

Modal Analysis of Loudspeaker Diaphragms

by

Ryan Miller

An Engineering Project Submitted to the Graduate

Faculty of Rensselaer Polytechnic Institute

in Partial Fulfillment of the

Requirements for the degree of

MASTER OF ENGINEERING IN MECHANICAL ENGINEERING

Approved:

Ernesto Gutierrez-Miravete, Project Adviser

Rensselaer Polytechnic Institute
Hartford, Connecticut

December, 2010

© Copyright 2010
by
Ryan Miller
All Rights Reserved

CONTENTS

LIST OF TABLES	iv
LIST OF FIGURES	v
NOMENCLATURE.....	vi
ACKNOWLEDGMENT.....	vii
ABSTRACT.....	viii
1. Introduction.....	1
1.1 Background	1
1.2 Problem Description.....	2
2. Methodology	4
2.1 Modal Analysis.....	4
2.2 Exact Solution for the Modes of Vibration of a Circular Plate with Fixed Edge Boundary Conditions.....	4
2.3 Finite Element Model Development	5
2.3.1 3-D Modeling	5
2.3.2 Importing Geometries.....	6
3. Results and Discussion.....	7
3.1 Baseline Model.....	7
3.2 Autodesk Inventor Geometry Development.....	8
3.3 COMSOL Boundary Conditions and Mesh Determination	8
3.4 Baseline Model.....	10
3.5 Annular Plate.....	11
3.6 Cone Shape Diaphragm.....	12
3.7 Curve Shape Diaphragm	14
4. Conclusion	16
5. References.....	17

LIST OF TABLES

Table 1: Material Properties of Diaphragm	3
Table 2: α_{ns}	7
Table 3: Theoretical Frequencies; Flat Circular Plate	7

LIST OF FIGURES

Figure 1: Loudspeaker Components and Oscillation Pattern.....	1
Figure 2: Cross Sectional Sketches.....	5
Figure 3: 3D Model of Diaphragms, 180 Degrees.....	8
Figure 4: Mesh of Baseline Diaphragm, 180 Degrees.....	9
Figure 5: Boundary Conditions.....	10
Figure 6: Modal Shapes; Flat Circular Plate.....	11
Figure 7: Modal Shapes; Flat Annular Plate.....	12
Figure 8: Sweep Mesh Approach; Cone Shape.....	13
Figure 9: Modal Shapes; Cone shape.....	14
Figure 10: Modal Shapes; Curve Shape.....	15

NOMENCLATURE

ρ	Density (kg/m ³)
E	Modulus of Elasticity (Pa)
ν	Poisson's Ratio (-)
η	Loss Factor (-)
D	Flexural Rigidity (Pa·m ³)
h	Thickness (m)
a	Radius (m)
P	Frequency (Hz)
α_{ns}	Frequency Constant (-)

ACKNOWLEDGMENT

The author would like to thank the Rensselaer Polytechnic Institute faculty for sharing their knowledge and expertise during this Masters Project and Masters of Engineering curriculum. Also great thanks to Professor and Advisor Ernesto Gutierrez-Miraverte for the guidance throughout this projects duration.

ABSTRACT

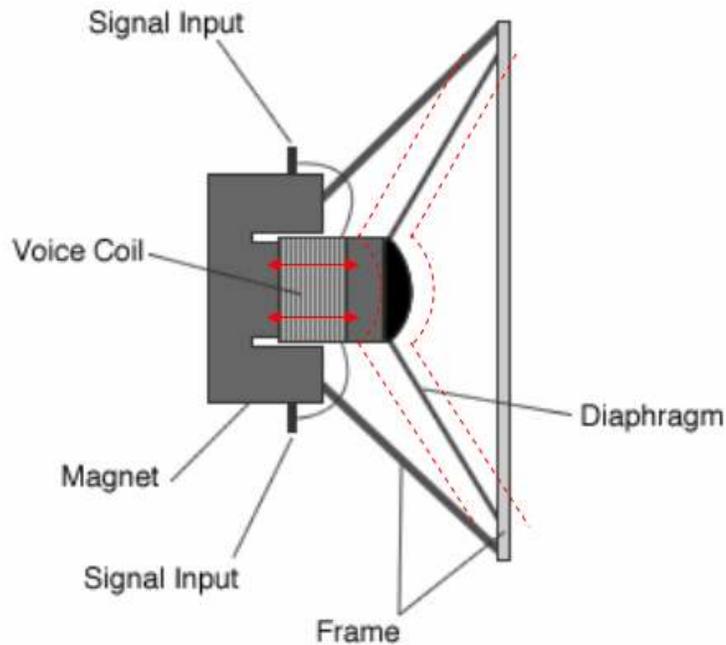
This report describes a study of the modes of vibration of different shapes of diaphragms commonly used in loudspeakers. Diaphragms are used in loudspeakers for producing a wide range of frequencies with the interest in reducing the amount of distortion. In this study, various diaphragm geometries were created using Autodesk Inventor 5 and then modal analysis was carried out using COMSOL Multiphysics 3.5a, eigenfrequency module. The finite element models were validated by comparison with theoretical solutions when available. Results obtained show that diaphragms that utilize a flat or shallow height produce higher frequencies, as opposed to deeper heights which are better for lower frequencies.

1. Introduction

1.1 Background

Loudspeakers are commonly comprised of an electromechanical driving system that converts an electrical power into an acoustic output power. An electrical signal is sent through a coil surrounding a magnet, producing oscillating forces. This driving system oscillates the coil at different frequencies depending on the electrical signal it receives. The coil is directly attached to the center of the loudspeaker diaphragm, and these coil oscillations are transferred, exerting a vibration force against a diaphragm. The outer edge of the diaphragm is attached to a flexible surrounding which has fixed boundary conditions. The diaphragm is essentially an annular plate with a skewed geometry. Oscillating axial forces are exerted on the inner diameter, and is held fixed on the outer diameter. Figure 1 represents the components generally found in a typical cone loudspeaker and the oscillation motion that is imposed on the diaphragm by the voice coil. [1]

Figure 1: Loudspeaker Components and Oscillation Pattern



The diaphragm may be designed using different diameter, height and curvature to achieve different ranges of frequencies. Diaphragms are often desired to be made thin and light, though also have a specific rigidity to prevent buckling. Even though the simple diaphragm loudspeaker design has not changed much over the years, efforts to improve clarity and efficiency are constantly being made. As industry becomes more competitive and the need to design more lightweight and stiff materials increases, developing an optimized design is becoming more important. The use of more detailed analyses must be conducted for more complex geometries. [2]

1.2 Problem Description

In loudspeaker design, it is desired to build a small diameter diaphragm for high frequency response and larger diameters for low frequency response. A deep cone shaped diaphragm is recommended for low frequency piston mode operation (rigid body motion). In contrast, shallow cone shapes are more beneficial for higher frequencies. This is because maximum acoustic energy is transmitted to the surrounding media when the flexural wave velocity through the speaker diaphragm coincides with the velocity of air, which requires a relatively small height. Diaphragms in loudspeakers that are intended to be operated over a wide frequency range often have a curvilinear shape. This provides a deep cross section around the inner portion of the diaphragm and shallow cross section towards the outer periphery. [2] The objective for this project is to analyze the mode shapes present in different diaphragm geometries using finite element analysis. The changes in the mode shape will be noted while changing the frequency excitation within the geometry. Linear and non-linear vibrations of flat annular plates have been studied [3-5]. The first step of the present study will investigate a theoretical method of solving the eigenfrequencies of a circular membrane developed in literature. Then a baseline model will be created, solved using finite element analysis and compared to the theoretical values. Upon verification of the baseline models results, this study will then investigate the effects on more complex geometries.

A somewhat similar study [6] was completed by M. Petyl and P.N. Gelat. Their study investigated the shape of the loudspeaker diaphragm using the dynamic stiffness method.

Their study was conducted using different cone shape heights and analyzing the modal shapes of the cross sections for specific exerted frequencies. The basis of the models in the present study will utilize the previously studied geometry and its material properties for Polypropylene. The properties are given in Table 1:

Table 1: Material Properties of Diaphragm

Material	ρ	E	ν	η
Polypropylene	900	1.510^9	0.33	0.02

2. Methodology

2.1 Modal Analysis

Modal analysis is the study of the natural characteristics, or dynamic properties, of structures. Understanding both the natural frequency and mode shapes will assist the design of structures for noise and vibration applications. When applying an oscillation to a known structure and changing the rate of oscillation, the amplitude changes. The response amplifies as an applied force with a rate of oscillation gets close to the natural frequency of the structure and reaches a maximum when the rate of oscillation is at the resonant frequency of the structure. The structure will experience multiple natural frequencies. When looking at the deformation of the structure at each of these natural frequencies, the structure will take on a variety of different shapes depending on which is used for the excitation force. These deformation patterns are referred as the mode shape of the structure.

2.2 Exact Solution for the Modes of Vibration of a Circular Plate with Fixed Edge Boundary Conditions

A flat circular plate with fixed boundary conditions on the outer diameter is a common problem of interest for various applications. In literature, vibration calculation methods have been used to predict the modal frequencies of simple geometries (i.e. rectangular or circular flat plates). It is necessary to calculate the frequencies of the given circular plate studied in this paper numerically, and then compare those to the results given by using finite element analysis in COMSOL. This will verify that the physical and material properties specified in the finite element model are correct. Upon verification, the complex models shall be modeled using the same characteristics. Using the theoretical method of transverse vibrations of plates provided in reference [7], the frequency of vibration will be determined by the following equation:

$$P = \frac{\alpha_{ns}}{a^2} \sqrt{\frac{D}{\rho h}} \quad (1)$$

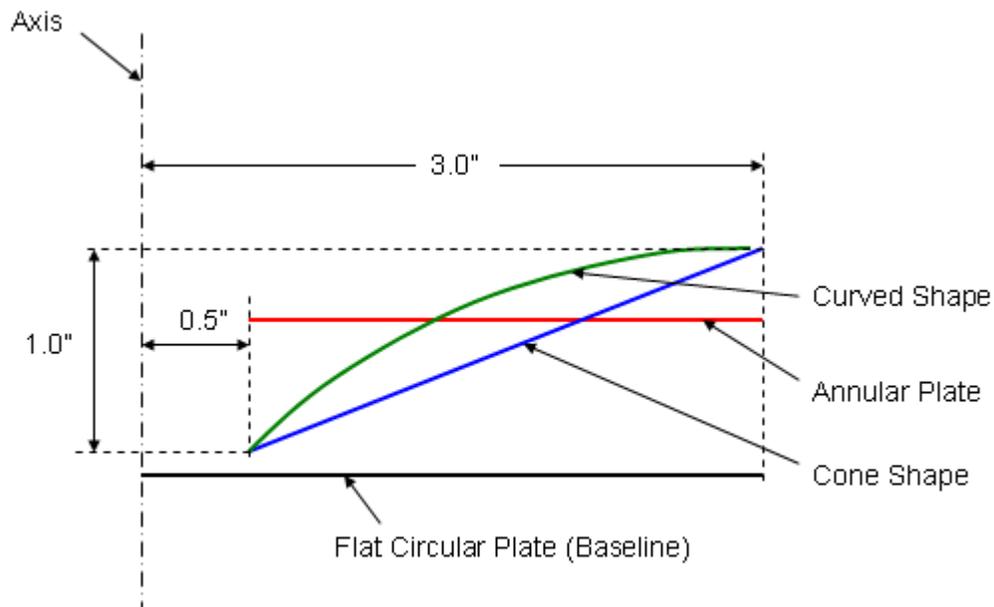
Where $D = Eh^3 / [12(1 - \nu^2)]$ is the flexural rigidity of the plate, and α_{ns} is a constant, in which n represents the number of nodal diameters and s represents the number of nodal circles.

2.3 Finite Element Model Development

2.3.1 3-D Modeling

The process of creating the 3D model for the diaphragm geometries was done using Autodesk Inventor 5. This is one of many 3D modeling tools that aid the development of more complex geometries. A cross section of the diaphragm geometry was produced and then revolved about the axis. The models generated featured the general shape of the diaphragm, reducing the small complex features and therefore simplifying the meshing and post-processing. Figure 2 depicts the cross sectional sketches that are used in the creation of each geometry.

Figure 2: Cross Sectional Sketches



Thickness of geometries are 0.02"

Each cross section was drawn in Inventor as a 2-D sketch. Then the features module was used to revolve the cross sections about a specified axis, positioned 0.5” for geometries with an annular shape. These models were revolved 180 degrees rather than 360 degrees for clarity of modal shapes.

2.3.2 Importing Geometries

The diaphragm models were analyzed using the eigenfrequency module in COMSOL Multiphysics 3.5a. As the geometries were created in Inventor, they were saved as a universal file type, e.g. .STEP, and then translated into COMSOL. Using COMSOL, the 3D model was applied with its material properties in the sub-domain settings. These material properties are given in Table 1. Following the sub-domain setting, the boundary conditions were specified. To represent a loudspeaker diaphragm, the outer diameter is fixed, while the remaining surfaces and edges are allowed to move freely. When using a 180 degree revolved 3D model, the cross section will be applied a plane symmetry boundary condition.

3. Results and Discussion

3.1 Baseline Model

The problem of a circular plate fixed at the edges has been studied in “Vibration Problems in Engineering” by S. Timoshenko. Some of the values for α_{ns} were listed for the proposed problem, given in Table 2. [7]

Table 2: α_{ns}

	n=0	n=1	n=2
s=1	10.21	21.22	34.84
s=2	39.78	-	-
s=3	88.9	-	-

Using the material properties of polypropylene (Table 1), the flexural rigidity can be calculated:

$$D = \frac{(1.5e9)(5.08e-4)^3}{12(1-0.33^2)} = 0.0184 \quad (2)$$

Solving for the frequencies for the given geometry and converting to Hertz, these values are show in Table 3. These values were compared to the finite element analysis results of the baseline model using COMSOL.

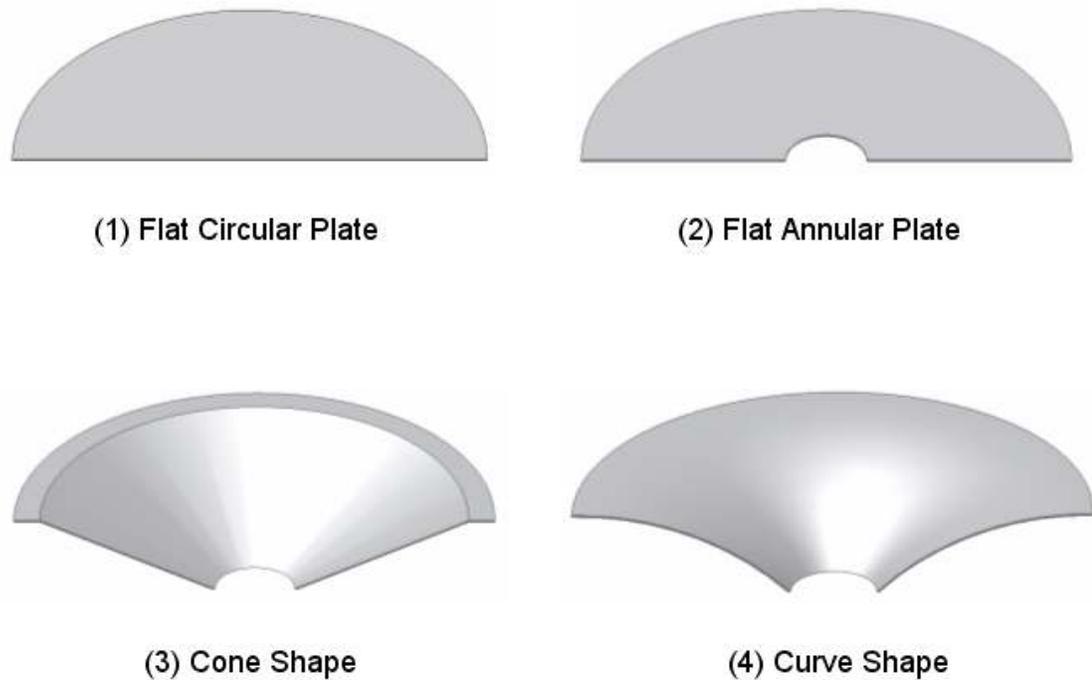
Table 3: Theoretical Frequencies; Flat Circular Plate

Frequency (Hz)			
	n=0	n=1	n=2
s=1	56.12679	116.6514	191.5237
s=2	218.6801	-	-
s=3	488.7043	-	-

3.2 Autodesk Inventor Geometry Development

Figures 3 show each of the diaphragms modeled using Inventor. Note that the cone shape geometry has a lip feature on its outer diameter. This was created to ensure that the fixed edge is consistent with the other geometries.

Figure 3: 3D Model of Diaphragms, 180 Degrees

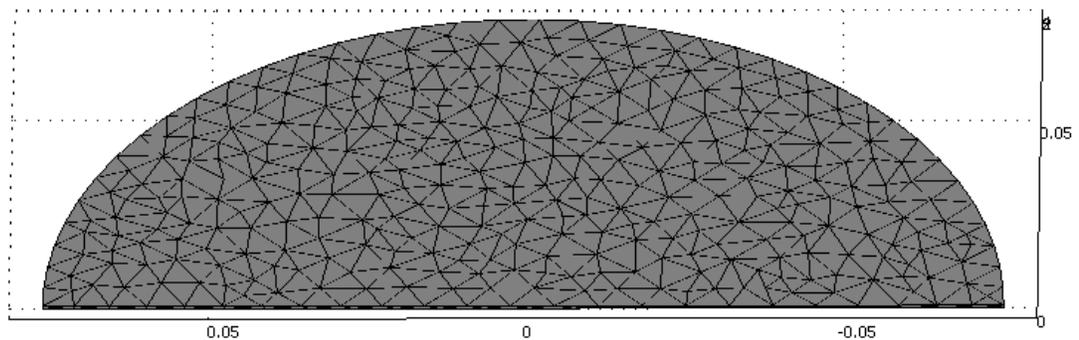


3.3 COMSOL Boundary Conditions and Mesh Determination

COMSOL was opened in the 3-D model eigenfrequency module to be able to support the previously modeled geometries. The models developed in Inventor were imported into COMSOL, and then needed to be defined for the material properties, boundary conditions, and mesh before the model can be analyzed. First, the material properties were applied for polypropylene (See Table 1). This was done by opening the sub-domain properties and editing the default values to change the density, ρ , modulus of elasticity, E , Poisson's ration, ν , and loss factor, η , accordingly.

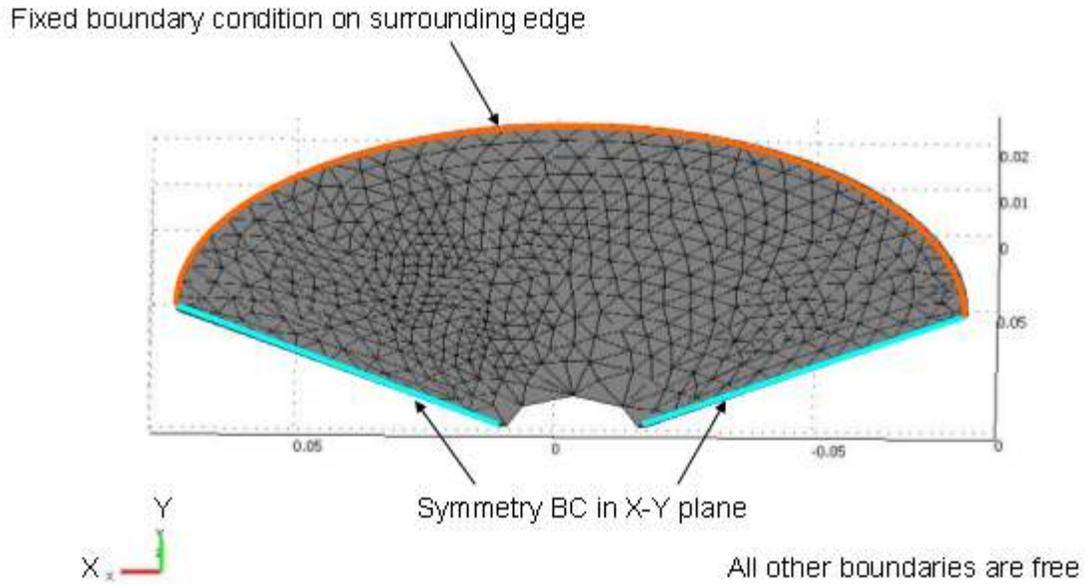
The elements chosen are Lagrange – Quad, and each model was initially meshed using the free mesh parameters option with an extremely course predefined mesh size due to fast computational duration. This mesh style is typical for all other models. After each geometry mesh was completed, they were inspected for any imperfections that could potentially cause inaccurate results. It is also important to verify that the results obtained from the finite element analysis are accurate and reliable by iterating on the element size or meshing approach. In order to ensure accurate results, the baseline finite element analysis was conducted first and compared to the theoretical results provided in Table 3. Upon acceptable results, the mesh size is use throughout the other diaphragm geometries. The meshed geometry of the baseline circular flat plate is shown in Figure 4.

Figure 4: Mesh of Baseline Diaphragm, 180 Degrees



Next the boundary conditions were applied under boundary settings. It was chosen to apply a symmetry boundary condition in the x-y plane along the surface that would represent the mirrored geometry, thus completing the full 360 degree revolution in theory. This is shown by the blue line in Figure 5. Next, the surface of the outer diameter is applied a fixed boundary condition, shown by the orange line in Figure 5. All other surfaces, edges, or points are set free, which is the default boundary condition in COMSOL.

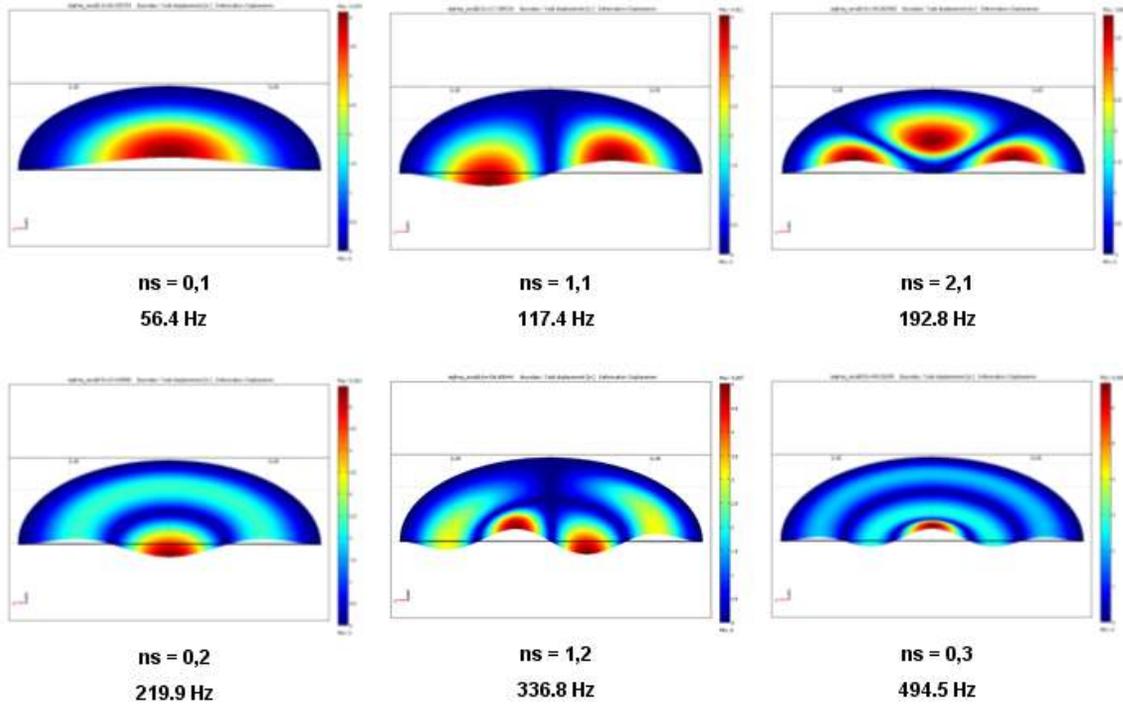
Figure 5: Boundary Conditions



3.4 Baseline Model

The baseline model (Flat Circular Plate) was modeled first to ensure that the boundary conditions and mesh were correctly applied for the remaining geometries. The geometry was meshed using the extremely course mesh size and solved. It was apparent that the initial mesh density was not sufficient and needed more refinement. The mesh was refined to consist of 23,743 elements. It was chosen to plot six (6) eigenfrequencies for this model. Figures 6 shows each of these modal displacements for each eigenfrequency.

Figure 6: Modal Shapes; Flat Circular Plate

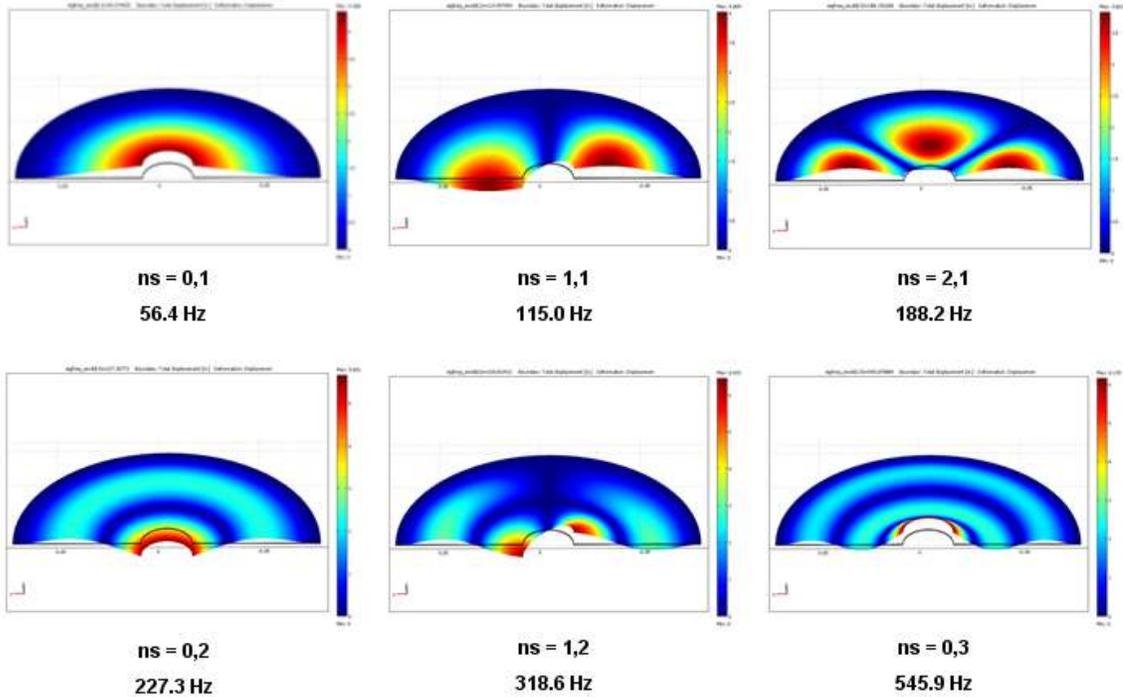


These models are equivalent to the modal shapes given from the supporting calculations for flat circular plates (See Table 3); therefore the models physical conditions are acceptable. The next geometries will then be based from these characteristics of the baseline model to predict accurate results.

3.5 Annular Plate

The second geometry is very similar to the Flat Circular Plate, with the exception of a center hole. The model was solved with a mesh consisting of 17,297 elements. The modal results for this geometry are shown in Figures 7, with the respective eigenfrequency.

Figure 7: Modal Shapes; Flat Annular Plate

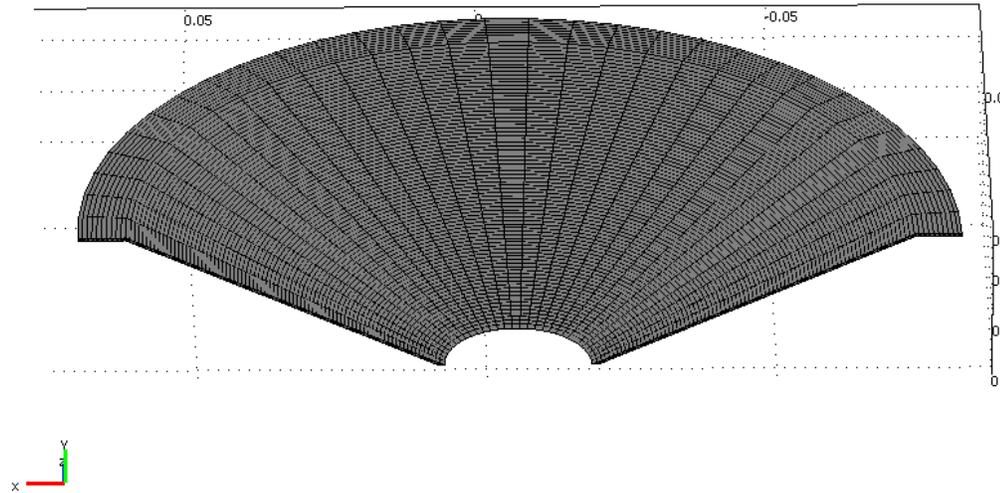


Since the geometry is very similar to that of the flat circular plate, the eigenfrequencies are similar.

3.6 Cone Shape Diaphragm

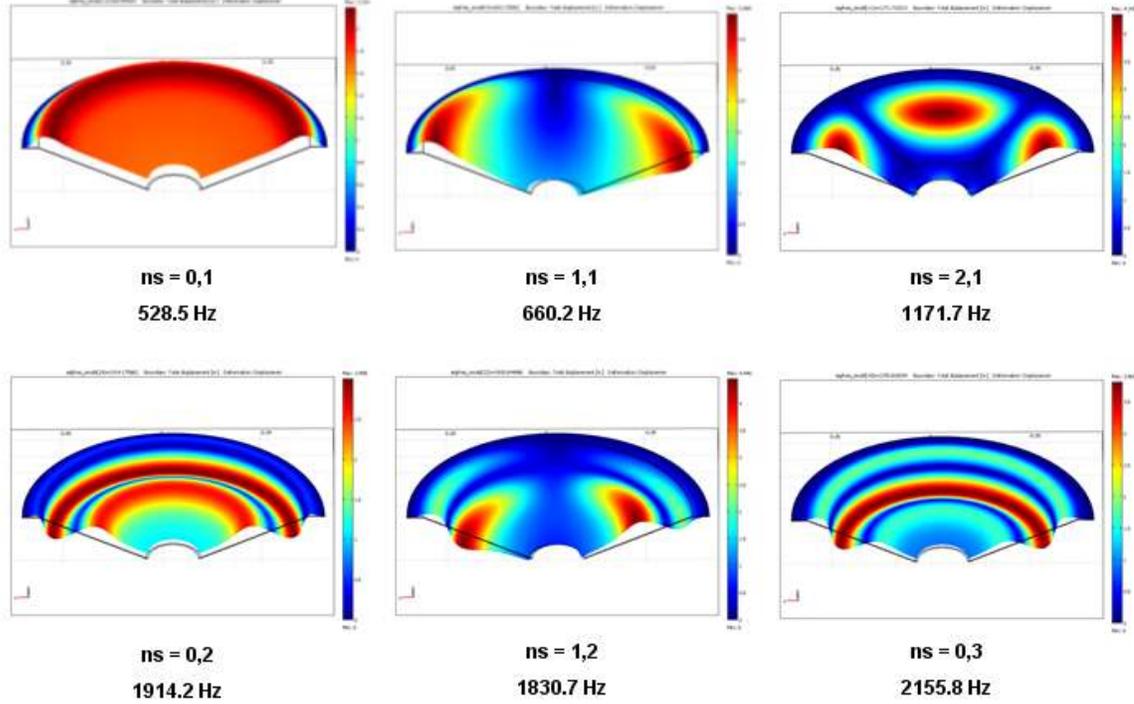
The next geometry now introduces a height parameter to the previous annular plate, shown in a cone shape. This model also features a lip around the outer diameter that has its fixed boundary condition. As the cone shape diaphragm results were plotted using the meshing technique previously determined for the baseline model, the mode shapes did not show a common pattern. The mesh present did not look symmetrical and was thought that this could be the lead cause. A different meshing approach using the sweep mesh tool was then utilized to see an improvement in results. This new technique was checked using the baseline model to ensure that this new set of results were still reasonable. The new mesh is shown in Figure 8.

Figure 8: Sweep Mesh Approach; Cone Shape



The model was solved with a mesh consisting of 5611 elements using the extra fine mesh size. The number of elements is significantly less than when meshed freely. The modal results for this geometry are shown in Figure 9, with the respective eigenfrequency.

Figure 9: Modal Shapes; Cone shape

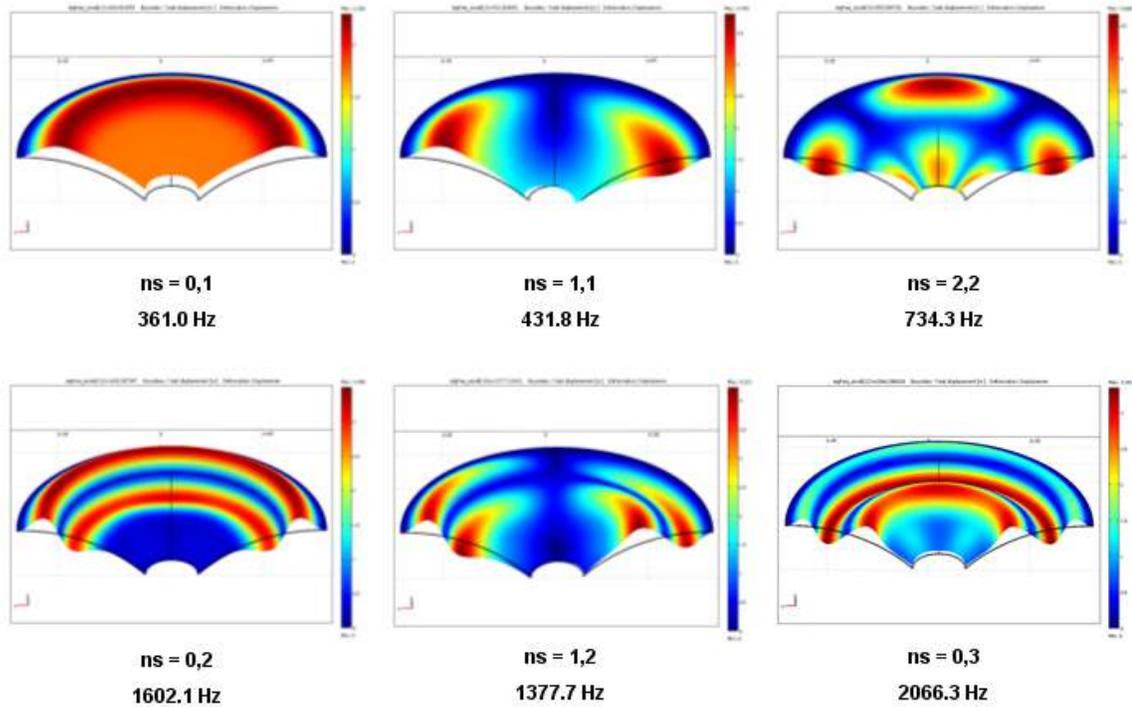


The modal results for this analysis are very similar to those previously obtained, except with much higher exciting frequencies. The cone shape presents a higher stiffness in the diaphragm structure. It was also evident that the modal shape for ns = 0,2 had a higher exciting frequency than ns = 1,2.

3.7 Curve Shape Diaphragm

The last geometry adds curvature to the cross sectional shape. This finite element model utilized the sweep mesh technique as well; with 1092 elements using the extra fine mesh size. The results are shown in Figure 10.

Figure 10: Modal Shapes; Curve Shape



The modal results for the curved diaphragm have slightly less exciting frequencies than the cone shape. The modal shape for $ns = 2,1$ was not achievable when solving this analysis. Instead, there was a modal shape for $ns = 2,2$. Like the cone shape, the modal shape for $ns = 0,2$ had a higher exciting frequency than $ns = 1,2$.

4. Conclusion

The expected outcomes of this study were met. It is evident that shallow diaphragms or flat diaphragms, as analyzed by the baseline model, have lower frequencies above the piston mode operation and are more effective towards the high end of the frequency spectrum. As the height factor is introduced, the frequency response needed to achieve the same modal shapes increases, allowing the diaphragms to produce lower frequency output in the piston mode operation.

The baseline model resulted in a solution within 1% of the theoretical solution. By refining the mesh of the finite element model, the outcome became more accurate and reliable. However, an increase in elements contributes to longer durations of time to solve, and potentially will cause the system to run out of memory with no results.

With the introduction of height, the geometry becomes more complex, which in turn makes meshing much more difficult. The initial approach of meshing freely using COMSOL was appropriate for the baseline and annular plate models, but not for the cone and curve shape models. The sweep meshing approach created a more efficient mesh for these complex models.

5. References

- [1] E. Skrodzka, P. Aleksander, Comparison of Modal Parameters of Loudspeakers in Different Working Conditions. Applied Acoustics 60, pages 267-277, 2000.
- [2] H.W. Schafft, High Stiffness Speaker Cone. United States Patent 4,112,168, 5 September 1978.
- [3] N. Quaegebeur, C. Antoine, Nonlinear vibrations of loudspeaker-like structures. Journal of Sound and Vibration 309, pages 178-196, 2008.
- [4] F. Tornabene, Free vibration analysis of functionally graded conical, cylindrical shell and annular plate structures with a four-parameter power-law distribution. Computer Methods in Applied Mechanics and Engineering Volume 198, Issues 37-40, Pages 2911-2935, 1 August 2009.
- [5] L. Cheng, Y.Y. Li, L.H. Yam, Vibration analysis of annular-like plates. Journal of Sound and Vibration 262, 2003.
- [6] M. Petyt, P.N. Gelat, Vibration of Loudspeaker Cones Using the Dynamic Stiffness Method. Applied Acoustics, Vol. 53, N0. 4, pages 313-332, 1998.
- [7] S. Timoshenko, D.H. Young, W. Weaver Jr., Vibration Problems in Engineering. Forth Edition. John Wiley & Sons, pages 481-502, 1974.
- [8] C.F. Liu, Y.T. Lee, Finite element analysis of three dimensional vibrations of thick circular and annular plates. Journal of Sound and Vibration 233, 2000.