1993

Structural analysis and design of a plastic electric guitar neck

Edward John Grasso

Lehigh University

Follow this and additional works at: http://preserve.lehigh.edu/etd

Recommended Citation

This Thesis is brought to you for free and open access by Lehigh Preserve. It has been accepted for inclusion in Theses and Dissertations by an authorized administrator of Lehigh Preserve. For more information, please contact preserve@lehigh.edu.
AUTHOR: Grasso, Edward John

TITLE: Structural Analysis and Design of A Plastic Electric Guitar Neck

DATE: May 30, 1993
Structural Analysis and Design
of a Plastic Electric Guitar Neck

by
Edward John Grasso

A Thesis
Presented to the Graduate Committee
of Lehigh University
in Candidacy for the Degree of
Master of Science
in
Mechanical Engineering

Lehigh University
May 1993
Acknowledgements

I would like to thank my advisor, Professor John B. Ochs, of Lehigh University's Mechanical Engineering and Mechanics Department for his continued guidance throughout my graduate work at Lehigh.

I would also like to thank Richard Roland, CEO of Neo Products Inc., for making such research available to me, and for his assistance in helping the project move along smoothly.

Special thanks go out to my parents, Fred and Anne Grasso, for always showing patience, support and encouragement throughout my entire education.

Finally, I would like to dedicate this thesis to my fiancee, Amy Kipp, who has been my emotional support throughout my graduate education.
Table of Contents

ABSTRACT .......................................................... 1

1. INTRODUCTION .................................................. 2
   1.1 Statement of Need ........................................... 2
   1.2 Purpose of the Study ....................................... 3
   1.3 State of the Art ........................................... 3
   1.4 Geometric Modeling ......................................... 10
   1.5 Introduction to FEM/FEA ................................... 13
   1.6 Organization of Thesis ................................... 14

2. TWO DIMENSIONAL MODELING ...................................... 16
   2.1 Design Requirements ....................................... 16
   2.2 Cross-Sectional Analysis .................................. 18
   2.3 Two Dimensional Model ..................................... 26
   2.4 Finite Element Model ...................................... 26
      2.4.1 Thin Shell Element .................................... 29
      2.4.2 Restraints ............................................ 29
      2.4.3 Structural Loads ...................................... 31
      2.4.4 Material Properties .................................. 31
      2.4.5 Stresses and Displacements. .......................... 33
   2.5 Experimental Model ........................................ 44
      2.5.1 Test Bench and Specimen .............................. 45
      2.5.2 Test Procedure ....................................... 48
      2.5.3 Results and Verification of FEA ................. 50

3. THREE DIMENSIONAL MODELING ................................... 53
   3.1 Design Geometry ........................................... 53
   3.2 Finite Element Model ...................................... 53
      3.2.1 Solid Element ....................................... 55
      3.2.2 Restraints ............................................ 56
      3.2.3 Structural Loads ...................................... 58
      3.2.4 Material Properties .................................. 59
      3.2.5 Stresses and Displacements. .......................... 59
         3.2.5.1 String Tension Cases ............................ 61
         3.2.5.2 User Torsion Case ............................... 66
      3.2.6 Linear Element vs. Parabolic Elem. ............... 67
      3.2.7 String Distance to Fretboard ....................... 69
   3.3 Experimental Comparison ................................... 70
      3.3.1 Test Specimen ........................................ 73
      3.3.2 Results and Comparison .............................. 73

4. MANUFACTURING METHOD .......................................... 76
   4.1 Existing Manufacturing Methods ........................... 76
      4.1.1 Casting ................................................ 76
      4.1.2 CNC Machining ........................................ 77
4.1.3 Injection Molding .......... 79
4.1.4 Composite Layup .......... 82
4.2 Mold Design Considerations .......... 82
4.2.1 Neo Molding Technique .......... 85
4.3 Material .................. 87
4.4 Recommended Material and Manufacturing Method ........ 88

5. CONCLUSIONS .............. 92
5.1 Conclusion of Study .......... 92
5.2 Future Work ............... 93

REFERENCES .................. 95
APPENDIX A. - Neo Products Inc. Information Booklet 97
APPENDIX B - Guitar Geometry .......... 100
APPENDIX C. - Three Dimensional Finite Results .......... 107
APPENDIX D. - Three Dimensional Experimental Results .......... 132
VITA ............................. 137
List of Tables

Table 2.1 - Cross-sectional Beam Analysis Results 25
Table 2.2 - Material Properties of Steel and Aluminum Alloy 33
Table 2.3 - Case 1, Displacement and Stresses 40
Table 2.4 - Case 2, Displacement and Strain Data 44
Table 2.5 - 2-D Experimental and FEA Results (Case 2) 52
Table 3.1 - Cases 1-7, Displacement and Stresses 66
Table 3.2 - Case 8, Displacement and Stresses 67
Table 3.3 - Linear Element vs. Parabolic Element Results 69
Table 3.4 - Three Dimensional Experimental Results 75
Table 4.1 - Material Properties 89
Table D.1 - Neo Products Experimental Data 134
Table D.2 - Martin Neck Experimental Data 135
Table D.3 - Sekova Neck Experimental Data 136
List of Figures

Figure 1.1 - Circular Two-Way Adjustable Truss Rod . 4
Figure 1.2 - Another Two-Way Adjustable Truss Rod . 5
Figure 1.3 - Gibson Style Adjustable Truss Rod . 6
Figure 1.4 - S-shaped Gibson Adjustable Truss Rod . 6
Figure 1.5 - Rickenbacker Adjustable Truss Rod . 7
Figure 1.6 - Martin Style Adjustable Truss Rod . 8
Figure 1.7 - Compression Rod . 8
Figure 1.8 - Non-Adjustable Truss Rods . 9
Figure 1.9 - Graphite Epoxy Composite . 10
Figure 1.10 - Typical I-DEAS Screen . 12
Figure 2.1 - Exploded View of Guitar Design . 17
Figure 2.2 - Minimum Allowable Modulus of Elasticity for Stiffener with Various Cross-Sectional Shapes (A-J) . 20
Figure 2.3 - Minimum Allowable Modulus of Elasticity for Stiffener with Various Cross-Sectional Shapes (K-R) . 21
Figure 2.4 - Cross-section Dimensions . 23
Figure 2.5 - I-DEAS Beam Analysis Model . 24
Figure 2.6 - Wooden Neck Dimensions . 27
Figure 2.7 - Two Dimensional Model Dimensions . 28
Figure 2.8 - Case 1 Restraints and Structural Loads . 30
Figure 2.9 - Case 2 Restraints and Structural Loads . 32
Figure 2.10 - Case 1, Displacement in Y-dir . 35
Figure 2.11 - Case 1, Deformed vs Undeformed Stiffener . 36
Figure 2.12 - Case 1, Maximum Principal Stresses . 37
Figure 2.13 - Case 1, Minimum Principal Stresses . 38
Figure 2.14 - Case 1, Maximum Shearing Stresses . 39
Figure 2.15 - Case 2, Displacement in y direction . 41
Figure 2.16 - Case 2, Strain in x direction, Top Nodes . 42
Figure 2.17 - Case 2, Strain in x dir, Bottom Nodes . 43
Figure 2.18 - Experimental Apparatus . 46
Figure 2.19 - Strain Gage & Dial Indicator Locations . 51
Figure 3.1 - TapI, TapU, and TapT Designs . 54
Figure 3.2 - 3-D Restraints and Structural Loads . 57
Figure 3.3 - Case 8, Structural Loads . 60
Figure 3.4 - Case 3, Displacement in the Y-Dir . 62
Figure 3.5 - Case 3, Maximum Principal Stresses . 63
Figure 3.6 - Case 3, Minimum Principal Stresses . 64
Figure 3.7 - Case 3, Maximum Shearing Stresses . 65
Figure 3.8 - String Fretboard Displacement . 71
Figure 3.9 - Location of Maximum String Deflection 72

vii
List of Equations

2.1 - Minimum Deflection of Candlever Beam Equation. ........................................... 19
2.2 - Percent Error Equation .......................................................... 50
3.1 - Percent Difference Equation ...................................................... 68
D.1 - Mean Equation .............................................................. 132
D.2 - Standard Deviation Equation .................................................... 133
D.3 - Signal to Noise Equation ......................................................... 133
ABSTRACT

The primary structural component of an electric guitar is its neck. The stiffening member for a hollow plastic guitar neck has been designed using solid modeling, assembly modeling, and finite element modeling. Both two and three dimensional analyses were conducted. Experiments were conducted to verify the finite element analysis results. Changes in geometry, material and manufacturing methods were used to improve the strength and stiffness capabilities of various designs while still keeping within the geometric restrictions. The final design was a single-piece, stiffening member that connects to the strings at the pegs and the tailpiece. The material selected to be injection molded was the chopped fiber form of Graphite/epoxy.
1. INTRODUCTION

1.1 Statement of Need

Neo Products Inc. is a company that specializes in the production of plastic violins and guitars. The instruments that are created are made of a Polymethylmethacrylate (PMMA) skin, better known as Plexiglas. The rest of the instrument is hollow and therefore allows for an array of possibilities. For the interior space some of the uses of this space are the placement of neon tubes, the insertion of gumballs, or crinkled dollar bills, just to name a few. These designs are patent pending, where the U.S. patent application number is 760956, and the international P.C.T. number is U292-07872. For a more detailed description of the Neo product line see Appendix A.

For the guitar, Neo Products Inc. originally used a plastic body and a wooden neck. It was in their interest to make the entire guitar out of plastic, yet the low strength and stiffness of PMMA made the plastic incapable of withstanding the forces induced by the string tension. Therefore, the need for an added stiffening member that would take this load became evident. Most wooden guitars have an imbedded metal support of various designs. The
study presented in this thesis is the investigation of the technical feasibility of a plastic neck with some kind of internal support incorporated into the plastic guitar.

1.2 Purpose of the Study

This study was conducted to determine various design choices that would increase the stiffness of the guitar and allow it to maintain its hollow plastic skin. In doing so, ease and cost of manufacturing, ability to assemble with other parts, and availability of materials all had to be considered. To help determine if a design was acceptable both theoretical finite element analyses and experimental tests were conducted. It was necessary to design some new parts, as well as, redesign some old ones so assembly would be possible. Two and three dimensional models have been created. The final step of this study was to create production drawings of all the parts for manufacturing. This investigation also included the design of the mold for the proposed neck stiffener.

1.3 State of the Art

For all guitars, the most critical design location is the neck. This is due to the large string forces which generate high stresses and cause bowing. Therefore strengthening of the neck is required. In the music
industry today there are many different designs and materials that are used for strengthening the neck of a wooden guitar. There are three major categories of support mechanisms; adjustable truss rods, non-adjustable truss rods and compression rods. The adjustable truss rod is used most often.

There are many different adjustable truss rods that are used in the music industry, the following is a majority of them:

Figure 1.1 - Circular Two-Way Adjustable Truss Rod

1. A circular two-way adjustable truss rod that has no bend or bow to it. The rod is fourteen inches long and made of tempered stainless steel. As shown in Figure 1.1 there is a brass stop block (A) and an allen head keyway (B) used for adjustment. The rod is wrapped with fiberglass reinforced tape to damp vibrations that might be absorbed into the rod.
2. Another circular two-way adjustable truss rod is shown in Figure 1.2 [1]. It uses an eighteen inch long by 3/16 inch diameter rod (A) with a 1/4 inch hex head (B) and two stop blocks (C). Since the rod rotates, both compression and tension forces can be produced, thereby allowing the rod to correct for both directions of bow.

3. The Gibson style adjustable truss rod in Figure 1.3 is the same as the rod explained in number one but the rod is curved and the rod is only one-way adjustable. This was the first truss to be used.

4. An S-shaped Gibson style adjustable truss rod exists and can be seen in Figure 1.4 [2]. It was patented for use in a plastic molded guitar. Its exact dimensions
Figure 1.3 - Gibson Style Adjustable Truss Rod

Figure 1.4 - S-shaped Gibson Adjustable Truss Rod

and the purpose of the s-shape is not described in any available literature.

5. The Rickenbacker adjustable truss rod is a single rod that folds onto itself. As shown in Figure 1.5 one end is adhered to a stop block (A) and the other end is threaded and passed through the stop block and fastened by
the nut (B). This particular version is thirty six inches long and uses a 3/16 inch diameter steel rod. The rod is then wrapped with metal flash tape.

Figure 1.5 - Rickenbacker Adjustable Truss Rod

6. The Martin style adjustable truss rod, shown in Figure 1.6 [1], utilizes a 7/16 inch by 3/8 inch by 14-3/4 inch aluminum U-shaped channel (A) where a 3/16 inch steel rod (B) is placed inside it. The steel rod has an adjusting nut (C) and when it is tightened it forces the aluminum to bend in one direction.

Another alternative, shown in Figure 1.7, is the compression rod, this design is much less used and little information has been found about it. It is a 3/16 inch diameter steel rod that is bent to a right angle at the end. This bend keeps the rod immobile at that end and when
Figure 1.6 - Martin Style Adjustable Truss Rod

Figure 1.7 - Compression Rod

compressed at the other end by the adjusting screw the rod keeps the neck from bending.

Non-adjustable truss rods are less common. These designs are less complicated with respect to their assembly. The general concept for the non-adjustable truss
Figure 1.8 - Non-Adjustable Truss Rods

A truss rod is that it acts as a structural stiffener and prevents bowing due to its resistance to bending. Three different designs have been found, as shown in Figure 1.8. The different shapes are a square rod (A), a T-shaped rod (B) and a double T-shaped rod (C). All three are approximately fourteen inches long and no more than 3/8 inches high. They are all made of steel.

It has been found that other materials exist other than the conventional steel. There exists a graphite-epoxy composite, shown in Figure 1.9, that comes in the form of thin rectangular rods (A), sheet stock (B), and thin bars (C). The rods are used like truss rods, the sheet stock is put under the fingerboard and the thin bars are put under the neck/fingerboard surface. One of the notable
properties of this material is the fact that it is 80% as stiff as steel by cross-section but is much lighter.

Figure 1.9 - Graphite Epoxy Composite

When correlating this information, in relation to the stiffener designed for the plastic guitar, one main difference should be noted. The reinforcement described above does not get attached to the strings while the stiffener designed in this study does. Because of this critical detail all of the previously existing reinforcements are inapplicable. The closest similarity could be seen in the non-adjustable truss rod shown in Figure 1.7.

1.4 Geometric Modeling

The Computer Aided Design and Computer Aided Manufacturing (CAD/CAM) package used for the various types of modeling was Integrated Design Engineering Analysis
Software (I-DEAS) by Structural Dynamics Research Corporation (SDRC). I-DEAS is a fully functional three dimensional solid modeler that has many tasks that it can perform. Each division of the software is called a family. The different families that were used in this study were Solid Modeling, Finite Element Modeling and Analysis, and Drafting. Within each family there are smaller groupings that are called tasks. Some of the tasks used to perform this study were Object Modeling, Assembly Modeling, Mesh Generation, and Post Processing just to name a few.

The main advantage of using I-DEAS is the fact that a single database is used by all the families, therefore, if a change is made in one family it will automatically be transferred to another. Errors due to database exporting and importing are thereby avoided.

Many drawings, figures, and graphs that will follow in this paper have been generated from the SDRC software. Figure 1.10 shows a typical layout from the I-DEAS screen. In the top left window the graphical representation is shown of the design where many overlapping menus can be selected from in order to manipulate the software. In the top right window, many icons are visible which allow easy access to frequently used commands. In the lower left corner we have the prompt window, where keyboard input is directed. And finally in the lower right corner is the
Figure 1.10 - Typical I-DEAS Screen
list window which posts any typed output to the screen. In many figures the boarders have been removed.

The computer that was used to conduct this study was the Hewlett-Packard Apollo 9000 series workstation. It was suitably configured to run the I-DEAS software.

1.5 Introduction to FEM/FEA

The finite element analysis or finite element method, abbreviated FEA/FEM, has been used to solve engineering problems since its first formal introduction in 1943 by R. Courant. [3] The need for the method arose when engineers came across problems that were too complex in their geometry to be solved with existing methods.

The finite element method makes a complex geometric problem solvable by taking that complex geometry and dividing it up into smaller sections. Each section will be a shape that is easy to evaluate such as a rod, a 2-D plate, or a 3-D block. Each rod, plate or block is represented by a wireframe of points and lines, where each point is called a node. The rod, plate or block is then called an element. Many different elements are connected at nodes which are then shared. The connection of many elements then creates what is called a mesh and will match the shape of the object to be studied. For ease of explanation let us consider a two dimensional mesh. In
this case each node has two degrees of freedom, therefore allowing it to move in the x and y directions. For each element an equation can be assigned to each node that approximates its displacement in respect to the x and y coordinates of that node. Once all the nodes are assigned equations, the equations can be written in matrix form. The matrix is called the element stiffness matrix. This same procedure is then done on all the elements. Then all the element stiffness matrices are combined into one large matrix, called the structure stiffness matrix, by a procedure that matches up shared nodes from neighboring elements. Once the structure stiffness matrix is known it is then possible to solve for the nodal displacements by using the structural loads and boundary conditions of the entire model. And finally from the displacements the stresses are calculated. [4]

The FEM/FEA techniques and experimental measurements are used in this study to develop the optimum design for a plastic guitar neck.

1.6 Organization of Thesis

Chapter two contains all the design work that was conducted for this study using a two dimensional model. This includes the geometric models, the finite element
analyses of the stiffener, and the experimental testing for verification.

Chapter three contains all the three dimensional model design work. Again, this includes the geometric models, the finite element analyses of the stiffener, and the experimental testing for verification.

Chapter four describes the mold design that has been developed and gives a background on injection molding and material choice.

Chapter five gives suggestions on any future work that could be done on this design and makes conclusions about this work.
2. TWO DIMENSIONAL MODELING

2.1 Design Requirements

Our unique approach to the design of the guitar neck involved the development of a structural support member surrounded by a plastic, possibly clear plastic, skin as shown in Figure 2.1. Three main design requirements had to be met: geometry requirements, displacements requirements, and manufacturing requirements.

For the geometry requirements, the design was to preserve the outside shape of the guitar. The plastic wall of the outer shell (A), as shown in Figure 2.1, was to be 0.25 inches thick. It was also necessary to fit two 3/8 inch diameter neon tubes up the entire length of the neck. In the body of the guitar it was necessary to allow the neon to run from one side of the body to the other as well. At all times all parts created must be capable of being easily assembled.

Under the loading of the strings, all guitar necks will deflect or bend causing a separation of the string from the fretboard making the instrument harder to play. In the average guitar with a truss rod this displacement is 0.2 inches. Some displacement is due to creep, which is
Figure 2.1 - Exploded View of Guitar Design
defined as:

The slow deformation of solid materials over extended periods under load. The amount of deformation is dependent on the time, the load, the material, and the temperature. [5]

As a result of this creep, musicians require adjustment of the "action" on the string. In the design presented here, it was desired to create a stiffener that would be so stiff that no adjustments would be necessary.

The design goal for the maximum displacement of the loaded neck is 0.02 inches. Therefore, the shape and materials had to be selected to provide the necessary stiffness to the neck. It was also necessary to verify that all stresses in the stiffener and the plastic shell would not exceed the ultimate stress of the material of each part.

Finally, it was necessary to determine the most cost effective and practical method of manufacture. Among the types of fabrication choices that could be chosen from were machining, casting, injection molding, thermoforming, and manual composite lay-up. The material and the shape of the part had a large influence on the methods of fabrication capable.

2.2 Cross-Sectional Analysis

The first step in determining the design was to do some hand and computer calculations on different cross-
sections of the stiffener that would fit in the neck. These calculations were done to determine which cross sections would produce acceptable deflections. To do this two different analyses were done. The first was a simple calculation where the neck was modeled as a constant cross section cantilever beam and the deflection of the neck was determined at the end. The second was a simple finite element analysis done with beam elements where the cross section was kept constant as well.

For the first analysis Equation 2.1 [6] was used:

$$Y_{\text{min}} = \frac{1}{2} \frac{ML^2}{EI}$$

where;

- $Y_{\text{min}}$ - minimum deflection (inches)
- $M$ - moment at end of cantilever (lb-in)
- $L$ - distance from tailpiece to nut (ins)
- $E$ - modulus of elasticity (psi)
- $I$ - moment of inertia (inches$^4$)

With this equation different cross-sectional shapes were checked to find out which ones could be used in the neck. This was done by determining the minimum allowable modulus of elasticity, $E_{\text{min}}$, that would produce a maximum deflection of 0.1 inches. The value of 0.1 inches was used because at this stage of the analysis it was uncertain what the maximum deflection could be. The moment of inertia was determined from the geometry of the cross-section. The value for the moment was determined by multiplying the distance from the centroid to the top of the cross-section...
Figure 2.2 - Minimum Allowable Modulus of Elasticity for Stiffener with Various Cross-Sectional Shapes (A-J).
Figure 2.3 - Minimum Allowable Modulus of Elasticity for Stiffener with Various Cross-Sectional Shapes (K-R).
with the total force that the strings created, 180 lbs. The cross-sections used and the resulting moduli are shown in Figures 2.2 and 2.3. These cross-sections were not all created to allow neon to be run through the neck. Notice that the shapes are shown with the thinnest part of the neck enclosing it. This is to give a proportional representation of each cross-section.

From this analysis it was determined that for the filled neck cross-section the minimum modulus, \( E_{\text{filled neck}} = 0.75 \times 10^6 \text{ psi} \), was higher than the modulus of Plexiglas, \( E_{\text{plexiglas}} = 0.5 \times 10^6 \text{ psi} \). This meant that no cross-section made of plexiglas was acceptable. Therefore, it was then necessary to consider other materials. Notice that the modulus of steel, \( E_{\text{steel}} = 30 \times 10^6 \text{ psi} \), was larger than that required for all the cross-sections considered. In order to make comparisons of different cross-sections each was evaluated for the same material, steel.

The second analysis was a finite element analysis of the acceptable cross-sections. The cross-sections used were those that utilized the space most efficiently and yet still allowed space for the placement of neon tubes. These cross-sections were then input into the I-DEAS beam element function. Each element was then used to represent the stiffener as a connection of beams with the same cross-section. The stiffener was fixed at the bridge and a force
Figure 2.4 - Cross-section Dimensions
Figure 2.5 - I-DEAS Beam Analysis Model
of 180 lbs was placed at the head of the neck, which represented the summation of the 30 lbs created by each of the six strings. In Figure 2.4 the dimensioned cross-sections are shown. In Figure 2.5 a sample of the beam model created in I-DEAS is shown with deflections indicated, and in Table 2.1 the maximum deflections of all cross-sections are shown.

<table>
<thead>
<tr>
<th>Cross-Section</th>
<th>Max. Displacement (ins)</th>
</tr>
</thead>
<tbody>
<tr>
<td>U-channel</td>
<td>0.048</td>
</tr>
<tr>
<td>Wide - T</td>
<td>0.061</td>
</tr>
<tr>
<td>Inverted - T</td>
<td>0.073</td>
</tr>
<tr>
<td>Curved Inverted - T</td>
<td>0.082</td>
</tr>
<tr>
<td>Rectangle 0.8 x 0.3</td>
<td>0.087</td>
</tr>
<tr>
<td>Rectangle 0.8 x 0.25</td>
<td>0.104</td>
</tr>
<tr>
<td>Tapered I - Beam</td>
<td>0.111</td>
</tr>
</tbody>
</table>

Table 2.1 - Cross-sectional Beam Analysis Results

As it can be seen, deflections are larger than the acceptable deflection criteria of 0.02, but it should be noted that the cross-section used is continuous, and in the final design the cross-section varies, getting larger as the neck approaches the body.
2.3 Two Dimensional Model

In order to determine the internal stiffener dimensions, the external dimensions of the original wooden neck were used along with the assumption of a 0.25 inch wall thickness for the plastic shell. Figure 2.6 shows the wooden neck dimensions and Figure 2.7 shows the dimensions of the two dimensional model that was created and used in both the finite element analysis and the experimental testing. Notice that in this model the number of locations for the strings has been reduced from six to three. This has been done to maintain the planar geometry, and is allowable since the forces of the three strings that were removed were added to the other three. The resulting model is still representative of the three dimensional case for it is symmetrical and the forces in the z-direction would cancel each other.

2.4 Finite Element Model

The dimensions used for this model are those shown in Figure 2.7. The I-DEAS Finite Element Analysis software was used to calculate the results. The following two cases were analyzed;

Case 1 - Linear elements, 180 lbs, Steel, 3 strings
   224 Nodes, 165 Elements.

Case 2 - Linear elements, 90 lbs, Aluminum, 3 strings
   230 Nodes, 168 Elements.
Figure 2.6 - Wooden Neck Dimensions
Figure 2.6 - Wooden Neck Dimensions
Figure 2.7 - Two Dimensional Model Dimensions
Figure 2.7 - Two Dimensional Model Dimensions
2.4.1 Thin Shell Element

The type of element that all the 2-D cases use is the thin shell element. This element is a quadratic element that has a uniform thickness throughout. As stated in the I-DEAS Student Guide,

Thin shell elements can be effectively used for structures with relatively thin walls such as molded plastic or sheet metal parts where bending and in-plane forces are important. [7]

The only drawback to the thin shell element is that it cannot give the stresses that vary through the thickness of the element. For this case it is assumed that those stresses are negligible. Therefore, this choice of element is acceptable for this study.

2.4.2 Restraints

When creating a finite element model it is necessary to designate restraints on the different directions of the model preventing linear or rotational motion in a specified direction. In addition the model must be grounded to avoid rigid body motion (linear or rotational). For the two cases presented in this chapter, restraints were placed at the 3 nodes at the left end of the stiffener near the tailpiece. The closed arrows shown in Figure 2.8, (A) represent the different directions that have been restrained. Note that each node has been restrained in the x, y, and z direction from both translation and rotation.
Figure 2.8 - Case 1, Restraints and Structural Loads
2.4.3 Structural Loads

To simulate the force of the strings, loads were put at various points on the stiffener. Each of the six strings of the guitar were estimated to have a total of thirty pounds of force in it [8]. In case 1, as shown in Figure 2.8, five different node locations were used to model the six strings. An open arrow is used to signify the direction of the force. The locations and forces applied were as follows:

- B - 180.00 lbf in the +x direction at the tailpiece.
- C - 40.49 lbf in the -y direction at the nut.
- D, E, F - 13.49 lbf in the +y direction and 58.46 lbf in the -x direction at each peg location.

For case 2, as shown in Figure 2.9, the number of string connection points was reduced to one, therefore, a total of three load locations exist. The loading and directions for this case were as follows:

- A - 40.49 lbf in the -y direction at the nut.
- B - 40.49 lbf in the +y direction and 175.38 lbf in the -x direction at the peg location.

2.4.4 Material Properties

For the two dimensional models two different materials were used, aluminum and steel. The choice of steel was made and used in case 1 because it was a well known
Figure 2.9 - Case 2, Restraints and Structural Loads
material that had the required modulus of elasticity that the cross sectional analysis stipulated. By keeping the material constant it made it possible to compare the results from different geometric models. As for case 2, this case was used to compare the finite element results with the experimental test results. It was easier to machine the experimental specimen out of aluminum, so the finite element model was run with the aluminum properties to create comparable results. Table 2.2 shows the properties of both aluminum and steel used by I-DEAS.

<table>
<thead>
<tr>
<th></th>
<th>Steel</th>
<th>Aluminum</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modulus of Elasticity (psi)</td>
<td>29.9*10^6</td>
<td>10.1*10^6</td>
</tr>
<tr>
<td>Poissons Ratio</td>
<td>0.290</td>
<td>0.334</td>
</tr>
<tr>
<td>Yield Stress (psi)</td>
<td>36000</td>
<td>14000</td>
</tr>
<tr>
<td>Coefficient of Friction</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>Allow. Stress in Tension (psi)</td>
<td>218000</td>
<td>16000</td>
</tr>
<tr>
<td>Allow. Stress in Comp. (psi)</td>
<td>218000</td>
<td>16000</td>
</tr>
</tbody>
</table>

Table 2.2 - Material Properties of Steel and Aluminum Alloy

2.4.5 Stresses and Displacements

The two cases described in this chapter were created in the I-DEAS finite element pre-processor, run in I-DEAS
and the results were transmitted through the I-DEAS postprocessor.

In case 1, both displacement data and stress data were taken. The displacement direction of interest was that in the y direction, Figure 2.10, shows this displacement with respect to the global x coordinates. Note that the first point on the graph represents the first node at the left side near the tailpiece. Figure 2.11, shows the undeformed and deformed stiffener, note the deformation is grossly exaggerated with the maximum displacement actually being 0.199 inches. As for the stresses, the maximum principal stresses, Figure 2.12, the minimum principal stresses, Figure 2.13, and the maximum shear stresses, Figure 2.14, were recorded. All important information has been gathered and put in Table 2.3. Note that the maximum displacement is lower than 0.2 inches which is the standard deflection of a truss rod guitar. But also remember that our design goal is to get the deflection down to 0.02 inches.

This model and it's results show that the stresses that were previously predicted are negligible and that the critical factor in this design is the displacement. Trends in the locations of high stresses were also predicted by this model. Notice in Figures 2.12 - 2.14 that the largest stresses occur in the body section of the stiffener and at the end of the neck.
Figure 2.10 - Case 1, Displacement in Y-dir
Figure 2.11 - Case 1, Deformed vs Undeformed Stiffener
Figure 2.12 - Case 1, Maximum Principal Stresses
Figure 2.13 - Case 1, Minimum Principal Stresses
Figure 2.14 - Case 1, Maximum Shearing Stresses
In case 2, both displacement data and strain data were recorded. This case was done to use as a comparison to the experimental testing that was done as part of this study. The experimental displacement and strain were to be recorded through the use of dial gages and strain gages (see section 2.5 for details). Figure 2.15, shows the displacement in the y direction, Figure 2.16, shows the strain in the x direction of the top nodes, Figure 2.17, shows the strain in the x direction of the bottom nodes. Table 2.4 shows a summary of specific case 2 data. Comparison of this data with experimental results is described in Section 2.5.4.
Figure 2.15 - Case 2, Displacement in y direction
Figure 2.16 - Case 2, Strain in x direction, Top Nodes
Figure 2.17 – Case 2, Strain in x dir, Bottom Nodes
<table>
<thead>
<tr>
<th>Displacement (ins)</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>@ 18.13 inches in Global x dir</td>
<td>0.272</td>
</tr>
<tr>
<td>@ 15.59 inches in Global x dir</td>
<td>0.226</td>
</tr>
<tr>
<td>@ 10.63 inches in Global x dir</td>
<td>0.154</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Strain (in/in)</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>@ 17.82 inches in Global x dir</td>
<td>-189</td>
</tr>
<tr>
<td>@ 9.94 inches in Global x dir</td>
<td>-237</td>
</tr>
<tr>
<td>@ -3.24 inches in Global x dir</td>
<td>-575</td>
</tr>
<tr>
<td>@ 17.82 inches in Global x dir</td>
<td>127</td>
</tr>
<tr>
<td>@ 10.00 inches in Global x dir</td>
<td>149</td>
</tr>
<tr>
<td>@ -3.18 inches in Global x dir</td>
<td>479</td>
</tr>
</tbody>
</table>

Table 2.4 - Case 2, Displacement and Strain Data

2.5 Experimental Model

As stated earlier in the report, an experimental model has been constructed and tests were performed to verify the two-dimensional finite element model. The work of designing, creating and testing the experimental model and test bench was done by David A. Hult, a senior Lehigh University student. The rest of section 2.5 is a
paraphrased version of the paper, Structural Analysis of a Guitar Neck, by David A. Hult [9].

2.5.1 Test Bench and Specimen

Two types of measurements were of interest; strain in the x direction, and deflection in the y direction. The instruments selected for these measurements were resistance strain gages and mechanical dial indicators.

Figure 2.18, shows the test bench that was created for this testing. It employs a vise, (6), for the rigid fixturing of the specimen (4). The vise is a heavy duty vise effective for any type of specimen to be tested. This inherent flexibility makes future testing capable. The mechanism employs pulleys to insure the correct positioning of the wire cable (3). The load beam and support bar allow the large pulley (2) to hold the cable at a fixed height, at the bridge position, just as in a real guitar. This insures that the loads the specimen endures during the test will effectively mimic the loads of a real guitar. The load beam also permits the smaller pulley (1) to drape the cable over the edge of the work bench where it is connected to a T-bar. The weights to be added will be put onto this T-bar singularly or in pairs. The opposite end of the cable is fastened to the middle hole of the specimen head section by a screw. It should be noted that a notch was
Figure 2.18 - Experimental Apparatus
put at the top of the first fret to simulate the string touching the nut of the guitar.

The following is a list of the used test instruments;

2 Ames 1-inch total range dial indicators, Model 282
1 Ames 2-inch total range dial indicator, Model 2822
7 Micro Measurements, Model EA-13-125BT-120 strain gages
1 Micro Measurements strain gage switch box
1 Micro Measurements P-3500 strain indicator

The placement of strain gages in this test were -3.2, 10.0, and 17.8 inches in the x direction according to the theoretical specimen coordinate system. The gages used were general purpose gages, with a 1/16 inch compact grid for use with aluminum, brass or tin specimens. It was a self temperature compensating gage but an additional compensation gage was used for the test. The adhesive, M-Bond 200, was used to bond the gage to the specimen.

The dial indicators used were purchased with regular points and adjustable backs. The brackets created for use with these allowed a wider range of placement on the specimen. They were bolted to the work bench and were designed to allow for adjustment in the y direction and the z direction as well as rotation in the xz plane. The position requirement of the dial indicators is merely that they read three different points along the length of the
2.5.2 Test Procedure

Step 1: Six strain gages were applied to the specimen in the designated positions: top and bottom surface of the body midsection, top and bottom surfaces of the neck midsection, and the top and bottom surfaces of the neck end. One gage was applied to an unloaded specimen. Lead wires were soldered to tabs, two wires to one side and one wire to the other. The horizontal distance of the center of each grid from neck-body joint was recorded.

Step 2: The vise was positioned so that the specimen would extend over the workbench, parallel to table lines.

Step 3: The load beam was bolted in place, Figure 2.18 (A), and propped by pins and support bar, Figure 2.18 (B).

Step 4: The specimen was fastened in the vice tightly. Making sure the specimen was level, the inside edge of the bridge section was made flush with the edge of the vise. The top surface was set to one thirty-second inches below the top surface of the vise jaw.

Step 5: The cable was threaded through the pulleys and connected to the far end of the neck. The eye end was attached to the specimen head at the middle hole by a bolt with over and under washers.
Step 6: The switch box was connected to the strain gage indicators and individual gages using a quarter bridge circuit. A compensation gage was also connected. Gages were then zeroed.

Step 7: Dial indicators were fastened onto brackets. The indicator tip was pre-compressed in order to contact the middle of the specimen's bottom surface. Some vise and bracket repositioning were required.

Step 8: While the cable was loaded only by the weight of the T-bar, cable height relative to top surface of neck at neck-body joint and neck-head joint were measured and recorded. Making note that heights were equal. If necessary, the cable was repositioned to make the cable parallel to the top surface. The cable height was recorded.

Step 9: After having recorded the initial indicator readings, the cable was loaded with 45 lbs or half load.

Step 10: Strain values were recorded for all active gages. Loaded specimen displacements were recorded as well.

Step 11: The cable was loaded with full load or 90 lbs and strain and displacement values were recorded.

Step 12: The cable was completely unloaded, and measurements of strain and displacement of the unloaded
specimen were retaken. Returning to step 9, the procedure was then repeated for a total of 5 loadings.

Step 13: The specimen was removed from the vise once the test was completed and the procedure was restarted from step 4. This was repeated 5 times.

Notice that a load of 90 lbs was used due to the inability of aluminum to withstand a higher load.

2.5.3 Results and Verification of FEA

For ease of explanation the strain gage and dial indicators were each given names and these are labeled in Figure 2.19. DI indicates a dial gage and SG indicates a strain gage. The finite element results and the experimental results have been collected and put in Table 2.5. Equation 2.2 was used for the calculation of the percent error.

\[
\%e = \frac{|\text{Exp} - \text{FEA}|}{\text{FEA}} \times 100\% \tag{2.2}
\]

where:

\%-e - Percent error
Exp - Experimental result
FEA - Finite element analysis result

50
Figure 2.19 - Strain Gage & Dial Indicator Locations
<table>
<thead>
<tr>
<th></th>
<th>Ex. Results</th>
<th>FEA Results</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Displacement (ins)</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DI1 @ 18.1 in</td>
<td>0.289</td>
<td>0.272</td>
<td>7.48</td>
</tr>
<tr>
<td>DI2 @ 15.6 in</td>
<td>0.245</td>
<td>0.226</td>
<td>9.48</td>
</tr>
<tr>
<td>DI3 @ 10.6 in</td>
<td>0.162</td>
<td>0.154</td>
<td>6.47</td>
</tr>
<tr>
<td><strong>Strain (in/in)</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SG1 @ 17.8 in</td>
<td>-223</td>
<td>-189</td>
<td>17.8</td>
</tr>
<tr>
<td>SG2 @ 9.9 in</td>
<td>-241</td>
<td>-237</td>
<td>1.48</td>
</tr>
<tr>
<td>SG3 @ -3.2 in</td>
<td>-562</td>
<td>-575</td>
<td>2.18</td>
</tr>
<tr>
<td>SG4 @ 17.8 in</td>
<td>135</td>
<td>127</td>
<td>6.22</td>
</tr>
<tr>
<td>SG5 @ 10.0 in</td>
<td>159</td>
<td>149</td>
<td>7.07</td>
</tr>
<tr>
<td>SG6 @ -3.18 in</td>
<td>459</td>
<td>479</td>
<td>4.15</td>
</tr>
</tbody>
</table>

Table 2.5 - 2-D Experimental and FEA Results (Case 2)

The percent error ranges from 1.48 to 17.8, with an average of 6.93 percent error. It can be concluded from these results that the finite element model for the two dimensional testing is within an acceptable accuracy.
3. THREE DIMENSIONAL MODELING

3.1 Design Geometry

The two dimensional analysis of the various three dimensional cross sections has been described in section 2.2. These cross-sections were constructed to fill the largest amount of available space in the neck as possible and still leave room for one or two neon tubes. Based on the results of the two dimensional finite element modeling and finite element analysis three cross-sections have been considered. Figure 3.1 shows the three different cross-sections that were named, TapI, TapT, and TapU, because of the fact that they were tapered representations of the letter used in their name. A multi-viewed drawing of the TapI model is represented in Appendix B. The other two, TapT and TapU, were exactly the same as the TapI except for the location of the channel of the neon tubes in the neck.

3.2 Finite Element Model

A full three dimensional solid FEM/FEA has been developed to obtain accurate deflections, stresses and strains created by the string tension. The I-DEAS Finite Element Analysis software was used to calculate the
Figure 3.1 - TapI, TapU, and TapT Designs
displacement, stresses, and strains for the following cases:

Case 1 - Fullneck, Linear Elements, 3 strings
Case 2 - Fullneck, Parabolic Elements, 3 strings
Case 3 - TapI, Linear Elements, 3 strings
Case 4 - TapI, Parabolic Elements, 3 strings
Case 5 - TapT, Linear Elements, 3 strings
Case 6 - TapT, Parabolic Elements, 3 strings
Case 7 - TapU, Linear Elements, 3 strings
Case 8 - TapI, Linear Elements, torsion at neck.

For all the cases the material used was steel. The structural loads used for all cases are explained in Section 3.2.3. Notice that in cases 1-7 the number of strings was reduced from six to three, for modeling purposes. The force of each string in the model was twice that of what a normal six string guitar generates. This was allowed because of the symmetric geometry of the head of the neck, the strings that were adjacent to each other were combined into one string that was located at the centerline, as shown in Figure 3.2.

3.2.1 Solid Element

The type of element that all the 3-D cases use is the solid brick element. This element is an eight cornered block with either eight or sixteen nodes, depending on
whether it is a linear or parabolic element respectively. Each node of the solid element has six degrees of freedom, three translation and three rotation. Due to the changing cross section and the desire to see if the stresses were changing through the cross section, solid elements were the only choice. Due to the number of nodes and degrees of freedom of solid elements, a considerable amount of computer memory is required. In addition the process of obtaining results in the three dimensional cases take approximately five times longer than for the two dimensional cases.

3.2.2 Restraints

When creating a finite element model it is necessary to designate restraints on the different directions of the model preventing linear or rotational motion in a specified direction. In addition the model must be grounded to avoid rigid body motion (linear or rotational). For all the cases in this chapter, restraints were placed at 6 nodes at the left end of the stiffener near the tailpiece. The closed arrows shown in Figure 3.2 (A) represent the directions that have been restrained. Each node was restrained in the x, y, and z direction from both translation and rotation.
Figure 3.2 - 3-D Restraints and Structural Loads
3.2.3 Structural Loads

To simulate the strings, loads were put at various points on the stiffener. Each of the six strings of the guitar were estimated to have a total of thirty pounds of force in it. For cases 1-7 of this chapter the same loading conditions were used. As shown in Figure 3.2, five different node locations were used to model the six strings. An open arrow is used to signify the direction of the force. Notice that each string touches the stiffener at three locations, the tailpiece, the nut and the peg. To calculate the force that was generated at each of these points of contact simple static analysis techniques were used. At the tailpiece (B) and the nut (C) the forces generated by all six strings were combined into one force, but for the peg locations (D, E, F) only the forces from two strings were combined at each location. The locations and forces applied were as follows:

B - 180.00 lbf in the +x direction at the center of the tailpiece.

C - 40.49 lbf in the -y direction at the center of the nut.

D, E, F - 13.50 lbf in the +y direction and 58.46 lbf in the -x direction at each peg location.

For case 8, this case was being used to test the resistance to the user placing a torque at the top of the
neck. As shown in Figure 3.3 the forces applied were the following:

A - 50 lbf in the $+y$ direction at the first fret.

B - 50 lbf in the $-y$ direction at the first fret.

Forces A and B are two inches apart therefore creating a torque of 100 in-lbs. This value was determined as the maximum torque that a user could instill on the guitar during use.

3.2.4 Material Properties

For the three dimensional models only one material was used, steel. The choice of steel was made because it was a well known material that had the required modulus of elasticity that the cross sectional analysis stipulated. By keeping the material constant it made it easier to compare the different models and to reference the models in chapter two. Table 2.2, in section 2.4.4, shows the properties of steel that were used by I-DEAS.

3.2.5 Stresses and Displacements

The eight cases described in this chapter were created in the I-DEAS finite element pre-processor, run by the I-DEAS processor, and the results were transmitted through the I-DEAS post-processor.
Figure 3.3 - Case 8, Structural Loads
3.2.5.1 String Tension Cases

In all string tension cases, Cases 1-7, both displacement and stress data were taken. The displacement direction of interest was in the y direction. As an example Figure 3.4 shows this displacement with respect to the global x coordinates for case 3. Note that the first point on the graph represents the first row of nodes at the left side of the stiffener near the tailpiece. All the string tension cases show a similar deformation pattern, therefore, output results have been restricted to case 3 only. (A complete output of each case has been included in Appendix C.)

As for the stresses, the maximum principle stresses, the minimum principle stresses and the maximum shear stresses were recorded. Figures 3.5 - 3.7 show the stresses at all nodes of the stiffener for the three types of recorded stresses. All of the string tension case results have been collected and included in Table 3.1.

Note that the displacement varies between 0.041 and 0.065 inches. These numbers are considerably lower than that of the two dimensional case, this is due to the added material in the cross-section. The displacements here are almost 4 times less than that of the conventional wooden truss rod system. Tests were conducted on different conventional wooden truss rods (see Section 3.3.).
Figure 3.4 - Case 3, Displacement in the Y Direction
Figure 3.5 - Case 3, Maximum Principal Stresses
Figure 3.6 - Case 3, Minimum Principal Stresses
Figure 3.7 - Case 3, Maximum Shearing Stresses
After presenting these results it was determined that the best aesthetic and practical design of the three different cross-sections was the TapI. This was determined for its ability to incorporate two neon tubes in a highly desired area of the neck. Even though this is the worst of all the cases its deflection and stresses are still within the acceptable range.

3.2.5.2 User Torsion Case

In case 8, the TapI neck with linear elements previously used in case 3 was adapted to this case. For a full description of restraints and structural loads see Section 3.2.2 and 3.2.3 respectively. This case was used to determine what sort of deflections the user could instill on the stiffening member due to torsion. It was assumed that the greatest force a player could instill on the guitar would be 50 lbs at the top of the neck, the

<table>
<thead>
<tr>
<th>Case Number</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacement</td>
<td>0.041</td>
<td>0.038</td>
<td>0.065</td>
<td>0.063</td>
<td>0.052</td>
<td>0.052</td>
<td>0.043</td>
</tr>
<tr>
<td>Max. Princ. Stress</td>
<td>1850</td>
<td>2110</td>
<td>2280</td>
<td>2290</td>
<td>2190</td>
<td>2380</td>
<td>1640</td>
</tr>
<tr>
<td>Max. Shear Stress</td>
<td>2200</td>
<td>3010</td>
<td>2400</td>
<td>3130</td>
<td>2310</td>
<td>2690</td>
<td>1580</td>
</tr>
</tbody>
</table>

Table 3.1 - Cases 1-7, Displacement and Stresses
weakest part of the stiffener. Table 3.2 shows the results of the displacement in the y and z directions as well as the maximum principal stress, minimum principal stress and the maximum Von-Mises stress. Noticing that the maximum displacement in the y direction is 0.0053 inches and the maximum displacement in the z direction is 0.0052 inches, it can be said that the amount of twist caused by the user will be small enough that it can be neglected. The stresses are also so minimal that they can be neglected as well.

<table>
<thead>
<tr>
<th>Case 8</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacement in Y direction (inches)</td>
</tr>
<tr>
<td>Displacement in Z direction (inches)</td>
</tr>
<tr>
<td>Max. Principal Stress (psi)</td>
</tr>
<tr>
<td>Min. Principal Stress (psi)</td>
</tr>
<tr>
<td>Max. Von Mises Stress (psi)</td>
</tr>
</tbody>
</table>

Table 3.2 - Case 8, Displacement and Stresses

3.2.6 Linear Element vs. Parabolic Element

When choosing what type of element that should be used it was necessary to determine between a linear eight noded solid brick element or the parabolic sixteen noded solid brick element. The advantage of the eight noded element is
that it can most of the time give adequate results while taking up less matrix space therefore making calculations faster. The disadvantage to the linear element is that it is not very effective in determining shear stress due to bending. In comparison the parabolic element takes up much more matrix space but allows for a more accurate calculation of shear stresses.

It was in our best interest to determine which element would serve our means the best. For the Fullneck, TapI, and TapT designs both a linear element and a parabolic element mesh was constructed and tested under the same restraints and loading. Table 3.3 shows the results and the percent difference between the two. Percent difference was calculated with the following equation:

$$\%\text{diff} = \frac{\text{Linear}-\text{Parabolic}}{\text{Parabolic}} \times 100\%$$  \hfill (3.1)

When conducting this test it took 3 times longer for the parabolic meshes to solve than the linear meshes. The percent difference of the maximum displacement was between 0 and 7.89 percent. The percent difference of the maximum shear ranged from 14.0 and 26.8 percent. It was noticed that the maximum shear from both linear and parabolic elements was more than a factor of ten less than the allowable shear of steel. Therefore, any inaccuracies in shear would not be critical. As a result, it was felt that
the linear element was allowable for it produced comparable results and quicker solution time.

<table>
<thead>
<tr>
<th></th>
<th>Linear</th>
<th>Parabolic</th>
<th>% diff</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Tapfull</strong></td>
<td>Disp. (ins)</td>
<td>0.041</td>
<td>0.038</td>
</tr>
<tr>
<td></td>
<td>Max. Shear (psi)</td>
<td>2200</td>
<td>3010</td>
</tr>
<tr>
<td><strong>TapT</strong></td>
<td>Disp. (ins)</td>
<td>0.052</td>
<td>0.052</td>
</tr>
<tr>
<td></td>
<td>Max. Shear (psi)</td>
<td>2310</td>
<td>2690</td>
</tr>
<tr>
<td><strong>TapI</strong></td>
<td>Disp. (ins)</td>
<td>0.065</td>
<td>0.063</td>
</tr>
<tr>
<td></td>
<td>Max. Shear (psi)</td>
<td>2400</td>
<td>3130</td>
</tr>
</tbody>
</table>

Table 3.3 - Linear Element vs. Parabolic Element Results

### 3.2.7 String Distance to Fretboard

One of the reasons why the displacement of the neck is such a critical factor is because the distance from the string to the fretboard is desired to be constant along the neck. If the distance varies the result can be buzzing in the frets or poor action. When the neck deflects, the distance between the string and the fretboard is changed. Since the displacement of the deflecting stiffener is known from the three dimensional finite element model, it was desired to calculate what the resulting string-fretboard displacement would be. To accomplish this a fortran
program was created. The program used simple geometry to calculate the distance. Four different cases were input into the program, cases one, three, five, and seven. These were the linear element cases for Tapfull, TapI, TapU, and TapT. Figure 3.8 shows the resulting displacement of the string in the y direction due to the displacement of the neck. The coordinate system used is the same as that used for the I-DEAS models. The origin is placed at the connection point of the neck and the body. Figure 3.9 shows the location on the neck that the maximum displacement occurs. Notice that, as would be expected, the larger cross section cases result in smaller deflections. Another point worth noting is that the gap is the largest in the middle of the neck, while the largest stiffener deflections occurred at the nut.

3.3 Experimental Comparison

To compare the three dimensional case results it was necessary to do some experimental testing on conventional wooden truss rod supported necks. Three different necks were tested; a Neo Products neck, a Martin neck D-18MB, and a Sekova neck.
Figure 3.8 - String Fretboard Displacement

Displacement in Y dir

0.020

0.015

0.010

0.005

0.000

-15.00 -10.00 -5.00 0.00 5.00 10.00 15.00 20.00

X Coordinate along String

x=7.32
y=0.0158

úmeros

x=5.99
y=0.0090

x=5.99
y=0.0084

***** TapI
x×x×x TapFull
+++ TapT
○○○○○ TapU
Figure 3.9 - Location of Maximum String Deflection
3.3.1 Test Specimen

Each specimen was tested with the test bench that was used for the two dimensional experimental testing, in section 2.5.1. The Neo Products neck had a Gibson-style truss rod, the Martin neck had a Martin style truss rod and the Sekova neck had a traditional truss rod. Explanations of different styles of truss rods can be found in Section 1.3.

The test bench was altered only slightly, meaning instead of one cable being used to hold the weights two were employed. The cables were connected to the two middle peg locations and run parallel to each other through the pulley system. The load that was used was 180 lbs. The rods were tested with the maximum allowable tension and without tension in the rods.

3.3.2 Results and Comparison

A total representation of the results can be seen in Appendix D. In Table 3.4 the mean displacement is shown for the three different necks as well as for the finite element TapI case. And in Figure 3.10 this data is graphically represented. Note that all necks that were tested produced results that were higher than that of the finite element results. Therefore it can be safely assumed that if the material that is chosen to make the stiffener
Figure 3.10 - Experimental vs. Finite Results
is stiffer than steel than it too will produce results better than any truss rod.

<table>
<thead>
<tr>
<th>Location of Gage in X Dir (in)</th>
<th>Dial Gage 1</th>
<th>Dial Gage 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tap1 Results (in)</td>
<td>0.0090</td>
<td>0.0450</td>
</tr>
<tr>
<td>Neo Neck - no tension (in)</td>
<td>0.0181</td>
<td>0.0905</td>
</tr>
<tr>
<td>Neo Neck - max. tension (in)</td>
<td>0.0180</td>
<td>0.0685</td>
</tr>
<tr>
<td>Martin Neck - no tension (in)</td>
<td>0.0422</td>
<td>0.1078</td>
</tr>
<tr>
<td>Martin Neck - max. tension (in)</td>
<td>0.0360</td>
<td>0.0475</td>
</tr>
<tr>
<td>Sekova Neck - no tension (in)</td>
<td>***</td>
<td>0.2266</td>
</tr>
</tbody>
</table>

Table 3.4 - Three Dimensional Experimental Results
4. Manufacturing Method

4.1 Existing Manufacturing Methods

Many different types of manufacturing methods exist today. For metal materials, there are casting, welding, forming, and computer numerically controlled (CNC) machining, to name a few. For plastic materials, including composites, there are casting, injection molding, thermoforming, and hand layup to name a few. Considering the geometric constrictions of the stiffener design, only a few manufacturing methods would be capable of making the stiffening member. Among these methods are casting, CNC machining, injection molding, and hand layup. To help understand the motives in the selection of the method that has been recommended, it is first necessary to briefly describe all the choices.

4.1.1 Casting

Casting is a general name for many different types of casting. Sand casting, shell molding, plaster molding, investment casting, are die casting, are just a few of the different types of casting. The similar thread that runs through all types of casting is that by some method a cast
or mold is created that is a female representation of the part that is to be produced. Into this mold the material in a liquid state is poured into the mold and allowed to cool. Figure 4.1 shows a sectional view of a typical casting mold design [10]. Most all metals and epoxy or nylon plastics can be used. One major concern in using this method is to make sure that there are no large changes in cross sections for air pockets will form if there are, decreasing the strength of the part. Careful placement of the parting line can sometimes alleviate this problem.

4.1.2 CNC Machining

CNC machining is just like any other type of machining but it is controlled by a computer instead of by a user, therefore insuring a more precise and repeatable part. The designer uses software to create the tooling patterns, tool choice, speeds, and feeds. This information is then sent from the computer to the machine and the part is cut from a block of material. Many different types of machining procedures can be numerically controlled, including, milling, drilling, and even work done on a lathe. One disadvantage to this procedure is the large amount of material that is lost due to the cutting. All metals and plastics can be machined, but composite machining should be limited to chopped fiber composites only. This is due to
Figure 4.1 - Section View of a Casting Mold
large loss in strength and shortened lifetime in machined continuous fiber composites.

4.1.3 Injection Molding

Injection molding is a procedure used for plastic parts only. Figure 4.2 [10] shows a typical injection molding machine and the reciprocating screw injection system. The general procedure followed in injection molding is as follows. Pellets of the plastic compound are placed into a hopper and then fed into an extrusion screw where it is chopped finer and then melted by the time it reaches the end of the screw. Then it is forced into the mold which is at the end of the screw. The mold is allowed to cool and the molded part is then ejected. Both pure plastics and chopped fiber impregnated plastics can be injected. Two main concerns with using fiber impregnation are fiber orientation after cooling, and weld line adhesion. The problem with fiber orientation is that the flow of the plastic can orient the fibers in one direction, if this direction is not a desired direction of increased strength then this will create a problem. As for weld lines, see Figure 4.3 [10], this is also related to fiber orientation but occurs when two flow fronts meet and then the fibers turn 90 degrees. If proper gating and placement of wells is done then both problems can be avoided.
Figure 4.2 - Injection Molding Machine
Weld Line Formation

Resin injected into mold

Fibers align parallel to resin flow

Mold configuration breaks injected resin into two flow paths

Weld line formed when flow fronts meet and fibers turn 90°

Figure 4.3 - Weld Line Formation
4.1.4 Composite Layup

A continuous fiber composite layup consists of three procedures. First, a female mold is to be created. This can be done by machining, casting or however you choose. Second, in each mold, layers of fiber impregnated sheets are laid down in various directions, depending on the desired directions of increased strength. Finally once the mold is completely filled it is then placed in an autoclave where it is subjected to high temperature and pressure. The part is then removed and any excess material is cut off. Some of the disadvantages of this are the extremely long production time and the inability to machine the part after layup. On the other hand an advantage is that strength can be placed in desired directions. Another advantage to continuous fiber layups is that they have higher strengths than all plastics and almost all metals.

4.2 Mold Design Considerations

Out of the four manufacturing techniques three require a mold to be used, therefore a description of mold design requirements follow. Many different criteria need to be considered when designing a mold such as location of parting line, inserts, draft angles, and porting.

The parting line is the line that divides the top and the bottom halves of the mold. If the parting line is
placed correctly it can minimize machining after the part is made, minimize the work necessary in making the mold, and can eliminate porosity from occurring.

Inserts are any metal pieces that are added to the mold to prevent the material from taking up that space. Inserts are usually necessary for internal cavities or complex holes that could not be incorporated into the mold due to its geometry. Inserts make the mold more complex and expensive yet if no other method is available it is then necessary.

As for the mold for the stiffener that is described in this thesis, no inserts will be necessary, yet multi-porting will. The reason that no inserts will be necessary for the peg location is that each half of the mold will be able to incorporate half of the peg, this still will allow for easy removal of the part. Figure 4.4 shows a preliminary design of a mold, generated in I-DEAS solid modeling, that could be used to create the stiffener. The consultation of expert mold designers has been undertaken and the recommendations from their analysis still awaits.

Draft angles are angles of relief on walls that are perpendicular to the parting plane. This allows the part to be removed from the mold easily. Draft angles vary from 1 - 10 degrees depending on the type of molding procedure being used.
Figure 4.4 - Stiffener Mold Design
Porting is the placement of port or gates that the material will flow into the mold from. If many well located ports are used the probability of porosity or weld lines occurring is greatly reduced. Figure 4.5 [10] shows some of the various types of gating.

4.2.1 Neo Molding Technique

At present, Neo Products Inc. uses a plastic mold to make the neck of their violin. It is possible that a similar method could be used to create the stiffener for the guitar. If this were done the procedure for creating the mold and stiffener would be as follows:

1. A scale model is created out of an engineering material such as HIS (high impact styrene), ABS (acrylonitrile-butadiene styrene), or a similar material. From a block the material is formed into the scale model by use of hand tools and machining.

2. The model is then placed on a wooden board where it is glued down to. This will represent the lid. Walls are then created several inches from the model totally surrounding it. Silicon rubber (i.e. RTV reticulated vinyl) is then poured into the box and allowed to cure for 12 hours.

3. Once the rubber has cured, the walls are removed and the model is removed from the rubber mold. Any
A single, distinct weld line is produced opposite the gate.

Dissipated weld lines are produced between each pair of runners.

Distinct weld lines that are stronger than that of the single gate are produced between each pair of gates.

Weld lines are totally eliminated.

Figure 4.5 - Types of Porting
4. If any fiber reinforcement is to be used, it is laid into the mold at this time. A 1/4 inch piece of plexiglass is then sprayed with a separation agent and attached to the mold, which will serve as a lid.

5. A vent and an inlet channel is then cut into the mold, to allow for the material to be poured into the mold and for the air and any excess material to exit.

6. The mold is then placed into a pressure tank with the vent and sprue in the up position. The epoxy resin or similar casting material is then mixed and poured into the mold. The pressure tank is sealed and the pressure is raised to ninety pounds per square inch. Allow to cure for 4 to 12 hours depending on material.

7. The pressure is then removed and the mold is removed from the tank. The lid is removed and the casting is then removed by flexing the mold. The vent, sprue, and any excess flash are then trimmed away.

8. The casting is complete unless any painting is necessary for aesthetic reasons.

4.3 Material Properties

Taking into consideration the results that were found in chapter two and three, stating that a modulus of
elasticity was required to be as high as if not higher than steel, only a few metals and plastics could be chosen from. Among the acceptable metals are: all steels, tungsten, and wrought iron. Among the plastics none are acceptable by themselves unless strengthened by composite fibers. The acceptable composites are: graphite fibers, graphite whiskers, carbon-steel fibers, beryllium fibers, boron fibers, silicon carbide fibers, chromium whiskers, iron whiskers, nickel whiskers, aluminum oxide whiskers, beryllium oxide whiskers, and boron carbide whiskers. Table 4.1 shows all the acceptable materials and their respective modulus of elasticity, density, and tensile strength [10,11]. Make special note that of the materials described only carbon fibers, graphite fibers, and carbon steel fibers come in a chopped form. This is important if the fibers are to be poured or injected into a cast or mold with a matrix material, for this can only be done with chopped fibers.

4.4 Recommended Material and Manufacturing Method

Note that this recommendation is only one of the possible choices of material and manufacturing methods that would work for the stiffener design.

The recommended material is graphite chopped fibers imbedded in any injectable plastic. When considering all
<table>
<thead>
<tr>
<th>Material</th>
<th>Modulus of Elast. (Mpsi)</th>
<th>Density lb/in³</th>
<th>Tensile Strength (ksi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Graphite whiskers</td>
<td>102</td>
<td>0.060</td>
<td>2850</td>
</tr>
<tr>
<td>Boron carbide whiskers</td>
<td>70</td>
<td>0.091</td>
<td>2000</td>
</tr>
<tr>
<td>Silicon carbide whisker</td>
<td>70</td>
<td>0.116</td>
<td>3000</td>
</tr>
<tr>
<td>Aluminum oxide whiskers</td>
<td>62</td>
<td>0.143</td>
<td>3000</td>
</tr>
<tr>
<td>Graphite fibers</td>
<td>55</td>
<td>0.068</td>
<td>340</td>
</tr>
<tr>
<td>Boron fibers</td>
<td>53</td>
<td>0.081</td>
<td>475</td>
</tr>
<tr>
<td>Tungsten</td>
<td>51</td>
<td>0.699</td>
<td>500</td>
</tr>
<tr>
<td>Beryllium oxide whisker</td>
<td>50</td>
<td>0.103</td>
<td>1900</td>
</tr>
<tr>
<td>Beryllium fibers</td>
<td>35</td>
<td>0.660</td>
<td>185</td>
</tr>
<tr>
<td>Chromium whiskers</td>
<td>35</td>
<td>0.260</td>
<td>1290</td>
</tr>
<tr>
<td>Nickel whiskers</td>
<td>31</td>
<td>0.324</td>
<td>560</td>
</tr>
<tr>
<td>Iron whiskers</td>
<td>29</td>
<td>0.283</td>
<td>1900</td>
</tr>
<tr>
<td>Steel - high carbon</td>
<td>30</td>
<td>0.284</td>
<td>200</td>
</tr>
<tr>
<td>Carbon steel fibers</td>
<td>30</td>
<td>0.280</td>
<td>400</td>
</tr>
<tr>
<td>Iron - wrought</td>
<td>29</td>
<td>0.264</td>
<td>40</td>
</tr>
</tbody>
</table>

Table 4.1 - Material Properties
the different types of materials that could be used it was noticed that the graphite chopped fibers were readily available and provided strength and stiffness that ranked in the top five of the available materials. Graphite is widely used in today's marketplace for composite layups of airplane wings, tennis rackets, golf clubs, and even stiffening guitar necks. It has also been shown that graphite will not transmit any unwanted vibrations into the pickups. The density of graphite is only 0.068 lbs/in³, where most woods are approximately 0.02 lbs/in³. This will make the neck heavier than most wooden necks yet it will be much lighter than if a metal were to be used.

The choice of material was made in conjunction with the choice of the manufacturing method. The method of manufacturing that is recommended is injection molding. This is recommended because it will produce an almost finished part right from the mold in a very short time, each mold will provide at least 10,000 parts, and little material will be lost for any excess can be reused.

The other ways that a graphite stiffener can be made is by a composite layup or by machining. Both alternatives are very time consuming as well as costly. Large amounts of material will be scrapped in both methods.

If a steel were chosen then the method for manufacture would be casting. The main drawback to this method is that
steel has a relatively low strength compared to most composites and is at times three times heavier than some composites. The cost of making the mold is cheaper than in injection molding yet it will only produce a maximum of 10,000 parts.
5. CONCLUSIONS

5.1 Conclusion of Study

The main objective of this thesis was to determine if there are viable designs that would provide adequate support for a plastic guitar where neon is incorporated into the neck. This investigation was completed by using finite element analysis and experimental testing on a statically loaded model. The results have shown that many various designs are possible, and these are stronger and stiffer than the conventional truss rod system used in the industry today. The design of choice, TapI, has passed the structural analysis testing as well as providing an aesthetically pleasing look, that fits with the NEO Products Inc.'s design look.

The area of the guitar that is most vulnerable to large deflections and high stresses is the top of the neck before the headpiece. This is due to the decreased cross-section. Therefore, if any modifications were to be made to the design it would be suggested that the cross-section at that point be increased to improve the strength.

The fully plastic guitar design will allow for an increase in production, for all parts incorporated in the
design will be machine-made and not man-made. Thus, the
time to make, assemble and package the guitar will be
reduced. The decrease in production time will help keep
product cost low. Due to the fact that no parts need to be
hand made the number of skilled workers necessary to
assemble the guitar will be reduced, also reducing the cost
of the final product.

The design presented offers many future capabilities.
The incorporation of neon is but one, anything that the
mind can think of putting inside the guitar, this design
will allow. This design brings to the music industry an
entirely new method of manufacturing guitars, and will set
the ground work for musical instrument designs to come.

5.2 Future Work

Considering that this work is only the design phase of
the entire project, many more steps will be necessary
before production can begin.

First, an injection molding facility that will inject
chopped fiber composites needs to be identified. Upon
identification any specifications of the machine to be used
should be secured and a mold for the TapI design should be
made. A short run of parts should then be made.

Second, the parts created should then be tested in the
test bench that has been used to initially determine the
experimental two dimensional and three dimensional test results. Testing should consist of static loading tests where deflection, stresses and creep are analyzed.

Third, upon approval of the test results, a fully assembled guitar should be made. Any other parts that need to be modified or created should also be done. Once assembled, testing for the guitars acoustic ability should follow.

Finally upon approval of all results the guitar then will be ready for manufacturing.
REFERENCES


<table>
<thead>
<tr>
<th>APPENDIX</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Neo Products Inc. Info. Booklet</td>
<td>97</td>
</tr>
<tr>
<td>B</td>
<td>Guitar Geometry</td>
<td>100</td>
</tr>
<tr>
<td>C</td>
<td>Three Dimensional Finite Results</td>
<td>107</td>
</tr>
<tr>
<td>D</td>
<td>Three Dim. Experimental Results</td>
<td>132</td>
</tr>
</tbody>
</table>
APPENDIX A. - Neo Products Inc. Information Booklet

Used for promotional purposes, the booklet shown in Figures A.1 and A.2, provides a visual representation of the varied musical instruments that Neo Products Inc. have on the market today. In Figure A.1, on the left are shown the aquarium guitar and violin, and on the right are the standard neon guitar and violin. In Figure A.2, on the left the NeoVarius violins™ are shown, and on the right from top to bottom are the Seedless guitar™, the Chameleon™/Greenback™ guitar and the Gumball guitar.
APPENDIX B - Guitar Geometry

This appendix provides drawings of the major parts used in the proposed design. No dimensions are presented due to confidentiality of this information. In Figure B.1, an exploded view of all the parts of the guitar are presented and labeled. In Figure B.2, an assembly view is shown, representing all the parts of the guitar and where they are placed after assembly. In Figure B.3 and B.4, the TapI version of the stiffener is shown. In Figure B.5, a drawing of the neck is shown. And finally in Figure B.6, the body is shown.
Figure B.1 - Exploded View of Entire Guitar
Figure B.1 - Exploded View of Entire Guitar
Figure B.2 - Assembly View of Entire Guitar
Figure B.3 - TapI Drawing (1 of 2)
SECTION A-A

SECTION B-B

SECTION C-C

Figure B.4 - TapI Drawing (2 of 2)
Figure B.5 - Neck Drawing
Figure B.6 - Body Drawing
This appendix contains the displacement and stress data recorded for all cases presented in Section 3.2 except cases 3 and 8. Case 3 results are presented and explained in Section 3.2.5.1. and case 8 results can be found in Section 3.2.5.2. For each case, displacement, maximum principal stress, minimum principal stress, and maximum shear stress are shown in graph form. Each point represents the value at a specific node. Due to the vast amount of nodes and therefore the vast amount of results at each cross section, only the maximum value at each cross section has been recorded.

The following figures correlate to the designated case:

Case 1, Fullneck, Linear Element  -  Figures C.1-C.4
Case 2, Fullneck, Parabolic Elem.  -  Figures C.5-C.8
Case 4, TapI, Parabolic Element  -  Figures C.9-C.12
Case 5, TapT, Linear Element  -  Figures C.13-C.16
Case 6, TapI, Parabolic Element  -  Figures C.17-C.20
Case 7, TapU, Linear Element  -  Figures C.21-C.24
Figure C.1 - Case 1, Max. Displacement in Y Direction
Figure C.2 - Case 1, Maximum Principal Stresses
Figure C.3 – Case 1, Minimum Principal Stresses
Figure C.4 - Case 1, Maximum Shearing Stresses
Figure C.5 - Case 2, Max. Displacement in y Direction
Figure C.6 - Case 2, Maximum Principal Stresses
Figure C.7 - Case 2, Minimum Principal Stresses
Figure C.8 - Case 2, Maximum Shearing Stresses
Figure C.9 - Case 4, Max. Displacement in Y Direction
Figure C.10 - Case 4, Maximum Principal Stresses
Figure C.11 - Case 4, Minimum Principal Stresses
Figure C.12 - Case 4, Maximum Shearing Stresses
Figure C.13 - Case 5, Max. Displacement in Y Direction
Figure C.14 - Case 5, Maximum Principal Stresses
Figure C.15 - Case 5, Minimum Principal Stresses
Figure C.16 - Case 5, Maximum Shearing Stresses
Figure C.17 - Case 6, Max. Displacement in Y Direction
Figure C.18 - Case 6, Maximum Principal Stresses
Figure C.19 - Case 6, Minimum Principal Stresses
Figure C.20 - Case 6, Maximum Shearing Stresses
Figure C.21 - Case 7, Max. Displacement in Y Direction
Figure C.22 - Case 7, Maximum Principal Stresses
Figure C.23 - Case 7, Minimum Principal Stresses
Figure C.24 - Case 7, Maximum Shearing Stresses
APPENDIX D. - Three Dimensional Experimental Results

As explained in Section 3.3, three specimens were tested. The three specimens were a Neo Products neck, a Martin neck and a Sekova neck. The Neo neck and the Martin neck were tested with both tension and no tension in the truss rod. In Table D.1, the results of the testing of the Neo neck are shown. Notice that the mean, standard deviation and the signal to noise ratio has been calculated for each set of data. The equations used for each of these calculations follows;

\[
\mu = \frac{\sum_{j=1}^{n} x_j}{n} \quad (D.1)
\]

where:
\( \mu \) = mean or average of data
\( x \) = value of the data point
\( j \) = counter signifying which data point
\( n \) = total number of data points

\[
\sigma = \left[ \left( \frac{\sum_{j=1}^{n} x_j^2}{n} - \left( \frac{\sum_{j=1}^{n} x_j}{n} \right)^2 \right) \frac{1}{n-1} \right]^{1/2} \quad (D.2)
\]

where:
\( \sigma \) = standard deviation
All other variables are same as for Equation D.1.
\[ SNA = 10 \log_{10} \frac{\sigma}{\mu} \quad \text{(D.3)} \]

where:
\[ SNA = \text{signal to noise ratio} \]

Table D.2 shows the data recorded and the calculations from the testing of the Martin neck. While Table D.3 shows the results from testing the Sekova neck. Note that all tables use Equations D.1-3 for the necessary calculations.
### Neo Neck - No Tension in the Rod

<table>
<thead>
<tr>
<th>Dial Gage 1</th>
<th>Dial Gage 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>No Load</td>
<td>No Load</td>
</tr>
<tr>
<td>180 lbs</td>
<td>180 lbs</td>
</tr>
</tbody>
</table>

<p>| | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.018</td>
<td>0</td>
<td>0.089</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0.018</td>
<td>0</td>
<td>0.09</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0.018</td>
<td>0</td>
<td>0.09</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0.018</td>
<td>0</td>
<td>0.091</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0.018</td>
<td>0.0005</td>
<td>0.09</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0.018</td>
<td>0</td>
<td>0.091</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0.019</td>
<td>0.001</td>
<td>0.09</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0.018</td>
<td>0.001</td>
<td>0.091</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0.018</td>
<td>0</td>
<td>0.091</td>
<td></td>
</tr>
</tbody>
</table>

---

<table>
<thead>
<tr>
<th>Mean</th>
<th>0</th>
<th>0.0181</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.0035</td>
<td>0.0905</td>
</tr>
<tr>
<td>Std Dev</td>
<td>0.0003</td>
<td>0.0045</td>
</tr>
<tr>
<td>SNA ERR</td>
<td>17.8057</td>
<td>1.09144</td>
</tr>
</tbody>
</table>

### Neo Neck - Maximum Tension in the Rod

<table>
<thead>
<tr>
<th>Dial Gage 1</th>
<th>Dial Gage 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>No Load</td>
<td>No Load</td>
</tr>
<tr>
<td>180 lbs</td>
<td>180 lbs</td>
</tr>
</tbody>
</table>

<p>| | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.018</td>
<td>-0.023</td>
<td>0.068</td>
<td></td>
</tr>
<tr>
<td>-0.001</td>
<td>0.018</td>
<td>-0.024</td>
<td>0.068</td>
<td></td>
</tr>
<tr>
<td>-0.001</td>
<td>0.0175</td>
<td>-0.024</td>
<td>0.068</td>
<td></td>
</tr>
<tr>
<td>-0.001</td>
<td>0.018</td>
<td>-0.024</td>
<td>0.068</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0.018</td>
<td>-0.022</td>
<td>0.069</td>
<td></td>
</tr>
<tr>
<td>-0.001</td>
<td>0.018</td>
<td>-0.023</td>
<td>0.069</td>
<td></td>
</tr>
<tr>
<td>-0.001</td>
<td>0.018</td>
<td>-0.023</td>
<td>0.069</td>
<td></td>
</tr>
<tr>
<td>-0.001</td>
<td>0.018</td>
<td>-0.024</td>
<td>0.069</td>
<td></td>
</tr>
<tr>
<td>-0.001</td>
<td>0.018</td>
<td>-0.024</td>
<td>0.069</td>
<td></td>
</tr>
</tbody>
</table>

---

<table>
<thead>
<tr>
<th>Mean</th>
<th>-0.0007</th>
<th>0.01795</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>-0.0234</td>
<td>0.0685</td>
</tr>
<tr>
<td>Std Dev</td>
<td>0.000458</td>
<td>0.00015</td>
</tr>
<tr>
<td>SNA</td>
<td>16.39884</td>
<td>20.77973</td>
</tr>
</tbody>
</table>

**Table D.1 - Neo Products Experimental Data**
Martin Neck - No Tension in the Rod

<table>
<thead>
<tr>
<th>Dial Gage 1</th>
<th>Dial Gage 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>No Load</td>
<td>180 lbs</td>
</tr>
<tr>
<td>No Load</td>
<td>180 lbs</td>
</tr>
<tr>
<td>===========</td>
<td>==========</td>
</tr>
<tr>
<td>0</td>
<td>0.033</td>
</tr>
<tr>
<td>0.006</td>
<td>0.038</td>
</tr>
<tr>
<td>0.01</td>
<td>0.04</td>
</tr>
<tr>
<td>0.009</td>
<td>0.042</td>
</tr>
<tr>
<td>0.01</td>
<td>0.043</td>
</tr>
<tr>
<td>0.012</td>
<td>0.044</td>
</tr>
<tr>
<td>0.014</td>
<td>0.044</td>
</tr>
<tr>
<td>0.014</td>
<td>0.045</td>
</tr>
<tr>
<td>0.015</td>
<td>0.046</td>
</tr>
<tr>
<td>0.015</td>
<td>0.047</td>
</tr>
</tbody>
</table>

Mean: 0.0105 0.0422 0.001 0.1078
Stnd Dev: 0.004478 0.003995 0.001375 0.005363
SNA: 3.701321 10.23796

------------------------

Martin Neck - Maximum Tension in the Rod

<table>
<thead>
<tr>
<th>Dial Gage 1</th>
<th>Dial Gage 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>No Load</td>
<td>180 lbs</td>
</tr>
<tr>
<td>No Load</td>
<td>180 lbs</td>
</tr>
<tr>
<td>===========</td>
<td>==========</td>
</tr>
<tr>
<td>0</td>
<td>0.033</td>
</tr>
<tr>
<td>-0.055</td>
<td>0.038</td>
</tr>
<tr>
<td>0.005</td>
<td>0.035</td>
</tr>
<tr>
<td>-0.056</td>
<td>0.045</td>
</tr>
<tr>
<td>0.006</td>
<td>0.035</td>
</tr>
<tr>
<td>-0.054</td>
<td>0.045</td>
</tr>
<tr>
<td>0.007</td>
<td>0.036</td>
</tr>
<tr>
<td>-0.053</td>
<td>0.049</td>
</tr>
<tr>
<td>0.007</td>
<td>0.037</td>
</tr>
<tr>
<td>-0.053</td>
<td>0.05</td>
</tr>
<tr>
<td>0.008</td>
<td>0.037</td>
</tr>
<tr>
<td>-0.051</td>
<td>0.051</td>
</tr>
<tr>
<td>0.008</td>
<td>0.037</td>
</tr>
<tr>
<td>-0.051</td>
<td>0.052</td>
</tr>
<tr>
<td>0.009</td>
<td>0.037</td>
</tr>
<tr>
<td>-0.051</td>
<td>0.051</td>
</tr>
<tr>
<td>0.009</td>
<td>0.038</td>
</tr>
<tr>
<td>-0.051</td>
<td>0.051</td>
</tr>
</tbody>
</table>

Mean: 0.0064 0.036 -0.054 0.0475
Stnd Dev: 0.002538 0.001414 0.00405 0.004342

Table D.2 - Martin Neck Experimental Data
### Table D.3 - Sekova Neck Experimental Data

<table>
<thead>
<tr>
<th>Dial Gage 1</th>
<th>Dial Gage 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>No Load</td>
<td>180 lbs</td>
</tr>
<tr>
<td>0</td>
<td>0.002</td>
</tr>
<tr>
<td>-0.004</td>
<td>0.002</td>
</tr>
<tr>
<td>-0.003</td>
<td>0.002</td>
</tr>
<tr>
<td>-0.005</td>
<td>0.002</td>
</tr>
<tr>
<td>0</td>
<td>0.003</td>
</tr>
<tr>
<td>-0.007</td>
<td>0.003</td>
</tr>
<tr>
<td>-0.005</td>
<td>0.002</td>
</tr>
<tr>
<td>-0.004</td>
<td>0.003</td>
</tr>
<tr>
<td>-0.005</td>
<td>0.003</td>
</tr>
</tbody>
</table>

| Mean       | -0.0038    | 0.0034   | 0.2266  |
| Stnd Dev   | 0.002135   | 0.002332 | 0.001281 |
| SNA        | 2.503012   | 6.901056 | 1.636795 22.47838 |
Vita

Edward John Grasso was born on June 11, 1969, in New Hyde Park, New York. He is the son of Fred and Anne Grasso, and has three other siblings.

He obtained his high school diploma at Chaminade High School in Mineola, New York in 1987. He received his bachelor’s degree in mechanical engineering at Villanova University in 1991. While at Villanova he was initiated into the Tau Beta Pi, Pi Tau Sigma, and Phi Kappa Phi honor societies as well as being awarded the Dean’s Award of Academic Excellence and the Dean’s Award of Meritorious Service.

At Lehigh he has served as a research assistant for Neo Products Inc. and plans on receiving his master’s degree in mechanical engineering in May of 1993. After graduation he plans on working for Allied Signal Inc. as a design engineer in Morristown, New Jersey.