Chapter 4

Finite Element Analysis of Mufflers

As described in the previous chapter for multi-dimensional approach analytical methods become very complicated and greater disagreement appears between the analytical and experimental TL trend after cut-on frequencies due to deviation from plane wave theory. Numerical methods provide solution to these multi-dimensional problems which are covered in this chapter.

With the ever-increasing computational speed and storage capacity of computers, the numerical simulation methods are greatly accepted and become popular. Thus the use of the finite element method (FEM) in design is growing rapidly. One area that lends itself very well to these methods is the design of silencer systems for noise control. There is much work that has been done for smaller systems such as those used in automobiles and small engines, however, the design of much larger systems is still largely based on empirical extensions of previous results. Due to the large size, difficulties in testing and high costs of these silencer systems, the ability to accurately predict the performance before construction and commissioning would be very beneficial. To properly predict the performance of a silencer system, many factors need to be involved in the calculation [64].

The numerical methods have some advantages over conventional methods as follows:

- Substantial reduction of lead time and cost of new design.
- Ability to study the systems where experiments are difficult or impossible to perform.
- Practical unlimited level of details of result.
Computational modeling using a Finite Element Analysis (FEA) software package provides a cost effective way to investigate the performance of a muffler/silencer device in a virtual environment. This is practically the same as building a prototype device and testing it.

The Finite Element Method (FEM), sometimes referred to as finite element analysis (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. Boundary value problems are also sometimes called field problems. The field is the domain of interest and most often represents a physical structure. The field variables are the dependent variables of interest governed by the differential equation. The boundary conditions are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field. All acoustic problems considered in this work are governed by dimensionless equations with appropriate boundary and initial conditions. Numerical results are obtained using standard computational code.

Principle Steps of Finite Element Application (FEA) [93, 94]
Modern mechanical design involves complicated shapes, sometimes made of different materials that as a whole cannot be solved by existing mathematical tools. Engineers need the FEA to evaluate their designs. Following are the principle steps involved in FEA.

- **Model Definition**
  Building a finite element model is the first step in the analysis. The purpose of the geometric modeling phase is to represent in terms of points, lines, areas and volume. Complicated or smooth objects can be represented by geometrically simple pieces (Elements).

- **Model Type**
  The basic two-dimensional (or plane) elements are loaded in their own plane. They are triangular or quadrilateral elements.
  They are often used to model a wide range of engineering problems.
CHAPTER 4. FINITE ELEMENT ANALYSIS OF MUFFLERS

The most common three-dimensional elements are tetrahedral and hexahedral (or brick) elements; they are used when it becomes necessary to perform a three-dimensional analysis. Judgment concerning the appropriateness of one, two, or three-dimensional idealizations is necessary.

- **Meshing the geometry**
  The process of dividing the model into small pieces is called meshing. Choosing the most appropriate element type is essential to model most closely the actual physical behavior of the system and a very crucial part of FEA. Using sufficient number of elements is an important step as the accuracy of the analysis is dependent on it. The elements must be made small enough to give usable results and yet large enough to reduce computational effort and avoid numerical rounding errors. Small elements (and possibly higher order elements) are generally desirable where the results are changing rapidly, such as where changes in geometry occur. Large elements can be used where results are relatively constant. It also depends on the physical makeup of the body under actual loading conditions and on how close to the actual behavior the analyst wants the results to be. In acoustics, the mesh size is determined by the maximum size of the applied wave length.

- **Selecting interpolation functions**
  The next step is to choose the interpolation function to represent the variation of the field variable over the element.

- **Materials and Properties**
  Material properties such as density, sonic velocity etc. are provided as input to the program. Once the finite element model has been established it is necessary to determine the matrix equations expressing the properties of the individual elements.

- **Boundary Conditions**
  After assigning properties boundary conditions are imposed. To study its behaviour, some degrees of freedom must be restricted for some of the nodes. Such constraints are termed boundary conditions. Harmonic response analysis is performed which calculates the pressure distribution in the fluid due to a harmonic load.
CHAPTER 4. FINITE ELEMENT ANALYSIS OF MUFFLERS

- Use of FEM solvers
  FEA solvers are available for solving matrix equations. Each analysis type has various solvers to choose from.

- Post processing
  The output from the Finite Element Analysis is primarily in the numerical form. It usually consists of nodal values of the field variables and its derivatives. Graphic outputs and displays are usually more informative. The curves and contours of the field variable can be plotted. The output is primarily in the form of color-coded maps. Post processing is used to extract required information. From the extracted information results are plotted.

4.1 The Finite Element Method by using ANSYS

ANSYS is a comprehensive general purpose finite element computer program that contains more than 100,000 lines of code. It has been a leading FEA program for more than 36 years [www.ansys.com]. Today ANSYS can be found in use in many engineering fields including aerospace, automotive, electronics, acoustics and nuclear. The acoustic analysis in ANSYS involves modeling of a fluid medium and surrounding structure. Typical quantities of interest are the pressure distribution in the fluid at different frequencies, pressure gradient, particle velocity, the sound pressure level, as well as scattering, diffraction, transmission, radiation, attenuation and dispersion of acoustic waves.

The ANSYS program assumes that the fluid is compressible, but allows only relatively small pressure changes with respect to the main pressure. Furthermore, the fluid is assumed non-flowing and inviscid, meaning that viscosity causes no dissipative effects. The mean density and mean pressure are assumed uniform, with the pressure solution being deviated from the mean pressure and not the absolute pressure.

There are two types of acoustic analysis in ANSYS, coupled acoustic analysis and uncoupled acoustic analysis. Coupled acoustic analysis considers the fluid structure interaction whereas in the uncoupled acoustic analysis only the fluid is taken into account. The fluid-structure interaction is ignored and the solid walls are considered as rigid.
CHAPTER 4. FINITE ELEMENT ANALYSIS OF MUFFLERS

4.1.1 Geometry of Simple Expansion Chamber Muffler

The three-dimensional simple expansion chamber muffler is modeled in ProE 2. The muffler is subjected to a harmonic input velocity and boundary conditions are imposed. The acoustic performance of the muffler is then obtained using the transmission loss equation. Figure 4.1 shows the 3-D geometry of simple expansion chamber. If muffler is symmetrical about its own axis, complete chamber is not required to be modeled, but only part of it serves the purpose.

![Figure 4.1: 3-D model of simple expansion chamber](image)

4.1.2 Building the Finite Element Model

This first step starts off with defining the model’s geometry, placement of the structural nodes and sizing of elements. During pre-processing the element type, material properties and constants are defined. In addition the mesh size, shape and distribution are defined. The second step involves applying the loads and boundary conditions to the model. Initial numerical analysis and solution is obtained in this step. The relevant acoustic components available in ANSYS library are FLUID29 for two dimensional and FLUID30 for three-dimensional analyses [www.ansys.com]. Density and speed of sound are required as material properties for the acoustic element. Real constants specify properties for the element type that are not geometrically defined or specific to the material. For fluid elements, ambient pressure is defined as a real constant that is equivalent to atmospheric pressure. Prior to meshing the physical boundaries have to be defined on the geometrical model. Using sufficient number of elements is an important step as the accuracy of the analysis is dependent on it. The mesh size is determined by the maximum size of the
applied wave length. To be able to solve the problem, the analysis type has to be defined followed by the applications of all loads acting on the model. The ANTYPE command was chosen to specify that a harmonic analysis has to be performed. Next, the solution method for the acoustic analysis was specified as (HROPT, FULL), which selects the full solution option (real and imaginary parts) and can handle unsymmetrical matrices. Having chosen harmonic analysis (ANTYPE), it was assumed that all applied loads vary harmonically (sinusoidal). In acoustic problems, the input loads are required to be velocity or pressure, and they may be stepped or ramped. By default the loads are ramped, which means that the load values increase gradually with each sub step [www.ansys.com]. If the loads are stepped, the loads maintain the same load value for all sub steps in a given frequency range. At the inlet surface of three-dimensional finite element model input is given as unit displacement, as ANSYS does not accept velocity and it should be converted into displacement. The other end of outlet being free to radiate into space. Hence, for the boundary condition at the outlet end, the pressure is set to zero. It is used for modeling the fluid medium and the fluid structure interaction.

The Figure 4.2 shows the meshed model of simple expansion chamber. The density = 1.12 kg/m$^3$ and speed of sound = 346 m/s (values depends upon temperature) are input as material properties for the acoustic element. For fluid elements, ambient pressure is defined as a real constant that is equivalent to atmospheric pressure. To calculate the acoustic performance of the muffler using the transmission loss equation,
two boundary conditions need to be satisfied.

Boundary conditions
At the inlet velocity \( v_1 = 1 \text{ m/sec} \)
At the outlet velocity \( v_2 = 0 \)
for the first pass determination of constants \( A_{11} \) and \( A_{21} \)
At the outlet pressure \( p_2 = 0 \)
for the second pass determination of constants \( A_{12} \) and \( A_{22} \)

### 4.1.3 Post Processing

The final step of the analysis in post processing is the review of results. This involves listing values at points of interest, mathematical operations and graphical presentation of the results. M.L. Munjal [87] described the use of transfer matrix method applied to acoustic system. The essence of the transfer matrix method is its ability to break the system into discrete elements which can be modeled by using basic acoustic principles. This is facilitated by taking two state variables, acoustic pressure \( p \) and acoustic mass velocity \( v \), at the input and the output sides of an element and relating them by defining a \( 2 \times 2 \) matrix.

\[
\begin{pmatrix}
  p_{\text{inlet}} \\
  v_{\text{inlet}}
\end{pmatrix} =
\begin{pmatrix}
  A_{11} & A_{12} \\
  A_{21} & A_{22}
\end{pmatrix}
\begin{pmatrix}
  p_{\text{outlet}} \\
  v_{\text{outlet}}
\end{pmatrix}
\]  

(4.1)

Above equation shows the relation between two state variables at either side of the element. The elements of the transfer matrix \( A_{11}, A_{12}, A_{21}, A_{22} \) are determined by applying the relationships between the pressure and mass velocity of the element of interest. The elements \( A_{11}, A_{22} \) are dimensionless: \( A_{12} \) has the dimensions of impedance and \( A_{21} \) the dimensions of admittance.

The acoustic performance, the transmission loss of the muffler is calculated using elements of transfer matrix. The resulting pressures are imaginary and the velocities are real.
For evaluating each term, the following expressions are used:

\[
A_{11} = \frac{p_{\text{inlet}}}{p_{\text{outlet}}} \bigg| v_{\text{outlet}} = 0
\]

\[
A_{12} = \frac{p_{\text{inlet}}}{v_{\text{outlet}}} \bigg| p_{\text{outlet}} = 0
\]

\[
A_{21} = \frac{v_{\text{inlet}}}{p_{\text{outlet}}} \bigg| v_{\text{outlet}} = 0
\]

\[
A_{22} = \frac{v_{\text{inlet}}}{v_{\text{outlet}}} \bigg| p_{\text{outlet}} = 0
\]

\[
TL = 20\log_{10}\left(\frac{1}{2}\sqrt{(A_{11} + \frac{A_{12}}{\rho c} + A_{21}\rho c + A_{22})^2}\right) \tag{4.2}
\]

Acoustic problems can be solved by using harmonic response analysis. The FEM analysis determines the pressure distribution in the fluid due to a harmonic load at the fluid structure interface. Models in ANSYS can be generated in two ways. The most common means is the graphical user interface which consists of accessing a selection of menus and placement of model features and commands by the use of the mouse. This is a relatively simple approach and is most commonly used. The second one is writing an algorithm in command files. This method which is being used in this thesis has many advantages having an entire analysis done with a small text file. Hence, any changes in geometries and properties can be done quickly and efficiently.
CHAPTER 4. FINITE ELEMENT ANALYSIS OF MUFFLERS

4.2 The Finite Element Method by using COMSOL

COMSOL Multiphysics is a finite element analysis solver and simulation software / FEA software package for various physics and engineering applications, especially coupled phenomena, or multiphysics problems. COMSOL multiphysics also offers an extensive interface to MATLAB and its toolboxes for a large variety of programming, preprocessing and post-processing possibilities. The packages are cross-platform (Windows, Mac, Linux). In addition to conventional physics-based user interfaces, COMSOL multiphysics also allows for entering coupled systems of partial differential equations (PDEs). The PDEs can be entered directly or using the so-called weak forms. Out of the several add-on products available, Acoustics Module is used in this work.

Transmission loss is calculated directly in COMSOL using the acoustic power at the inlet and outlet of the acoustic system. Sound power is defined as (Reynolds 1981) [95].

\[ W = \int \frac{p_{rms}^2}{\rho c} dS \]  

(4.3)

\( p_{rms} \) is the root mean square pressure and \( S \) is the area of the surface through which the sound power is passing. The corresponding form of the transmission loss equation is

\[ TL = 10 \log_{10} \left( \frac{W_i}{W_t} \right) \]  

(4.4)

\( W_i \) and \( W_t \) are the incident and transmitted sound power, respectively. This method of calculating transmission loss offers a significant advantage over the ANSYS method in that only one load case is required, thereby reducing computational effort. Second advantage is that ANSYS uses transfer matrix method for the calculations of TL. Hence results in only planar region are accurately predicted by ANSYS but COMSOL predicts result by taking the incident and transmitted sound power for surfaces at inlet and outlet respectively. Thus planar condition is satisfied at inlet and outlet surfaces while many frequencies are cut-on inside.

Thus a three-dimensional finite element method is implemented to evaluate the transmission loss (TL) for all mufflers assuming zero Mach number. The numerical analysis was carried out using COMSOL multiphysics software without fluid structure interaction. This software requires air model of the silencer which can be created or imported in var-
ious formats and processed. This program is capable of applying FEM solution to the muffler problem. Parametric solver and linear solver are used.

Each model was meshed using the default (Lagrange-quadratic) elements. The element size for the finite element domain was chosen to provide a minimum resolution of 12 elements per wavelength to ensure that the resolution requirements were met and consequently, accuracy was maintained. A harmonic pressure of 1 \( P_a \) was specified at the inlet and a radiation condition applied at inlet and outlet. In these cases a frequency of 50-3400 Hz is considered with a frequency resolution of 6.25 Hz and the frequency response of the sound pressure (transmission loss) is observed. The speed of sound is taken as 349.12 \( m/sec \) at 30 degrees ambient temperature.

Figure 4.3: Typical expansion chamber

Figure 4.4: Meshed model of typical expansion chamber
Figure 4.3 shows a typical model and Figure 4.4 shows same model meshed and processed in COMSOL with 104086 tetrahedral elements and 15870 triangular boundary elements.

Figure 4.5 shows slice plot of same muffler with pressure distribution inside the chamber.

Figure 4.5: Slice plot of typical expansion chamber
Figure 4.6 shows Comparison of FEM results with analytical method for the given configuration. Analytical formula gives results in planar range only. Hence the results agree quite well in planar range and after cut-on frequencies variation is observed in TL curve.

The results achieved by numerical tool i.e. by FEM may not be accurate due to many reasons such as modeling errors, meshing errors, assumptions while solving the partial differential equations (solution errors), specifications of approximate boundary conditions, insufficient constraints, selection of meshing elements, types of meshing. Irrespective of these drawbacks numerical methods can be used for prediction of performance of model of complicated shaped muffler/silencers with reasonable accuracy and great speed.

The trial and error method in the enhancement of the muffler/silencer design is considered tedious and expensive. The problem of space constraint in muffler/silencer design frequently occurs in practical design work. Predicting and optimizing the performance of muffler/silencer under space constraint is the need of global competitive world under stringent legal restrictions. Therefore the interest to optimize muffler performance under space constraint is increasing in the field. Therefore next chapter emphasizes on various optimization techniques which are easy to implement, economical and quite effective.