Theses and Dissertations

2006-07-13

Design of a Linear Ultrasonic Piezoelectric Motor

Scott L. Sharp

Brigham Young University - Provo

Follow this and additional works at: <https://scholarsarchive.byu.edu/etd>

Part of the [Mechanical Engineering Commons](#)

BYU ScholarsArchive Citation

Sharp, Scott L., "Design of a Linear Ultrasonic Piezoelectric Motor" (2006). *Theses and Dissertations*. 997.
<https://scholarsarchive.byu.edu/etd/997>

This Thesis is brought to you for free and open access by BYU ScholarsArchive. It has been accepted for inclusion in Theses and Dissertations by an authorized administrator of BYU ScholarsArchive. For more information, please contact scholarsarchive@byu.edu, ellen_amatangelo@byu.edu.

DESIGN OF A LINEAR ULTRASONIC PIEZOELECTRIC MOTOR

by

Scott L. Sharp

A thesis submitted to the faculty of

Brigham Young University

Master of Science

Department of Mechanical Engineering

Brigham Young University

August 2006

BRIGHAM YOUNG UNIVERSITY

GRADUATE COMMITTEE APPROVAL

of a thesis submitted by

Scott L. Sharp

This thesis has been read by each member of the following graduate committee and by majority vote has been found to be satisfactory.

Date

Jonathan D. Blotter, Chair

Date

Craig C. Smith

Date

Kenneth W. Chase

BRIGHAM YOUNG UNIVERSITY

As chair of the candidates graduate committee, I have read the thesis of Scott L. Sharp in its final form and have found that (1) its format, citations, and Bibliographical style are consistent and acceptable and fulfill university and department style requirements; (2) its illustrative materials including figures, tables, and charts are in place; and (3) the final manuscript is satisfactory to the graduate committee and is ready for submission to the university library.

Date

Jonathan D. Blotter
Chair, Graduate Committee

Accepted for the Department

Matthew R. Jones
Graduate Coordinator

Accepted for the College

Alan R. Parkinson
Dean, Ira A. Fulton College of Engineering and
Technology

ABSTRACT

DESIGN OF A LINEAR ULTRASONIC PIEZOELECTRIC MOTOR

Scott L. Sharp

Department of Mechanical Engineering

Masters of Science

A new geometrically unique ultrasonic motor (USM) was designed using finite element modeling (FEM). A USM operates by vibrating a drive tip in an elliptical motion while it is in periodic contact with a driven surface. Piezoelectric elements are used to create the elliptical motions and are driven near a resonant frequency to create the needed displacements for the motor to operate. An idea for a motor frame was conceived that consisted of an arch, a center ground, and two piezoelectric elements connected to the center ground. End caps were added between the frame and the piezoelectric elements to reduce the stress of the elements. Legs located at the bottom of were used to increase the rigidity. Several FEM programs were written to design the motor and to predict its performance. The FEM motor model exceeded the performance characteristics of the benchmark Nanomotion HR1. The model predicted a linear motor capable of pushing up to 5 N and a maximum speed of 0.4 m/s.

A prototype frame was built out of tool steel and run against an oxide ceramic plate. The USM prototype's piezoelectric elements did not provide the expected displacements in the motor frame as determined by the FEM. The discrepancy was determined to be caused manufacturing errors. Soft glue layers were thicker than expected on each side of the piezoelectric elements causing a large amount of compliance inline with the piezoelectric motion. An additional unexpected layer of glue between the end cap and frame increased the compliance inline with the piezoelectric elements even more. It was also determined that even if the motor had been assembled properly that Hertzian displacement would have caused a $1/3$ decrease in piezoelectric motion. The prototype frame's steady state displacements were approximately 20% of the expected output from the FEM models. The motor was still able to achieve a maximum speed of 55.6 mm/s and a push force of 0.348 N at a preload of 6 N. It is expected that a motor assembly correctly dimensioned and manufactured and designed to minimize Hertzian displacement would result in a significantly better performing prototype.

ACKNOWLEDGEMENTS

I would like to thank all those who have helped me in my efforts and contributed in any way to the completion of this thesis. I first would like to recognize my advising professor and committee chair, Jonathan Blotter. I also wish to thank the other two members of my graduate committee, Craig Smith, and Kenneth Chase for their help and support. I also thank Jeff Paine and Dynamic Structures & Materials LLC (DSM) who made this project possible. Jeff proved to be invaluable resource for completing this project. I would especially like to thank my family and my wife Kalie for their love and support. I am thankful for inspiration from heaven that helped me throughout this project.

TABLE OF CONTENTS

TITLE PAGE	vii
SIGNATURE PAGES	ix
ABSTRACT.....	ixiii
ACKNOWLEDGEMENTS	xvii
TABLE OF CONTENTS	vii
LIST OF TABLES	iiix
LIST OF FIGURES.....	xi
1 INTRODUCTION.....	1
1.1 OVERVIEW OF PIEZOELECTRIC MOTORS.....	1
1.2 ULTRASONIC MOTORS.....	2
1.2.1 History of Ultrasonic Motors	6
1.2.2 Current USM Design	8
1.3 PROBLEM STATEMENT.....	11
1.4 OBJECTIVE AND GOALS	11
1.4.1 Thesis Contribution.....	12
1.5 HYPOTHESIS	14
1.6 THESIS OUTLINE.....	15
2 CONTACT INTERFACE TESTING	17
2.1 OVERVIEW OF THE USMs CONTACT INTERFACE.....	17
2.2 TESTING METHODS	19
2.3 USM FRICTION INTERFACE TEST RESULTS	22
2.3.1 Profilometer Results and Discussion of Results	22
2.3.2 Friction Test Results and Discussion of Results.....	24
3 FINITE ELEMENT MODELING OF THE ULTRASONIC MOTOR.....	27

3.1	OVERVIEW.....	27
3.2	PROCESSING MODELS.....	29
3.3	MODAL AND HARMONIC ANALYSIS	30
3.4	STATIC ANALYSIS	34
3.5	DYNAMIC ANALYSIS	38
4	ULTRASONIC MOTOR DESIGN.....	45
4.1	OVERVIEW OF DESIGN PROCESS.....	45
4.2	PRELIMINARY DESIGN CONCEPTS	49
4.3	FPA WITH GROUNDED CENTER CONCEPT	51
4.3.1	Materials	51
4.3.2	Arches	53
4.3.3	Legs.....	56
4.3.4	Center Block	58
4.3.5	Piezoelectric Element Mounting.....	58
4.4	FINAL DESIGN	59
4.4.1	Final Geometry	61
4.4.2	Finite Element Modeling Performance Predictions.....	62
5	FINAL DESIGN PROTOTYPE RESULTS.....	65
5.1	EXPERIMENTAL TEST SETUP	65
5.2	PROTOTYPE TEST RESULTS	69
5.2.1	FEM / Prototype Discrepancies	74
6	CONCLUSIONS	79
6.1	SUMMARY	79
6.2	RECOMMENDATIONS FOR FURTHER RESEARCH.....	80
	REFERENCES.....	83
	APPENDIX.....	89
	MOTOR FEA CODE	89
	Modal and Harmonic Analysis Code	89
	Static Analysis Code.....	103
	Transient Analysis Code.....	124
	ANSYS BATCH FILES TUTORIAL	144

LIST OF TABLES

Table 1-1	Advantages and Design Challenges of USMs	4
Table 1-2	USM Development	14
Table 2-1	Surface Roughness Results	24
Table 2-2	Static and Dynamic Friction Coefficients Results	25
Table 4-1	Material Properties of Motor Materials	51
Table 4-2	FPA with Grounded Center Specifications.....	60
Table 4-3	Final FEM Design Performance Specifications.....	63
Table 5-1	USM Prototype Steady State Displacements.....	69
Table 5-2	Motor Force – Speed Performance Results	72
Table 5-3	Comparison of NM HR1, Model, & Prototype	73
Table 5-4	Comparison of Model & Prototype.....	73
Table 5-5	Motor Size Comparison	73
Table 5-6	Piezoelectric Element Measurements	75

LIST OF FIGURES

Figure 1-1	Piezoelectric Motors:	2
Figure 1-2	Ultrasonic Motor	3
Figure 1-3	Two Types of USMs	5
Figure 1-4	Ultrasonic Motor by H.V. Barth	7
Figure 1-5	Ring Type Traveling Wave USM	8
Figure 1-6	Single-Element Bidirectional USMs	9
Figure 1-7	Single Drive Signal USMs	10
Figure 1-8	Thesis Contribution	13
Figure 2-1	Profilometer Setup	20
Figure 2-2	Measurement System for Obtaining Friction Coefficients	21
Figure 2-3	Sample Holder for Friction Measurements	22
Figure 2-4	Profilometer Scans of Potential Friction Bars	23
Figure 2-5	Friction Testing of Potential Contact Materials	25
Figure 3-1	Basic USM Design Components	28
Figure 3-2	First Three Mode Shape	31
Figure 3-3	Frequency Response Plots	32
Figure 3-4	Motor Structure Vertical Stiffness	34
Figure 3-5	Drive Tip Displacement Caused by Actuated PZT	35
Figure 3-6	Static Stress Analysis of the USM	37
Figure 3-7	Drive Tip Clearance	38
Figure 3-8	Drive Tip Path Analysis	39

Figure 3-9	Motor Performance Graphs.....	40
Figure 3-10	Drive Tip Motion with Slider Interaction	41
Figure 3-11	Dynamic Stress Analysis of the USM	42
Figure 4-1	Motor Concept Generation Qualification Process	46
Figure 4-2	USM Development Process	48
Figure 4-3	Single-Element Motor Concepts.....	49
Figure 4-4	Multi-Element Motor Concepts	50
Figure 4-5	Desired Natural Modes of USM Structure.....	53
Figure 4-6	Single and Double Arch Designs.....	54
Figure 4-7	Arch Dimensions	55
Figure 4-8	Three Types of Arch Bases.....	56
Figure 4-9	Leg Types and Dimensions.....	57
Figure 4-10	Motor Center Frame Dimensions.....	58
Figure 4-11	Piezoelectric Element Mount.....	59
Figure 4-12	Final Ultrasonic Motor Design Frame Dimensions	61
Figure 4-13	Final Ultrasonic Motor End cap Design Dimensions	62
Figure 4-14	FEM Speed and Force Performance Estimations	64
Figure 5-1	USM Prototype	65
Figure 5-2	USM Prototype Test Fixture.....	66
Figure 5-3	USM Prototype Test Setup	67
Figure 5-4	USM Prototype Free-Space Testing	68
Figure 5-5	Frequency Response of Drive Tip from PZT 1	70
Figure 5-6	Frequency Response of Drive Tip from PZT 2	71
Figure 5-7	Drive Tip Horizontal Resonances Using PZT 1	71
Figure 5-8	Drive Tip Horizontal Resonances Using PZT 2	71

Figure 5-9	Piezoelectric Element Measurement Method	74
Figure 5-10	USM Ball and Socket Joint.....	76
Figure 5-11	Hertzian Stress	77

1 INTRODUCTION

This chapter presents a brief overview of piezoelectric motors along with the history of ultrasonic motors with an emphasis placed on single-element and single drive signal ultrasonic piezoelectric motors. The main limitations of current ultrasonic motors are presented. The problems that this work addresses and the specified goals of this research are also discussed.

1.1 OVERVIEW OF PIEZOELECTRIC MOTORS

Conventional electromagnetic motors were invented over 150 years ago. Through the years, their designs have evolved to achieve greater efficiencies that dominate the electric motor industry today. However, one of the current limitations with small electromagnetic motors is their lower efficiency. This is especially true for electromagnetic motors smaller than one cubic centimeter¹. Piezoelectric motors are a feasible solution to achieve greater small motor efficiency and can offer many other advantages for specific applications that conventional electromagnetic motors cannot.

Piezoelectric motors can be divided into three basic categories: impact drive mechanisms, inchworm mechanisms, and ultrasonic motors². Figure 1-1 illustrates the operation of a simple impact drive and inchworm motor. Impact drive mechanisms use a piezoelectric element to drive a mass abruptly. The inertia of the mass drives the actuator forward. The inchworm works by clamping the back section, stretching the middle, and

then clamping the front section of the mechanism. The device repeats this action to move forward.

Figure 1-1 Piezoelectric Motors: a) Impact Drive Motor, b) Inchworm Motor

1.2 ULTRASONIC MOTORS

Ultrasonic motors (USM) use one or more piezoelectric elements to resonate a drive tip or surface in an elliptical motion. The motor is driven at an ultrasonic frequency (>20 kHz) which is inaudible to the human ear. A simple USM design proposed by T. Sashida is shown in Figure 1-2¹. The combination of the oscillating piezoelectric element

and the resonating stator creates an elliptical motion at the drive tip of the stator. In this simple unidirectional design, the stator's resonance is excited by the fluctuating contact resistance of the rotor. Approximately half of the drive tip's path is in contact with the rotor creating forward movement of the rotor. This frictional drive interface between the stator and rotor is what transfers the motors vibratory energy to the rotor. Many USMs can be directly applied to linear or rotary applications by just changing the driven surface. Since the piezoelectric element in the motor are driven above 20 kHz, a high frequency power source must be used to drive the motor.

Figure 1-2 Ultrasonic Motor (Exaggerated Elliptical Path Shown)

Ultrasonic motors have several advantages and a few design challenges over conventional motors as listed in Table 1-1. Two great advantages of USMs are their high speed and high positional accuracy. Speeds of motors can reach velocities over 1.0 m/s. Typical motor positional resolutions are in the submicron to micron range. However,

motors can have positional accuracies down to the sub nanometer range with a maximum speed of 0.3 meters per second³. Once stopped, the motors have a brake force of approximately their maximum push force.

Table 1-1 Advantages and Design Challenges of USMs

Advantages	Challenges
<ul style="list-style-type: none"> • Direct drive (no gears) • Low speed and high torque • High brake force when stopped • High power/weight ratio • High efficiency • Fine positional accuracy • Quick response without backlash • Absence of magnetic interference • No magnetic field generation • Silent operation 	<ul style="list-style-type: none"> • Wear and tear caused by frictional drive • Need for a high frequency power source • Sagging force vs. speed characteristics

The need for a high frequency power source and the frictional drive are the largest challenges facing USM design. While many motors use multiple high frequency input signals, it should be avoided if a more economical motor is desired. Each high frequency source used to drive a motor increases the total cost of an USM.

A second challenge with USMs is their frictional drive surfaces. High forces during sliding can cause considerable wear on the drive tip and motor limiting its useful life. Not only must the drive surface and the friction bar (the driven surface on the rotor) be wear resistant, they also must have a fine surface roughness. The surface roughness must be smooth enough to allow the drive tip enough clearance to travel over the surface of the rotor during part of its elliptical path.

Ultrasonic motors can be divided into two categories: standing wave and traveling wave mechanisms. Figure 1-3 illustrates the basic operating principles of both types of USMs. Traveling wave motors operate by propagating a wave through the stator, which creates an elliptical motion on the stator surface. The stator surface's elliptical motion propels the rotor in the opposite direction as the wave. Equations (1.1 & 1.2) describe a traveling and standing wave respectively. Equation (1.3) is equivalent to Eq. (1.2) and demonstrates that a traveling wave can be created by two standing waves that are shifted by 90° in space and time. Because of the need for two standing waves, traveling wave motors in general require two or more vibration sources to create the propagating wave.

Figure 1-3 Two Types of USMs: a) Traveling-Wave Motor, b) Standing-Wave Motor

$$u_t(x, t) = A \cdot \cos(k \cdot x - \omega \cdot t) \quad (1.1)$$

$$u_s(x, t) = A \cdot \cos(k \cdot x) \cdot \cos(\omega \cdot t) \quad (1.2)$$

$$u_t(x, t) = A \cdot \cos(k \cdot x) \cdot \cos(\omega \cdot t) + A \cdot \cos(k \cdot x - \frac{\pi}{2}) \cdot \cos(\omega \cdot t - \frac{\pi}{2}) \quad (1.3)$$

Standing wave motors operate by producing the elliptical motion at a single drive point. Because of the need for only a standing wave, these motors can be designed to operate with a single high frequency power source and single piezoelectric element. Reducing the drive signals and piezoelectric elements can greatly reduce the cost of piezoelectric motors.

The performance of ultrasonic motors is characterized by their maximum no-load speed or RPM, maximum push force or torque, and maximum efficiency. Typical load versus speed curves linearly decrease with an increasing load. Equation (1.4) is used to calculate USMs' efficiency curves where V & f are the output velocity and force, and v & i are the input voltage and current.

$$\eta = \frac{P_{out}}{P_{in}} = \frac{V \cdot f}{v \cdot i} \quad (1.4)$$

1.2.1 History of Ultrasonic Motors

The first ultrasonic motor was proposed by Williams and Brown in 1942⁴. It consisted of a cantilever-bar with piezoelectric elements bonded to the four sides. When the piezoelectric elements were excited on orthogonal surfaces with a 90° phase shift, a wobble motion was created at one tip of the motor.

The first practical USM design was designed by H.V. Barth of IBM in 1973 and is shown in Figure 1-4. It consisted of two horns that push against a cylinder. When one of the horns was excited by the piezoelectric element, it rotated in one direction. To drive in the opposite direction, the second horn/piezoelectric element was used. Russian engineers, V.V. Lavrinenko and P.E. Vasiliev, followed by creating various other designs based on the same concept of Barth's USM. At this time, the motors were of little practical use because of excessive wear of the motor, piezoelectric inefficiency, and non-constant vibration amplitude^{5, 6}.

Figure 1-4 Ultrasonic Motor by H.V. Barth

The first major push for USM development came in the 1980's as a request from the semiconductor industry for high precision positioners that did not generate electric field noise. In 1980, T. Sashida developed a rotary USM that achieved 40% efficiency at 1500 rpm and a torque of 0.8 kgf cm. Hitachi Maxel significantly improved and simplified Sashida's motor by replacing several vibratory piezoelectric pieces with a torsional coupler. Maxel improved the efficiency to 80% at a torque of 13 kgf cm with a

30mm x 60mm motor. Sashida's and Maxel's prototypes helped to develop significant interest in developing ultrasonic motors^{5, 6}.

1.2.2 Current USM Design

Most current ultrasonic motors can be divided into 4 topological categories: ring, π , rod, and cylinder types⁵. Ultrasonic motors usually become successful in commercial applications for their high torque, simple design, very small size, linear motion, and/or high precision.

One successful USM, originally proposed by T. Sashida in 1982, was designed by Shinsei Co. Ltd. Shinsei's traveling wave motor is licensed to Canon Co. Ltd for their single-lens reflex cameras and is used in the majority of their EOS camera lenses for quick focusing. It is represented in Figure 1-5. The USM is an ideal motor for cameras due to its compact size, quiet and quick operation, and low power demand. It's unique shape is also beneficial because the camera's lens system is easily built through the empty motor core. The Shinsei motor has a maximum efficiency of 39%, maximum torque of 0.6 Nm, and a maximum speed of 100 rpm⁶.

Figure 1-5 Ring Type Traveling Wave USM

There have been a few notable simple bidirectional single-element USM designs. James R. Friend developed a π shaped motor that is represented in Figure 1-6 (a). It improved upon previous π shaped designs by inducing a leg lifting motion. By using the leg lifting motion along with a tuning fork resonance, the motor is able to achieve a maximum efficiency of 18.9%, a maximum push force of 1.86 N, and a maximum velocity of 16.5 cm/s⁷. Another USM noteworthy for its simple design is made by Elliptec Resonant Actuator AG and is represented in Figure 1-6 (b). The single-element rod-type bidirectional motor operates by exciting longitudinal and bending waves in its structure. It can push a maximum of 0.2-0.4 N and has a maximum speed of 5 cm/s⁸.

Figure 1-6 Single-Element Bidirectional USMs: a) Friend's π Shaped Motor, b) Elliptec's Rod Shaped Motor^{7,8}.

Several successful commercially sold motors use multiple piezoelectric elements driven with a single input signal. The three USMs represented in Figure 1-7 use

symmetry to drive in forward and reverse by only operating half of the piezoelectric elements to move in one direction (notice the exception in Figure 1-7 (b)).

The EDO Corp. motor represented in Figure 1-7 (a) is a simple design that expands one piezoelectric element at an angle to the horn or drive tip. The motor is excited near resonance to create an elliptical motion at the drive tip to propel the slider. The EDO model PDA130 motor has a top speed of 0.25 m/s and can push up to 2.5 N⁹.

The next two motors have similar designs in that they both use rectangular plates of piezoelectric material. Figure 1-7 (b) illustrates the configuration of Physik Instrumente (PI) L.P. motor. The PI motor is separated into five quadrants. The right section (top and bottom) and the middle section are used to drive the motor in one direction and the left section and middle section drives it the other way. A single piezoelectric element may have multiple excitation directions by polarizing the element in different directions. The PI model P-661 is 35 x 13.5 x 6 mm and has a maximum speed of 0.6 m/s and maximum push force of 2 N¹⁰.

Figure 1-7 Single Drive Signal USMs: a) EDO Motor, b) PI Motor, c) Nanomotion Motor. White Arrows Represent the Active Piezoelectric Elements for Movement in One Direction^{11, 12, & 13}.

The last motor represented in Figure 1-7 is produced by Nanomotion Inc. To operate, it drives two piezoelectric elements diagonal to each other. The elements excite bending and longitudinal oscillations in the motor structure, which creates an elliptical motion at the tip. The Nanomotion model HR1 motor measures 40.5 x 25.7 x 8 mm. It has a maximum velocity of 0.25 m/s, maximum push force of 4 N, and a maximum static hold force of 3.5 N¹⁴.

1.3 PROBLEM STATEMENT

Bidirectional ultrasonic motors usually consist of multiple piezoelectric elements and multiple drive signals. Bidirectional ultrasonic motors must not only produce motion in two orthogonal directions, but must also be able to change the relative phases of the two motions to create bidirectional motion. The easiest way to produce the necessary reversible elliptical motion is with multiple piezoelectric elements and drive signals. Piezoelectric elements and drive signal electronics affect the cost of ultrasonic motors, making most of the motors considerably expensive. A high-efficiency, high power to volume ratio USM with a single drive signal would be more economically realizable than other motor designs. To be economically realizable, the motor must be cost effective, simple to manufacture, and durable. While many past designs consist of simple vibratory structures, current computer technology allows one to create new complex designs that were not feasible before.

1.4 OBJECTIVE AND GOALS

The objective of this thesis is to design, analyze, construct, and test a bidirectional single-input ultrasonic motor. While many developers look at perfecting past designs,

this thesis will create a geometrically unique motor. The motor will be compared to the Nanomotion LTD model HR1 motor as a standard of performance to achieve. The HR1 is a 40.5 x 25.7 x 8 mm standing wave motor. The HR1 has a maximum no-load speed of 0.25 m/s and a maximum push force of 4 N¹⁴.

1.4.1 Thesis Contribution

The area to which this research will contribute is represented in Figure 1-8. Most small electric motors can be broken into two groups: conventional electromagnetic motors and piezoelectric motors. Conventional motors lack many of the advantages of piezoelectric and especially USMs in small motor designs. For this research, the standing wave USM is chosen over the traveling wave motor for its ability to use a single drive signal.

The anticipated contribution of this research is to design a geometrically unique bidirectional single input standing wave USM. The anticipated motor will be no larger than the Nanomotion HR1 and will attempt to exceed its speed and force performance characteristics. A small high-efficiency USM with a single input would be economically realizable in a large variety of small motor applications. An additional contribution will be to create a computer simulation of the motor to improve its design and improve upon future USM computer modeling techniques.

Figure 1-8 Thesis Contribution

1.5 HYPOTHESIS

A number of items must be accomplished in order to complete the objective and goals of this thesis. The goals associated with the thesis objective and the steps needed to accomplish these goals are listed in Table 1-2.

Table 1-2 USM Development

Goals & Description
1. Develop ANSYS analytical models of modal deformations <ol style="list-style-type: none">Develop several potential designsPerform modal analysis of designsFind promising configurations that create a reversible elliptical path
2. Develop dynamic ANSYS models of designs <ol style="list-style-type: none">Create dynamic models of designs plotting their elliptical pathsModify the designs to improve the desired path and frequenciesDetermine the best designCreate a full model of the final designPredict prototype performance
3. Verify friction coefficients of materials <ol style="list-style-type: none">Perform profilometer scans of sample materialsDesign friction coefficient measurement systemTest friction coefficients of samplesCompare experimental results to previous results
4. Create the USM prototype <ol style="list-style-type: none">Build final USM designVerify surface finish of drive tip and friction barDesign and build test configurationTest the prototypeCompare results to FEA model and USM benchmark

Analytical models created in the finite element analysis (FEA) package ANSYS will be used to examine mode shape deformations of various motor concepts. The concepts will be systematically modified until designs are generated that can produce a reversible elliptical path. The promising designs will then be analyzed further by doing more dynamic simulations of the concepts. Additional modification of the model will be

done to try to optimize the design. Using the model, the motor's performance will be estimated.

A physical prototype will be built based on the final design. It will be necessary to verify that the prototype was created accurately. Profilometer tests will verify the surface finish on the drive tip and the friction bar of the USM. After verifying the motor's specifications, its performance will be measured and compared to the FEA model and the Nanomotion HR1.

Because of the small elliptical motions of USMs, it is imperative that the drive tip and friction bar have fine surface finishes. The motors frictional interface surfaces will be tested to determine if the surface finishes are acceptable.

To create accurate models, friction coefficients will be measured between various potential materials for the USM. The surface finish of these materials will be verified before the friction test by performing profilometer scans.

1.6 THESIS OUTLINE

The remainder of this thesis describes the motor development in detail. Chapter 2 presents surface finish testing and friction coefficient results for potential motor contact interfaces. Chapter 3 presents the use of the FEA package, ANSYS, in developing the USM presented in this thesis. Chapter 4 provides descriptions of various USM concepts. Motor parameters of the final design are then presented in detail and the performance estimations are presented for the experimental prototype. Chapter 5 presents the results of the prototyped USM. The experimental results are then compared to the FEA model predictions. Conclusions for this research are made in Chapter 6 as well as recommendations for possible further work in the development of the prototyped motor.

A list of references and the Appendix are then given, which includes a brief ANSYS tutorial using batch files and ANSYS code used to design the motor for this thesis.

2 CONTACT INTERFACE TESTING

This chapter presents an overview of the frictional interface used in ultrasonic motors. Testing is performed to determine appropriate friction coefficients for several potential frictional interfaces for the USM. Surfaces are also tested on the samples to verify that surface roughnesses are within acceptable levels. The procedures followed for testing the surface roughness and friction coefficients are explained. The results are then presented and discussed.

2.1 OVERVIEW OF THE USMs CONTACT INTERFACE

Ultrasonic motors operate by converting electric energy into forward movement through two energy conversion processes. The first is when the piezoelectric elements convert electric energy into small mechanical motions that are amplified by the stator into larger mechanical oscillations using one or more resonance modes. The second conversion takes place as the mechanical oscillations are changed into unidirectional motion. This second conversion is achieved through a frictional interface between the USM's drive tip and a friction bar. Contact properties of materials greatly influence the conversion of elliptical oscillations into linear motion. Before properties are measured, a model needs to be proposed and relevant properties need to be determined.

The general Coulomb static friction model is represented in Equation (2.1). The maximal tangential resisting force, F_s , in a contact area between two bodies before sliding

is proportional to the perpendicular force, F_N , where μ_s is the proportional constant known as the static coefficient of friction.

$$F_s \leq \mu_s \cdot F_N \quad (2.1)$$

Once the maximum static force is reached, the two bodies slide against each other and the friction is described by a second equation. Equation (2.2) describes dynamic Coulomb friction model where F_d is the tangential force in the contact area, F_N is the perpendicular force between the two bodies, and μ_d is the dynamic friction proportionality coefficient.

$$F_d = \mu_d \cdot F_N \quad (2.2)$$

While many frictional models have been developed for ultrasonic motors, this thesis will use the generalized Coulomb model, which has been shown to have reasonable agreement with experimentally and empirically obtained data¹⁶. Equations (2.3) & (2.4) represent the generalized Coulomb friction model, where p_N is the normal pressure between two contacting surfaces, μ_s is the static friction proportionality coefficient, and τ is the maximum shear stress before sliding will occur. Once sliding starts, the dynamic friction is modeled by Equation (2.4), where τ is the shear stress at the contacting surfaces.

$$|\tau| \leq \mu_s \cdot p_N \quad \text{if } v_{rel.} = 0 \quad (2.3)$$

$$\tau = -\mu_d \cdot p_N \cdot \text{sign}(v_{rel.}) \quad \text{if } v_{rel.} \neq 0 \quad (2.4)$$

In addition to using a proper friction model, there are several contact interface requirements for ultrasonic standing-wave motors which include; surface finish, wear resistivity, and low inherent damping⁶. Surface finishes of the contacting surfaces in USMs have several consequences. For most contact materials (with their fine surface

finishes) used in USMs, the static and dynamic friction coefficients are the same value so $\mu_s = \mu_d$ may be assumed¹⁶. Next, vibration amplitudes of ultrasonic motor are on the order of a micron or less, which can be on the same order of magnitude of the surface finishes of the stator and slider. One of the consequences of the similar magnitudes of vibration amplitude and surface roughness is motor stall for small stator vibration amplitudes^{17, 18}. If a motor is to perform well at low speeds, the stator and friction bar must have fine surface finishes.

Several materials have been considered for use in the ultrasonic standing-wave motor contact interface. Lifetime and efficiency of the motor are affected by the contact materials resistiveness to wear and low inherent damping. These properties are found in hard contact materials like alumina oxide, Al_2O_3 , and other ceramics and steels. Alumina has been successfully used in several USM's contact interfaces including the motors developed by PI and Nanomotion^{14, 19}.

2.2 TESTING METHODS

In this thesis, several hard materials with low damping will be considered which include; oxide ceramics, croblox[®], and stainless steel. These materials will be tested in a two-step process. First, the surface roughness will be tested to verify that they are in an acceptable range for use with an USM. After the surface finish is verified that it is acceptable, the samples will then be tested to determine the coefficients of static and dynamic friction, μ_s and μ_d . Coefficient values found in this testing will be used in the modeling of the ultrasonic motor in this thesis.

The first test consists of verifying that contact surface samples are within acceptable values. A profilometer is used to test for surface roughness and other surface

profile properties. A profilometer is a measurement device that traces a sharp metal stylus across a surface to measure surface height and is shown in Figure 2-1. These measurements are then used to determine such things as surface roughness, texture, waviness, and other surface properties. The measured materials were securely taped to a steel stage mounted on the granite block. A simple program was then run that pulls the profilometer's stylus 10 mm across the sample as it traces the surface height. Several traces were completed on each sample. The height profile and surface property results are then outputted in the form of a text file. Graphs are then created in Matlab to better visualize the results.

Figure 2-1 Profilometer Setup

The second test to determine contact interface properties is used to estimate the coefficients of friction, μ_s and μ_d , between the various potential interface materials. A

simple force measurement device is used and is shown in Figure 2-2. To use the measurement system, two sample materials are cleaned with acetone and placed in the sample holders. The bottom holder is bolted to the measurement system's frame and the top is pulled with a wire when measurements are taken. Figure 2-3 shows a close-up of the sample holders, which were designed to hold a variety of sample sizes. Once the sample materials are placed properly, a 10 lb weight was placed on top of the sample holders. A linear drive mechanism pulls the top sample holder, with the 10 lb weight on top, at a constant velocity. A force transducer in series and a position transducer in parallel measure the force and position throughout the measurement period. Data from the transducers is low-pass filtered and recorded using a laptop computer via an analog to digital converter. The data is then graphed in Matlab and the friction coefficients, μ_s and μ_d , are determined.

Figure 2-2 Measurement System for Obtaining Friction Coefficients

Figure 2-3 Sample Holder for Friction Measurements

2.3 USM FRICTION INTERFACE TEST RESULTS

This section presents the results of surface roughness and friction testing of four wear resistant sample materials. The materials used in the testing include; stainless steel, croblox[®], ceramic oxide 1, ceramic oxide 2. Friction coefficient testing was performed with all four materials against stainless steel.

2.3.1 Profilometer Results and Discussion of Results

Figure 2-4 shows the profilometer scans of the four materials. The steel, croblox and oxide ceramic 2 samples' scans are fairly flat and would work well with an ultrasonic motor. Looking at the oxide ceramic 1 sample's scan, it has excessive waviness that would not be desired considering the small mechanical oscillations of an USM's drive

tip. An USM using oxide ceramic 1 would probably have inconsistent speed and force characteristics and would very likely have a lot of motor stall.

Figure 2-4 Profilometer Scans of Potential Friction Bars: a) Stainless Steel, b) Croblos, c) Oxide Ceramic 1, and d) Oxide Ceramic 2

Table 2-1 gives the surface roughness results in a numeric form, where R_a is the average surface roughness and R_t is the distance from the highest peak to lowest valley. Croblos has the finest surface roughness followed by oxide ceramic 2, then stainless steel, and finally oxide ceramic 1. The croblos and oxide ceramic 2 smooth surface roughnesses would be helpful for a smoother operating USM. The R_t values rank in the

same order as the R_a values. Like the R_a values, the larger R_t values leave concerns that an USM would stall and not be able to move past a surface aberration.

Table 2-1 Surface Roughness Results

Sample	R_a	R_t	
Stainless Steel	0.108	1.337	μm
Croblox	0.038	0.454	μm
Oxide Ceramic 1	0.201	2.075	μm
Oxide Ceramic 2	0.071	0.803	μm

2.3.2 Friction Test Results and Discussion of Results

Figure 2-5 shows the graphical results of the friction test. The croblox on stainless steel, seen in Figure 2-5 (a), has the least variation of frictional force throughout its pull length followed by the oxide ceramic 2. This is somewhat expected due to their finer surface roughness than the stainless steel and oxide ceramic 1 measured previously. The horizontal dashed lines across the graph indicate the average dynamic friction coefficient, μ_d .

Numerical results for the friction coefficients, μ_s and μ_d , are shown in Table 2-2. As mentioned earlier, the static and dynamic friction coefficients have similar values for the hard materials with fine surface finishes used with USMs¹⁶. The only material that shows a significantly different static versus dynamic friction coefficient is ceramic oxide 1, which has a rougher surface than the other samples. Its rough surface is probably why it has the highest friction coefficients out of all the samples. The other three samples have very similar results with the stainless steel on stainless steel being slightly higher.

Because of the fine surface finish and the frequent use of oxide ceramic in USMs, the friction results of oxide ceramic 2 will be used to model the motor in this thesis.

Figure 2-5 Friction Testing of Potential Contact Materials: a) Stainless Steel on Stainless Steel, b) Crobiox on Stainless Steel, c) Oxide Ceramic 1 on Stainless Steel, and d) Oxide Ceramic 2 on Stainless Steel

Table 2-2 Static and Dynamic Friction Coefficients Results

Friction Samples	μ_s	μ_d
Stainless Steel on SS	0.20	0.20
Croblox on SS	0.18	0.18
Oxide Ceramic 1 on SS	0.27	0.23
Oxide Ceramic 2 on SS	0.18	0.17

3 FINITE ELEMENT MODELING OF THE ULTRASONIC MOTOR

This chapter provides an overview of finite element modeling (FEM) and describes its use in creating the motor in this thesis. This chapter does not explain specific design features of the proposed USM, but presents the use of several programs written for the development of the motor. Examples are included, which demonstrate how the different analysis and simulation programs were used during the motor design. The programs that are described in this chapter may be found in the Appendix and are separated into three categories: modal and harmonic analysis, static analysis, and dynamic analysis.

3.1 OVERVIEW

The advent of modern computing has made it possible to engineer complex structures not possible before. FEM allows one to calculate the strength and behavior of structures through computer modeling. Models usually consist of thousands of equations that must be solved simultaneously, too large to compute by hand. Today's FEA programs can solve very complex nonlinear problems that include; large deformations, nonlinear material properties, intricate geometries, and contact interfaces. These nonlinear models cannot be solved in one step, but must be computed through an iterative process, which can be computationally expensive.

This thesis uses the finite element modeling program ANSYS to analyze the behavior of the proposed ultrasonic motor. ANSYS will be used to calculate mode frequencies, mode shapes, frequency responses, stress contours, displacements, motions, and other quantities of the proposed ultrasonic motor. The results of this analysis will be used to construct the final ultrasonic motor design.

To explain the programs presented in this chapter, Figure 3-1 identifies the basic design and components of the USM proposed in this thesis. There are five parts of the USM; the frame (including the middle pillar), the two piezoelectric elements, and the two end caps. Sliding is allowed between the contact interface of the end caps and frame. The frame is also loaded with a force (represented by the heavy arrows) to preload the piezoelectric elements. The drive tip and ground points are also identified. The slider is the motor's output and is located above the drive tip

Figure 3-1 Basic USM Design Components

The vibratory motor, shown in Figure 3-1, operates by driving one of the piezoelectric element at the correct resonant frequency of the motor assembly. The drive tip vibrates in an elliptical motion against the slider propelling the slider in one direction. One piezoelectric element is used to drive the slider forward and the other is used to drive it in reverse.

3.2 PROCESSING MODELS

FEA consists of three steps; preprocessing, solving, and postprocessing. The preprocessor is used to create or import the geometry, define material properties, declare the element types and associated options, mesh the geometry, declare the boundary/initial conditions, and apply external loads. Next, the type of analysis and the associated options are declared in the solver. Load steps may be created at this time, which allow the model to undergo several loading phases when the problem is being solved. Between each load step boundary/initial conditions and external loads may be added or deleted. Several load steps may be used to properly prestress an object. The postprocessor then provides numerical and graphical results of the solved problem. The software package, ANSYS, provides all three of these functions. The preprocessor, solver, and postprocessor may be operated through a graphical user interface (GUI) or through ANSYS Parametric Design Language (APDL) written in a text file.

This analysis work in this thesis was performed through ANSYS APDL files, also called log, text, batch, or command line files, which contain APDL command code and user defined variables. The batch file created includes code for all of the preprocessing, solving, and postprocessing used for the motor development with a few exceptions. The

exceptions being when additional postprocessing information was desired and not output through the written program, the graphical postprocessing interface in ANSYS was used.

Once the batch files were written, they were executed by opening the ANSYS interactive session, selecting *Utility Menu > File > Read Input Data*, and selecting the desired batch file. A basic tutorial of how to create and run a batch file with contact interfaces is provided in the Appendix. For additional help in learning ANSYS commands and batch files, ANSYS Help Menu, *ANSYS Tutorial: Release 6.1* by K.L. Lawrence, and University of Alberta ANSYS Tutorial webpage are recommended^{20,21,22}.

3.3 MODAL AND HARMONIC ANALYSIS

The first analysis used for the investigation of potential ultrasonic motor designs is modal analysis. This is the obvious fundamental starting point in creating an USM because of its vibratory nature. Motor concepts must be able to amplify small motions of a piezoelectric element into larger mechanical motions. At this beginning step, one is mainly concerned with producing motion in two directions at the drive point of the motor and not yet with the necessary elliptical motion. The slider interaction with the drive tip is neglected for this analysis, but will be added in the dynamic simulation model. Even though the slider interaction is neglected for the modal and harmonic analysis programs, they will give a good indication of the motor behavior before more time costly simulations are performed.

The program, F_mode_24.txt, is used to obtain the mode shape and natural frequencies of the USM structure. The program will output up to the first 10 modes shapes of a structure with their corresponding frequencies. Figure 3-2 shows the first three modes of a structure.

Like all the final programs written for this thesis, F_mode_24.txt uses a preloading solution step before the mode shapes are calculated. This first solution step loads the frame and piezoelectric elements with the preload force, seen in Figure 3-1, to accurately represent an assembled motor. Contact elements are added between the endcaps and frame, while the piezoelectric elements are glued to the center frame post and the endcaps. The glued boundary condition simulates a firm bonded connection. All of the models have fixed nodes in the vertical and horizontal directions at the bottom of the frame and center frame post as boundary conditions.

Figure 3-2 First Three Mode Shape: a) 1st Mode (30,185 Hz), b) 2nd Mode (41,297 Hz), c) 3rd Mode (53,321 Hz)

The mode shapes in Figure 3-2 are an example of a good USM motor frame. The first two modes provide horizontal and vertical amplification at the drive tip. In addition, by examining the mode shape displacement, one could conclude that both of the first two modes could be excited by the actuation of the piezoelectric elements at the proper frequencies. Less desirable results would consist of a structure with modes that could not be directly excited by the piezoelectric elements or a structure with several undesired

modes in close proximity with the desired mode. The motion resulting from such structures would likely not result in the formation of elliptical motion.

The next program, F_harm_24.txt, is used to estimate the drive tip vertical and horizontal steady-state displacements and relative phases. The user sets the program to analyze the frequencies near the desired mode found by using F_mode_24.txt. The piezoelectric elements are given a thermal coefficient of expansion to simulate the piezoelectric expansion that would happen with a 60-volt sine wave. The 60-volt sine wave may then be simulated by applying a 1 degree temperature change to either element. The program applies a step load of 1 degree to one of the piezoelectric elements which acts as the input for the frequency response. The harmonic analysis varies the load sinusoidally and calculates the horizontal and vertical displacement to generate the steady-state frequency response plots shown in Figure 3-3.

(a)

(b)

Figure 3-3 Frequency Response Plots: a) Magnitude Response of Vertical and Horizontal Drive Tip Motion (Cyan and Purple Respectively), b) Phase Response of Drive Tip Motion and Relative Phase to Each Other (Red)

The magnitude of the frequency response plot in Figure 3-3 (a) represents actual displacements since the input load is equal to one degree which represent 60 volts. The preload forces must be deleted after the preloading step of the solution phase, because ANSYS will vary all applied loads sinusoidally while calculating the frequency response²⁰. If the preload forces remained, the frequency response would include the effects of a sinusoidally changing preload, which would give an incorrect solution.

The results of the harmonic analysis are used to determine the desired drive frequency of the USM and estimate the motor speed. To create an elliptical motion at the drive tip, a frequency must be chosen that provides a phase lag between the horizontal or vertical component of the motion. This is achieved by running the motor near or at resonance as seen in Figure 3-3 (b). The relative phase of the vertical and horizontal motion is shown in red. The motor speed can be estimated by using the magnitude of the horizontal motion of the drive tip at the drive frequency, $UX_{f_{drive}}$, and multiplying it by the drive frequency, f_{drive} , as shown in Equation (3.1).

$$v_{motor} = UX_{f_{drive}} \cdot f_{drive} \quad (3.1)$$

It should be noted that ANSYS assumes all nonlinearities as linear in modal and harmonic analyses. The stiffness of the contact interfaces is calculated at the beginning of the modal and harmonic analysis and is never changed²⁰. Once the sliding contact interfaces between the motor frame and end caps were added to F_harm_24.txt, the displacement magnitude overpredicted what was later seen using the transient analysis programs. Thus, Equation (3.1) overpredicted the motor speed, but was still useful to gage the change in motor velocities in varying design alterations.

3.4 STATIC ANALYSIS

Three static analysis programs were written to help develop and understand the USM presented in this thesis. While the harmonic analysis could be used to gage motor velocity, the first two programs of the static analysis are used to estimate the potential push force of the motor. Both programs use a piezoelectric preloading step before other loads are applied.

The first file, F_vert_stiff_24.txt, is used to obtain the vertical stiffness of the USM structure when a downward force is applied to the drive tip. As shown in Figure 3-4, a force, F , of 50 N is placed on the drive tip and the vertical displacement is measured, UY_F . The stiffness, K_Y , is then calculated by using Equation (3.2).

$$K_Y = \frac{F}{UY_F} \quad (3.2)$$

Figure 3-4 Motor Structure Vertical Stiffness

The next program, F_UY_24.txt, is used to find the vertical and horizontal displacements of the drive tip when one piezoelectric element is fully expanded, as shown in Figure 3-5. The horizontal displacement, UX_{PZT} , had little use because the horizontal magnitude of the motor is more accurately determined by the harmonic analysis because the proposed motor will use a horizontal acting resonance. However, the vertical displacement results are better determined by the static and not the harmonic analysis because of the interaction of the drive tip and slider. The drive tip's vertical motion is perpendicular to the slider causing a damping effect that does not allow the drive tip's vertical motion to significantly build over time. The static vertical displacement, UY_{PZT} , more accurately predicts the displacement seen in the motor simulation.

Figure 3-5 Drive Tip Displacement Caused by Actuated PZT

Using the results of F_stiff_24.txt and F_UY_24.txt, the push force of the motor can be estimated. The maximum change in normal force of the drive tip, N_{MAX} , against the slider can be found by Equation (3.3) using the vertical drive tip displacement due to an actuated piezoelectric element and the vertical motor stiffness (UY_{PZT} and K_Y). The maximum potential motor push force is then found using Equation (3.4), where μ_d is the dynamic friction coefficient for the contact interface.

$$N_{MAX} = UY_{PZT} \cdot K_Y \quad (3.3)$$

$$F_{SLIDER\ MAX} = \mu_d \cdot N_{MAX} \quad (3.4)$$

The third static analysis program, F_int_chk_24.txt, is used for setting up and checking the interference of the drive tip with the slider prior to running the dynamic model F_slide_24.txt. The program is also used to understand the static stress in the piezoelectric elements before further investigation with the dynamic model. The program is run in three steps: 1) the piezoelectric elements are preloaded, 2) the slider is moved against the motor frame with the preset drive tip interference, 3) one piezoelectric element is actuated. The first goal of this program is accomplished by using the GUI and checking the stress after the 2nd and 3rd stages of the solution. By looking at the static stress at these stages, a general comprehension of the effects of the preloading and the piezoelectric actuation on the stress distribution can be understood. Figure 3-6 shows an example output of the stress distribution after preloading. The individual components can be plotted separately for better resolution.

Figure 3-6 Static Stress Analysis of the USM: Separated into the Motor Frame, Piezoelectric Elements, and End Caps

In preparation for the dynamic motor simulation program F_slide_24.txt, F_int_chk_24.txt also calculates the drive tip clearance with respect to the slider once the motor is placed next to the slider. Figure 3-7 shows the measurement of $TIP_{CLEARANCE}$. The program is adjusted until $TIP_{CLEARANCE}$ is well below the order of magnitude of the drive tip motion. The $TIP_{CLEARANCE}$ value changes with meshing and geometry and must be adjusted to obtain accurate results from the motor simulation program F_slide_24.txt.

Figure 3-7 Drive Tip Clearance

3.5 DYNAMIC ANALYSIS

The Transient analysis code is an essential part to of the USM development process and consists of two programs. Both programs preload the piezoelectric elements before running the transient analysis. The transient programs are significantly more computationally expensive and thus are run after motor characteristics are estimated using the other previously explained programs.

The first analysis program, F_dyn_24.txt, is used to trace the drive tip motion running in free space. Figure 3-8 (a) shows a good motion which has a clear elliptical projectory. The width of the elliptical path can be used to estimate the drive speed of the motor by multiplying it by the drive frequency. Figure 3-8 (b) demonstrates an inconsistent drive path, which would probably not produce the desired mechanical output of an USM once a slider is added. While considering different potential designs, F_dyn_24.txt was run right after the harmonic analysis to check for clean clockwise and counter clockwise elliptical motion.

(a)

(b)

Figure 3-8 Drive Tip Path Analysis: a) Good Elliptical Motion at Drive Tip, b) Inconsistent Motion at Drive Tip

Up to this point in the analysis programs, all of the programs have neglected the slider/drive tip interaction because of modeling restrictions and time constraints. Modeling the interaction is an important part of understanding the USM dynamics and must be done to fully understand a motor's behavior¹⁶. The next program F_slide_24.txt includes the slider/drive tip interaction.

F_slide_24.txt models the USM driving a slider and is used to approximate motor performance. The program is run in three stages: 1) the piezoelectric elements are preloaded, 2) the slider is preloaded with an interference against the motor, and 3) an external force is added to the slider to push against the motor's driving force and one piezoelectric element is driven with a sine wave for a specified number of cycles. F_slide_24.txt is capable of running loops, which can vary the preload interference against the slider and the opposing slider push force. Contact elements located between the slider & drive tip and encaps & frame are created to allow for sliding and separation

of the surfaces. The contacts elements are also given friction coefficients as measured in Chapter 2.

Figure 3-9 shows an example output from the program, where the red line is the slider position, the cyan and purple lines being the horizontal and vertical drive tip displacement, respectively. Figure 3-9 (a) shows a motor with zero push force and Figure 3-9 (b) shows a motor pushing at its maximum push force that can be seen by the slow response and wavy motion. The motor speed is estimated by the slope of the linear portion of the slider displacement.

Figure 3-9 Motor Performance Graphs: a) Low Push Force, b) Maximum Push Force. Slider Position (Red), Drive Tip Horizontal and Vertical Displacement (Cyan and Purple Respectively)

F_slide_24.txt also outputs the drive tip path of the motor. Figure 3-10 (a) shows a good drive tip motion against the slider. Notice the flat smooth top region of the path where the drive tip would be in contact with the slider. Figure 3-10 (b) shows an inconsistent drive tip motion that would result in poor motor performance. There is not a

constant pressure against the slider during the upper half of the drive tip motion. The tip motion also appears to be inconsistent from loop to loop.

Figure 3-10 Drive Tip Motion with Slider Interaction: a) Good Circular Drive Tip Motion, b) Inconsistent Drive Tip Motion

F_slide_24.txt is also used to analyze the stress and contact status in the USM structure throughout a full drive cycle. When stress or other analysis information is wanted, F_slide_24.txt should be run with a single slider interference and push force. The program erases all analysis data while running loops, so the '/clear' command also needs to be commented out with a '!' near the end of the code.

Figure 3-11 illustrates the stress of the USM structure during different times in one cycle of the piezoelectric drive signal. Particular interest is given to the piezoelectric elements that are softer and more fragile than the rest of the motor. Maximum and minimum stresses are noted throughout the cycle. Other parameters that were examined

at different times during the drive signal to further understand the motor are the contact statuses (sticking, sliding, open) and localized displacements.

Figure 3-11 Dynamic Stress Analysis of the USM: Sample of Stress Plots Throughout a Full Input Cycle with Emphasis on Piezoelectric Elements

The transient codes also use a few subroutines to output graphs and data with specified titles. The subroutine, filenamecreate.tcl, in F_dyn_24.txt gives ANSYS an output name that consists of the driving frequency and the activated piezoelectric element's area number. The subroutine filenameSL.tcl in F_slide_24.txt gives ANSYS an output name consisting of the driving frequency, interference, and the file number.

Similarly, the filenameSLF.tcl returns a filename consisting of the driving frequency, interference, push force, and the file number.

4 ULTRASONIC MOTOR DESIGN

This chapter presents in detail the proposed ultrasonic motor design. An overview of the design process followed in this thesis is provided. After the design process is laid out, this chapter addresses the motor concepts leading to the final design. The final design concept is explained and individual features are examined in detail. All of the conclusions in this chapter are made by analyzing the results of the FEM programs explained in Chapter 3. A final design is proposed and FEM motor performance estimations are made.

4.1 OVERVIEW OF DESIGN PROCESS

Designing an ultrasonic motor can be a very iterative process. Minor changes of the motor's geometry can significantly change the vibratory properties the motor. Even if the motor concept can create elliptical motion in free space, it does not mean the motor concept will perform as expected when contacting a frictional interface¹⁶. This section will outline the process by which a concept was generated and then developed into the proposed ultrasonic motor prototype.

The first step of the design process was to generate a concept that had a reversible elliptical motion at a drive point. Figure 4-1 shows the process that was used to qualify USM concepts for further devolvment. The process starts with brainstorming for unique motor geometries. Once the motor concepts are created, the program F_mode_24.txt is

used to plot the first several modes of the motor concept structure. Modes are looked for that show potential in providing an elliptical motion at a drive point and that could also provide a reversible motion. If the concept does not show promise, then the process is started over. If the concept does show potential usefulness as a reversible motor, then the dynamic program F_dyn_24.txt is run at several frequencies, which plots the free motion of the drive tip. If reversible elliptical motion is not found, the concept is altered and retried. All successful concepts are considered for further development.

Figure 4-1 Motor Concept Generation Qualification Process

After a final motor concept is determined to be a viable candidate for the proposed ultrasonic motor, the concept is developed further using other FEM programs. Figure 4-2 shows the iterative USM development process. After every stage of the process, it was decided whether the motor design passed, failed, or just needed improvement. If the process step failed or needed improvement, the design was modified and the process restarted. The process became very iterative because every modification of the motor could cause significant changes to the different motor characteristics such as mode shapes, natural frequencies, modal displacements, forced displacements, and/or motor stiffness. These motor characteristic changes would have a direct effect on other motor properties like stress, drive tip motion, and motor performance.

Figure 4-2 guides one through the motor design process. The first stage checks for the desired modes of the motor and their natural frequencies. In the next stage, the driving frequency/frequencies are determined along with an estimation of the motor speed calculated by measuring the horizontal drive tip displacement at the drive frequency/frequencies. The third stage consists of measuring the motor vertical stiffness from the drive tip and the vertical displacement caused by a fully expanded piezoelectric element. The results are used to determine if there is sufficient vertical displacement and to estimate the motor stall force. Next, the motor is run in free space to check for a reversible elliptical motion at the drive tip. The fifth step checks the motors static stress and is used to eliminate the drive tip clearance with the slider. The final step is to run the motor simulation at various slider preloads, slider push forces, and drive frequencies. The results are then evaluated including motor speed, stall force, stress, and contact statuses.

Figure 4-2 USM Development Process

4.2 PRELIMINARY DESIGN CONCEPTS

The primary objective of this thesis was to create a unique single drive signal motor. In addition, it was originally desired to use only one piezoelectric element in the design. Figure 4-3 illustrates several single-element concepts that were considered as potential USM designs. Several of the concepts shown make use of a flex-tensional piezoelectric actuator (FPA) design. A FPA design can create motion in two orthogonal directions with the expansion of the piezoelectric element in a single direction.

Figure 4-3 Single-Element Motor Concepts: a) Fourbar, b) FPA with legs, c) finger, d) FPA with arch ground. Piezoelectric elements are represented in dark gray.

None of the motor concepts in Figure 4-3 were used for several reasons. Several could not produce a reversible elliptical motion, others were similar to USM designs already created, and some had resonance frequencies well below the minimum desired level. Since several single-element motor concepts failed to meet the goals of this thesis, several multi-element motor concepts were developed. Figure 4-4 illustrates two multi-element motors concepts developed. Both concepts use a FPA-like design with piezoelectric elements connected to ground or close to it.

Figure 4-4 Multi-Element Motor Concepts: a) Side Actuated FPA, b) FPA with Grounded Center

Both of the multi-element concepts could create elliptical motion with a single signal to one piezoelectric element. The reverse direction would make use of the second piezoelectric element. The FPA with the grounded center, shown in Figure 4-4 (b), was chosen over the side actuated FPA because of its superior elliptical motion and more compact design. The USM concept, illustrated in Figure 4-4 (b), is also a unique design from any current ultrasonic motor.

4.3 FPA WITH GROUNDED CENTER CONCEPT

This section explains motor features and properties considered for the proposed USM. Several modifications to the FPA with a grounded center motor concept were experimented with during the development of the proposed motor. General trends of the motor's modifications will be discussed along with their consequences to the motors characteristics. Many motor properties must be considered for each alteration due to the strong interactionary effects each modification can have on the motor.

4.3.1 Materials

The motor materials have a significant effect on the motor vibratory characteristics and should be considered early in the design process. Several materials have already been discussed in Chapter 2 in relation to their contact interaction properties. Tool steel was chosen over stainless steel for the stator frame because of its better wear properties. Table 4-1 shows the material properties of tool steel for the stator frame, PZT for the piezoelectric elements, and oxide ceramic 2 for the friction bar.

Table 4-1 Material Properties of Motor Materials: Modulus of Elasticity, Poisson's Ratio, Density, & Damping Ratio^{23, 24}

Materials	E (GPa)	ν	ρ (kg/m³)	ξ
Tool steel	203	0.30	7820	0.001
PZT	44	0.34	7700	0.0005
Oxide Ceramic 2	303	0.21	8720	

The damping properties of the materials significantly change the mechanical amplification amplitudes of the stator because USMs operate at or near resonance

frequencies. The damping ratio for the PZT was calculated by using Equation (4.1) which is valid when driving at a resonance frequency, where Q and ξ are the quality factor and damping ratio respectively²⁵. The damping ratio of the PZT was calculated assuming a standard quality factor of 1000.

$$\xi = \frac{1}{2 \cdot Q} \quad (4.1)$$

For use in the analysis programs of this thesis, the FEA program ANSYS requires that Rayleigh damping be used²⁰. Rayleigh damping is described by Equation (4.2) where C is the damping matrix, M & K are the structural mass and stiffness matrices, and α & β are the constant mass and stiffness matrix multipliers.

$$C = \alpha M + \beta K \quad (4.2)$$

The constant matrix multipliers α and β can be calculated if one knows the modal damping ratios at several modes. If the damping ratio is only known for the fundamental mode it is an acceptable procedure in structural problems to assume $\alpha = 0$ and β can be estimated by Equation (4.3) where ξ_1 is the fundamental mode damping ratio and f_1 is the frequency of the fundamental mode^{20, 24}.

$$\beta = \frac{\xi_1}{\pi \cdot f_1} \quad (4.3)$$

By using Equation (4.3) and the damping ratios in Table 4-1, the material damping coefficient, β , was calculated to be approximately 1×10^{-8} and 5×10^{-9} for the steel and PZT respectively assuming the fundamental mode was around 30 kHz. It should be noted that even though a user may define material damping to individual material sets, ANSYS only allows β to be defined once for each element type used in the analysis²⁰.

For this reason, the larger material damping value of 1×10^{-8} was used for both the steel and PZT materials.

4.3.2 Arches

The arch/arches are one of the fundamental features of the proposed USM design. As seen by the mode shapes in Figure 4-5, the top arch can provide a mechanical amplification at the drive tip in both the vertical and horizontal directions. Alterations to the arches cause changes to every characteristic of the motor including motor performance.

Figure 4-5 Desired Natural Modes of USM Structure

One of the first decisions made in the design process was to choose the use of one or two arches as shown in Figure 4-6. Two arches provided a more symmetric design, which was good considering that a more even loading of the fragile piezoelectric elements could be accomplished. While the loading characteristics of two arches were a large plus, there were also a couple of drawbacks. The first drawback was that the double

arch introduced several other natural modes around the operating frequency while the single arch design only had the two desired modes near the driving frequency shown in Figure 4-5. The addition of several modes could cause erratic behavior when operating the motor. The second drawback was that the center frame could not be grounded at the bottom making the design slightly more complex.

Figure 4-6 Single and Double Arch Designs

Once the single arch design was decided upon, the arch geometry was modified to determine the general effects of the arch on motors performance. The first dimensions to be analyzed were the arch thickness, height, and width as defined in Figure 4-7. As the arch height or thickness increased, the motor vertical stiffness, K_Y , increased but the non-modal amplified vertical drive tip displacement, UY_{PZT} , decreased. As explained in Section 3.4, the motor vertical stiffness, K_Y , and vertical displacement, UY_{PZT} , determine the motor's push force potential by determining the maximum normal force range that can be exerted by the motor on the friction bar. When the arch became taller and skinnier, the horizontal modal displacement increased which would increase the motors

speed. The mode's relative natural frequencies to each other could be altered simply by increasing the arch height or decreasing the arch thickness, which would decrease the difference between their frequencies. When the width of the arch was increased, keeping the rest of the motor constant, the vertical stiffness and displacement (K_Y & UY_{PZT}) decreased, greatly decreasing the motors push potential as explained in Chapter 3.4.

Figure 4-7 Arch Dimensions

Figure 4-8 illustrates three arch base geometries that were considered: thick base, thin outer base, and thin inner base. The design's push force potentials were compared while keeping a constant vertical and horizontal frequency difference of 1 kHz. At the desired vertical displacement, UY_{PZT} , of two microns, the thick arch had the highest force output potential while the thin outer and thin inner arches were 7.7% and 3.8% less respectively. It should also be noted that if the vertical displacement, UY_{PZT} , is allowed to decrease more, the motor force output potential could increase at a rate of

approximately 40% per micron. The limiting factor being when the vertical displacement is decreased too far, motor stall will occur due to the surface roughness of the drive tip and friction bar and the motor will never move.

Figure 4-8 Three Types of Arch Bases: a) Thick, b) Thin Outer, c) Thin Inner.

While it was the original intent to use both the vertical and horizontal modes shown in Figure 4-5, little to no vertical amplification of the drive tip was seen during the motor simulations. This was thought to be caused by the contacting interface of the motor with the friction bar. It was determined that the motor's maximum push force could be increased by raising the arch, which increased the motor's vertical stiffness. The modification also raised the vertical mode frequency and lowered the useful horizontal mode frequency.

4.3.3 Legs

The next geometric feature of the motor that was considered was the legs. Figure 4-9 illustrates the dimensions and positions of the legs that were considered. Three

dimensions were altered (leg slant, width, and height) and two leg positions were considered (outside and inside). Shorter wider legs were determined to increase the motor speed by increasing the horizontal mechanical amplification at the drive tip. The motor's outside legs were thought to be more advantageous because they caused lower stress in the piezoelectric elements and less rotation of the side of the motor than the inside legs.

Figure 4-9 Leg Types and Dimensions: a) Outside Leg, b) Inside Leg.

While the legs were originally planned on being vertical, by slanting them at an angle the vertical displacement, UY_{PZT} , was increased and the vertical stiffness was fairly unchanged as long as the slant was not too large. The net result was an increase in the motor's push force potential.

A model was also considered eliminating the legs altogether. However, the structure had no way of supporting itself from rocking and had a low vertical stiffness.

Even when a bottom arch was added and grounded at the center, the motor structure lacked the stiffness needed to produce the desired push force of the motor.

4.3.4 Center Block

The motor center block affects the motor speed and motor stiffness. Figure 4-10 shows the two dimensions of the center block of the frame. The motor speed could be increased by increasing the width and decreasing the height. However, when the width is increased the arch width is also increased which reduces the motor potential push force. The center block is best when it is the most rigid it can be without increasing the width, but the center must be rigid enough to achieve the desired speed of the motor.

Figure 4-10 Motor Center Frame Dimensions

4.3.5 Piezoelectric Element Mounting

The final geometric feature considered was the piezoelectric mountings. The original design concept consisted of gluing both sides of the piezoelectric elements to the frame as shown in Figure 4-11 (a). The stress in piezoelectric elements was too large for

the mounting configuration to work. To reduce the stress, a rotating end cap and a flexure end cap were tried. The flexure end cap was quickly eliminated because it widened the arch so much that the motor had very little push force potential.

Figure 4-11 Piezoelectric Element Mounts: a) Glued - Glued, b) Rotating End Cap - Glued, c) Flexure End Cap - Glued.

The rotating end cap was chosen as the best mounting because of its compact design and lower stress on the piezoelectric elements. The sliding interface in the ball and socket joint decouples the vertical stretching of the frame from the piezoelectric elements, which significantly reduced the stress in the elements. However, during simulation the end cap would separate from the frame causing a large detrimental effect to the motor performance. To stop the separation, the legs were moved to the inside position which causes more of a rotation effect on the motor ends than the outside legs.

4.4 FINAL DESIGN

After the piezoelectric mounting was decided upon, the arch, legs, and center dimensions were chosen to yield the desired performance characteristics. Table 4-2 gives

the final specifications and dimensions of the motor as described in Section 4.3. The frame was designed for piezoelectric elements that are 5 x 5 x 4 mm in size. The thin-inner arch was chosen over the thick arch to reduce the oscillating mass of the motor. The thin-inner arch and inner legs allowed additional material to be cut off the motor sides to even further reduce the mass of the motor sides. The reduced mass lessens the amount of inertia that the piezoelectric element preload force must overcome to keep the elements in compression and to keep the contacts from separating.

Table 4-2 FPA with Grounded Center Specifications

Materials	
Motor	Tool Steel
Slider	Oxide Ceramic 2
Piezo Elements	PZT 5 x 5 x 4 mm
Arch	
Type	Single – Thin Inner
Height	2.8 mm
Thickness	1.3 mm
Width	11 mm
Legs	
Type	Inner Leg w/ Slant
Height	3 mm
Thickness	1.5 mm
Slant	1.5 mm
Center Block	
Width	3 mm
Height	0.5 mm
Piezo Mounting	
Type	Ball and Socket End Cap
Thickness	0.5 mm
Socket Radius	12.5 mm

4.4.1 Final Geometry

The final geometry includes the chosen dimensions of the motor features previously described and is shown in Figure 4-12. Additional fillets were added to allow machining by wire EDM (electrical discharge machining). Also as noted earlier, large chamfers were added to the motor's sides to reduce their inertial effects while the motor is operating. The base is made to easily bolt onto a fixture for testing.

Figure 4-12 Final Ultrasonic Motor Design Frame Dimensions

Figure 4-13 Final Ultrasonic Motor End cap Design Dimensions

Figure 4-13 shows the final design of the piezoelectric end cap for the ball and socket joint. They are made thin to reduce the inertia at the end of the piezoelectric elements and to keep the arch width to a minimum.

The final main motor body (not including the base used for mounting during testing) is approximately 17.4 x 12.5 x 10 mm, which is well within the desired size limit of 40.5 x 25.7 x 8 mm of the Nanomotion HR1 motor¹⁴. Of course, a mounting package would also have to be designed for the motor if it were to be sold commercially, which would slightly increase the overall dimensions.

4.4.2 Finite Element Modeling Performance Predictions

Table 4-3 shows the final motor performance specifications estimated by using the FEM motor simulation program (F_slide_24.txt). The motor is made to operate at 60

volts at 30.1 kHz; however, it is expected that the driving frequency will be slightly different once the prototype frequency response is tested. The motor has a very large static hold force of 12.9 N at a motor preload of 76 N. The motor should have a maximum sliding force of 5 N and a maximum no-load velocity of 0.4 N. The motor's performance predictions exceed the desired push force of 4 N, no-load velocity of 0.25 m/s, and static hold force of 3.5 N¹⁴.

Table 4-3 Final FEM Design Performance Specifications

Motor Specifications	
Energized Vertical Displacement	2.17 μm
Non-Energized Vertical Stiffness	36.3 N/ μm
Operating Voltage	75 V
Operating Frequency	30.1 kHz
Nominal Preload on Stage	76 N
Static Holding Force	12.9 N
Maximum Sliding Force	5 N
Maximum Velocity	0.4 m/s

Figure 4-14 shows the velocity versus force curves created from the FEM motor simulation. Several different motor preloads on the stage are plotted. The highest preload that still allows the ultrasonic motor's drive tip to leave the slider surface performed the best. Not only is it capable of pushing a higher force, it also has the same no-load velocity as the other preloads.

Figure 4-14 FEM Speed and Force Performance Estimations at Several Motor Preloads

5 FINAL DESIGN PROTOTYPE RESULTS

In this chapter, the final design prototype results will be presented. Fixtures for testing of the motor will be described along with measuring techniques used to measure the motor's characteristics and performance. The prototype test results are compared to the finite element modeling presented in Chapters 3 & 4 and to the benchmark Nanomotion HR1 ultrasonic motor.

5.1 EXPERIMENTAL TEST SETUP

The prototype motor is shown in Figure 5-1. It consists of the design described in Chapter 4. The piezoelectric elements are glued into place with a low modulus epoxy to keep the elements and end caps from vibrating out of the frame. The drive tip was polished to ensure a fine surface finish. The red and black wires are attached to piezoelectric element 1 and 2 and represent the positive and negative leads of the motor.

Figure 5-1 USM Prototype

Figure 5-2 shows the fixture designed to test the motor. The USM prototype is bolted to an aluminum piece and is mounted on a linear slide. The linear slide permits the motor to move freely into the output slider allowing adjustment to the motor preloading. The motor preloading is controlled by a screw and spring in series and is shown in Figure 5-2 as the preload adjuster. A load cell is placed between the preload adjuster and the motor to measure the preload force on the motor. The motor may be operated in either direction by activating piezoelectric element 1 or 2 by flipping the switch in the proper direction. The ceramic drive surface was attached to the output slider by applying epoxy around the outside edges of the ceramic. The output slider also has holes on each side where a resistive force may be applied during motor operation.

Figure 5-2 USM Prototype Test Fixture

Figure 5-3 shows the complete test setup for the USM prototype. A HP Multifunction Synthesizer is used to generate a sine wave at the desired frequency. A Crown DC300A Amplifier then amplifies the signal. An oscilloscope monitors the amplified signal as it is being directed to the motor. A switch before the motor allows the signal to be directed to either piezoelectric element. The load cell indicator is used to view the current motor preload on the motor.

A scanning laser vibrometer is used to measure mode shapes, low and high frequency motor frame displacements and output slider speed. The vibrometer controller is used to setup vibrometer measurements and to record all the data.

Figure 5-3 USM Prototype Test Setup

The vibrometer was used to measure the velocity amplitudes of the motor drive tip in two orthogonal directions and the end caps while the motor is not preloaded against the output slider. From the velocity measurements, the displacements of these points of interest can be computed. Figure 5-4 (a) illustrates the measurement of the vertical motor tip displacement. Figure 5-4 (b) shows the measurement of the motor end cap displacement and the horizontal drive tip displacement. These measurements are used to determine if the motor was modeled correctly and to determine if the piezoelectric elements are operating correctly.

(a)

(b)

Figure 5-4 USM Prototype Free-Space Testing: a) Measurements Taken at Motor Tip, b) Measurements Taken on the Motor's Side

5.2 PROTOTYPE TEST RESULTS

The USM prototype was first tested at a low frequency (200 Hz) to see if the expected steady state amplitudes were achieved. The low frequency signal allows the piezoelectric elements to expand and contract while minimizing any dynamic forces. When measuring the drive tip and end caps, the measured displacement amplitudes were about $1/10^{\text{th}}$ of what was expected. Thinking that the piezoelectric elements could have been depolarized, the elements were repolarized by applying 120 volt DC to each element for 10 minutes. After repolarization, the end cap displacement of the motor increased by 29.7% indicating that some depolarization had occurred. The depolarization was likely caused by overheating when the piezoelectric elements were run too long.

Table 5-1 shows the steady state displacements for each piezoelectric element at different locations on the motor frame. As can be seen, the best displacement is 29% of the model result, while most are approximately 19% of the model results. Piezoelectric element 2 has a larger range of displacement ratios than element 1, which probably indicates improper contact between the end cap and frame. The cause of the discrepancies in the model and the prototype are discussed later in this chapter.

Table 5-1 USM Prototype Steady State Displacements

Steady State Motor Displacements	FEM Model	Piezoelectric Element 1	Ratio	Piezoelectric Element 2	Ratio
Vertical Tip Disp.	1.75 μm	0.298 μm	0.170	0.187 μm	0.107
Horizontal Tip Disp.	1.39 μm	0.314 μm	0.226	0.405 μm	0.291
End Cap Disp.	1.85 μm	0.368 μm	0.199	0.317 μm	0.171

After the steady state amplitudes were measured, the laser vibrometer was used to scan the motor tip in the vertical and horizontal directions. Random noise was fed into one piezoelectric element at a time and the drive tip was scanned. Figure 5-5 and Figure 5-6 show the averaged frequency response plots of the scanned drive tip. Using the 2-D FEA model, the first two expected resonances were at 30.1 and 41.2 kHz. The response plots give a good indication of which frequencies the motor might operate. Figure 5-7 and Figure 5-8 show the horizontal velocities (blue) across the drive tip for several of the lower peaks of the frequency response plots. Both the second and third peaks seem suitable for driving the motor because of the fairly even horizontal displacements across the drive tip seen in Figure 5-7 and Figure 5-8. The motor was tested around both of these frequencies. It should also be noted that it is expected that the resonant peaks could shift during operation of the piezoelectric elements and when the slider interference is added.

Figure 5-5 Frequency Response of Drive Tip from PZT 1 (Vertical and Horizontal Response, Blue and Purple Respectively)

Figure 5-6 Frequency Response of Drive Tip from PZT 2 (Vertical and Horizontal Response, Blue and Purple Respectively)

Figure 5-7 Drive Tip Horizontal Resonances Using PZT 1

Figure 5-8 Drive Tip Horizontal Resonances Using PZT 2

After the initial testing was completed, the motor fixture was assembled and the motor was preloaded against the output slider. The motor was run at a light preload of 6 N and piezoelectric elements were driven through a range of frequencies of 24 to 36 kHz. Motion of one complete stroke of the output slider was achieved at 34.6 and 36 kHz while driving element 2. However, element 1 could complete the full output stroke of the slider from 27 to 29 kHz and around 35 kHz. At a higher preload of 45 N, motion was seen occasionally at around 29.7 kHz, but was very inconsistent. The motor ran the best using element 1 at 27.7 kHz. Table 5-2 shows the fastest speed as 55.6 mm/s and the highest load as 0.3 N, which was obtained by using element 1.

Table 5-2 Motor Force – Speed Performance Results

Push Force (N)	Speed (mm/s)
0	55.6
0.070	36.0
0.139	23.0
0.209	7.2
0.278	3.48
0.348	2.58

Table 5-3 compares the results of the Nanomotion HR1, the FEM model, and the prototype. Because of the low displacements, the prototype preload had to be reduced greatly for the motor to operate. The maximum motor speed of the prototype was 13.9% of what was expected and 22.2% of the desired goal of the Nanomotion HR1. The force is 7.0% and 8.7% of the expected result and goal, respectively. The prototype results suffer because of the unexpected low displacements of the motor.

Table 5-3 Comparison of NM HR1, Model, & Prototype

Motor Specification	NM HR1	FEM Model	Prototype	Units
Operating Frequency	39.6	30.1	27.7	kHz
Nominal Preload on Stage	18	76	6.3	N
Static Holding Force	3.5	12.9	1.05	N
Maximum Sliding Force	4	5	0.348	N
Maximum Velocity	0.25	0.4	0.0556	m/s

Table 5-4 Comparison of Model & Prototype

Motor Specification	FEM Model	Prototype	Ratio
Operating Frequency	30.1 kHz	27.7 kHz	0.920
Nominal Preload on Stage	76 N	6.3 N	0.083
Static Holding Force	12.9 N	1.05 N	0.081
Maximum Sliding Force	5 N	0.348 N	0.070
Maximum Velocity	0.4 m/s	0.0556 m/s	0.139

Table 5-4 shows the ratios of the results of the model and prototype. Most of the results are approximately 8% of the expected results from the modal. Table 5-5 shows the size comparison of the prototype and the Nanomotion HR1. Of course, a mounting/preloading package would also have to be designed for the motor if it were to be sold commercially, which would increase the overall dimensions.

Table 5-5 Motor Size Comparison

Motor	Size	Volume
USM Prototype	17.4 x 12.5 x 10 mm *	2175 mm ³ *
Nanomotion HR1	40.5 x 25.7 x 8 mm	8327 mm ³

* Motor size does not include motor casing for mounting and preloading

5.2.1 FEM / Prototype Discrepancies

To determine the problem with the lower than expected steady state displacements, the piezoelectric elements were measured to check for proper expansion. The PZTs should expand $1.9\text{ }\mu\text{m}$ at 60 volts unbounded. With a small increase of load (caused by the frame) of 33 N, the expected expansion can be calculated to be $1.78\text{ }\mu\text{m}$. By taking the PZT expansion and multiplying it times its Poisson's ratio of 0.34, the sides are calculated to move approximately $0.605\text{ }\mu\text{m}$. Figure 5-9 shows the measurement technique used to measure the PZT's side displacements. The angled measurements in Figure 5-9 (b) were corrected for angle. Table 5-6 shows the measurements taken of the piezoelectric elements sides. By examining the ratios of the expected results to the measurements, it is clear that the elements were expanding correctly.

Figure 5-9 Piezoelectric Element Measurement Method

Table 5-6 Piezoelectric Element Measurements

Location	Displacement (nm)	Expected (nm)	Ratio
PZT 2 (Top/Bottom)	569	605	0.940
PZT 2 (Front/Back)	588	605	0.972
PZT 1 (Top/Bottom)	not measurable	605	N/A
PZT 1 (Front/Back)	670	605	1.107

Given that the piezoelectric elements are operating correctly, there has to be compliance inline with the elements where the motion is lost. There are three potential sources of compliance in the motor assembly. The first one could be caused by glue and insulation on each side of the PZTs. The glue and insulation have a modulus of approximately 2.5 GPa. The stiffness of the glue layer can be conservatively calculated by Equation (5.1), where k is the stiffness, E is the modulus of elasticity, A is the area, and L is the length.

$$k = \frac{E \cdot A}{L} \quad (5.1)$$

Figure 5-10 shows the frame socket joint next to piezoelectric element 2. Glue layers on both sides of the end cap can be seen. The end cap has a flatter radius than specified in the original design. It was reduced during assembly when it was sanded. The total thickness of the glue layers on both sides of piezoelectric element 2 and in between the frame and end cap was measured to be approximately 0.8 mm. The area of the PZT facing the socket is $25\text{e}^{-6} \text{ m}^2$, by examining the glue layer under the microscope the glue is seen not to cover the entire area. Assuming the contact area to have an area of $16\text{e}^{-6} \text{ m}^2$ and given a frame and PZT stiffness of $75 \text{ N}/\mu\text{m}$ and $275 \text{ N}/\mu\text{m}$ respectively, the thick glue layers would account for $1.21 \mu\text{m}$ of lost displacement, which is approximately 71% of the total motion. From Figure 5-10 and the estimated loss displacement, it is

theorized that the motor's low-frequency displacements are caused by manufacturing error, which primarily include the thick glue layers.

Figure 5-10 USM Ball and Socket Joint

The second source of the of compliance inline with the PZTs could be caused by uneven pressure or clearances across the endcap and frame socket. Very small clearances in part of the socket could cause a significant reduction in piezoelectric motion. Because of the assembly difficulties, it would not be surprising if such clearances or uneven

pressures existed across the end cap and frame joint. Varying thicknesses of glue on each side of the end cap would also create uneven pressure across the frame socket joint when a piezoelectric element is actuated.

The third source of compliance inline with the PZTs could be caused by the end cap joint. If the motor had been manufactured correctly, Hertzian stress in the ball and socket joint (end cap and frame joint) would also have caused a loss of motion. Pressure between two curved elastic bodies causes surface stresses and surface displacements larger than what would be seen between to flat-sided bodies. Equations (5.2)-(5.5) are the Hertzian cylindrical ball and socket equations and can be used to calculate the deflection of the end cap and socket²⁶. In these equations, 1 and 2 represent the socket and ball respectively, ν is Poisson's ratio, E is the modulus of elasticity, D is the ball or socket diameter, and p is the distributed load, as shown in Figure 5-11.

Figure 5-11 Hertzian Stress

$$C_E = \frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2} \quad (5.2)$$

$$K_D = \frac{D_1 \cdot D_2}{D_1 - D_2} \quad (5.3)$$

$$b = 1.60 \cdot \sqrt{p \cdot K_D \cdot C_E} \quad (5.4)$$

$$\delta = \frac{2 \cdot p(1-\nu^2)}{\pi \cdot E} \cdot \left(\frac{2}{3} + \ln \frac{2 \cdot D_1}{b} + \ln \frac{2 \cdot D_2}{b} \right) \quad (5.5)$$

When the motor was being manufactured, the end caps were sanded to help assembly. Assuming the socket to have been assembled without sanding or glue between the end cap and frame, the Hertzian displacement could be calculated by assuming a diameter of 24.5 mm and 25 mm for the end cap and frame socket respectively, a load of 128 N, and the load distributed over 5 mm; the loss of motion would be 0.649 μm , approximately 34% of the total motion. The sanding of the end caps and frame give a reduced and unknown contact radius and width, which would increase the Hertzian displacement further. While the Hertzian stress did not cause the total lack of displacement with the prototype, it would have caused a loss of about 1/3rd of the PZT motion. If a second prototype was produced, the Hertzian displacement could be reduced by using better tolerances on the ball and socket joint, increasing the ball and socket diameters, and using a higher modulus end cap.

6 CONCLUSIONS

This chapter summarizes the development of a unique ultrasonic motor created for this thesis. Results from the finite element modeling and from the motor prototype are summarized. Recommendations are made for future work and development.

6.1 SUMMARY

A new geometrically unique ultrasonic motor was designed using the FEM package ANSYS. An idea for a simple motor frame was conceived that consisted of an arch with a center ground with two piezoelectric elements connecting the frame to the center ground. Legs were added to the frame to increase its rigidity. Several FEM programs were written to design the motor and to predict its characteristics and performance. End caps were added between the frame and the piezoelectric elements to reduce the stress of the elements. The FEM motor model exceeded the performance characteristics of the benchmark Nanomotion HR1. The model predicted a linear motor capable of pushing up to 5 N and a maximum speed of 0.4 m/s.

Tool steel and oxide ceramic were chosen for the final motor prototype and slider. The prototype was built as designed. The USM prototype's piezoelectric elements did not provide the expected displacements in the motor frame. The cause of the lack of displacement was the thick soft glue layers inline with the piezoelectric motion. Had the

motor been manufactured correctly, Hertzian displacement would have absorbed $1/3^{\text{rd}}$ of the piezoelectric elements motion.

The prototype frame's steady state displacements were approximately 19% of the expected output from the FEM models. Given the low steady state output, the motor still reached a maximum speed of 55.6 mm/s and a maximum push force of 0.348 N at a preload of 6 N.

6.2 RECOMMENDATIONS FOR FURTHER RESEARCH

There are several recommendations to further the development of the prototype motor presented in this thesis. First, it is recommended that the next prototype be built at a larger scale. A larger motor would be easier to tolerance, to test, and would not be significantly affected by surface roughness effects. The second recommendation would be to eliminate or reduce the several locations of compliance found in the motor. The insulation and glue inline with the piezoelectric elements should have a higher modulus and therefore more stiffness. Glue layers should also be kept as thin as possible. The Hertzian stress needs to be reduced in the ball and socket joint in the motor. The Hertzian displacement may be reduced by increasing the ball and socket diameters, decreasing the difference in ball and socket diameters, increasing the modulus of the end caps by using a different material, and improving tolerances to ensuring proper assembly. It is recommended that the motor tip be made flat to reduce the Hertzian stress on the tip. The end cap joint should be analytically analyzed to determine the Hertzian stress and then the model should be adjusted accordingly.

With the correct amplitudes, further testing could be done on the motor to better determine its damping properties and improve the FEM models. The motor then could be modeled better and developed for commercial applications.

REFERENCES

- ¹ Uchino, K. and Koc, B., “Compact Piezoelectric Ultrasonic Motors,” *International Center for Actuators and Transducers*.
- ² Ngol, B. K. A., Lim, L. E. N., and Lin, W., “Linear Piezoelectric Motor with Self-locking Means”, U.S. Patent 6,611,080, Aug 26, 2003.
- ³ Feinstein, A. and Teschler, L., ”Coarse and Fine Combined”, *Machine Design* vol. 76, Issue 13 (2004).
- ⁴ Koc, B., Basaran, D., Akin, T, and Erden, A., “Design of a Piezoelectric Ultrasonic Motor for Micro-Robotic Application”.
- ⁵ Uchino, K., *Piezoelectric Actuators and Ultrasonic Motors*. Boston: Kluwer Academic Publishers, 1997.
- ⁶ Ueha, S. and Tomikawa, Y., *Ultrasonic Motors: Theory and Application*. New York: Oxford University Press, 1993.
- ⁷ Friend, J. R., Satonobu, J., Nakamura, K., Ueha, S., and Stutts, D. S., “A Single-Element Tuning Fork Piezoelectric Linear Actuator,” *IEEE Transactions on Ultrasonics, Ferroelectrics, and Frequency Control* vol. 50, no. 2 (2003).
- ⁸ Elliptec Resonant Actuator AG, “X15G Data Sheet” [Online], Oct 2004 [cited July 1, 2005]. Available from URL: <http://www.elliptec.com>.
- ⁹ EDO corp. “EDO Piezoelectric Motor”, EDO Catalog 2002, <http://www.edoceramic.com>.
- ¹⁰ Physik Instrumente (PI) L.P., “P-661” [Online], Dec 2005 [cited April 12, 2006]. Available from URL: <http://www.physikinstrumente.com>.
- ¹¹ Teschler, L., “Precision Moves with Pint-Sized Motors”, *Machine Design* vol. 77, Issue 7 (2005).

- 12 Waldimir Wischnewski, W., “Piezoelectric Adjusting Element”, US Patent, 6765355, July 20, 2004.
- 13 Zumeris, J., “Ceramic Motor”, US Patent, 5453653, September 6, 1995.
- 14 Nanomotion LTD., “HR1 Ultrasonic Motor User Manual” [Online], Dec 2005 [cited April 12, 2006]. Available from URL: <http://www.nanomotion.com>.
- 15 Sashida, T. and Kenjo, T., *An Introduction to Ultrasonic Motors*. New York: Oxford University Press, 1993.
- 16 Wallaschek, J., “Contact Mechanics of Piezoelectric Ultrasonic Motors”, *Smart Materials and Structures* vol. 7 (1998).
- 17 Nakamura, K., Kurosawa, M., and Ueha, S., “Characteristics of a Hybrid Transducer-Type Ultrasonic Motor”, *IEEE Transactions on Ultrasonic, Ferroelectrics, and Frequency Control* vol. 38 (1991).
- 18 Kamano, T., Suzuki, T., and Otoi, E., “Characteristics and Model of Ultrasonic Motor”, *Proceedings of 8th Symposium on Ultrasonic Electronics* (1987).
- 19 Physik Instrumente (PI) L.P., “User Manual MP 58E: P-661 & P-665”.
- 20 ANSYS Incorporated, “ANSYS Release 8.0 Documentation”, <http://www.ansys.com>.
- 21 Lawrence, K.L., *ANSYS Tutorial: Release 6.1*. Schroff Development Corporation Publications, 2002.
- 22 “UofA ANSYS Tutorials” [Online], April 2004 [cited April 24, 2006]. Available from URL: <http://www.mece.ualberta.ca/tutorials/ansys/>.
- 23 Callister, W. D. Jr, *Materials Science and Engineering an Introduction (5th edition*. New York: John Wiley & Sons, Inc., 2000.
- 24 Smith, J. W., *Vibration of Structures: Applications in Civil Engineering Design*. London: Chapman and Hall, 1988.
- 25 Harris, C. M., *Shock and Vibration Handbook (5th edition.)*. Boston: McGraw-Hill, 2002.

- ²⁶ Roark, R.J., Young, W.C., *Formulas for Stress and Strain (5th edition)*. New York: McGraw-Hill, 1975.

APPENDIX

APPENDIX

This Appendix contains the ANSYS batch code developed for this research and a basic ANSYS batch files tutorial. The codes for the ultrasonic motor development are provided for those interested in understanding the modeling techniques used in this thesis or wishing to use them as a starting point for further research. The tutorial is provided as a starting point for those who desire to learn to how to use ANSYS batch files.

MOTOR FEA CODE

This section contains the ANSYS batch files used in the experiment modeling of the ultrasonic motor developed in this thesis.

Modal and Harmonic Analysis Code

The USM development process began with modal and harmonic analysis of the motor geometry. The first file, F_mode_24.txt is used to obtain the mode shape and natural frequencies of the USM structure. The second file, F_harm_24.txt is used to estimate the drive tip vertical and horizontal displacements and relative phases. It should be noted that ANSYS assumes all nonlinearities as linear in modal and harmonic analyzes and the results may differ from the dynamic analysis. The uses of these results and the nonlinearities involved are explained in Section 3.3.1.

F_mode_24.txt

```
/FILENAME,mode
/TITLE,FPA 2-D Modal

/PREP7
ET,1,PLANE82      !element type

KEYOPT,1,3,3
KEYOPT,1,5,0      !extra element output
KEYOPT,1,6,0      !extra surface output
R,1,.008,         !thickness snow
R,2,.010,         !thickness slider
R,3,.005,         !thickness pzt

MP,EX,1,203e9     !Tool Steel
MP,PRXY,1,.3
MP,DENS,1,7820
MP,DAMP,1,1e-8

MP,EX,2,44e9      !Piezo
MP,PRXY,2,.34
MP,DENS,2,7700
MP,DAMP,2,1e-8
MP,CTEX,2,.0006

wp = .004         !Width piezo
hp = .005         !Height piezo
hs = .006         !Height side
ws = .003         !width sides
ll = .003         !Length Legs
wl = .0015        !Width legs
wm = .003         !width middle
wpp = .0005       !width push plate
rpp = .0125       !diameter push plate
olpp = .0003      !overlap push plate
tarc = .0013      !thickness mid arc
harc = .0028      !height mid arc

lpos= .0015       !leg position from outside
loff= .0015       !leg offset

pradin = harc-tarc/2    !Position radius inner
pradout = harc+tarc/2   !Position radius outer

RECTNG, -wp-wm/2-ws-wpp+olpp, -wp-wm/2-wpp+olpp, 0, hs/2    !Frame side
KL,3,0.5,38
KL,4,2/3,39
A,4,38,39
ASBA,1,2
ARSYM,Y,3
AADD,ALL

CYL4,rpp-wp-wm/2-wpp,0,rpp    !Frame socket
ASBA,2,1
ARSYM,X,3
```

```

K,40,0,hs/2+pradin                                !Top arc
K,41,0,hs/2+pradout
LARC,3,1,40
LARC,5,38,41
AL,2,21,9,22

legpt = -wp-wm/2-ws-wpp+olpp+lpos                !Legs
K,50,legpt,-hs/2
K,51,legpt+wl,-hs/2
K,52,legpt+loff,-hs/2-11
K,53,legpt+wl+loff,-hs/2-11
A,50,51,53,52
ARSYM,X,4

CYL4,0,hs/2+pradout-.0001,.0004                  !Frame tip
AADD,ALL

ASEL,NONE                                           !End caps
CYL4,rpp-wp-wm/2-wpp,0,rpp
RECTNG,-wp-wm/2,rpp*2,-rpp*2,rpp*2
RECTNG,-wp*2,0,hp/2,hp
RECTNG,-wp*2,0,-hp/2,-hp
ASBA,1,2
ASBA,5,3
ASBA,1,4
ARSYM,X,2

RECTNG,-wp-wm/2,-wm/2,-hp/2,hp/2                  !Piezos
RECTNG,wm/2,wp+wm/2,-hp/2,hp/2

RECTNG,-wm/2,wm/2,-hs/2,hs/2                      !Steel center
AGLUE,ALL
ALLSEL,ALL

ASEL,S,AREA,,7                                     !Meshing flexture body
AATT,1,1,1,0,
SMRT,3
AMESH,ALL

ASEL,S,AREA,,6
ASEL,A,AREA,,8                                     !Meshing PZT
AATT,2,3,1,0,
SMRT,1
AMESH,ALL

ASEL,S,AREA,,9                                     !Meshing steel center
AATT,1,1,1,0,
SMRT,3
AMESH,ALL

ASEL,S,AREA,,1,2                                   !Meshing steel push plates
AATT,1,1,1,0,
SMRT,6
AMESH,ALL

```



```

*GET,nodetiploc,NODE,,MXLOC,Y          !Name top node on frame tip
NSEL,S,LOC,Y,nodetiploc
NSEL,R,LOC,X,0
*GET,nodetip,NODE,,NUM,MIN

! /COM, CONTACT PAIR CREATION - START
CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
MP,MU,1,.2
MAT,1
R,5
REAL,5
ET,3,169
ET,4,172
KEYOPT,4,9,1
R,5,
RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
LSEL,S,,,2
LSEL,A,,,22
CM,_TARGET,LINE
TYPE,3
NSLL,S,1
ESLN,S,0
ESURF,ALL
CMSEL,S,_ELEMCM
! Generate the contact surface
LSEL,S,,,15
LSEL,A,,,16
CM,_CONTACT,LINE
TYPE,4
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,6
REAL,6
ET,5,169
ET,6,172
KEYOPT,6,9,1
R,6,
RMORE,
RMORE,,0
RMORE,0
TYPE,5
ESEL,S,TYPE,,4
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,6
ESEL,S,TYPE,,3

```

```

NSLE,S
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - End
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,3
ESEL,A,TYPE,,4
ESEL,R,REAL,,5
ESEL,ALL
ESEL,S,TYPE,,3
ESEL,A,TYPE,,4
ESEL,R,REAL,,5
ESEL,A,TYPE,,5
ESEL,A,TYPE,,6
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END

! /COM, CONTACT PAIR CREATION - START
CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
MP,MU,1,0.2
MAT,1
R,7
REAL,7
ET,7,169
ET,8,172
KEYOPT,8,9,1
R,7,
RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
LSEL,S,,5
LSEL,A,,24
CM,_TARGET,LINE
TYPE,7
NSLL,S,1
ESLN,S,0
ESURF,ALL

```

```

CMSEL,S,_ELEMCM
! Generate the contact surface
LSEL,S,,,19
LSEL,A,,,20
CM,_CONTACT,LINE
TYPE,8
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,8
REAL,8
ET,9,169
ET,10,172
KEYOPT,10,9,1
R,8,
RMORE,
RMORE,,0
RMORE,0
TYPE,9
ESEL,S,TYPE,,8
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,10
ESEL,S,TYPE,,7
NSLE,S
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - End
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,7
ESEL,A,TYPE,,8
ESEL,R,REAL,,7
ESEL,ALL
ESEL,S,TYPE,,7
ESEL,A,TYPE,,8
ESEL,R,REAL,,7
ESEL,A,TYPE,,9
ESEL,A,TYPE,,10
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END

```

```

FINISH

/SOL

LSEL,S,LINE,,25                !BCs on frame
LSEL,A,LINE,,29
LSEL,A,LINE,,46
NSLL,S,1
D,ALL,UX,0
D,ALL,UY,0
ALLSEL,ALL

LSEL,S,LINE,,10                !Preload PZTs
LSEL,A,LINE,,12
NSLL,S,1
NSEL,R,LOC,Y,-hs/8,hs/8
*GET,nodes_selected,NODE,,COUNT
F,ALL,FX,400/nodes_selected

LSEL,S,LINE,,4
LSEL,A,LINE,,8
NSLL,S,1
NSEL,R,LOC,Y,-hs/8,hs/8
*GET,nodes_selected,NODE,,COUNT
F,ALL,FX,-400/nodes_selected
ALLSEL,ALL

ANTYPE,0
NLGEOM,0
NSUBST,20,1000,20
AUTOTS,1
OUTRES,ERASE
OUTRES,ALL,LAST
PSTRES,ON
TIME,1
SOLVE
FINISH

/SOL
ANTYPE,2
MSAVE,0
MODOPT,LANB,10
EQSLV,SPAR
MXPAND,10,,0
LUMPM,0
PSTRES,ON
MODOPT,LANB,10,0,0,,OFF
SOLVE
FINISH

/POST1
/SHOW,JPEG
JPEG,QUAL,75,
JPEG,ORIENT,HORIZ
JPEG,COLOR,2
JPEG,TMOD,1
/GFILE,800,

```

```

SET,FIRST
PLDISP,2      !1st mode output pic
SET,NEXT
PLDISP,2      !2nd
SET,NEXT
PLDISP,2      !3rd
!SET,NEXT
!PLDISP,2     !4th
!SET,NEXT
!PLDISP,2     !5th
!SET,NEXT
!PLDISP,2     !6th
!SET,NEXT
!PLDISP,2     !7th
!SET,NEXT
!PLDISP,2     !8th
/SHOW,CLOSE

```

F_harm_24.txt

```

/FILNAME,Harmonic
/TITLE,FPA 2-D Freq Response

FStart = 28000      !Frequency Start Range
FreqRange = 30000   !Frequency Stop Range
points = 100

/PREP7
ET,1,PLANE82        !Element type

KEYOPT,1,3,3
KEYOPT,1,5,0        !extra element output
KEYOPT,1,6,0        !extra surface output
R,1,.008,           !thickness frame
R,3,.005,           !thickness pzt

MP,EX,1,203e9       !Tool Steel
MP,PRXY,1,.3
MP,DENS,1,7820
MP,DAMP,1,1e-8

MP,EX,2,44e9        !Piezo
MP,PRXY,2,.34
MP,DENS,2,7700
MP,DAMP,2,1e-8
MP,CTEX,2,.0006

wp = .004           !Width piezo
hp = .005           !Height piezo
hs = .006           !Height side
ws = .003           !width sides
ll = .003           !Length Legs
wl = .0015          !Width legs

```

```

wm = .003           !width middle
wpp = .0005         !width push plate
rpp = .0125         !diameter push plate
olpp = .0003        !overlap push plate
tarc = .0013        !thickness mid arc
harc = .0028        !height mid arc

lpos= .0015         !leg position from outside
loff= .0015         !leg offset

pradin = harc-tarc/2  !Position radius inner
pradout = harc+tarc/2 !Position radius outer

RECTNG, -wp-wm/2-ws-wpp+olpp, -wp-wm/2-wpp+olpp, 0, hs/2  !sides
KL,3,0.5,38
KL,4,2/3,39
A,4,38,39
ASBA,1,2
ARSYM,Y,3
AADD,ALL

CYL4,rpp-wp-wm/2-wpp,0,rpp           !frame sockets
ASBA,2,1
ARSYM,X,3

K,40,0,hs/2+pradin                   !Top arc
K,41,0,hs/2+pradout
LARC,3,1,40
LARC,5,38,41
AL,2,21,9,22

legpt = -wp-wm/2-ws-wpp+olpp+lpos    !Legs
K,50,legpt,-hs/2
K,51,legpt+wl,-hs/2
K,52,legpt+loff,-hs/2-11
K,53,legpt+wl+loff,-hs/2-11
A,50,51,53,52
ARSYM,X,4

CYL4,0,hs/2+pradout-.0001,.0004     !Drive tip
AADD,ALL

ASEL,NONE                             !End caps
CYL4,rpp-wp-wm/2-wpp,0,rpp
RECTNG,-wp-wm/2,rpp*2,-rpp*2,rpp*2
RECTNG,-wp*2,0,hp/2,hp
RECTNG,-wp*2,0,-hp/2,-hp
ASBA,1,2
ASBA,5,3
ASBA,1,4
ARSYM,X,2

RECTNG, -wp-wm/2, -wm/2, -hp/2, hp/2  !Piezo
RECTNG, wm/2, wp+wm/2, -hp/2, hp/2

RECTNG, -wm/2,wm/2,-hs/2,hs/2         !Steel

```

```

AGLUE,ALL
ALLSEL,ALL

ASEL,S,AREA,,7           !Meshing flexture body
AATT,1,1,1,0,
SMRT,3
AMESH,ALL

ASEL,S,AREA,,6
ASEL,A,AREA,,8           !Meshing PZT
AATT,2,3,1,0,
SMRT,1
AMESH,ALL

ASEL,S,AREA,,9           !Meshing steel center
AATT,1,1,1,0,
SMRT,3
AMESH,ALL

ASEL,S,AREA,,1,2         !Meshing steel push plates
AATT,1,1,1,0,
SMRT,6
AMESH,ALL

*GET,nodetiploc,NODE,,MXLOC,Y
NSEL,S,LOC,Y,nodetiploc
NSEL,R,LOC,X,0
*GET,nodetip,NODE,,NUM,MIN
ALLSEL,ALL

! /COM, CONTACT PAIR CREATION - START
CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
MP,MU,1,.2
MAT,1
R,5
REAL,5
ET,3,169
ET,4,172
KEYOPT,4,9,1
R,5,
RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
LSEL,S,,,2
LSEL,A,,,22
CM,_TARGET,LINE
TYPE,3
NSLL,S,1
ESLN,S,0
ESURF,ALL
CMSEL,S,_ELEMCM

```

```

! Generate the contact surface
LSEL,S,,,15
LSEL,A,,,16
CM,_CONTACT,LINE
TYPE,4
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,6
REAL,6
ET,5,169
ET,6,172
KEYOPT,6,9,1
R,6,
RMORE,
RMORE,,0
RMORE,0
TYPE,5
ESEL,S,TYPE,,4
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,6
ESEL,S,TYPE,,3
NSLE,S
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - End
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,3
ESEL,A,TYPE,,4
ESEL,R,REAL,,5
ESEL,ALL
ESEL,S,TYPE,,3
ESEL,A,TYPE,,4
ESEL,R,REAL,,5
ESEL,A,TYPE,,5
ESEL,A,TYPE,,6
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END

! /COM, CONTACT PAIR CREATION - START

```



```

CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
MP,MU,1,0.2
MAT,1
R,7
REAL,7
ET,7,169
ET,8,172
KEYOPT,8,9,1
R,7,
RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
LSEL,S,,,5
LSEL,A,,,24
CM,_TARGET,LINE
TYPE,7
NSLL,S,1
ESLN,S,0
ESURF,ALL
CMSEL,S,_ELEMCM
! Generate the contact surface
LSEL,S,,,19
LSEL,A,,,20
CM,_CONTACT,LINE
TYPE,8
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,8
REAL,8
ET,9,169
ET,10,172
KEYOPT,10,9,1
R,8,
RMORE,
RMORE,,0
RMORE,0
TYPE,9
ESEL,S,TYPE,,8
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,10
ESEL,S,TYPE,,7
NSLE,S
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - End
ALLSEL
ESEL,ALL

```

```

ESEL,S,TYPE,,7
ESEL,A,TYPE,,8
ESEL,R,REAL,,7
ESEL,ALL
ESEL,S,TYPE,,7
ESEL,A,TYPE,,8
ESEL,R,REAL,,7
ESEL,A,TYPE,,9
ESEL,A,TYPE,,10
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END
ALLSEL,ALL
FINISH

```

```

/SOL

```

```

LSEL,S,LINE,,25
LSEL,A,LINE,,29
LSEL,A,LINE,,46
NSLL,S,1
D,ALL,UX,0
D,ALL,UY,0
ALLSEL,ALL

```

```

LSEL,S,LINE,,10
LSEL,A,LINE,,12
NSLL,S,1
NSEL,R,LOC,Y,-hs/8,hs/8
*GET,nodes_selected,NODE,,COUNT
F,ALL,FX,400/nodes_selected

```

```

LSEL,S,LINE,,4
LSEL,A,LINE,,8
NSLL,S,1
NSEL,R,LOC,Y,-hs/8,hs/8
*GET,nodes_selected,NODE,,COUNT
F,ALL,FX,-400/nodes_selected
ALLSEL,ALL

```

```

ANTYPE,0
NLGEOM,0
NSUBST,20,1000,20
AUTOTS,1
OUTRES,ERASE

```

```

OUTRES,ALL, LAST
PSTRES,ON
TIME,1
SOLVE
FINISH

/SOL
ANTYPE,3
HROPT,FULL
HROUT,ON
PSTRES,ON
OUTRES,ERASE
KBC,1

FDELE,ALL,FX

ASEL,S,AREA,,6
NSLA,S
BF,ALL,TEMP,1
ALLSEL,ALL

HARFRQ,FStart,FreqRange,
NSUBST,points,
SOLVE
FINISH

/POST26
NUMVAR,210
NSOL,2,nodetip,U,X,UX_2
NSOL,3,nodetip,U,Y,UY_2
STORE,MERGE

PLCPLX,0      !amplitude
PLVAR,2,3
/GROPT,LOGY,ON
/SHOW,JPEG
JPEG,QUAL,75,
JPEG,ORIENT,HORIZ
JPEG,COLOR,2
JPEG,TMOD,1
/GFILE,800,
/REPLOT

PLCPLX,2      !phase angle
ATAN,4,2
ATAN,5,3
PROD,4,4,,,Phase_UX,,,180/3.1416
PROD,5,5,,,Phase_UY,,,180/3.1416
ABS,6,4
ABS,7,5
ADD,8,6,7,,,Phase_diff,,,1,-1
/GROPT,LOGY,OFF
PLVAR,4,5,8
/SHOW,CLOSE

```

Static Analysis Code

Three static analysis programs were written to help develop and understand the USM presented in this thesis. The first file, F_vert_stiff_24.txt, is used to obtain the vertical stiffness of the USM structure when a downward force is applied to the drive tip. The next file, F_UY_24.txt, is used to get the vertical drive tip displacement when one of the piezoelectric elements is fully expanded. The last file, F_int_chk_24.txt, is used for setting up and checking the interference of the drive tip with the slider prior to running the dynamic model F_slide_24.txt. F_int_chk_24.txt is also used to understand the static stress in the piezoelectric elements before further investigations with dynamic models. The use of these programs and their results are explained further in Section 3.3.2.

F_vert_stiff_24.txt

```
/FILENAME,vert_stiff !Change Post loading name too
/TITLE,2-D FPA stiff

Force = 50      !N

/PREP7
ET,1,PLANE82

KEYOPT,1,3,3
KEYOPT,1,5,0      !extra element output
KEYOPT,1,6,0      !extra surface output
R,1,.008,         !thickness frame
R,3,.005,         !thickness pzt

MP,EX,1,203e9      !Tool Steel
MP,PRXY,1,.3
MP,DENS,1,7820
MP,DAMP,1,1e-8

MP,EX,2,44e9       !Piezo
MP,PRXY,2,.34
MP,DENS,2,7700
MP,DAMP,2,1e-8
MP,CTEX,2,.0006

wp = .004          !Width piezo
hp = .005          !Height piezo
```

```

hs = .006           !Height side
ws = .003           !width sides
ll = .003           !Length Legs
wl = .0015          !Width legs
wm = .003           !width middle
wpp = .0005         !width push plate
rpp = .0125         !diameter push plate
olpp = .0003        !overlap push plate
tarc = .0013        !thickness mid arc
harc = .0028        !height mid arc

lpos= .0015          !leg position from outside
loff= .0015          !leg offset
pradin = harc-tarc/2 !Position radius inner
pradout = harc+tarc/2 !Position radius outer

RECTNG, -wp-wm/2-ws-wpp+olpp, -wp-wm/2-wpp+olpp, 0, hs/2 !Left side
KL,3,0.5,38
KL,4,2/3,39
A,4,38,39
ASBA,1,2
ARSYM,Y,3
AADD,ALL

CYL4,rpp-wp-wm/2-wpp,0,rpp
ASBA,2,1
ARSYM,X,3

K,40,0,hs/2+pradin !Top arc
K,41,0,hs/2+pradout
LARC,3,1,40
LARC,5,38,41
AL,2,21,9,22

legpt = -wp-wm/2-ws-wpp+olpp+lpos
K,50,legpt,-hs/2
K,51,legpt+wl,-hs/2
K,52,legpt+loff,-hs/2-ll
K,53,legpt+wl+loff,-hs/2-ll

A,50,51,53,52
ARSYM,X,4

CYL4,0,hs/2+pradout-.0001,.0004
AADD,ALL

ASEL,NONE
CYL4,rpp-wp-wm/2-wpp,0,rpp
RECTNG,-wp-wm/2,rpp*2,-rpp*2,rpp*2
RECTNG,-wp*2,0,hp/2,hp
RECTNG,-wp*2,0,-hp/2,-hp
ASBA,1,2
ASBA,5,3
ASBA,1,4
ARSYM,X,2

RECTNG, -wp-wm/2, -wm/2, -hp/2, hp/2 !Piezo

```

```

RECTNG, wm/2, wp+wm/2, -hp/2, hp/2

RECTNG, -wm/2,wm/2,-hs/2,hs/2 !Steel Center
AGLUE,ALL
ALLSEL,ALL

ASEL,S,AREA,,7 !Meshing flexture body
AATT,1,1,1,0,
SMRT,3
AMESH,ALL

ASEL,S,AREA,,6
ASEL,A,AREA,,8 !Meshing PZT
AATT,2,3,1,0,
SMRT,1
AMESH,ALL

ASEL,S,AREA,,9 !Meshing steel center
AATT,1,1,1,0,
SMRT,3
AMESH,ALL

ASEL,S,AREA,,1,2 !Meshing steel push plates
AATT,1,1,1,0,
SMRT,6
AMESH,ALL

*GET,nodetiploc,NODE,,MXLOC,Y
NSEL,S,LOC,Y,nodetiploc
NSEL,R,LOC,X,0
*GET,nodetip,NODE,,NUM,MIN

! /COM, CONTACT PAIR CREATION - START
CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
MP,MU,1,.2
MAT,1
R,5
REAL,5
ET,3,169
ET,4,172
KEYOPT,4,9,1
R,5,
RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
LSEL,S,,,2
LSEL,A,,,22
CM,_TARGET,LINE
TYPE,3
NSLL,S,1
ESLN,S,0

```

```

ESURF,ALL
CMSEL,S,_ELEMCM
! Generate the contact surface
LSEL,S,,,15
LSEL,A,,,16
CM,_CONTACT,LINE
TYPE,4
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,6
REAL,6
ET,5,169
ET,6,172
KEYOPT,6,9,1
R,6,
RMORE,
RMORE,,0
RMORE,0
TYPE,5
ESEL,S,TYPE,,4
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,6
ESEL,S,TYPE,,3
NSLE,S
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - End
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,3
ESEL,A,TYPE,,4
ESEL,R,REAL,,5
ESEL,ALL
ESEL,S,TYPE,,3
ESEL,A,TYPE,,4
ESEL,R,REAL,,5
ESEL,A,TYPE,,5
ESEL,A,TYPE,,6
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END

```

```

! /COM, CONTACT PAIR CREATION - START
CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
MP,MU,1,0.2
MAT,1
R,7
REAL,7
ET,7,169
ET,8,172
KEYOPT,8,9,1
R,7,
RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
LSEL,S,,5
LSEL,A,,24
CM,_TARGET,LINE
TYPE,7
NSLL,S,1
ESLN,S,0
ESURF,ALL
CMSEL,S,_ELEMCM
! Generate the contact surface
LSEL,S,,19
LSEL,A,,20
CM,_CONTACT,LINE
TYPE,8
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,8
REAL,8
ET,9,169
ET,10,172
KEYOPT,10,9,1
R,8,
RMORE,
RMORE,,0
RMORE,0
TYPE,9
ESEL,S,TYPE,,8
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,10
ESEL,S,TYPE,,7
NSLE,S
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - End

```



```

ALLSEL
ESEL,ALL
ESEL,S,TYPE,,7
ESEL,A,TYPE,,8
ESEL,R,REAL,,7
ESEL,ALL
ESEL,S,TYPE,,7
ESEL,A,TYPE,,8
ESEL,R,REAL,,7
ESEL,A,TYPE,,9
ESEL,A,TYPE,,10
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END
FINISH

/SOL

LSEL,S,LINE,,25
LSEL,A,LINE,,29
LSEL,A,LINE,,46
NSLL,S,1
D,ALL,UX,0
D,ALL,UY,0
ALLSEL,ALL

LSEL,S,LINE,,10
LSEL,A,LINE,,12
NSLL,S,1
NSEL,R,LOC,Y,-hs/8,hs/8
*GET,nodes_selected,NODE,,COUNT
F,ALL,FX,400/nodes_selected

LSEL,S,LINE,,4
LSEL,A,LINE,,8
NSLL,S,1
NSEL,R,LOC,Y,-hs/8,hs/8
*GET,nodes_selected,NODE,,COUNT
F,ALL,FX,-400/nodes_selected
ALLSEL,ALL

ANTYPE,0
NLGEOM,1
NSUBST,10,1000,10
AUTOTS,1

```

```

TIME,1
OUTRES,ERASE
OUTRES,ALL,LAST
LSWRITE,1

NSUBST,20,1000,20
AUTOTS,1
TIME,2
OUTRES,ALL,ALL
F,nodetip,FY,-Force
LSWRITE,2
LSSOLVE,1,2,1
FINISH

/POST26
NUMVAR,200
NSOL,2,nodetip,U,Y,UY
STORE,MERGE
PROD,2,2,,,,,,,,-1

FILLDATA,195,,,,1,0
ADD,3,195,1,,FY,,, -1,1
PROD,3,3,,,FY,,,Force
/axlab,x,UY (m)
/axlab,y,FORCE (N)
XVAR,2
PLVAR,3,
FINISH

/POST1
SET,FIRST
*GET,UY_A,NODE,nodetip,U,Y
SET,,,1,,2
*GET,UY_B,NODE,nodetip,U,Y
K_TOTAL = 50/(UY_A-UY_B)/1000000
FINISH

```

F_UY_24.txt

```

/FILNAME,UY !Change Post loading name too
/TITLE,2-D FPA UY

/PREP7
ET,1,PLANE82

KEYOPT,1,3,3
KEYOPT,1,5,0      !extra element output
KEYOPT,1,6,0      !extra surface output
R,1,.008,         !thickness frame
R,3,.005,         !thickness pzt

MP,EX,1,203e9     !Tool Steel
MP,PRXY,1,.3
MP,DENS,1,7820
MP,DAMP,1,1e-8

```

```

MP,EX,2,44e9      !Piezo
MP,PRXY,2,.34
MP,DENS,2,7700
MP,DAMP,2,1e-8
MP,CTEX,2,.0006

wp = .004          !Width piezo
hp = .005          !Height piezo
hs = .006          !Height side
ws = .003          !width sides
ll = .003          !Length Legs
wl = .0015         !Width legs
wm = .003          !width middle
wpp = .0005        !width push plate
rpp = .0125        !diameter push plate
olpp = .0003       !overlap push plate
tarc = .0013       !thickness mid arc
harc = .0028       !height mid arc

lpos= .0015        !leg position from outside
loff= .0015        !leg offset
pradin = harc-tarc/2 !Position radius inner
pradout = harc+tarc/2 !Position radius outer

RECTNG, -wp-wm/2-ws-wpp+olpp, -wp-wm/2-wpp+olpp, 0, hs/2 !Left side
KL,3,0.5,38
KL,4,2/3,39
A,4,38,39
ASBA,1,2
ARSYM,Y,3
AADD,ALL

CYL4,rpp-wp-wm/2-wpp,0,rpp
ASBA,2,1
ARSYM,X,3

K,40,0,hs/2+pradin !Top arc
K,41,0,hs/2+pradout
LARC,3,1,40
LARC,5,38,41
AL,2,21,9,22

legpt = -wp-wm/2-ws-wpp+olpp+lpos
K,50,legpt,-hs/2
K,51,legpt+wl,-hs/2
K,52,legpt+loff,-hs/2-ll
K,53,legpt+wl+loff,-hs/2-ll

A,50,51,53,52
ARSYM,X,4

CYL4,0,hs/2+pradout-.0001,.0004
AADD,ALL

ASEL,NONE
CYL4,rpp-wp-wm/2-wpp,0,rpp

```

```

RECTNG, -wp-wm/2, rpp*2, -rpp*2, rpp*2
RECTNG, -wp*2, 0, hp/2, hp
RECTNG, -wp*2, 0, -hp/2, -hp
ASBA, 1, 2
ASBA, 5, 3
ASBA, 1, 4
ARSYM, X, 2

RECTNG, -wp-wm/2, -wm/2, -hp/2, hp/2      !Piezo
RECTNG, wm/2, wp+wm/2, -hp/2, hp/2

RECTNG, -wm/2, wm/2, -hs/2, hs/2          !Steel Center

AGLUE, ALL
ALLSEL, ALL

ASEL, S, AREA, , 7      !Meshing flexture body
AATT, 1, 1, 1, 0,
SMRT, 3
AMESH, ALL

ASEL, S, AREA, , 6, 8   !Meshing PZT
AATT, 2, 3, 1, 0,
SMRT, 1
AMESH, ALL

ASEL, S, AREA, , 9      !Meshing steel center
AATT, 1, 1, 1, 0,
SMRT, OFF
AMESH, ALL

ASEL, S, AREA, , 1, 2   !Meshing steel push plates
AATT, 1, 1, 1, 0,
SMRT, 6
AMESH, ALL

*GET, nodetiploc, NODE, , MXLOC, Y
NSEL, S, LOC, Y, nodetiploc
NSEL, R, LOC, X, 0
*GET, nodetip, NODE, , NUM, MIN
ALLSEL, ALL

! /COM, CONTACT PAIR CREATION - START
CM, _NODECM, NODE
CM, _ELEMCM, ELEM
CM, _KPCM, KP
CM, _LINECM, LINE
CM, _AREACM, AREA
CM, _VOLUCM, VOLU
MP, MU, 1, .2
MAT, 1
R, 5
REAL, 5
ET, 3, 169
ET, 4, 172
KEYOPT, 4, 9, 1
R, 5,

```

```

RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
LSEL,S,,,2
LSEL,A,,,22
CM,_TARGET,LINE
TYPE,3
NSLL,S,1
ESLN,S,0
ESURF,ALL
CMSEL,S,_ELEMCM
! Generate the contact surface
LSEL,S,,,15
LSEL,A,,,16
CM,_CONTACT,LINE
TYPE,4
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,6
REAL,6
ET,5,169
ET,6,172
KEYOPT,6,9,1
R,6,
RMORE,
RMORE,,0
RMORE,0
TYPE,5
ESEL,S,TYPE,,4
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,6
ESEL,S,TYPE,,3
NSLE,S
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - End
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,3
ESEL,A,TYPE,,4
ESEL,R,REAL,,5
ESEL,ALL
ESEL,S,TYPE,,3
ESEL,A,TYPE,,4
ESEL,R,REAL,,5
ESEL,A,TYPE,,5
ESEL,A,TYPE,,6
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM

```

```

CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END

! /COM, CONTACT PAIR CREATION - START
CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
MP,MU,1,0.2
MAT,1
R,7
REAL,7
ET,7,169
ET,8,172
KEYOPT,8,9,1
R,7,
RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
LSEL,S,,,5
LSEL,A,,,26
CM,_TARGET,LINE
TYPE,7
NSLL,S,1
ESLN,S,0
ESURF,ALL
CMSEL,S,_ELEMCM
! Generate the contact surface
LSEL,S,,,19
LSEL,A,,,20
CM,_CONTACT,LINE
TYPE,8
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,8
REAL,8
ET,9,169
ET,10,172
KEYOPT,10,9,1
R,8,
RMORE,
RMORE,,0
RMORE,0
TYPE,9

```

```

ESEL,S,TYPE,,8
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,10
ESEL,S,TYPE,,7
NSLE,S
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - End
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,7
ESEL,A,TYPE,,8
ESEL,R,REAL,,7
ESEL,ALL
ESEL,S,TYPE,,7
ESEL,A,TYPE,,8
ESEL,R,REAL,,7
ESEL,A,TYPE,,9
ESEL,A,TYPE,,10
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END
FINISH

/SOL

LSEL,S,LINE,,25
LSEL,A,LINE,,29
LSEL,A,LINE,,48
NSLL,S,1
D,ALL,UX,0
D,ALL,UY,0
ALLSEL,ALL

offband = 0
LSEL,S,LINE,,10
LSEL,A,LINE,,12
NSLL,S
NSEL,R,LOC,Y,-hs/8+offband,hs/8+offband
*GET,f_nodemax,NODE,,NUM,MAX
*GET,f_nodemin,NODE,,NUM,MIN
F,ALL,FX,400/(f_nodemax-f_nodemin+1)

```

```

LSEL,S,LINE,,4
LSEL,A,LINE,,8
NSLL,S
NSEL,R,LOC,Y,-hs/8+offband,hs/8+offband
*GET,f_nodemax,NODE,,NUM,MAX
*GET,f_nodemin,NODE,,NUM,MIN
F,ALL,FX,-400/(f_nodemax-f_nodemin+1)
ALLSEL,ALL

```

```

ANTYPE,0
NLGEOM,1
NSUBST,30,1000,30
AUTOTS,1
TIME,1
OUTRES,ALL,LAST
LSWRITE,1

```

```

NSUBST,10,1000,10
AUTOTS,1
TIME,2
OUTRES,ALL,LAST
ASEL,S,AREA,,6
NSLA,S,1
BF,ALL,TEMP,1
ALLSEL,ALL
LSWRITE,2

```

```

LSSOLVE,1,2,1
FINISH

```

```

/POST26
NUMVAR,200
NSOL,2,nodetip,U,Y, UY
NSOL,3,nodetip,U,X, UX
STORE,MERGE
/axlab,x,LOADING
/axlab,y,U (m)
XVAR,1
PLVAR,2,3
FINISH

```

```

/POST1
SET,FIRST
*GET,UY_A,NODE,nodetip,U,Y
SET,NEXT
*GET,UY_B,NODE,nodetip,U,Y
UY_TOTAL = UY_A-UY_B
FINISH

```

F_int_chk_24.txt

```

/FILNAME,dyn_int_chk
/TITLE,2-D FPA Dynamic Int CHK

```

```

Interference = 1.7          !um

```



```

/PREP7
ET,1,PLANE82

KEYOPT,1,3,3
KEYOPT,1,5,0      !extra element output
KEYOPT,1,6,0      !extra surface output
R,1,.008,          !thickness snow
R,2,.008,          !thickness slider
R,3,.005,          !thickness pzt

MP,EX,1,203e9      !Tool Steel
MP,PRXY,1,.3
MP,DENS,1,7820
MP,DAMP,1,1e-8

MP,EX,2,44e9       !Piezo
MP,PRXY,2,.34
MP,DENS,2,7700
MP,DAMP,2,1e-8
MP,CTEX,2,.0006

MP,EX,3,303e9      !Al Oxide Ceramic
MP,PRXY,3,.21
MP,DENS,3,3720

wp = .004          !Width piezo
hp = .005          !Height piezo
hs = .006          !Height side
ws = .003          !width sides
ll = .003          !Length Legs
wl = .0015         !Width legs
wm = .003          !width middle
wpp = .0005        !width push plate
rpp = .0125        !diameter push plate
olpp = .0003       !overlap push plate
tarc = .0013       !thickness mid arc
harc = .0028       !height mid arc

lpos= .0015        !leg position from outside
loff= .0015        !leg offset
pradin = harc-tarc/2 !Position radius inner
pradout = harc+tarc/2 !Position radius outer

RECTNG, -wp-wm/2-ws-wpp+olpp, -wp-wm/2-wpp+olpp, 0, hs/2 !Left side
KL,3,0.5,38
KL,4,2/3,39
A,4,38,39
ASBA,1,2
ARSYM,Y,3
AADD,ALL

CYL4,rpp-wp-wm/2-wpp,0,rpp
ASBA,2,1
ARSYM,X,3

K,40,0,hs/2+pradin !Top arc

```

```

K,41,0,hs/2+pradout
LARC,3,1,40
LARC,5,38,41
AL,2,21,9,22

legpt = -wp-wm/2-ws-wpp+olpp+lpos
K,50,legpt,-hs/2
K,51,legpt+wl,-hs/2
K,52,legpt+loff,-hs/2-11
K,53,legpt+wl+loff,-hs/2-11

A,50,51,53,52
ARSYM,X,4

CYL4,0,hs/2+pradout-.0001,.0004
AADD,ALL

ASEL,NONE
CYL4,rpp-wp-wm/2-wpp,0,rpp
RECTNG,-wp-wm/2,rpp*2,-rpp*2,rpp*2
RECTNG,-wp*2,0,hp/2,hp
RECTNG,-wp*2,0,-hp/2,-hp
ASBA,1,2
ASBA,5,3
ASBA,1,4
ARSYM,X,2

RECTNG,-wp-wm/2,-wm/2,-hp/2,hp/2 !Piezo
RECTNG,wm/2,wp+wm/2,-hp/2,hp/2

RECTNG,-wm/2,wm/2,-hs/2,hs/2 !Steel
AGLUE,ALL
ALLSEL,ALL

RECTNG,-wp*2,wp+wm/2+ws,hs/2+pradout+.0003,hs/2+pradout+.0009

ASEL,S,AREA,,7 !Meshing flexture body
AATT,1,1,1,0,
SMRT,1!3
AMESH,ALL

ASEL,S,AREA,,6
ASEL,A,AREA,,8 !Meshing PZT
AATT,2,3,1,0,
SMRT,1
AMESH,ALL

ASEL,S,AREA,,9 !Meshing steel center
AATT,1,1,1,0,
SMRT,OFF
AMESH,ALL

ASEL,S,AREA,,1,2 !Meshing steel push plates
AATT,1,1,1,0,
SMRT,4!4
AMESH,ALL

```

```

*GET,nodetiploc,NODE,,MXLOC,Y
NSEL,S,LOC,Y,nodetiploc
NSEL,R,LOC,X,0
*GET,nodetip,NODE,,NUM,MIN
ALLSEL,ALL

! /COM, CONTACT PAIR CREATION - START
CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
MP,MU,1,.2
MAT,1
R,5
REAL,5
ET,3,169
ET,4,172
KEYOPT,4,9,1
R,5,
RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
LSEL,S,,2
LSEL,A,,22
CM,_TARGET,LINE
TYPE,3
NSLL,S,1
ESLN,S,0
ESURF,ALL
CMSEL,S,_ELEMCM
! Generate the contact surface
LSEL,S,,15
LSEL,A,,16
CM,_CONTACT,LINE
TYPE,4
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,6
REAL,6
ET,5,169
ET,6,172
KEYOPT,6,9,1
R,6,
RMORE,
RMORE,,0
RMORE,0
TYPE,5
ESEL,S,TYPE,,4
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,6

```

```

ESEL,S,TYPE,,3
NSLE,S
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - End
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,3
ESEL,A,TYPE,,4
ESEL,R,REAL,,5
ESEL,ALL
ESEL,S,TYPE,,3
ESEL,A,TYPE,,4
ESEL,R,REAL,,5
ESEL,A,TYPE,,5
ESEL,A,TYPE,,6
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END

! /COM, CONTACT PAIR CREATION - START
CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
MP,MU,1,0.2
MAT,1
R,7
REAL,7
ET,7,169
ET,8,172
KEYOPT,8,9,1
R,7,
RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
LSEL,S,,,5
LSEL,A,,,24
CM,_TARGET,LINE
TYPE,7
NSLL,S,1
ESLN,S,0

```

```

ESURF,ALL
CMSEL,S,_ELEMCM
! Generate the contact surface
LSEL,S,,,19
LSEL,A,,,20
CM,_CONTACT,LINE
TYPE,8
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,8
REAL,8
ET,9,169
ET,10,172
KEYOPT,10,9,1
R,8,
RMORE,
RMORE,,0
RMORE,0
TYPE,9
ESEL,S,TYPE,,8
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,10
ESEL,S,TYPE,,7
NSLE,S
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - End
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,7
ESEL,A,TYPE,,8
ESEL,R,REAL,,7
ESEL,ALL
ESEL,S,TYPE,,7
ESEL,A,TYPE,,8
ESEL,R,REAL,,7
ESEL,A,TYPE,,9
ESEL,A,TYPE,,10
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END

```

```

FINISH

/SOL

LSEL,S,LINE,,25
LSEL,A,LINE,,29
LSEL,A,LINE,,46
NSLL,S,1
D,ALL,UX,0
D,ALL,UY,0
ALLSEL,ALL

LSEL,S,LINE,,10
LSEL,A,LINE,,12
NSLL,S,1
NSEL,R,LOC,Y,-hs/8,hs/8
*GET,nodes_selected,NODE,,COUNT
F,ALL,FX,400/nodes_selected

LSEL,S,LINE,,4
LSEL,A,LINE,,8
NSLL,S,1
NSEL,R,LOC,Y,-hs/8,hs/8
*GET,nodes_selected,NODE,,COUNT
F,ALL,FX,-400/nodes_selected
ALLSEL,ALL

ANTYPE,0
NLGEOM,ON
NSUBST,30,1000,30
AUTOTS,1
OUTRES,ALL,LAST
TIME,0.5
SOLVE
FINISH

/POST1

*GET,NODETIP_UY,NODE,NODETIP,U,Y
FINISH

/PREP7

ASEL,S,AREA,,3           !Meshing slider
AATT,3,2,1,0,
SMRT,3
AMESH,ALL
ALLSEL,ALL

LSEL,S,LINE,,33
NSLL,S
D,ALL,UY,0.000010
D,ALL,UX,0
ALLSEL,ALL

! /COM, CONTACT PAIR CREATION - START
CM,_NODECM,NODE

```

```

CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
MP,MU,1,0.15
MAT,1
MP,EMIS,1,7.88860905221e-031
R,9
REAL,9
ET,11,169
ET,12,172
R,9,,,1.0,0.1,0,
RMORE,,,1.0E20,0.0,1.0,
RMORE,0.0,0,1.0,,1.0,0.5
RMORE,0,1.0,1.0,0.0,,1.0
KEYOPT,12,3,0
KEYOPT,12,4,0
KEYOPT,12,5,0
KEYOPT,12,7,0
KEYOPT,12,8,0
KEYOPT,12,9,1
KEYOPT,12,10,1
KEYOPT,12,11,0
KEYOPT,12,12,0
KEYOPT,12,2,0
! Generate the target surface
LSEL,S,,,40
LSEL,A,,,41
CM,_TARGET,LINE
TYPE,11
NSLL,S,1
ESLN,S,0
ESURF,ALL
CMSEL,S,_ELEMCM
! Generate the contact surface
LSEL,S,,,30
CM,_CONTACT,LINE
TYPE,12
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,10
REAL,10
ET,13,169
ET,14,172
R,10,,,1.0,0.1,0,
RMORE,,,1.0E20,0.0,1.0,
RMORE,0.0,0,1.0,,1.0,0.5
RMORE,0,1.0,1.0,0.0,,1.0
KEYOPT,14,3,0
KEYOPT,14,4,0
KEYOPT,14,5,0
KEYOPT,14,7,0
KEYOPT,14,8,0
KEYOPT,14,9,1

```

```

KEYOPT,14,10,1
KEYOPT,14,11,0
KEYOPT,14,12,0
KEYOPT,14,2,0
KEYOPT,13,3,0
TYPE,13
ESEL,S,TYPE,,12
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,14
ESEL,S,TYPE,,11
NSLE,S
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - End
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,11
ESEL,A,TYPE,,12
ESEL,R,REAL,,9
ESEL,ALL
ESEL,S,TYPE,,11
ESEL,A,TYPE,,12
ESEL,R,REAL,,9
ESEL,A,TYPE,,13
ESEL,A,TYPE,,14
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
! /GRES,cwz,gsav
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END
FINISH

/SOL
ANTYPE,0
NLGEOM,ON
NSUBST,30,1000,30
AUTOTS,1
OUTRES,ALL,LAST
TIME,1
LSWRITE,1

TIP_Clearance = 0.453
MOVE = (-TIP_Clearance-Interference)/1000000+NODETIP_UY
LSEL,S,LINE,,31

```



```

NSLL,S
D,ALL,UY,MOVE
ALLSEL,ALL

NSUBST,20,1000,20
AUTOTS,1
OUTRES,ALL,LAST
TIME,2
LSWRITE,2

ASEL,S,AREA,,8
NSLA,S,1
BF,ALL,TEMP,1
ALLSEL,ALL
NSUBST,20,1000,20
AUTOTS,1
OUTRES,ALL,LAST
TIME,3
LSWRITE,3

LSSOLVE,1,2,1
FINISH

/POST26
NUMVAR,200                                !Number of possible data vars
NSOL,2,NODETIP,U,Y, UY_D                 !Y displacement Tip
STORE,MERGE                               !Read results
PLVAR,2
/axlab,x,TIME (s)
/axlab,y,UY (m)
/replot
FINISH

```

Transient Analysis Code

The Transient analysis code is an essential part to of the USM development process. The first analysis program, F_dyn_24.txt, is used to trace the drive tip motion running in free space. The second file, F_slide_24.txt, models the USM driving a slider and is used to approximate motor performance. F_slide_24.txt is capable of running loops, which can vary the preload interference and the opposing push force. F_slide_24.txt is also used to analyze the stress in the USM structure and piezoelectric elements. When stress or other analysis information is wanted, F_slide_24.txt should be run with a single interference and push force. The program erases all analysis data except

for the drive tip's path and the slider movement, so the '/clear' command also needs to be commented out with a '!' near the end of the code.

The transient codes also use a few subroutines to output graphs and data with specified titles. The subroutine, filenamecreate.tcl, in F_dyn_24.txt gives ANSYS an output name that consists of the driving frequency and the activated piezoelectric element's area number. The subroutine filenameSL.tcl in F_slide_24.txt gives ANSYS an output name consisting of the driving frequency, interference, and the file number. Similarly, the filenameSLF.tcl returns a filename consisting of the driving frequency, interference, push force, and the file number. Further explanations of the use of the transient programs are explained in Section 3.3.3.

F_dyn_24.txt

```

/FILNAME,dyn
/TITLE,2-D FPA Dynamic Simulation

FREQ = 28900

LOOPS = 15
OFFSET = 6                !Driving PZT

/PREP7
ET,1,PLANE82

KEYOPT,1,3,3
KEYOPT,1,5,0              !extra element output
KEYOPT,1,6,0              !extra surface output
R,1,.008,                  !thickness frame
R,3,.005,                  !thickness pzt

MP,EX,1,203e9              !Tool Steel
MP,PRXY,1,.3
MP,DENS,1,7820
MP,DAMP,1,1e-8

MP,EX,2,44e9              !Piezo
MP,PRXY,2,.34
MP,DENS,2,7700
MP,DAMP,2,1e-8
MP,CTEX,2,.0006

wp = .004                  !Width piezo
hp = .005                  !Height piezo

```

```

hs = .006           !Height side
ws = .003           !width sides
ll = .003           !Length Legs
wl = .0015          !Width legs
wm = .003           !width middle
wpp = .0005          !width push plate
rpp = .0125          !diameter push plate
olpp = .0003         !overlap push plate
tarc = .0013         !thickness mid arc
harc = .0028         !height mid arc

lpos= .0015          !leg position from outside
loff= .0015          !leg offset
pradin = harc-tarc/2  !Position radius inner
pradout = harc+tarc/2 !Position radius outer

RECTNG, -wp-wm/2-ws-wpp+olpp, -wp-wm/2-wpp+olpp, 0, hs/2 !Left side
KL,3,0.5,38
KL,4,2/3,39
A,4,38,39
ASBA,1,2
ARSYM,Y,3
AADD,ALL

CYL4,rpp-wp-wm/2-wpp,0,rpp
ASBA,2,1
ARSYM,X,3

K,40,0,hs/2+pradin !Top arc
K,41,0,hs/2+pradout
LARC,3,1,40
LARC,5,38,41
AL,2,21,9,22

legpt = -wp-wm/2-ws-wpp+olpp+lpos
K,50,legpt,-hs/2
K,51,legpt+wl,-hs/2
K,52,legpt+loff,-hs/2-ll
K,53,legpt+wl+loff,-hs/2-ll

A,50,51,53,52
ARSYM,X,4

CYL4,0,hs/2+pradout-.0001,.0004
AADD,ALL

ASEL,NONE
CYL4,rpp-wp-wm/2-wpp,0,rpp
RECTNG,-wp-wm/2,rpp*2,-rpp*2,rpp*2
RECTNG,-wp*2,0,hp/2,hp
RECTNG,-wp*2,0,-hp/2,-hp
ASBA,1,2
ASBA,5,3
ASBA,1,4
ARSYM,X,2

RECTNG, -wp-wm/2, -wm/2, -hp/2, hp/2 !Piezo

```

```

RECTNG, wm/2, wp+wm/2, -hp/2, hp/2

RECTNG, -wm/2,wm/2,-hs/2,hs/2 !Steel
AGLUE,ALL
ALLSEL,ALL

ASEL,S,AREA,,7 !Meshing flexture body
AATT,1,1,1,0,
SMRT,1!3
AMESH,ALL

ASEL,S,AREA,,6
ASEL,A,AREA,,8 !Meshing PZT
AATT,2,3,1,0,
SMRT,1
AMESH,ALL

ASEL,S,AREA,,9 !Meshing steel center
AATT,1,1,1,0,
SMRT,OFF
AMESH,ALL

ASEL,S,AREA,,1,2 !Meshing steel push plates
AATT,1,1,1,0,
SMRT,4
AMESH,ALL

*GET,nodetiploc,NODE,,MXLOC,Y
NSEL,S,LOC,Y,nodetiploc
NSEL,R,LOC,X,0
*GET,nodetip,NODE,,NUM,MIN
ALLSEL,ALL

! /COM, CONTACT PAIR CREATION - START
CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
MP,MU,1,.2
MAT,1
R,5
REAL,5
ET,3,169
ET,4,172
KEYOPT,4,9,1
R,5,
RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
LSEL,S,,2
LSEL,A,,22
CM,_TARGET,LINE
TYPE,3
NSLL,S,1

```

```

ESLN,S,0
ESURF,ALL
CMSEL,S,_ELEMCM
! Generate the contact surface
LSEL,S,,,15
LSEL,A,,,16
CM,_CONTACT,LINE
TYPE,4
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,6
REAL,6
ET,5,169
ET,6,172
KEYOPT,6,9,1
R,6,
RMORE,
RMORE,,0
RMORE,0
TYPE,5
ESEL,S,TYPE,,4
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,6
ESEL,S,TYPE,,3
NSLE,S
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - End
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,3
ESEL,A,TYPE,,4
ESEL,R,REAL,,5
ESEL,ALL
ESEL,S,TYPE,,3
ESEL,A,TYPE,,4
ESEL,R,REAL,,5
ESEL,A,TYPE,,5
ESEL,A,TYPE,,6
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
CMDEL,_TARGET
CMDEL,_CONTACT

```

```

! /COM, CONTACT PAIR CREATION - END

! /COM, CONTACT PAIR CREATION - START
CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
MP,MU,1,0.2
MAT,1
R,7
REAL,7
ET,7,169
ET,8,172
KEYOPT,8,9,1
R,7,
RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
LSEL,S,,,5
LSEL,A,,,24
CM,_TARGET,LINE
TYPE,7
NSLL,S,1
ESLN,S,0
ESURF,ALL
CMSEL,S,_ELEMCM
! Generate the contact surface
LSEL,S,,,19
LSEL,A,,,20
CM,_CONTACT,LINE
TYPE,8
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,8
REAL,8
ET,9,169
ET,10,172
KEYOPT,10,9,1
R,8,
RMORE,
RMORE,,0
RMORE,0
TYPE,9
ESEL,S,TYPE,,8
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,10
ESEL,S,TYPE,,7
NSLE,S
ESLN,S,0
ESURF,ALL

```

```

!* Create Companion Pair - End
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,7
ESEL,A,TYPE,,8
ESEL,R,REAL,,7
ESEL,ALL
ESEL,S,TYPE,,7
ESEL,A,TYPE,,8
ESEL,R,REAL,,7
ESEL,A,TYPE,,9
ESEL,A,TYPE,,10
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END
ALLSEL,ALL
FINISH

/SOL

LSEL,S,LINE,,25
LSEL,A,LINE,,29
LSEL,A,LINE,,46
NSLL,S,1
D,ALL,UX,0
D,ALL,UY,0
ALLSEL,ALL

LSEL,S,LINE,,10
LSEL,A,LINE,,12
NSLL,S,1
NSEL,R,LOC,Y,-hs/8,hs/8
*GET,nodes_selected,NODE,,COUNT
F,ALL,FX,400/nodes_selected

LSEL,S,LINE,,4
LSEL,A,LINE,,8
NSLL,S,1
NSEL,R,LOC,Y,-hs/8,hs/8
*GET,nodes_selected,NODE,,COUNT
F,ALL,FX,-400/nodes_selected
ALLSEL,ALL

*DEL,_FNCNAME      !Add Active Piezo
*DEL,_FNCMTID

```

```

*DEL,_FNC_C1
*SET,_FNCNAME,'ELEMEN'
*DIM,_FNC_C1,,1
*SET,_FNC_C1(1),FREQ
*DIM,%_FNCNAME%,TABLE,6,11,1
! Begin of equation: 0.5+(sin(2*{PI})*{TIME}*FREQ)/2)
*SET,%_FNCNAME%(0,0,1), 0.0, -999
*SET,%_FNCNAME%(2,0,1), 0.0
*SET,%_FNCNAME%(3,0,1), %_FNC_C1(1)%
*SET,%_FNCNAME%(4,0,1), 0.0
*SET,%_FNCNAME%(5,0,1), 0.0
*SET,%_FNCNAME%(6,0,1), 0.0
*SET,%_FNCNAME%(0,1,1), 1.0, -1, 0, 2, 0, 0, 0
*SET,%_FNCNAME%(0,2,1), 0.0, -2, 0, 3.14159265358979310, 0, 0, -1
*SET,%_FNCNAME%(0,3,1), 0, -3, 0, 1, -1, 3, -2
*SET,%_FNCNAME%(0,4,1), 0.0, -1, 0, 1, -3, 3, 1
*SET,%_FNCNAME%(0,5,1), 0.0, -2, 0, 1, -1, 3, 17
*SET,%_FNCNAME%(0,6,1), 0.0, -1, 9, 1, -2, 0, 0
*SET,%_FNCNAME%(0,7,1), 0.0, -2, 0, 2, 0, 0, -1
*SET,%_FNCNAME%(0,8,1), 0.0, -3, 0, 1, -1, 4, -2
*SET,%_FNCNAME%(0,9,1), 0.0, -1, 0, 0.5, 0, 0, -3
*SET,%_FNCNAME%(0,10,1), 0.0, -2, 0, 1, -1, 1, -3
*SET,%_FNCNAME%(0,11,1), 0.0, 99, 0, 1, -2, 0, 0
! End of equation: 0.5+(sin(2*{PI})*{TIME}*FREQ)/2)

ASEL,S,AREA,,OFFSET
NSLA,S,1
BF,ALL,TEMP,%ELEMEN%
ALLSEL,ALL

ANTYPE,4
TRNOPT,FULL
KBC,0
TIMINT,0
LUMPM,0
NLGEOM,1
DELTIM,1/(FREQ*40),0,0
OUTRES,ERASE
OUTRES,ALL,1
AUTOTS,0
TIME,(1/FREQ)
LSWRITE,1,

KBC,1
TIMINT,1
TIME,(1/FREQ)*LOOPS
AUTOTS,1
OUTRES,ALL,1
LSWRITE,2,
LSSOLVE,1,2,1
FINISH

/POST26
NUMVAR,200
NSOL,2,NODETIP,U,X, UX_D
NSOL,3,NODETIP,U,Y, UY_D
NSOL,4,5,U,X, UX2
!Number of possible data vars
!X displacement Left Side
!Y displacement Left Side
!PZT displacement

```



```
STORE,MERGE                                !Read results
```

```
XVAR,2
PLVAR,3
/axlab,x,UX
/axlab,y,UY
/replot
FILEOUT = 1
~eui,'source [file join filenamecreate.tcl]'
/IMAGE,SAVE,filename,jpg

XVAR,1
PLVAR,2,3,4
/axlab,x,Time
/axlab,y,Displacement
/replot
FILEOUT = 2
~eui,'source [file join filenamecreate.tcl]'
/IMAGE,SAVE,filename,jpg
FINISH
```

F_slide_24.txt

```
/FILENAME,dyn_slide
/TITLE,2-D FPA Dynamic Simulation

!*****LOOP INPUT*****

*DO,I,1.75,1.75,          !Iterference
*DO,J,0,0,                !FORCE
PARSAV,SCALAR,loopcounter

Interference = I  !um
FREQ = 30100
LOOPS = 10
OFFSET = 6          !drive area

NODETIP_UY = 2.598653489E-06
TIP_Clearance = 0.453
MOVE = (TIP_Clearance+Interference)/1000000-NODETIP_UY
FORCE = J

/PREP7
ET,1,PLANE82
ET,2,COMBIN14

KEYOPT,1,3,3
KEYOPT,1,5,0      !extra element output
KEYOPT,1,6,0      !extra surface output
R,1,.008,         !thickness frame
R,2,.008,         !thickness slider
R,3,.005,         !thickness pzt

KEYOPT,2,1,0
KEYOPT,2,2,1      !1D UX DOF
R,4,0,0,0,        !K,CV1,CV2 (slide damping)
```

```

MP,EX,1,203e9      !Tool Steel
MP,PRXY,1,.3
MP,DENS,1,7820
MP,DAMP,1,1e-8

MP,EX,2,44e9       !Piezo
MP,PRXY,2,.34
MP,DENS,2,7700
MP,DAMP,2,1e-8
MP,CTEX,2,.0006

MP,EX,3,303e9      !Al Oxide Ceramic
MP,PRXY,3,.21
MP,DENS,3,3720

wp = .004          !Width piezo
hp = .005          !Height piezo
hs = .006          !Height side
ws = .003          !width sides
ll = .003          !Length Legs
wl = .0015         !Width legs
wm = .003          !width middle
wpp = .0005        !width push plate
rpp = .0125        !diameter push plate
olpp = .0003       !overlap push plate
tarc = .0013       !thickness mid arc
harc = .0028       !height mid arc

lpos= .0015        !leg position from outside
loff= .0015        !leg offset
pradin = harc-tarc/2 !Position radius inner
pradout = harc+tarc/2 !Position radius outer

RECTNG, -wp-wm/2-ws-wpp+olpp, -wp-wm/2-wpp+olpp, 0, hs/2 !Left side
KL,3,0.5,38
KL,4,2/3,39
A,4,38,39
ASBA,1,2
ARSYM,Y,3
AADD,ALL

CYL4,rpp-wp-wm/2-wpp,0,rpp
ASBA,2,1
ARSYM,X,3

K,40,0,hs/2+pradin !Top arc
K,41,0,hs/2+pradout
LARC,3,1,40
LARC,5,38,41
AL,2,21,9,22

legpt = -wp-wm/2-ws-wpp+olpp+lpos
K,50,legpt,-hs/2
K,51,legpt+wl,-hs/2
K,52,legpt+loff,-hs/2-ll
K,53,legpt+wl+loff,-hs/2-ll

```

```

A,50,51,53,52
ARSYM,X,4

CYL4,0,hs/2+pradout-.0001,.0004
AADD,ALL

ASEL,NONE
CYL4,rpp-wp-wm/2-wpp,0,rpp
RECTNG,-wp-wm/2,rpp*2,-rpp*2,rpp*2
RECTNG,-wp*2,0,hp/2,hp
RECTNG,-wp*2,0,-hp/2,-hp
ASBA,1,2
ASBA,5,3
ASBA,1,4
ARSYM,X,2

RECTNG,-wp-wm/2,-wm/2,-hp/2,hp/2          !Piezo
RECTNG,wm/2,wp+wm/2,-hp/2,hp/2

RECTNG,-wm/2,wm/2,-hs/2,hs/2              !Steel
AGLUE,ALL
ALLSEL,ALL

RECTNG,-wp*2,wp+wm/2+ws,hs/2+pradout+.0003,hs/2+pradout+.0009

ASEL,S,AREA,,7          !Meshing flexture body
AATT,1,1,1,0,
SMRT,1!3
AMESH,ALL

ASEL,S,AREA,,6
ASEL,A,AREA,,8          !Meshing PZT
AATT,2,3,1,0,
SMRT,1
AMESH,ALL

ASEL,S,AREA,,9          !Meshing steel center
AATT,1,1,1,0,
SMRT,OFF
AMESH,ALL

ASEL,S,AREA,,1,2        !Meshing steel push plates
AATT,1,1,1,0,
SMRT,4!4
AMESH,ALL

*GET,nodetiploc,NODE,,MXLOC,Y
NSEL,S,LOC,Y,nodetiploc
NSEL,R,LOC,X,0
*GET,nodetip,NODE,,NUM,MIN
ALLSEL,ALL

ASEL,S,AREA,,3          !Meshing slider
AATT,3,2,1,0,
SMRT,3

```

```

AMESH,ALL
ALLSEL,ALL

NSEL,NONE !Damping
N,,-wp*2,hs/2+pradout+.0009
D,ALL,UX,0
TYPE,2
REAL,4
NSEL,A,LOC,X,-wp*2
NSEL,R,LOC,Y,hs/2+pradout+.0009
*GET,nodedamp1,NODE,,NUM,MAX
*GET,nodedamp2,NODE,,NUM,MIN
E,nodedamp1,nodedamp2

! /COM, CONTACT PAIR CREATION - START
CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
MP,MU,1,.2
MAT,1
R,5
REAL,5
ET,3,169
ET,4,172
KEYOPT,4,9,1
R,5,
RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
LSEL,S,,,2
LSEL,A,,,22
CM,_TARGET,LINE
TYPE,3
NSLL,S,1
ESLN,S,0
ESURF,ALL
CMSEL,S,_ELEMCM
! Generate the contact surface
LSEL,S,,,15
LSEL,A,,,16
CM,_CONTACT,LINE
TYPE,4
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,6
REAL,6
ET,5,169
ET,6,172
KEYOPT,6,9,1
R,6,
RMORE,

```

```

RMORE,,0
RMORE,0
TYPE,5
ESEL,S,TYPE,,4
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,6
ESEL,S,TYPE,,3
NSLE,S
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - End
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,3
ESEL,A,TYPE,,4
ESEL,R,REAL,,5
ESEL,ALL
ESEL,S,TYPE,,3
ESEL,A,TYPE,,4
ESEL,R,REAL,,5
ESEL,A,TYPE,,5
ESEL,A,TYPE,,6
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END

! /COM, CONTACT PAIR CREATION - START
CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
MP,MU,1,0.2
MAT,1
R,7
REAL,7
ET,7,169
ET,8,172
KEYOPT,8,9,1
R,7,
RMORE,
RMORE,,0

```

```

RMORE,0
! Generate the target surface
LSEL,S,,,5
LSEL,A,,,24
CM,_TARGET,LINE
TYPE,7
NSLL,S,1
ESLN,S,0
ESURF,ALL
CMSEL,S,_ELEMCM
! Generate the contact surface
LSEL,S,,,19
LSEL,A,,,20
CM,_CONTACT,LINE
TYPE,8
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,8
REAL,8
ET,9,169
ET,10,172
KEYOPT,10,9,1
R,8,
RMORE,
RMORE,,0
RMORE,0
TYPE,9
ESEL,S,TYPE,,8
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,10
ESEL,S,TYPE,,7
NSLE,S
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - End
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,7
ESEL,A,TYPE,,8
ESEL,R,REAL,,7
ESEL,ALL
ESEL,S,TYPE,,7
ESEL,A,TYPE,,8
ESEL,R,REAL,,7
ESEL,A,TYPE,,9
ESEL,A,TYPE,,10
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM

```

```

CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END

! /COM, CONTACT PAIR CREATION - START
CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
MP,MU,1,0.15
MAT,1
MP,EMIS,1,7.88860905221e-031
R,9
REAL,9
ET,11,169
ET,12,172
R,9,,,1.0,0.1,0,
RMORE,,,1.0E20,0.0,1.0,
RMORE,0.0,0,1.0,,1.0,0.5
RMORE,0,1.0,1.0,0.0,,1.0
KEYOPT,12,3,0
KEYOPT,12,4,0
KEYOPT,12,5,0
KEYOPT,12,7,0
KEYOPT,12,8,0
KEYOPT,12,9,1
KEYOPT,12,10,1
KEYOPT,12,11,0
KEYOPT,12,12,0
KEYOPT,12,2,0
! Generate the target surface
LSEL,S,,,40
LSEL,A,,,41
CM,_TARGET,LINE
TYPE,11
NSLL,S,1
ESLN,S,0
ESURF,ALL
CMSEL,S,_ELEMCM
! Generate the contact surface
LSEL,S,,,30
CM,_CONTACT,LINE
TYPE,12
NSLL,S,1
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - Start
R,10
REAL,10
ET,13,169

```

```

ET,14,172
R,10,,,1.0,0.1,0,
RMORE,,,1.0E20,0.0,1.0,
RMORE,0.0,0,1.0,,1.0,0.5
RMORE,0,1.0,1.0,0.0,,1.0
KEYOPT,14,3,0
KEYOPT,14,4,0
KEYOPT,14,5,0
KEYOPT,14,7,0
KEYOPT,14,8,0
KEYOPT,14,9,1
KEYOPT,14,10,1
KEYOPT,14,11,0
KEYOPT,14,12,0
KEYOPT,14,2,0
KEYOPT,13,3,0
TYPE,13
ESEL,S,TYPE,,12
NSLE,S
ESLN,S,0
ESURF,ALL
TYPE,14
ESEL,S,TYPE,,11
NSLE,S
ESLN,S,0
ESURF,ALL
!* Create Companion Pair - End
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,11
ESEL,A,TYPE,,12
ESEL,R,REAL,,9
ESEL,ALL
ESEL,S,TYPE,,11
ESEL,A,TYPE,,12
ESEL,R,REAL,,9
ESEL,A,TYPE,,13
ESEL,A,TYPE,,14
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
! /GRES,cwz,gsav
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END
FINISH

/SOL

```



```

LSEL,S,LINE,,25
LSEL,A,LINE,,29
LSEL,A,LINE,,46
NSLL,S,1
D,ALL,UX,0
D,ALL,UY,0
ALLSEL,ALL

D,nodedamp2,UX,0

LSEL,S,LINE,,10
LSEL,A,LINE,,12
NSLL,S,1
NSEL,R,LOC,Y,-hs/8,hs/8
*GET,nodes_selected,NODE,,COUNT
F,ALL,FX,400/nodes_selected

LSEL,S,LINE,,4
LSEL,A,LINE,,8
NSLL,S,1
NSEL,R,LOC,Y,-hs/8,hs/8
*GET,nodes_selected,NODE,,COUNT
F,ALL,FX,-400/nodes_selected
ALLSEL,ALL

ANTYPE,4
TRNOPT,FULL
NLGEOM,0
OUTRES,ERASE
OUTRES,ALL,NONE
KBC,0
TIME,0.5
NSUBST,10,200,10
TIMINT,0
LSWRITE,1

LSEL,S,LINE,,33
NSLL,S
D,ALL,UY,-MOVE
ALLSEL,ALL

NLGEOM,0
OUTRES,ERASE
OUTRES,ALL,NONE
KBC,0
TIME,1
NSUBST,30,200,30
TIMINT,0
LSWRITE,2

DDELE,nodedamp2,UX
F,nodedamp2,FX,-FORCE

*DEL,_FNCNAME      !Add Active Piezo
*DEL,_FNCMTID
*DEL,_FNC_C1

```

```

*SET,_FNCNAME,'ELEMEN'
*DIM,_FNC_C1,,1
*SET,_FNC_C1(1),FREQ
*DIM,%_FNCNAME%,TABLE,6,11,1
! Begin of equation: 0.5+(sin(2*{PI}*{TIME}*FREQ)/2)
*SET,%_FNCNAME%(0,0,1), 0.0, -999
*SET,%_FNCNAME%(2,0,1), 0.0
*SET,%_FNCNAME%(3,0,1), %_FNC_C1(1)%
*SET,%_FNCNAME%(4,0,1), 0.0
*SET,%_FNCNAME%(5,0,1), 0.0
*SET,%_FNCNAME%(6,0,1), 0.0
*SET,%_FNCNAME%(0,1,1), 1.0, -1, 0, 2, 0, 0, 0
*SET,%_FNCNAME%(0,2,1), 0.0, -2, 0, 3.14159265358979310, 0, 0, -1
*SET,%_FNCNAME%(0,3,1), 0, -3, 0, 1, -1, 3, -2
*SET,%_FNCNAME%(0,4,1), 0.0, -1, 0, 1, -3, 3, 1
*SET,%_FNCNAME%(0,5,1), 0.0, -2, 0, 1, -1, 3, 17
*SET,%_FNCNAME%(0,6,1), 0.0, -1, 9, 1, -2, 0, 0
*SET,%_FNCNAME%(0,7,1), 0.0, -2, 0, 2, 0, 0, -1
*SET,%_FNCNAME%(0,8,1), 0.0, -3, 0, 1, -1, 4, -2
*SET,%_FNCNAME%(0,9,1), 0.0, -1, 0, 0.5, 0, 0, -3
*SET,%_FNCNAME%(0,10,1), 0.0, -2, 0, 1, -1, 1, -3
*SET,%_FNCNAME%(0,11,1), 0.0, 99, 0, 1, -2, 0, 0
! End of equation: 0.5+(sin(2*{PI}*{TIME}*FREQ)/2)

ASEL,S,AREA,,OFFSET
NSLA,S,1
BF,ALL,TEMP,%ELEMEN%
ALLSEL,ALL

KBC,1.0
TIMINT,ON
NLGEOM,1
DELTIM,1/(FREQ*40),0,0
OUTRES,ERASE
OUTRES,ALL,1
AUTOTS,0
TIME,(1/FREQ)*LOOPS+1
NCNV,0,0,0,0,0
LSWRITE,3
LSSOLVE,1,3

FINISH

/POST26

NUMVAR,200
NSOL,2,NODETIP,U,X, UX_D
NSOL,3,NODETIP,U,Y, UY_D
NSOL,4,nodedamp2,U,X, Slider
STORE,MERGE
!Number of possible data vars
!X displacement Left Side
!Y displacement Left Side
!X displacement Right Side
!Read results

FILLDATA,5,,,1,0
ADD,5,5,1,,PZT_TIME,,, -1,1

Interference = I*100
XVAR,5
PLVAR,2,3,4,

```

```

/axlab,x,TIME (s)
/axlab,y,Displacement (m)
/replot
FILEOUT = 1
*IF,FORCE,EQ,0,THEN
~eui,'source [file join filenameSL.tcl]' !SL_INTERF_DIR_File#
*ELSE
~eui,'source [file join filenameSLF.tcl]' !SL_INTERF_DIR_FORCE_File#
*ENDIF
/IMAGE,SAVE,filename,jpg

XVAR,2
PLVAR,3
/axlab,x,UX (m)
/axlab,y,UY (m)
/replot
FILEOUT = 2
*IF,FORCE,EQ,0,THEN
~eui,'source [file join filenameSL.tcl]' !SL_INTERF_DIR_File#
*ELSE
~eui,'source [file join filenameSLF.tcl]' !SL_INTERF_DIR_FORCE_File#
*ENDIF
/IMAGE,SAVE,filename,jpg
FINISH

!Comment out the following two lines (for a single run) to review other
!output such as stress.
/clear
PARRES,NEW,loopcounter

FINISH
*ENDDO
*ENDDO

```

filenamecreate.tcl

```

set FREQ [ans_getvalue PARM,FREQ,VALUE]
set OFFSET [ans_getvalue PARM,OFFSET,VALUE]
set FILEOUT [ans_getvalue PARM,FILEOUT,VALUE]

set tclfile FPA_$FREQ\_$_OFFSET\_$_FILEOUT
ans_sendcommand *set,filename,('$tclfile')

```

filenameSL.tcl

```

set FREQ [ans_getvalue PARM,FREQ,VALUE]
set Interference [ans_getvalue PARM,Interference,VALUE]
set FILEOUT [ans_getvalue PARM,FILEOUT,VALUE]

set tclfile SL_$FREQ\_$_Interference\_$_FILEOUT
ans_sendcommand *set,filename,('$tclfile')

```

filenameSLF.tcl

```

set FREQ [ans_getvalue PARM,FREQ,VALUE]

```

```
set Interference [ans_getvalue PARM,Interference,VALUE]
set FORCE [ans_getvalue PARM,FORCE,VALUE]
set FILEOUT [ans_getvalue PARM,FILEOUT,VALUE]

set tclfile SLF_$FREQ\_$_Interference\_$_FORCE\_$_FILEOUT
ans_sendcommand *set,filename,('$tclfile')
```

ANSYS BATCH FILES TUTORIAL

The ANSYS tutorial provided is meant to help students understand how batch files work and how to create one. The tutorial builds a batch file from the preprocessing all the way to the postprocessing. The tutorial guides the user through a written batch file that includes a contact interface.

Weight Drop on Beam:

An ANSYS Tutorial to Create a Log File with Contact Interfaces By Scott Sharp

Problem:

A steel weight is to be dropped onto an aluminum beam. The maximum beam deflection and the maximum bending stress of the beam need to be determined.

Audience:

This tutorial assumes some familiarity with ANSYS 8.0. The user is left to look up unknown commands in the ANSYS 8.0 help files to determine how to use them.

Log Files (also called Batch or Command files):

This tutorial will lead the user through an ANSYS analysis using command lines to create a log file. Further recommended instruction and examples of batch files can be found in ANSYS help menu, ANSYS Tutorial by Kent L. Lawrence, and on the University of Alberta's ANSYS tutorial webpage <http://www.mece.ualberta.ca/tutorials/ansys/>.

Tutorial Layout:

The tutorial will be laid out in 6 sections. The first section will describe the problem setup. The next three sections will create a log file using the three FEA steps: preprocessor, solution processor, and postprocessor. The forth section will show the analysis output compared to analytical solutions and the fifth will provide the finished batch file. Several ANSYS commands will be briefly explained and will appear inside text boxes. All log file lines will be shown in 10 pt. `courier text`. Hint and suggestions will be *italicized*.

Section 1 - Problem Setup

The weight will be made of steel and the beam will be aluminum. The geometry is shown in the figure below. The problem will be created in 2D, because there is no need to evaluate a larger 3D problem that gives the same results. The dimensions are shown below, the thickness of the beam and weight are 3 and 2 inches respectively. The weight will be dropped from 10 inches above the beam. To further reduce processing time, the weight will be placed 1 inch above the beam and an initial velocity will be given to the weight. *Slightly better results can be computed by placing the weight tip on the beam surface at the start.* The side of the weight will be bound in the horizontal direction so that it can only fall vertically. The end of the cantilever beam is bound in the horizontal direction and bound in the center in the vertical direction. This allows for Poisson's ratio contraction of the beam that will give more accurate stress results near the bounded end of the beam. The beam will be massless to simplify the analytical calculations that the results are compared to.

Section 2 - Preprocessor

The preprocessor is used to declare the element type and associated options, define the material properties, create or import the geometry, create meshing for the geometry including contact meshes, and applying the loads and initial conditions. It should be noted that the loads and initial conditions can also be applied in the solution processor.

- 1) Open a new text file. *Microsoft Notepad works great.*
- 2) Enter the analysis filename and title (the title will be on all output).

```
/FILENAME,Weight_Drop
/TITLE,Weight Dropped on Beam
```

- 3) Enter the preprocessor and declare the element type and associated options. Element type Plane82 is an 8-node quad. *The commands may be looked up by going to **ANSYS help**, clicking the **Index tab**, and typing in the command. ANSYS commands can also be learned by using the GUI and opening the **Main Menu** >**Session Editor** and looking at the command lines. By looking up 'Plane82' in the help menu, one can choose from the several options available for the element type. KEYOPT 3 is chosen which is the plane stress with thickness option. It will be left to the user to look up the majority of the commands. Comments may be made in the code by using a '!'.*

```
/PREP7                                !Entering the preprocessor
ET,1,PLANE82
KEYOPT,1,3,3
```

ET, ITYPE, Ename, ...

Defines a local element type from the element library.

ITYPE Arbitrary element set number

Fname Element type

KEYOPT, ITYPE, KNUM, VALUE

Sets elements key options.

ITYPE Associated element set number

Knum Number of KEYOPT to be defined

Value Value for KEYOPT

- 4) Enter the real constants values (real constants are defined by element type options). *Here they define thickness.*

```
R,1,2                                !Thickness of weight
R,2,3                                !Thickness of beam
```


R, Real constant set, VALUE
Sets real constants.

- 5) Define material sets and properties. *Set 1 is for steel. Set 2 is for aluminum. Be careful that the units match in the analysis, gravity is 386.4 in/sec and the density must be the mass density (lbf-sec²/in⁴).*

```
MP,EX,1,30e6
MP,PRXY,1,0.3
MP,DENS,1,0.283/386.4
```

```
MP,EX,2,10e6
MP,PRXY,2,0.34
```

MP, LAB, MAT, C0,...
Defines material properties.

LAB	Material property label
MAT	Material property set number
C0	Liner material property value

- 6) Create variables. *Other beam and weight sizes can easily be tried.*

```
BmL = 10                !beam length
BmH = 0.5               !beam height

dH = 3                  !drop weight height
dW = 2                  !drop weight width
dTRI = 0.5              !drop weight triangle height
Drop = 10               !drop distance
```

- 7) Create geometry. *Rectangles are used for the beam and part of the weight. The bottom area of the weight is made by connecting 3 keypoints. The two areas of the weight are added together with the 'AADD' command.*

```
RECTNG, -BmL, 1, -BmH, 0          !create beam

RECTNG, -dW/2, dW/2, dTRI+1, dTRI+dH+1
K, 10, 0, 1
K, 11, -dW/2, dTRI+1
K, 12, dW/2, dTRI+1
A, 10, 11, 12                    !create area from keypoints
AADD, 2, 3
```

- 8) Create hard point on line on top of the beam at point of impact. Copy the commands up to this point into the ANSYS command prompt input and then check line number. It should be number 3. We will give it the hard point number

9 so we can reference it easily later in the program. *This will create a node on the beam at exactly 10 inches out which will be used to measure the beam deflection.*

```
HPTCREATE,LINE,3,9,COORD,0,0
```

9) Set area attributes and mesh.

```
ASEL,S,AREA,,1           !select area
AATT,2,2,1,0,           !area attributes
SMRT,2                   !set free mesh size
AMESH,ALL                !mesh all current selections
```

```
ASEL,S,AREA,,4
AATT,1,1,1,0,
SMRT,4
AMESH,ALL
ALLSEL,ALL               !Select everything
```

ASEL, TYPE, ITEM, COMP, VMIN,...
Selects a subset of areas.

TYPE Type of set (S = new set, R = reselect from current set)
 ITEM Label identifying data
 COMP Component of the item (if required)
 VMIN Minimum or only range value

AATT, MAT, REAL, TYPE, ESYS, ...
Associates element attributes with the selected, unmeshed areas.

MAT Material property set
 REAL Real constant set
 TYPE Type number set
 ESYS Associated coordinate system

- 10) We are now going to create the contacts meshes. For this, we will use the GUI. Copy and Paste all of the code up to this point into the ANSYS command prompt. *All of the commands may be copied and pasted in one chunk.*
- 11) Go **ANSYS Main Menu > Preprocessor > Modeling > Create > Contact Pair**. *The Contact Manager should be open now.*
- 12) Click on the Contact Wizard Button shown in the figure below. *The contact wizard window should open.*

- 13) Select 'Lines' and then 'Flexible' under the 'Target Surface' and 'Target Type' options.
- 14) Click the 'Pick Target' button. Select the upper line on the beam as shown below, *after step 17*. Select 'OK' in the selection menu after picking the line.
- 15) Click 'Next' in the contact wizard window.
- 16) Select 'Points' and then 'Node-to-Surface' under the 'Contact Surface' and 'Contact Element Type' options.
- 17) Click the 'Pick Contact' button. Select the bottom center point of the weight as shown below. Select 'OK' in the selection menu after picking the point.

- 18) Unselect 'Include initial penetration' then click the 'Create' button and 'Finish' on the popup window.
- 19) Go **File > Write DB Log File ...**
- 20) Select 'Write non-essential cmds as comments'. Name the file and Save. *Pay attention to where the file is saved to.*
- 21) Open the new 'lgw' file that you just created. *It can be opened in Notepad or other word processors.*

- 22) Copy everything starting from ‘! /COM, CONTACT PAIR CREATION – START’ to ‘! /COM, CONTACT PAIR CREATION – END’.
- 23) Paste the selection onto the end of your log file.
- 24) Exit the preprocessor

FINISH

Section 3 – Solution Processor

The solution processor is used to define loads, if not done in the preprocessor, and define the solution type with its solution options. The FEA model is solved at the end of the solution processor stage.

- 25) Enter the solution processor.

/SOL

- 26) Add gravity to the model. *Remember units must be kept constant, $g = 386.4 \text{ in/s}^2$.*

ACEL,0,386.4,0

- 27) Constrain the end of the beam in the x-direction. *The line is selected, all nodes of the line are then selected, and then the constraint is created.*

```
LSEL,S,LOC,X,-BmL      !line select
NSLL,S,1               !select nodes from line
D,ALL,UX,0             !ground nodes in x-dir
```

- 28) Constrain the middle of the beam in the y-direction. *The center nodes are reselected from the set from step 27 for the vertical constraint.*

```
NSEL,R,LOC,Y,-BmH/2-BmH/20,-BmH/2+BmH/20
D,ALL,UY,0             !ground nodes in y-dir
ALLSEL,ALL
```

- 29) Add the initial condition (velocity) to the weight. *The speed is calculated 1 inch above the beam.*

```
ASEL,S,AREA,,4
NSLA,S,1               !node select from area
speed = sqrt(2*386.4*(Drop-1))!calculate initial speed
IC,ALL,UY,, -speed     !set initial condition
```

- 30) Constrain the weight to only move vertically. *The side is constrained here.*

```

NSEL,R,LOC,X,-dW/2
D,ALL,UX,0
ALLSEL,ALL

```

- 31) Declare the analysis type. *We will do a full transient or dynamic simulation here.*

```

ANTYPE,4                !transient analysis
TRNOPT,FULL             !full transient method

```

- 32) Input the appropriate analysis options and controls. *Non-linear, 200 substep, erase previous output, keep all output from every substep, and end at time = 0.02 seconds.*

```

NLGEOM,1                !non-linear analysis
NSUBST,200,0,0          !substeps
OUTRES,ERASE            !erase previous solution output
OUTRES,ALL,ALL          !keep all solution output
TIME,0.02               !end analysis at 0.02 sec

```

- 33) Enter the solve command. Exit the solution processor.

```

SOLVE
FINISH

```

Section 4 – Postprocessor

The postprocessor is used to review the solution and to create the useful output such as graphs, animations, or data list. There are two separate postprocessor. The first is the time history postprocessor that is used for data list or graphs over time, frequency, solution steps, substep, or etc. The second postprocessor is the general postprocessor for all other analysis output.

- 34) Enter the time history postprocessor

```

/POST26

```

- 35) Create a variable for the node number at the weight tip. *This will be used to look at the motion of the weight over time.*

```

KSEL,S,KP,,10           !select keypoint
NSLK,S                  !select node at selected kps
*GET,weight_tip,NODE,,NUM,MIN !name = minimum node #
ALLSEL,ALL

```

- 36) Create a variable for the node number at the hard point on the beam. *This will be used to look at the displacement of the beam point over time.*

```

KSEL,S,KP,,9

```

```
NSLK,S
*GET,beam_node,NODE,,NUM,MIN
ALLSEL,ALL
```

- 37) Using the created node number variables, create result variables at the two nodes. *First, the maximum number of variables is declared (200 is default). Next, the two variables are named and the desired output is declared. Last, the variables are stored.*

```
NUMVAR,200
NSOL,2,weight_tip,U,Y,Weight Displacement
NSOL,3,beam_node,U,Y,Beam Deflection
STORE,MERGE
```

- 38) Create variables for the maximum beam deflection and the time at which it occurred.

```
*GET,max_deflection,VARI,3,EXTREM,VMIN
*GET,max_deflection_time,VARI,3,EXTREM,TMIN
```

- 39) Create a time vs. displacement plot of the weight and beam. *Name the y-axis, add a text label stating the maximum beam deflection, plot time on x-axis (1 is always the time variable in the postprocessor), and plot the created variable weight displacement and beam deflection.*

```
/axlab,y,Displacement (in)
/TLABEL,0,0.6,Max Beam Deflection = %max_deflection%
XVAR,1
PLVAR,2,3
```

- 40) Save the graph as a picture.

```
/IMAGE,SAVE,Beam_Drop_Disp,jpg
```

- 41) Exit the time history postprocessor.

```
FINISH
```

- 42) Enter the general postprocessor.

```
/POST1
```

- 43) Read results at time of maximum deflection.

```
SET,,1,,max_deflection_time,
```

- 44) Plot the x-direction stress and save the graph as a picture.

```
PLNSOL,S,X,0,1
/IMAGE,SAVE,Beam_Drop_Stress,jpg
```

45) Exit the general postprocessor.

FINISH

46) Save the text file and it is ready to be run.

47) Run the text file by entering the ANSYS GUI > **File > Read Input From...**

48) Select the text file and click 'OK'

Section 5 – Analytical and ANSYS Solutions

This section first describes the analytical solution. Next, the program output is shown and the maximum bending stress and maximum beam deflection results are compared to the analytical solutions.

The deflection of a cantilever beam with a single point load on the end is described by, $\delta_{MAX} = \frac{l^3}{3EI} \cdot F$, where l is the length of the beam, E is the modulus, I is the moment of inertia about the bending axis, and F is the point load. Recognizing that the equation is a simple spring ($F = k \cdot x$) the spring constant, k , would be $\frac{3EI}{l^3}$. Using the conservation of energy and assuming the beam as massless, the maximum beam deflection could be found by $m_{weight} \cdot g \cdot (y + \delta) = \frac{1}{2} k \delta^2$ where g is gravity, y is the distance the weight is dropped above the beam, k is the spring constant of the beam, and δ is the maximum beam deflection.

The maximum stress for the beam would be located at the constrained end at the top or bottom of the beam and can be found by $\sigma = \frac{F \cdot l \cdot h / 2}{I}$, where h is the height of the beam. The spring equation may be substituted in to simplify the equation to $\sigma = \frac{k \cdot \delta \cdot l \cdot h / 2}{I}$.

The ANSYS outputs the following two graphs. The first graph shows the beam and weight displacement over time. The graph indicates that the maximum deflection of the beam was 0.28269 inches. The second graph shows the x-direction stress in the beam. The maximum bending stress in the beam is 21,241 psi.

The following table compares the ANSYS results to the analytical ones.

Measurement	ANSYS	Analytical	Ratio
Max Beam Deflection	0.28269	0.2785	0.985
Max Bending Stress	21,241	21,187	0.997

Section 5 – Final Log File for Tutorial

```
/FILNAME,Weight_Drop
/TITLE,Weight Dropped on Beam

/PREP7                                !enter the preprocessor

!DEFINE ELEMENT TYPES
ET,1,PLANE82                          !element 'plane82' 8 node quad
KEYOPT,1,3,3

!DEFINE REAL CONSTANTS
R,1,2                                 !thickness 1 = 2" (weight)
R,2,3                                 !thickness 2 = 3"(beam)

!DEFINE MATERIAL PROPERTIES

!steel properties - MP set 1
MP,EX,1,30e6                          !modulus of elasticity
MP,PRXY,1,0.3                         !posion ratio
MP,DENS,1,0.283/386.4                 !mass density

!aluminum properties - MP set 2
MP,EX,2,10e6
MP,PRXY,2,0.34

!CREATE VARIABLES

BmL = 10                              !beam length
BmH = 0.5                             !beam height

dH = 3                                !drop Weight height
dW = 2                                !drop Weight width
dTRI = 0.5                            !drop Weight triangle height
Drop = 10                             !drop distance

! CREATE GEOMETRY

!create beam
RECTNG,-BmL,1,-BmH,0                 !create rectangle
HPTCREATE,LINE,3,9,COORD,0,0         !create hardpoint on line

!create weight
RECTNG,-dW/2,dW/2,dTRI+1,dTRI+dH+1

K,10,0,1                              !create keypoints
K,11,-dW/2,dTRI+1
K,12,dW/2,dTRI+1
A,10,11,12                            !create area from keypoints

AADD,2,3                              !add areas (weight)

!AREA PROPERTIES AND MESHING

!mesh beam
```

```

ASEL,S,AREA,,1           !select area
AATT,2,2,1,0,           !area attributes
SMRT,2                   !set free mesh size
AMESH,ALL                !mesh all current selections

!mesh weight
ASEL,S,AREA,,4
AATT,1,1,1,0,
SMRT,4
AMESH,ALL
ALLSEL,ALL

! /COM, CONTACT PAIR CREATION - START      !contact start
CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
! /GSAV,cwz,gsav,,temp
MP,MU,1,
MAT,1
MP,EMIS,1,7.88860905221e-031
R,3
REAL,3
ET,2,169
ET,3,175
R,3,,,1.0,0.1,0,
RMORE,,,1.0E20,0.0,1.0,
RMORE,0.0,0,1.0,,1.0,0.5
RMORE,0,1.0,1.0,0.0,,1.0
RMORE,10.0
KEYOPT,3,3,0
KEYOPT,3,4,0
KEYOPT,3,5,0
KEYOPT,3,7,0
KEYOPT,3,8,0
KEYOPT,3,9,1
KEYOPT,3,10,1
KEYOPT,3,11,0
KEYOPT,3,12,0
KEYOPT,3,2,0
! Generate the target surface
LSEL,S,,,3
CM,_TARGET,LINE
TYPE,2
NSLL,S,1
ESLN,S,0
ESURF,ALL
CMSEL,S,_ELEMCM
! Generate the contact surface
KSEL,S,,,10
CM,_CONTACT,KP
TYPE,3
KATT,-1,3,3,-1
KMESH,ALL
ALLSEL

```

```

ESEL,ALL
ESEL,S,TYPE,,2
ESEL,A,TYPE,,3
ESEL,R,REAL,,3
ESEL,ALL
ESEL,S,TYPE,,2
ESEL,A,TYPE,,3
ESEL,R,REAL,,3
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
! /GRES,cwz,gsav
CMDEL,_TARGET
CMDEL,_CONTACT
! /COM, CONTACT PAIR CREATION - END          !contact end
FINISH                                       !exit preprocessor

/SOL                                       !enter solution processor

!DEFINE LOADS and FORCES

ACEL,0,386.4,0                            !add gravity units=in/s^2

!Bind beam end in X-dir
LSEL,S,LOC,X,-BmL                         !line select
NSLL,S,1                                  !select nodes from line
D,ALL,UX,0                                !ground nodes in x-dir

!Bind beam end in center in Y-dir
NSEL,R,LOC,Y,-BmH/2-BmH/20,-BmH/2+BmH/20 !node select
D,ALL,UY,0                                !ground nodes in y-dir
ALLSEL,ALL

!Give wieght an initial velocity
ASEL,S,AREA,,4
NSLA,S,1                                  !node select from area
speed = sqrt(2*386.4*(Drop-1))            !calculate initial speed of mass
IC,ALL,UY,, -speed                        !set initial condition

!Bind weight on side in X-dir
NSEL,R,LOC,X,-dW/2
D,ALL,UX,0
ALLSEL,ALL                                !select everything

ANTYPE,4                                  !transient analysis
TRNOPT,FULL                               !full transient method
NLGEOM,1                                  !non-linear analysis

```

```

NSUBST,200,0,0          !substeps
OUTRES,ERASE            !erase previous solution output
OUTRES,ALL,ALL          !keep all solution output at every
substep
TIME,0.02               !end anaylsis at 0.02 sec
SOLVE                   !solve
FINISH                   !exit solution processor

/POST26                  !enter time history postprocessor

KSEL,S,KP,,10           !select keypoint
NSLK,S                  !select node at selected kps
*GET,weight_tip,NODE,,NUM,MIN !name min node #
ALLSEL,ALL

KSEL,S,KP,,9
NSLK,S
*GET,beam_node,NODE,,NUM,MIN
ALLSEL,ALL

NUMVAR,200               !max number of time history variables
NSOL,2,weight_tip,U,Y, Weight Displacement      !create and define TH
variable
NSOL,3,beam_node,U,Y,Beam Deflection
STORE,MERGE              !store new TH variables

*GET,max_deflection,VARI,3,EXTREM,VMIN           !get min value of TH
variable
*GET,max_deflection_time,VARI,3,EXTREM,TMIN      !get time of min value
of TH variable

/axlab,y,Displacement (in)      !add y axis title
/TLABEL,0,0.6,Max Beam Deflection = %max_deflection%
XVAR,1                          !plot time as x-axis
PLVAR,2,3                       !plot variables as y-axis
/IMAGE,SAVE,Beam_Drop_Disp,jpg  !create jpeg of graph
FINISH                           !exit time history postprocessor

/POST1                         !enter general postprocessor
SET,,,1,,max_deflection_time,  !read results at indicated time
PLNSOL,S,X,0,1                 !plot X-dir stress
/IMAGE,SAVE,Beam_Drop_Stress,jpg
FINISH                         !exit general postprocessor

```


REFERENCES

- ¹ Uchino, K. and Koc, B., “Compact Piezoelectric Ultrasonic Motors,” *International Center for Actuators and Transducers*.
- ² Ngol, B. K. A., Lim, L. E. N., and Lin, W., “Linear Piezoelectric Motor with Self-locking Means”, U.S. Patent 6,611,080, Aug 26, 2003.
- ³ Feinstein, A. and Teschler, L., ”Coarse and Fine Combined”, *Machine Design* vol. 76, Issue 13 (2004).
- ⁴ Koc, B., Basaran, D., Akin, T, and Erden, A., “Design of a Piezoelectric Ultrasonic Motor for Micro-Robotic Application”.
- ⁵ Uchino, K., *Piezoelectric Actuators and Ultrasonic Motors*. Boston: Kluwer Academic Publishers, 1997.
- ⁶ Ueha, S. and Tomikawa, Y., *Ultrasonic Motors: Theory and Application*. New York: Oxford University Press, 1993.
- ⁷ Friend, J. R., Satonobu, J., Nakamura, K., Ueha, S., and Stutts, D. S., “A Single-Element Tuning Fork Piezoelectric Linear Actuator,” *IEEE Transactions on Ultrasonics, Ferroelectrics, and Frequency Control* vol. 50, no. 2 (2003).
- ⁸ Elliptec Resonant Actuator AG, “X15G Data Sheet” [Online], Oct 2004 [cited July 1, 2005]. Available from URL: <http://www.elliptec.com>.
- ⁹ EDO corp. “EDO Piezoelectric Motor”, EDO Catalog 2002, <http://www.edoceramic.com>.
- ¹⁰ Physik Instrumente (PI) L.P., “P-661” [Online], Dec 2005 [cited April 12, 2006]. Available from URL: <http://www.physikinstrumente.com>.
- ¹¹ Teschler, L., “Precision Moves with Pint-Sized Motors”, *Machine Design* vol. 77, Issue 7 (2005).
- ¹² Waldimir Wischniewskiy, W., “Piezoelectric Adjusting Element”, US Patent, 6765355, July 20, 2004.
- ¹³ Zumeris, J., “Ceramic Motor”, US Patent, 5453653, September 6, 1995.

-
- ¹⁴ Nanomotion LTD., “HR1 Ultrasonic Motor User Manual” [Online], Dec 2005 [cited April 12, 2006]. Available from URL: <http://www.nanomotion.com>.
- ¹⁵ Sashida, T. and Kenjo, T., *An Introduction to Ultrasonic Motors*. New York: Oxford University Press, 1993.
- ¹⁶ Wallaschek, J., “Contact Mechanics of Piezoelectric Ultrasonic Motors”, *Smart Materials and Structures* vol. 7 (1998).
- ¹⁷ Nakamura, K., Kurosawa, M., and Ueha, S., “Characteristics of a Hybrid Transducer-Type Ultrasonic Motor”, *IEEE Transactions on Ultrasonic, Ferroelectrics, and Frequency Control* vol. 38 (1991).
- ¹⁸ Kamano, T., Suzuki, T., and Otoi, E., “Characteristics and Model of Ultrasonic Motor”, *Proceedings of 8th Symposium on Ultrasonic Electronics* (1987).
- ¹⁹ Physik Instrumente (PI) L.P., “User Manual MP 58E: P-661 & P-665”.
- ²⁰ ANSYS Incorporated, “ANSYS Release 8.0 Documentation”, <http://www.ansys.com>.
- ²¹ Lawrence, K.L., *ANSYS Tutorial: Release 6.1*. Schroff Development Corporation Publications, 2002.
- ²² “UofA ANSYS Tutorials” [Online], April 2004 [cited April 24, 2006]. Available from URL: <http://www.mece.ualberta.ca/tutorials/ansys/>.
- ²³ Callister, W. D. Jr, *Materials Science and Engineering an Introduction (5th edition)*. New York: John Wiley & Sons, Inc., 2000.
- ²⁴ Smith, J. W., *Vibration of Structures: Applications in Civil Engineering Design*. London: Chapman and Hall, 1988.
- ²⁵ Harris, C. M., *Shock and Vibration Handbook (5th edition)*. Boston: McGraw-Hill, 2002.
- ²⁶ Roark, R.J., Young, W.C., *Formulas for Stress and Strain (5th edition)*. New York: McGraw-Hill, 1975.