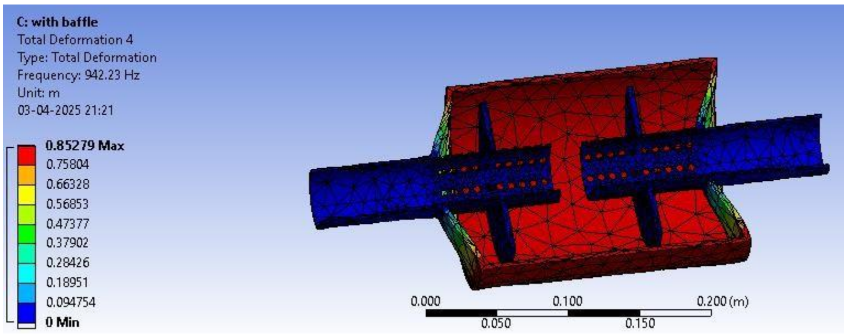


Figure 7.21**:**Mode4.[44]

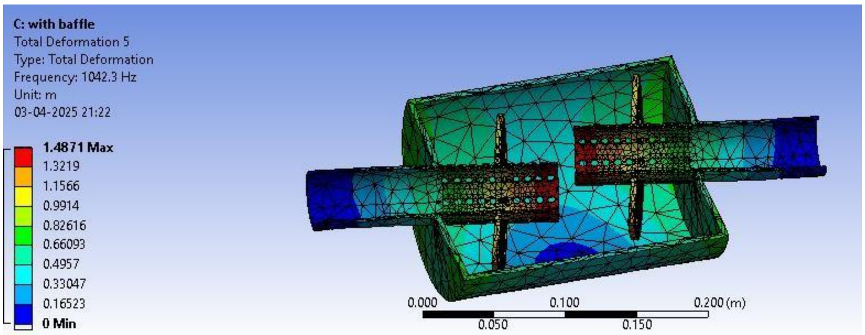


Figure 7.22**:**Mode5.[45]

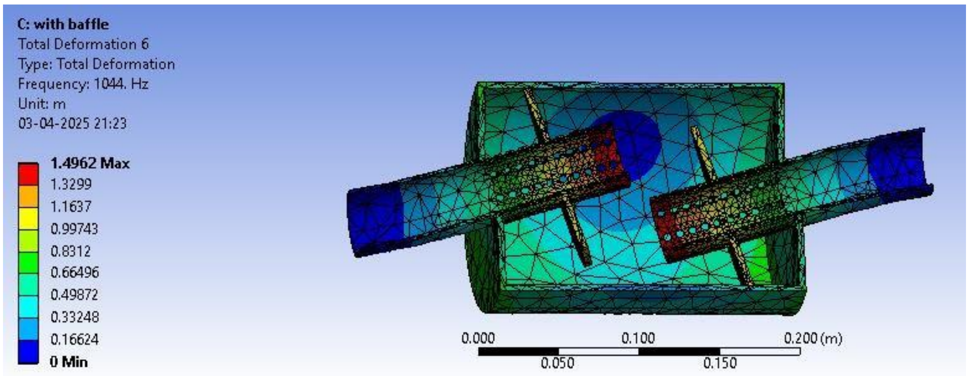


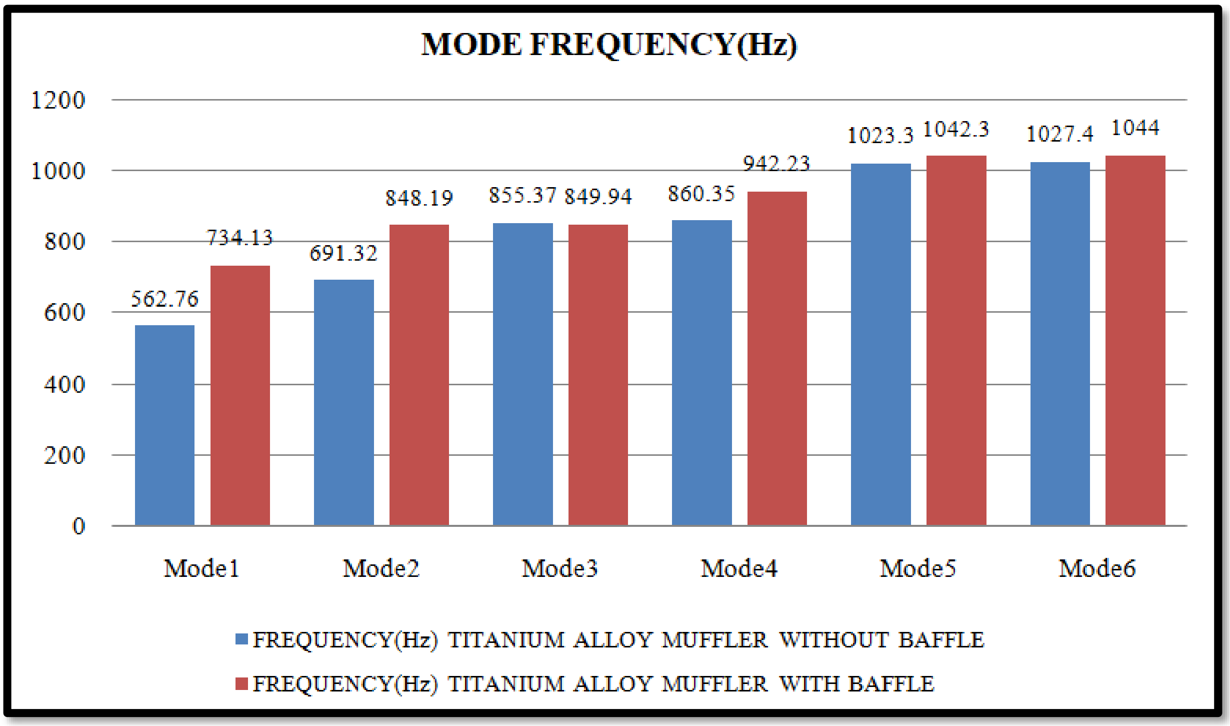
Figure 7.23**:**Mode6.[46]

7.4.1 GRAPH ANALYSIS: MODE FREQUENCY COMPARISON FOR TITANIUM ALLOY MUFFLER

After successful performing the modal analysis on the beat material concluded on the thermal analysis titanium alloy muffler with baffle plate and without plate the results are graphically represented below. The given bar chart represents the **mode frequencies (Hz)** for a **Titanium Alloy Muffler** in different vibration modes (**Mode 1 to Mode 6**) under two conditions:

1. **Without Baffle** (blue bars)
2. **With Baffle** (red bars)

- In **Mode 1**, the frequency increased from **562.76 Hz** (without baffle) to **734.13 Hz** (with baffle), indicating a significant rise in structural stiffness.
- In **Mode 2**, the frequency increased from **691.32 Hz** to **848.19 Hz**, again showing an improvement in vibration performance.
- In **Mode 3**, the frequency slightly decreased from **855.37 Hz** to **849.94 Hz**, indicating minimal change.
- In **Mode 4**, the frequency increased from **860.35 Hz** to **942.23 Hz**, showing improved vibration resistance.
- In **Mode 5**, the frequency increased from **1023.3 Hz** to **1042.3 Hz**, indicating better structural stability.
- In **Mode 6**, the frequency increased from **1027.4 Hz** to **1044 Hz**, reinforcing improved stiffness.



Graph 3: Mode Frequency.

Graph mode frequency of both models with proposed titanium alloy material

- **The addition of a baffle increases the mode frequencies in most cases**, indicating improved structural rigidity and reduced vibration effects.
- Higher frequencies suggest that **the model with a baffle is structurally superior**, as it is less prone to resonance at lower frequencies.
- The **Titanium Alloy Muffler with a Baffle is the best option** for applications requiring **better vibration resistance and structural integrity**.

7.5 STATIC STRUCTRUAL ANALYSIS ON MUFFLER

Static Structural Analysis in ANSYS is a type of simulation used to determine the response of a structure under static loading conditions. It helps engineers understand how structures like beams, frames, machines, buildings, or mechanical parts behave when subjected to loads that do not change over time.

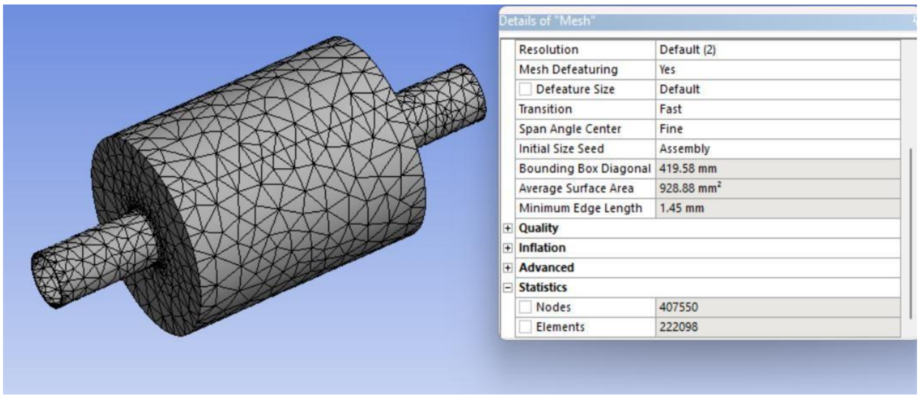


Figure 7.24:Meshing.[47]

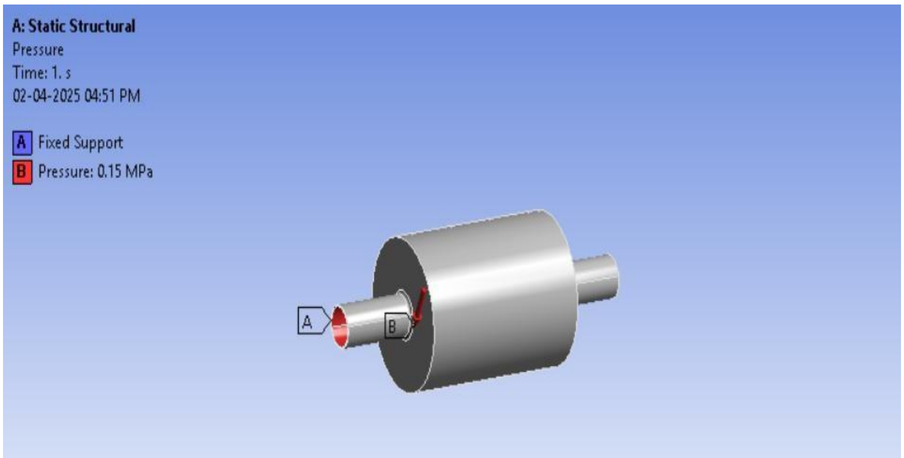


Figure 7.25**:**Static Structural Boundary Conditions.[48]

7.5.1 STAINLESS STEEL MATERIAL

7.5.2 STATIC STRUCTRUAL WITHOUT BAFFLE

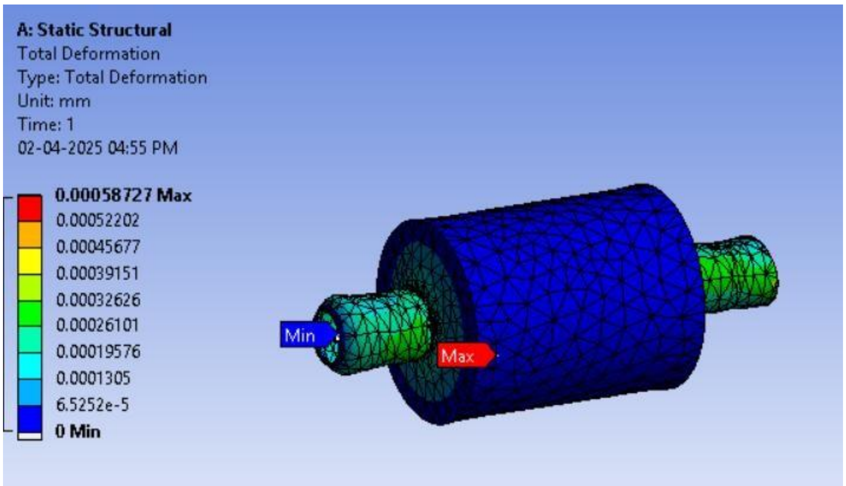


Figure 7.26:Total Deformation.[49]

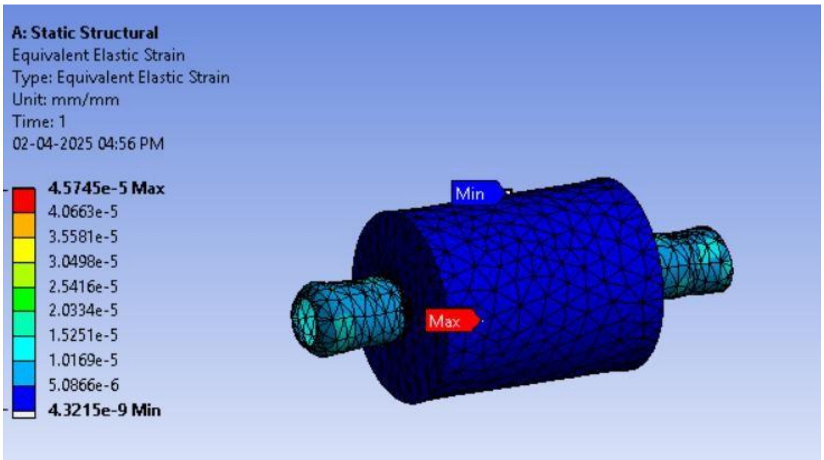


Figure 7.27:Strain.[50]

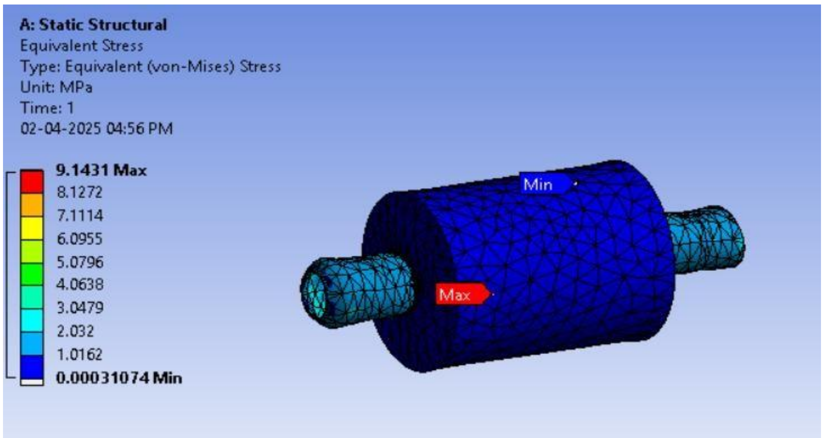


Figure 7.28:Stress.[51]

7.5.3 STATIC STRUCTRUAL WITH BAFFLE DESIGN 1

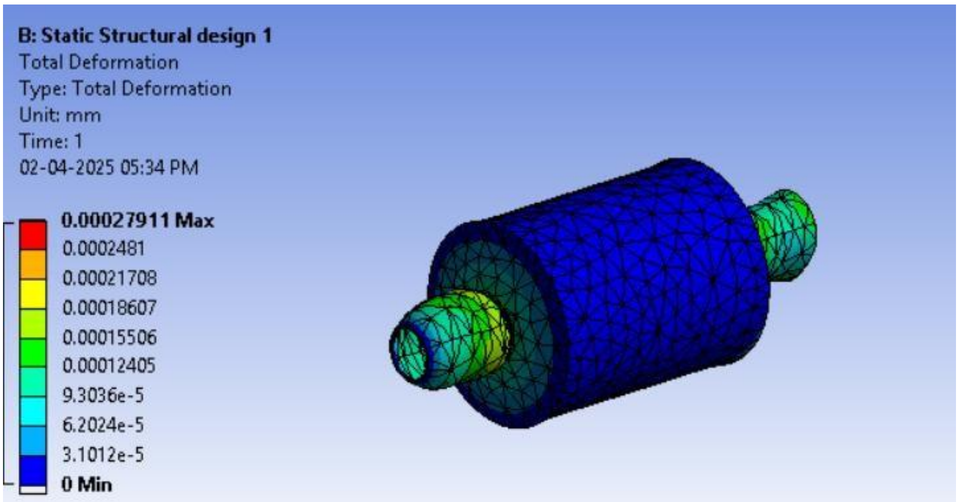


Figure 7.29:Total Deformation.[52]

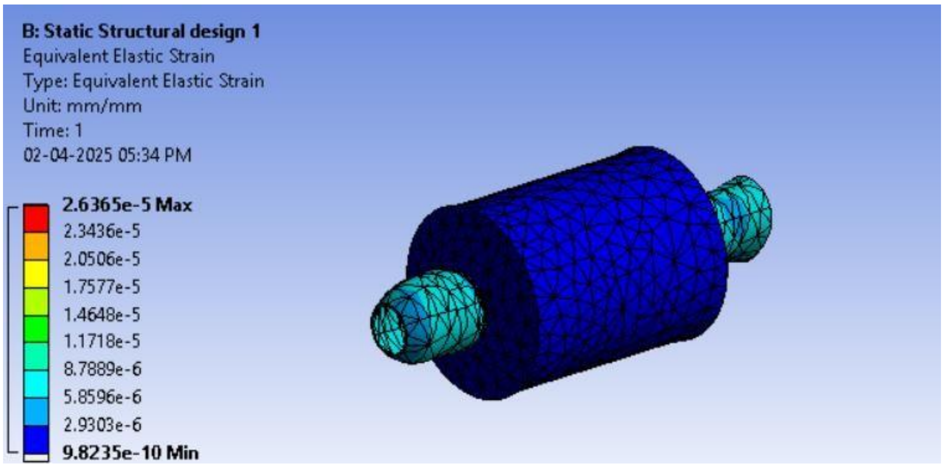


Figure 7.30:Strain.[53]

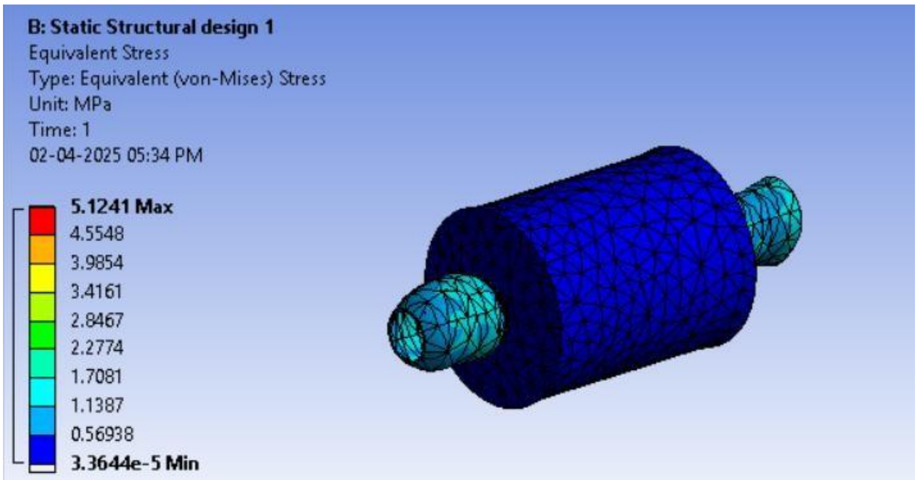


Figure 7.31:Stress.[54]

7.5.4 ALUMINIIUM ALLOY

7.5.5 STATIC STRUCTRUAL WITHOUT BAFFLE

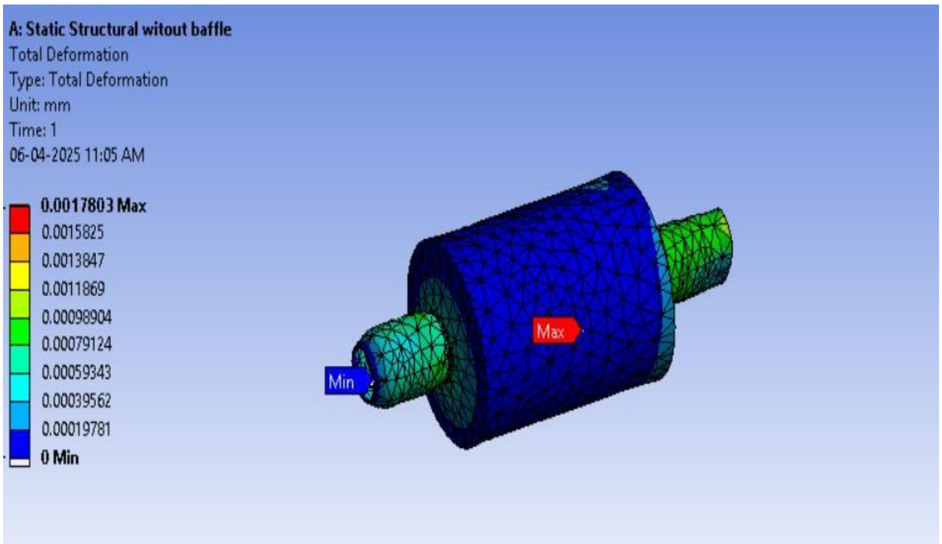


Figure 7.32:Total Deformation.[55]

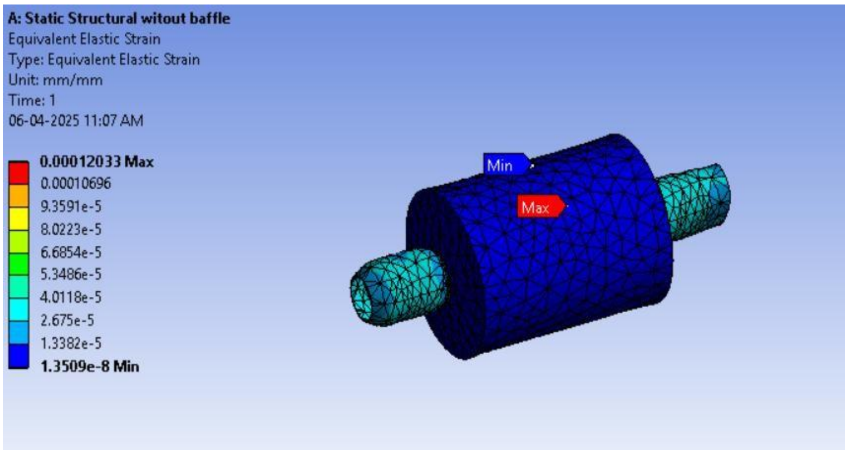


Figure 7.33:Strain.[56]

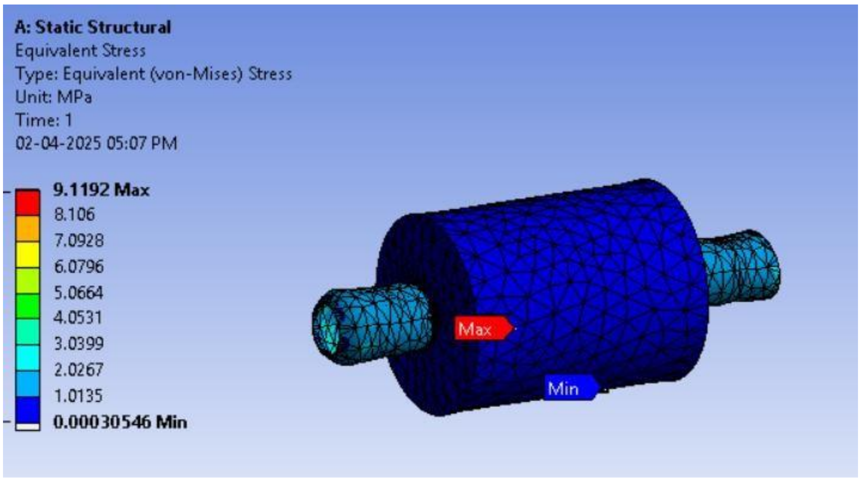


Figure 7.34:Stress.[57]

Advanced International Journal for Research (AIJFR)
E-ISSN: 3048-7641 ● Website: www.aijfr.com ● Email: editor@aijfr.com

7.5.6 STATIC STRUCTRUAL WITH BAFFLE DESIGN 1

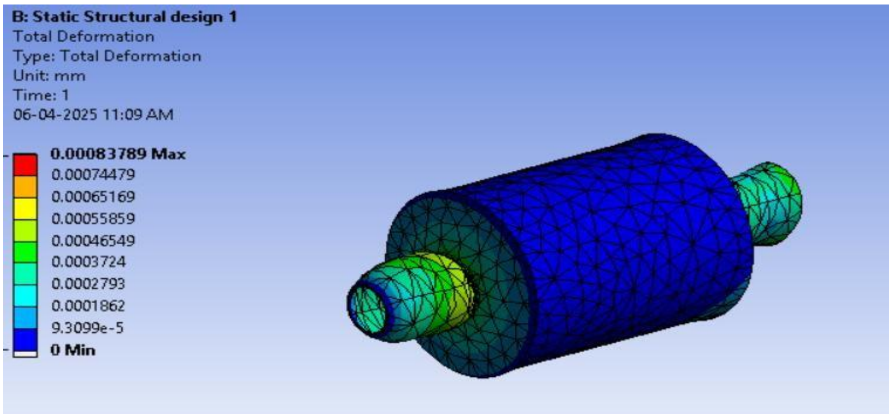


Figure 7.35:Total Deformation.[58]

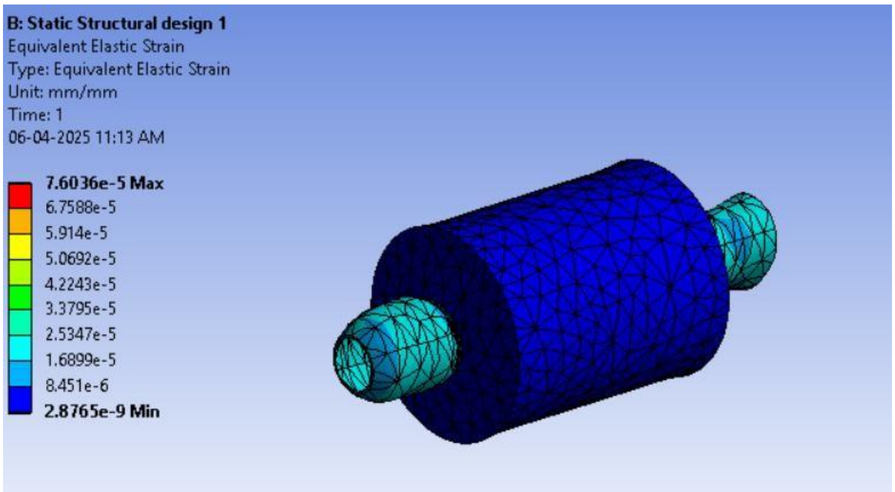


Figure 7.36:Strain.[59]

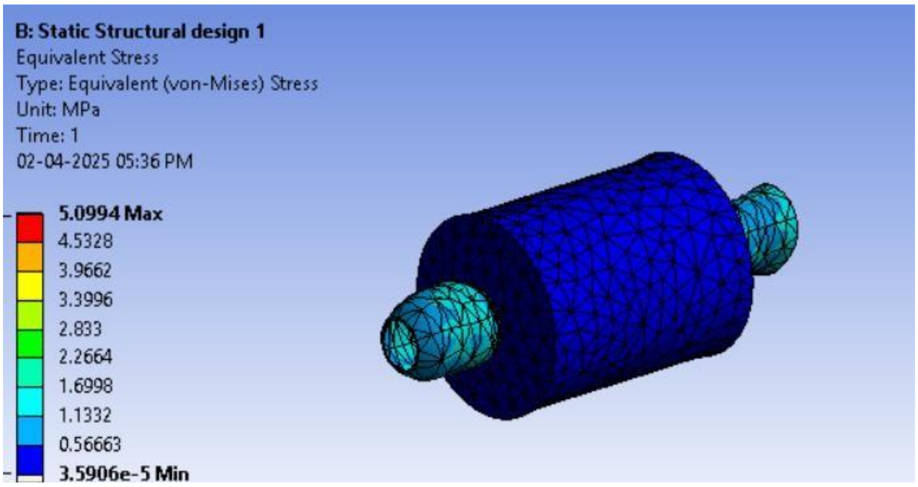


Figure 7.37:Stress.[60]

7.5.7 TITANIUM ALLOY

7.5.8 STATIC STRUCTRUAL WITHOUT BAFFLE

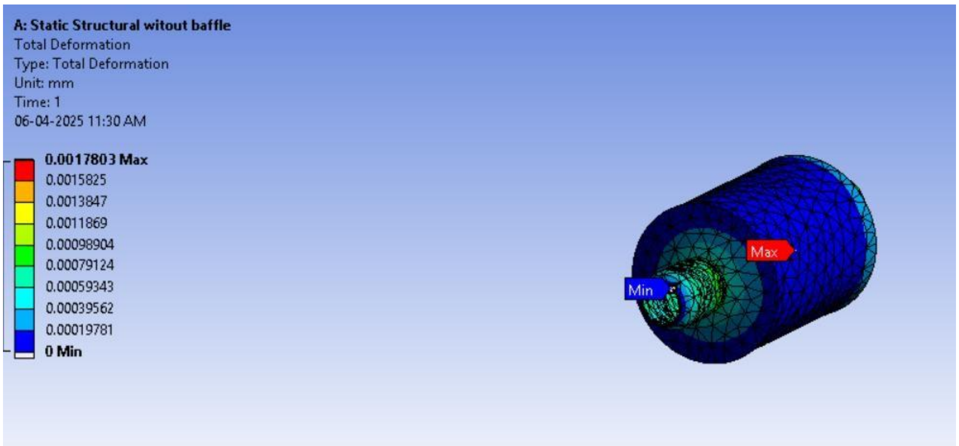


Figure 7.38:Total Deformation.[61]

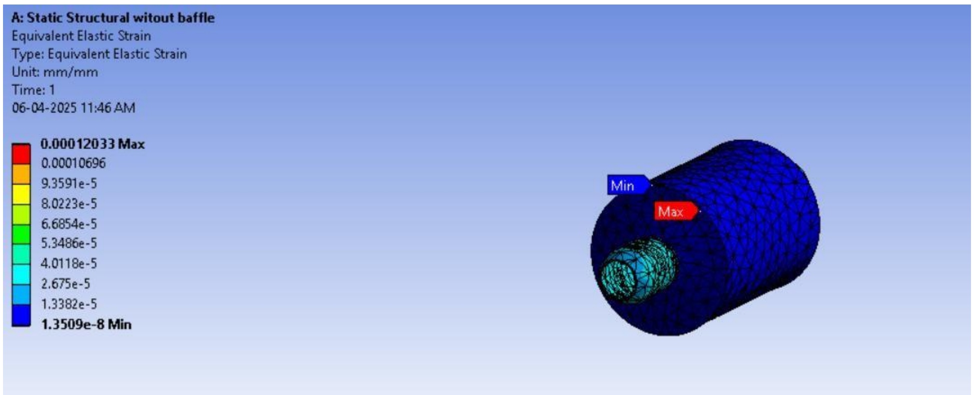


Figure 7.39:Strain.[62]

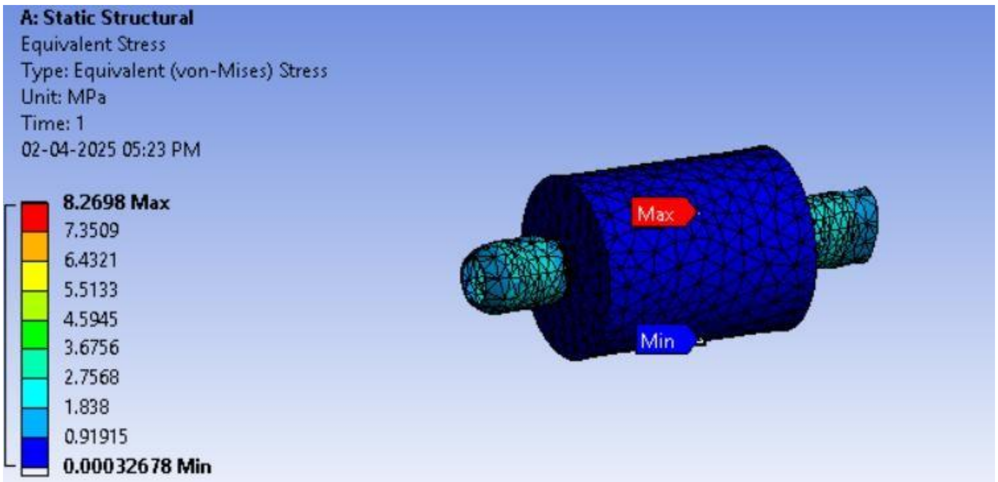


Figure 7.40:Stress.[63]

Advanced International Journal for Research (AIJFR)
E-ISSN: 3048-7641 ● Website: www.aijfr.com ● Email: editor@aijfr.com

7.5.9 STATIC STRUCTRUAL WITH BAFFLE DESIGN 1

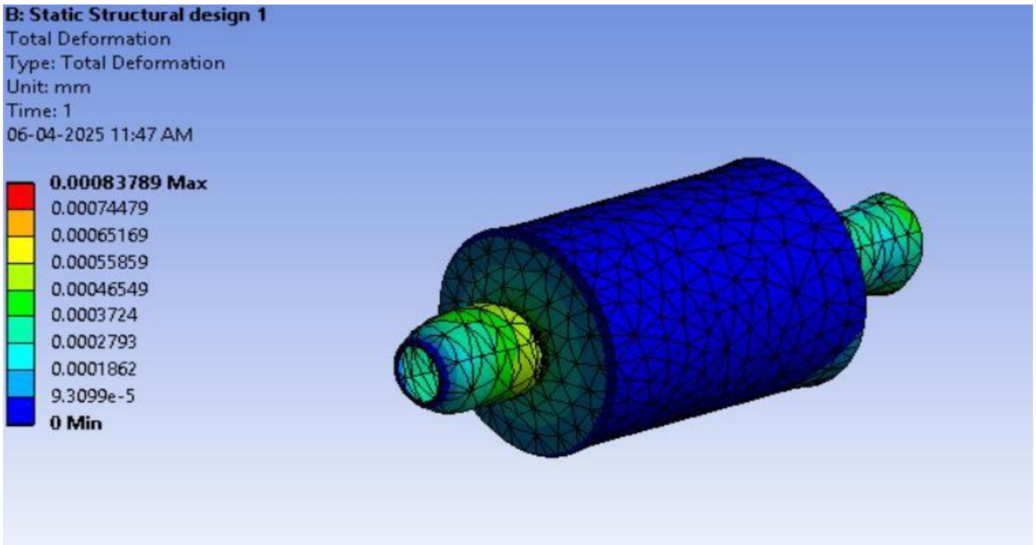


Figure 7.41:Total Deformation.[64]

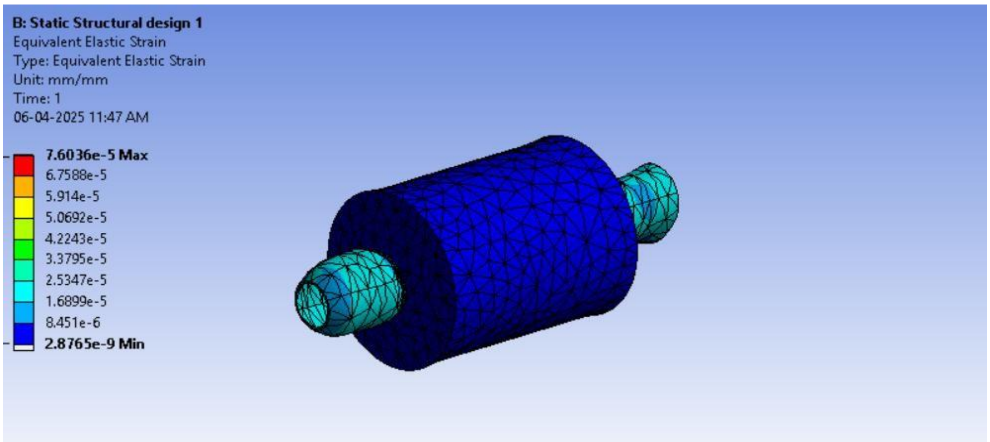


Figure 7.42:Strain.[65]

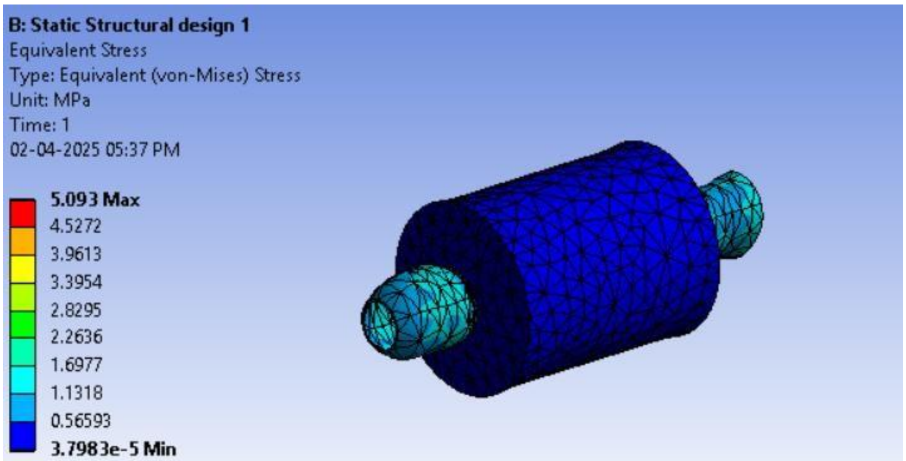


Figure 7.43**:**Stress.[66]

7.6 TABULAR DATA FOR EQUIVALENT STRESSES FOR BOTH MODELS

Equivalent vonmises stresses(Mpa)		
Materials	without baffle	with baffle design 1
Stainless steel	9.1431	5.1241
Aluminum alloy	9.1192	5.8994
Titanium alloy	8.269	5.093

Table 4: Tabular Data For Equivalent Stresses For Both Models

7.6.1 TABULAR DATA FOR EQUIVALENT STRAIN FOR BOTH MODELS

Equivalent Elastic strain		
Materials	without baffle	with baffle design
Stainless steel	0.00004574	0.000026365
Aluminum alloy	0.00012033	0.000076036
Titanium alloy	0.000041764	0.00004763

Table 5: Tabular Data For Equivalent Strain For Both Models

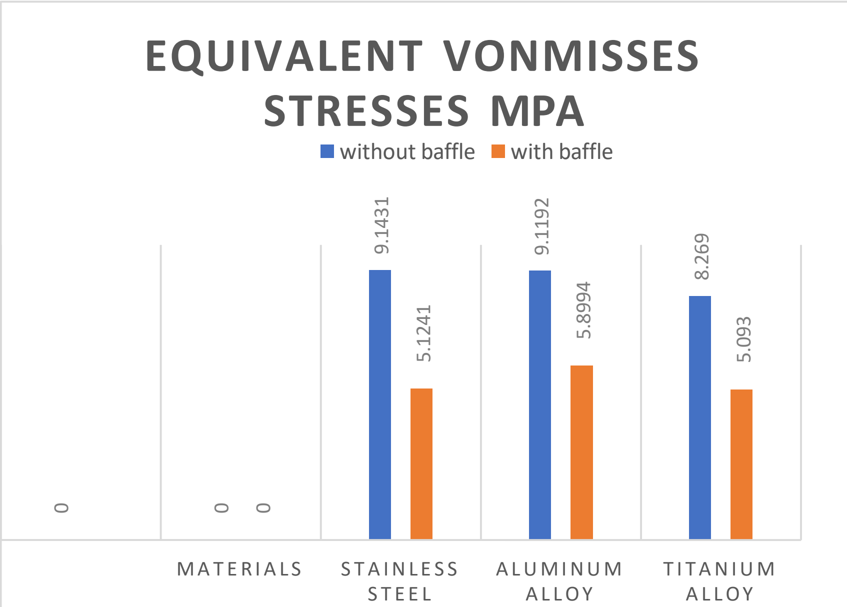
7.6.2 TABULAR DATA FOR TOTAL DEFORMATION(mm) FOR BOTH MODELS

Total Deformation(mm)		
Materials	without baffle	with baffle design 1
Stainless steel	0.00058727	0.00027911
Aluminum alloy	0.0017803	0.00083789
Titanium alloy	0.00059857	0.00053235

Table 6: Tabular Data For Total Deformation(Mm) For Both Models

7.6.3 GRAPH FOR EQUIVALENT STRESSES FOR BOTH MODELS

The given bar chart compares the Equivalent vonmises stresses(Mpa) for three materials— Stainless Steel, Aluminum Alloy, and Titanium Alloy—under two conditions: without baffle (blue bars) and with baffle (orange bars).



Graph 3: Equivalent Stresses For Both Models

OBSERVATIONS:

1. Stress Reduction with Baffle: All three materials show a significant reduction in equivalent von Mises stress when a baffle is used.

- Stainless Steel sees a reduction from 9.1431 MPa to 5.1241 MPa (about 44% drop).
- Aluminum Alloy drops from 9.1192 MPa to 5.8994 MPa (approx. 35% drop).
- Titanium Alloy decreases from 8.269 MPa to 5.903 MPa (about 29% drop).

2. Highest Stress Values: Stainless Steel and Aluminum Alloy both experience the highest stresses without a baffle, closely followed in magnitude.

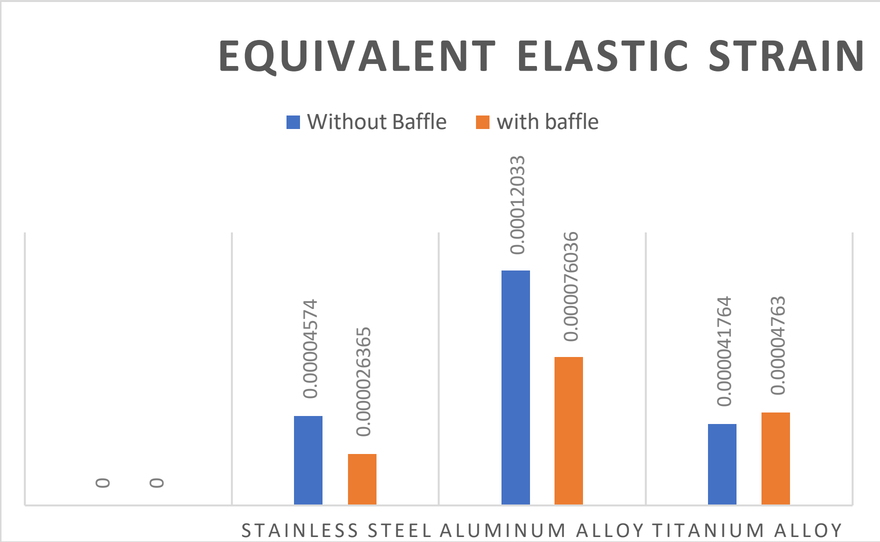
- **Titanium Alloy** shows the lowest stress values in both cases, making it more resistant under load.

3. Effectiveness of Baffle:

- The use of a baffle clearly improves stress performance across all materials, confirming its role in stress mitigation

7.6.4 GRAPH FOR EQUIVALENT ELASTIC STRAIN FOR BOTH MODELS

The given bar chart compares the Equivalent Elastic Strain for three materials—Stainless Steel, Aluminum Alloy, and Titanium Alloy—under two conditions: without baffle (blue bars) and with baffle (orange bars).



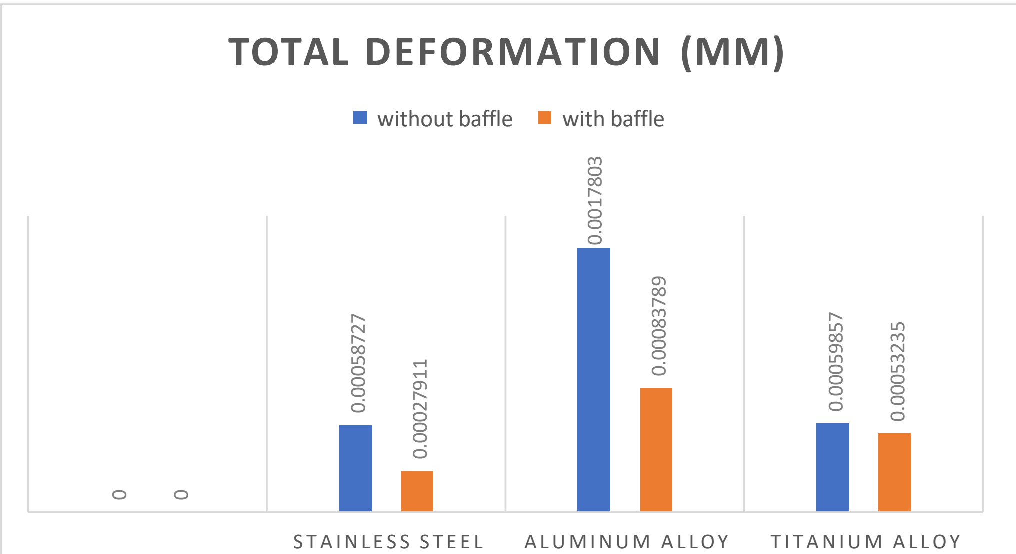
Graph 4: Equivalent Elastic Strain For Both Models

OBSERVATIONS:

- In all three materials, the equivalent elastic strain is lower with the baffle.
- Aluminum Alloy shows the highest strain in both cases.
- **Titanium Alloy** has the lowest strain, indicating it may handle the elastic deformation better than the other two.

7.6.5 GRAPH FOR TOTAL DEFORATION (mm) FOR BOTH MODELS

The given bar chart compares the total deforation for three materials—Stainless Steel, Aluminum Alloy, and Titanium Alloy—under two conditions: without baffle (blue bars) and with baffle (orange bars).



Graph 4: Total Deforation (Mm) For Both Models

OBSERVATION:

1. Stainless Steel:

Deformation reduced from 0.00052715 mm to 0.00026421 mm with the addition of a baffle. This indicates a 49.9% reduction, demonstrating the baffle's effectiveness.

2. Aluminum Alloy:

- Deformation decreased from 0.00147820 mm to 0.00083789 mm.
- A significant 43.3% reduction was observed, although deformation remained the highest among the three materials.
-

3. Titanium Alloy:

- Deformation changed slightly from 0.00059817 mm to 0.00053325 mm, a 10.9% reduction.
- The deformation values were relatively low, suggesting inherent material stability, with minimal improvement from the baffle.
- The presence of a baffle significantly reduces total deformation, especially in Stainless Steel and Aluminum Alloy.
- **Titanium Alloy** exhibits the least deformation overall, indicating it is the most resistant to structural distortion.
- For applications requiring high structural integrity with minimal deformation, **Titanium Alloy** is preferable

8 Conclusion

The purpose of this experiment was to conduct design and analysis of muffler system with and without baffles in order to determine the resonant frequencies of the system and suggest changes in the system design. For this analysis, ANSYS software was used, In order to determine the resonance frequencies, were then compiled to determine which peaks were the most significant for the system. From the data, side baffles were selected as weak parts of the muffler. In order to minimize the effects of these resonance frequencies, the suggested design improvement is to add thickness and also add damping to the system. Thermal analysis results on all three materials of stainless steel, aluminum alloy and titanium alloy. The total heat flux temperature distribution ,Equivalent stresses,elastic strain and total deformation values are more favorable for the titanium alloy, so on conducting the thermal analysis finally concluded the titanium alloy is suitable for manufacturing the mufflers in real world. Then conducting modal analysis on the proposed titanium alloy material for both designs at six different modes. The Titanium Alloy Muffler with a Baffle is the best option for applications requiring better vibration resistance and structural integrity.

References

1. Sibin Babu, Akhildev VP, Jerin Sabu, “design optimization of hybrid muffler and acoustic transmission loss prediction” international Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 07 Issue: 07 | July 2020 www.irjet.net p-ISSN: 2395-0072.
2. Kannan Chidambaram, Design and Optimization of a Performance Muffler for a Formula SAE Vehicle, January 2021SAE International Journal of Engines 14(2),DOI:10.4271/03-14-02-0014 .
3. Raj C N Thiagarajan Acoustic Performance Design of Automotive Muffler. October Conference: COMSOL Conference 2015At: Pune, India
4. Satyanarayana “Modelling and Analysis of Diesel Engine Muffler using CFD Analysis” Journal of Information and Computational Science ISSN: 1548-7741.
5. Sahil S Ambavane “Design and Thermal Analysis of Vehicle Exhaust Muffler’’ International Research Journal of Engineering and Technology (IRJET) e- ISSN:23950056, Volume: 08 Issue: 12 | Dec 2021,p-ISSN: 2395-00728.
6. Praveen. R “Design and analysis of automobile muffler” International Journal of Pure andAppliedMathematic,Volume118,2018,10531060,ISSN:13118080(printedversion);IS S N:1314-3395(on-lineversion) .
7. Ankit J. Desai, “Design and Analysis of Automotive Muffler”, International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181 Vol. 5 Issue 05, May2016.
8. Tejas j.kalange,Design and analysis of muffler for two wheeler”,vol-4,issue-4,2018,IJARIIE,ISSN(0)-2395-4396.
9. Ajay Kumar Jangir, “Practically Design and Analysis of Economically Exhaust System for Automobile” International Journal of Research in Engineering, Science and Management Volume-2, Issue-6, June-2019.
10. K Archana, “Design And Model Analysis Of Muffler” International Journal of Scientific Research in Engineering and Management (IJSREM) Volume: 05 Issue: 06 | June - 2021 ISSN: 2582-3930.
11. A. S. Mate, “Thermal analysis of silencer pipe”, International Journal of Mechanical Engineering and Technology (IJMET) Volume 8, Issue 3, March 2017, pp. 269–278 Article ID: IJMET_08_03_030.
12. Dattatray Dilip Giripunje, “Thermal Analysis Of Motor Bike Exhaust Silencer For Enhancing Heat Transfer Rate By Adding Dimple As Enhancing Features” International Journal of Scientific Research and Management Studies (IJSRMS) ISSN: 23493771 Volume 2 Issue 2, pg: 125-135.
13. Anesha B B, “Design and Thermal Analysis of Motor Bike Exhaust Silencer” International Journal of Engineering Research & Technology (IJERT) http://www.ijert.org ISSN: 2278-0181 IJERTV6IS090125 (This work is licensed under a Creative Commons Attribution 4.0 International License.) Published by : www.ijert.org Vol. 6 Issue 09, September – 2017.
14. 14.Anjan Pal, “Design, Analysis & Optimization of Muffler for Four Stroke Petrol Engine Motorcycle”, y (IJRASET) ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538 Volume 11 Issue V May 2023- Available at www.ijraset.com.
15. 15.Mr. Mohan Kumar G R, “ Finite Element Analysis of Automotive Muffler” July 2021| IJIRT | Volume 8 Issue 2 | ISSN: 2349-6002.

10 ANNEXURE CATIA TRAINING

Figure 10.1: Catia Training.[67]

Figure 10.2: Catia Training.[68]

ANSYS TRAINING

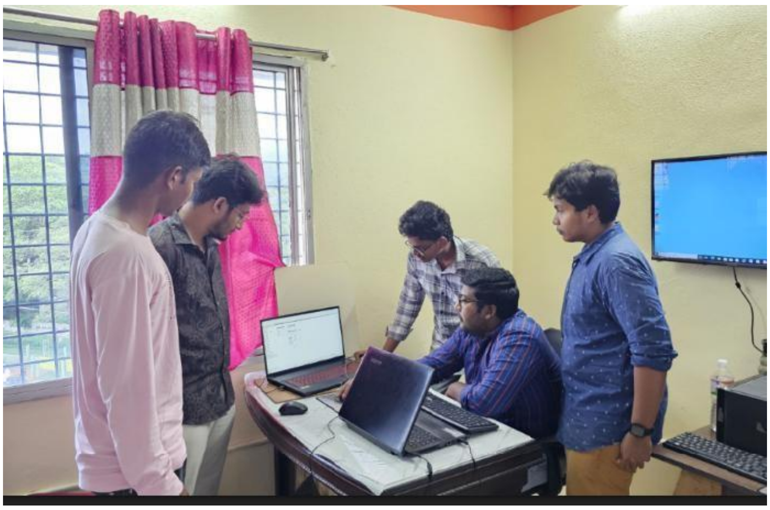


Figure 10.3: Ansys Training.[69]

CATIA COURSE CERTIFICATES

Figure 10.4: Catia Course Certificate. [70]

Figure 10.5: Catia Course Certificate. [71]

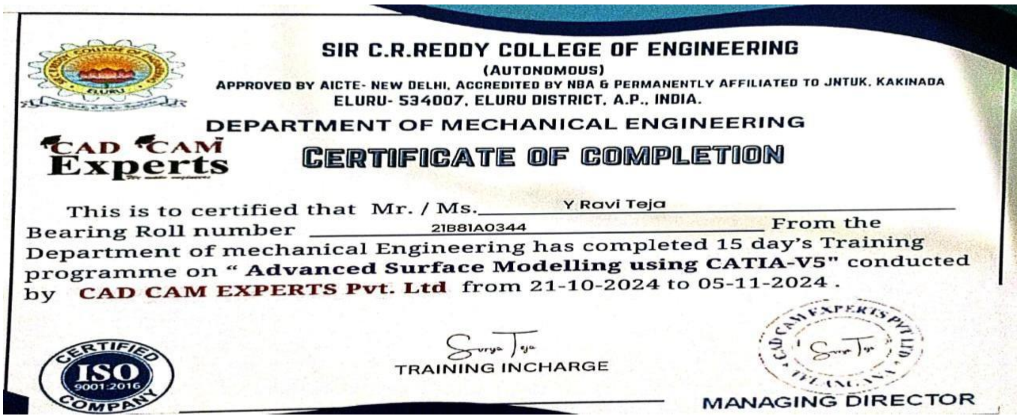

SIR C.R.REDDY COLLEGE OF ENGINEERING
(AUTONOMOUS)
APPROVED BY AICTE- NEW DELHI, ACCREDITED BY NBA & PERMANENTLY AFFILIATED TO JNTUK, KAKINADA
ELURU- 534007, ELURU DISTRICT, A.P., INDIA.
DEPARTMENT OF MECHANICAL ENGINEERING
CAD CAM Experts
CERTIFICATE OF COMPLETION
This is to certified that Mr. / Ms. Y Ravi Teja
Bearing Roll number 21B81A0344 From the Department of mechanical Engineering has completed 15 day's Training programme on " Advanced Surface Modelling using CATIA-V5" conducted by CAD CAM EXPERTS Pvt. Ltd from 21-10-2024 to 05-11-2024 .
CERTIFIED ISO 9001:2016 COMPANY
TRAINING INCHARGE
MANAGING DIRECTOR

Figure 10.6: Catia Course Certificate. [72]

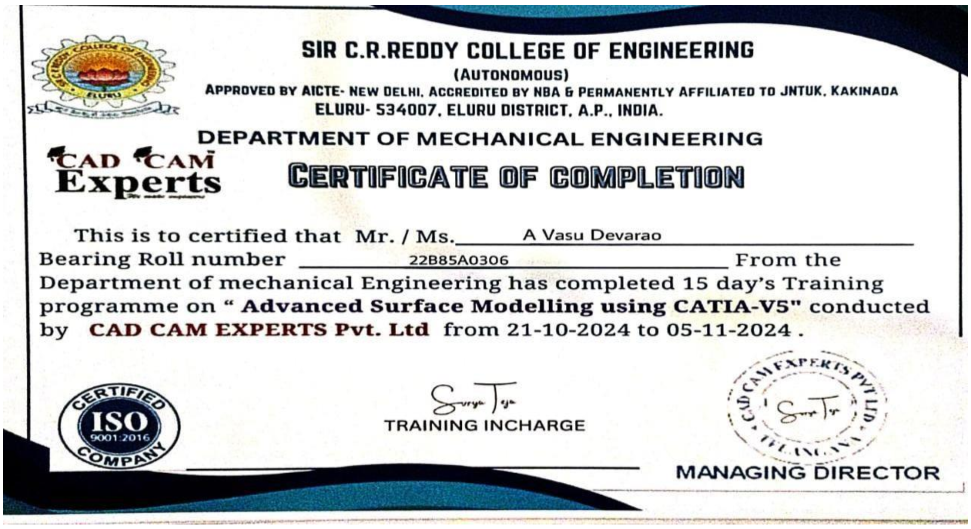

SIR C.R.REDDY COLLEGE OF ENGINEERING
(AUTONOMOUS)
APPROVED BY AICTE- NEW DELHI, ACCREDITED BY NBA & PERMANENTLY AFFILIATED TO JNTUK, KAKINADA
ELURU- 534007, ELURU DISTRICT, A.P., INDIA.
DEPARTMENT OF MECHANICAL ENGINEERING
CAD CAM Experts
CERTIFICATE OF COMPLETION
This is to certified that Mr. / Ms. A Vasu Devarao
Bearing Roll number 22B85A0306 From the Department of mechanical Engineering has completed 15 day's Training programme on " Advanced Surface Modelling using CATIA-V5" conducted by CAD CAM EXPERTS Pvt. Ltd from 21-10-2024 to 05-11-2024 .
CERTIFIED ISO 9001:2016 COMPANY
TRAINING INCHARGE
MANAGING DIRECTOR

Figure 10.7:Catia Course Certificate. [73]

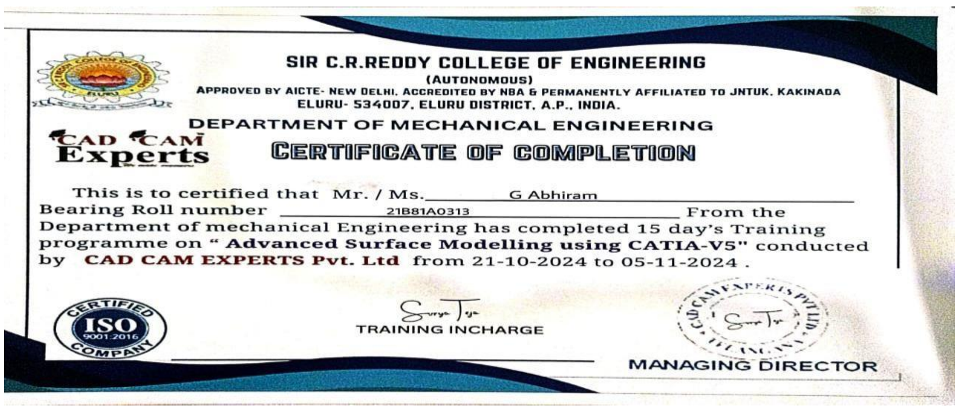
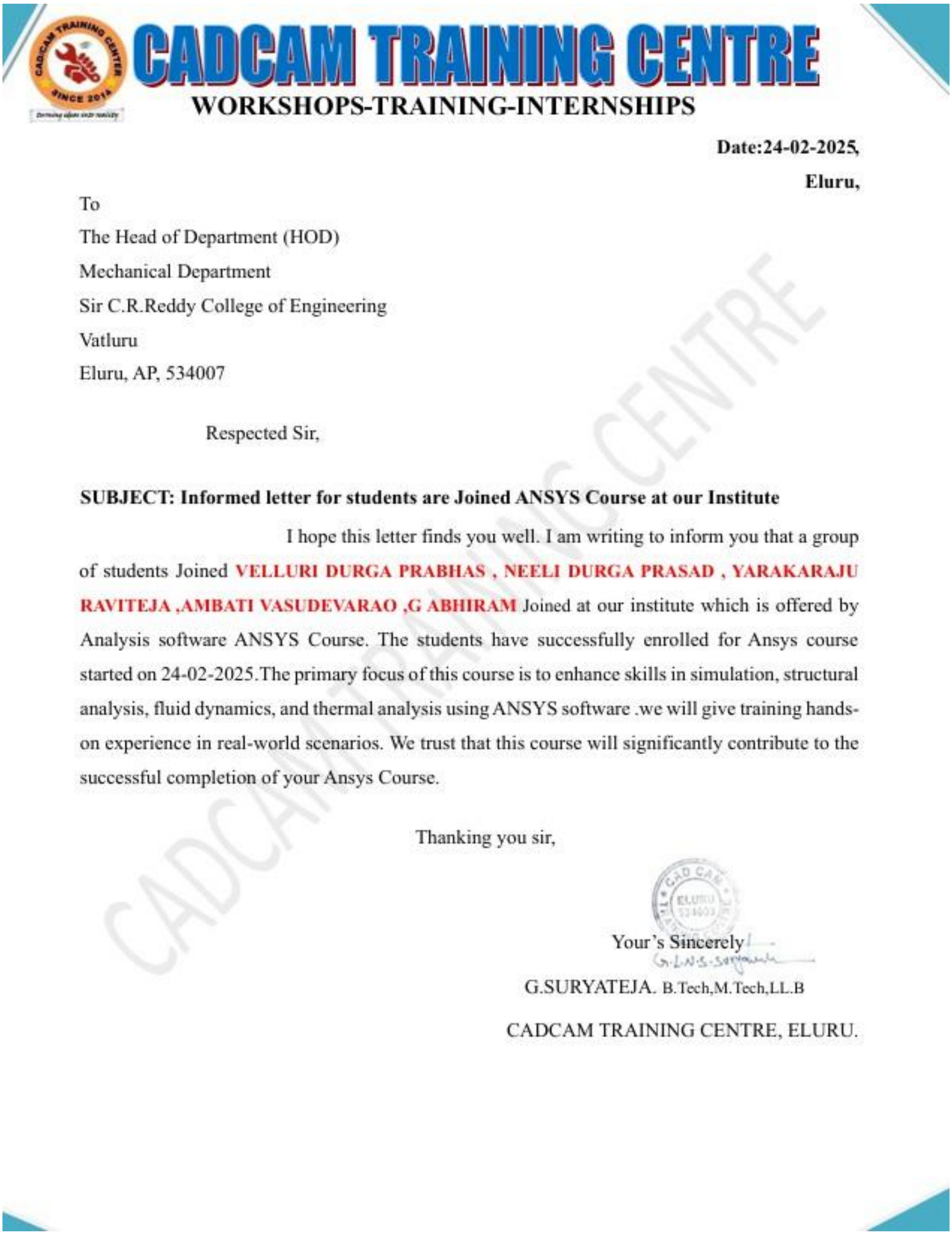

SIR C.R.REDDY COLLEGE OF ENGINEERING
(AUTONOMOUS)
APPROVED BY AICTE- NEW DELHI, ACCREDITED BY NBA & PERMANENTLY AFFILIATED TO JNTUK, KAKINADA
ELURU- 534007, ELURU DISTRICT, A.P., INDIA.
DEPARTMENT OF MECHANICAL ENGINEERING
CAD CAM Experts
CERTIFICATE OF COMPLETION
This is to certified that Mr. / Ms. G Abhiram
Bearing Roll number 21B81A0313 From the Department of mechanical Engineering has completed 15 day's Training programme on " Advanced Surface Modelling using CATIA-V5" conducted by CAD CAM EXPERTS Pvt. Ltd from 21-10-2024 to 05-11-2024 .
CERTIFIED ISO 9001:2016 COMPANY
TRAINING INCHARGE
MANAGING DIRECTOR

Figure 10.8: Catia Course Certificate. [74]

Advanced International Journal for Research (AIJFR)

E-ISSN: 3048-7641 ● Website: www.aijfr.com ● Email: editor@aijfr.com

ANSYS COURSE JOINING LETTER



CADCAM TRAINING CENTRE

WORKSHOPS-TRAINING-INTERNSHIPS

Date:24-02-2025,

Eluru,

To

The Head of Department (HOD)

Mechanical Department

Sir C.R.Reddy College of Engineering

Vatluru

Eluru, AP, 534007

Respected Sir,

SUBJECT: Informed letter for students are Joined ANSYS Course at our Institute

I hope this letter finds you well. I am writing to inform you that a group of students Joined **VELLURI DURGA PRABHAS , NEELI DURGA PRASAD , YARAKARAJU RAVITEJA ,AMBATI VASUDEVARAO ,G ABHIRAM** Joined at our institute which is offered by Analysis software ANSYS Course. The students have successfully enrolled for Ansys course started on 24-02-2025.The primary focus of this course is to enhance skills in simulation, structural analysis, fluid dynamics, and thermal analysis using ANSYS software .we will give training hands-on experience in real-world scenarios. We trust that this course will significantly contribute to the successful completion of your Ansys Course.

Thanking you sir,

Your's Sincerely

G.SURYATEJA. B.Tech,M.Tech,LL.B

CADCAM TRAINING CENTRE, ELURU.

Figure 10.9: Ansys Course Joining Letter. [75]