5-1-2009

Stress analysis of a glued timber beam

Walter Ray Williams

STRESS ANALYSIS OF A GLUED TIMBER BEAM

By

Walter Ray Williams

A Thesis
Submitted to the Faculty of
Mississippi State University
in Partial Fulfillment of the Requirements
for the Degree of Master of Science
in Aerospace Engineering
in the Department of Aerospace Engineering

Mississippi State, Mississippi

May 2009

STRESS ANALYSIS OF A GLUED TIMBER BEAM

By

Walter Ray Williams

Approved:

Gregory D. Olsen
Assistant Professor of
Aerospace Engineering
(Director of Thesis)

Thomas E. Hannigan III
Instructor of Aerospace Engineering
(Committee Member)

Rani Warsi Sullivan
Assistant Professor of
Aerospace Engineering
(Committee Member)

Pasquale Cinnella
Professor of Aerospace Engineering
Graduate Coordinator for Aerospace
Engineering

Sarah A. Rajala
Dean of the Bagley College of Engineering

Name: Walter Ray Williams

Date of Degree: May 2, 2009

Institution: Mississippi State University

Major Field: Aerospace Engineering

Major Professor: Dr. Gregory D. Olsen

Title of Study: STRESS ANALYSIS OF A GLUED TIMBER BEAM

Pages in Study: 42

Candidate for Degree of Master of Science

The Forestry Department at Mississippi State University has been contracted to design and test a novel beam to be used to create crossing platforms for cranes operating in muddy, swampy areas. To date, they have performed stress analyses on 30 beams, but their physical testing method requires costly amounts of material and man hours. It is theorized that the finite element method may be used as an alternative method of analysis in order to reduce costs.

The focus of this study is to create models of tested beams using the finite element solver, ANSYS, and verify the accuracy of these models using the results of the Forestry Department's physical testing.

Key Words: glued timber, stress analyses, ANSYS, finite element solver

DEDICATION

I would like to dedicate this research to my parents, Larry and Linda Williams.

ACKNOWLEDGEMENTS

The author would like to thank Dr. Greg Olsen, my committee chairman, for graciously providing the time, effort and patience required to guide me through the hours of research and writing involved with this thesis. Appreciation is also due to Dr. Rubin Shmulsky for graciously providing the background information without which this study would not have been possible. The author would also like to thank Mr. Thomas Hannigan III, and Dr. Rani Sullivan for their time and aid as thesis committee members. Finally, the author expresses sincere appreciation for the encouragement and support provided by family and friends, namely Larry and Linda Williams, Nicholas Chambers, and Samantha Ryan.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

CHAPTER I

INTRODUCTION

The Mississippi State University Forestry Department has been contracted to design beams that will aid cranes in crossing muddy, swampy terrain. In the past, when such terrain was encountered, large solid cedar beams were laid across the ground creating a crossing platform for the cranes. These cedar beams were able to carry a maximum bending stress of about 3100 psi. The company which contacted the Forestry Department was interested in reducing the amount of material used to create these platforms. The proposed solution was to create laminated timber beams using sweetgum lumber [1].

The individual sweetgum boards have an average depth and height of 0.86 in. and 5.75 in. respectively, but their lengths will vary. By gluing these boards side by side, beams consisting of 14 layers of boards are used to create a laminated beam. Each beam has average dimensions of 120.75 x 12.00 x 5.75 inches as shown in Figure 1. When a ditch is encountered, the beams are laid in such a way as to create a 12 inch wide, 120.75 inch long platform. Multiple beams are bolted side by side to create wider platforms for the crossing of equipment or men [1]. Figure 2 depicts a typical use of the beam and the bending that it creates.

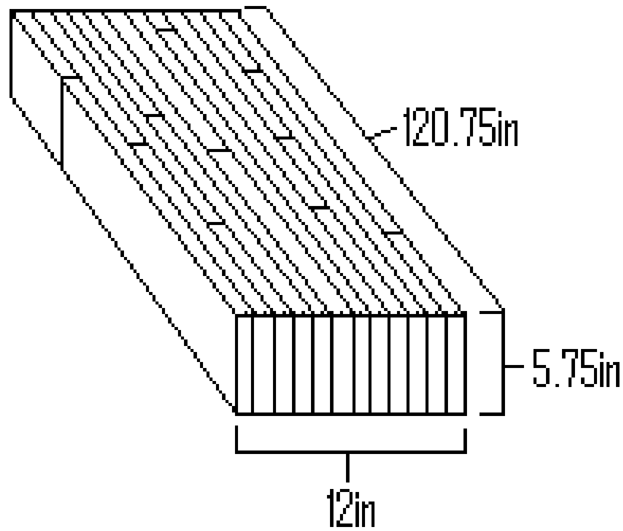

Figure 1

Average beam dimensions

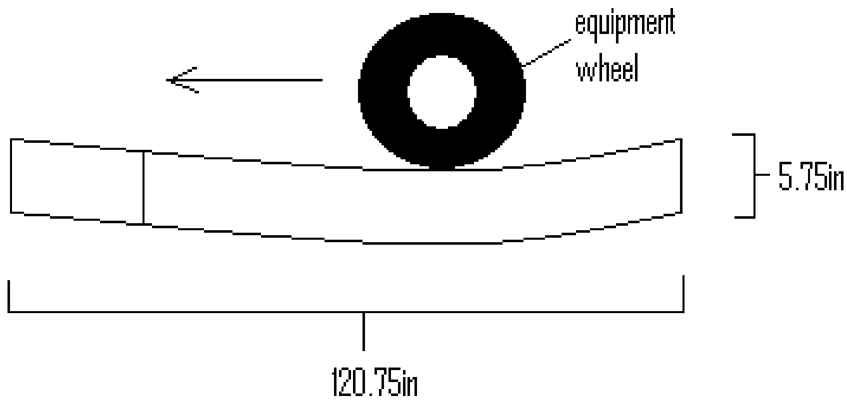

Figure 2

Typical beam usage

In order to ensure the laminated beams could sustain the same bending stress as the solid timbers, several mock beams have been constructed and tested to determine the maximum bending stress. The standard for evaluation of structural composite lumber products is given by the American Society for Testing and Materials (ASTM) standard D5456-01 [1].

ASTM D5456-01 states that the proper way to determine maximum bending stress of structural glued timber is via 4-point bend testing. This testing will be discussed in greater detail in Chapter 2.

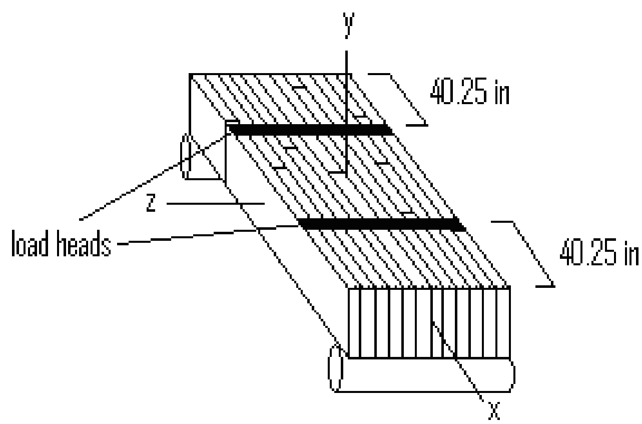

Figure 3

A beam in set up for 4-point bending

The maximum bending stress is found via the mathematical analysis also seen in ASTM D5456-01. After conducting the 4-point bend tests using the setup shown in Figure 3, the classical equation for maximum bending stress in a rectangular beam, Equation 1, is used to find the maximum bending stress in thirty beams.

$$S = \frac{PL}{bd^2} \tag{1}$$

where:

S	=	maximum bending stress in psi
P	=	maximum load in lbf
L	=	length of the beam in inches
b	=	width of the beam in inches
d	=	depth of the beam in inches

The main problem with using this method is that it requires that several mock beams be constructed and tested. This is expensive in regard to material and man hours required to conduct the testing. If possible, the Forestry Department would like to use a different method to perform stress analysis on the beams. The alternative method should ideally use less material and man hours. It is theorized that the best alternative form of analysis, in this case, would be the finite element method.

The goal of this thesis is to construct finite element models of the beams using ANSYS, a finite element solver, and verify the accuracy of the models using the results of the physical testing. Since no glue failures were encountered during testing, the glue itself will not be modeled.

There are three objectives for this thesis. First, a control model based on all the assumptions of Equation 1 will be constructed in ANSYS. If the results of the control model match the results of Equation 1, then the finite element model has been set up correctly. Second, two intermediate models will also be constructed. Each intermediate model will sequentially take away assumptions made by Equation 1. This will provide a better understanding of how the assumptions affect the maximum stress results. Lastly, two finite element models of actual tested beams will be constructed. The results of these models will be used to determine whether the estimates of Equation 1 are accurate enough to be accepted.

Chapter 2 will explain the testing conducted by the Forestry Department, and present the results of the tests. In Chapter 3, the modeling methodology will be presented including a description of ANSYS, the setup for the models, and the material models. The Final Chapter will present the conclusions that can be made from the models and will also present several parameters that should be kept in mind before building beams in the field.

CHAPTER II

PHYSICAL TESTING AND STRESS ANALYSIS

The main concern of the Forestry Department and of this thesis is the amount of loads carried by the bridge beams and the resultant stresses developed. Stress analyses are how engineers determine the stresses present in a component like the bridge beams. There are several different types of stress analyses such as classical, physical, and finite element. This chapter will focus on the physical testing and classical analysis performed by the Forestry Department.

The first step in performing any type of stress analysis is to have an understanding of the basic mechanical properties of the material and the forces that the material will see. The beams will be driven over by vehicles and the contact point between the vehicle and the beams will be the tires. The weight of the vehicle will produce two internal reactions in the beams: a bending moment, illustrated in Figure 4, and a vertical shear force, illustrated in Figure 5. These internal reactions will cause the stresses in the beams.

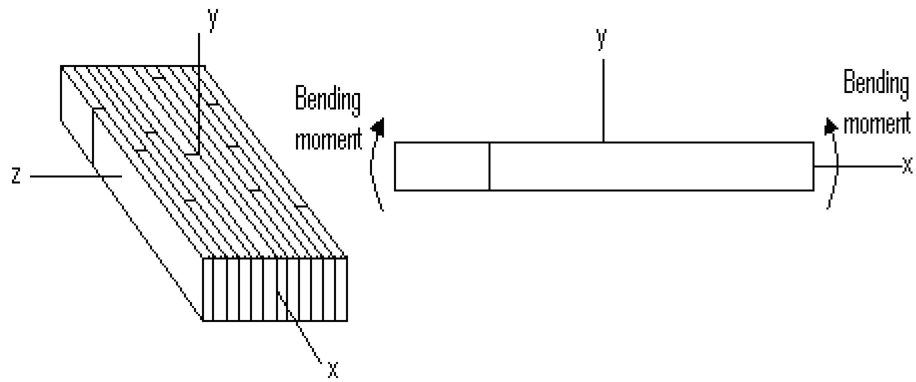

Figure 4

Bending moment

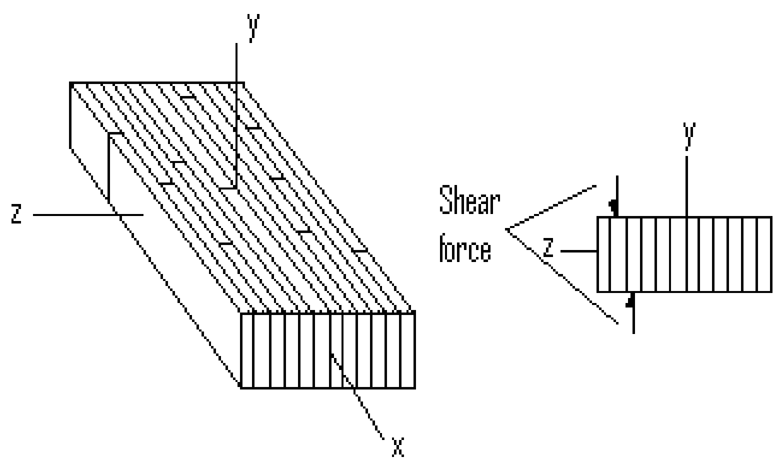

Figure 5

Vertical shear force

There are three main mechanical properties that characterize mechanical behavior: modulus of elasticity, Poisson's ratio, and shear modulus. The modulus of elasticity is the measure of a material's ability to return to its original shape after being deformed. Mathematically it is the slope of a material's stress strain curve in the elastic region. Figure 6 depicts a typical stress strain curve and has the modulus of elasticity labeled. The higher the modulus of elasticity, the less the material will deform before it becomes plastic, meaning it cannot return to its original shape. Poisson's ratio is a measure of the transverse strain with respect to the longitudinal strain as given by Equation 2.

$$\mu = \frac{-\varepsilon_{lateral}}{\varepsilon_{longitudinal}} \qquad (2)$$

where:

μ	=	Poisson's ratio
$\varepsilon_{lateral}$	=	Transverse strain
$\varepsilon_{longitudinal}$	=	Longitudinal strain

Shear modulus is a derived property defined by Equation 3.

$$G = \frac{E}{2(1+\mu)} \qquad (3)$$

where:

G	=	Shear Modulus in psi
E	=	Modulus of Elasticity in psi
μ	=	Poisson's Ratio

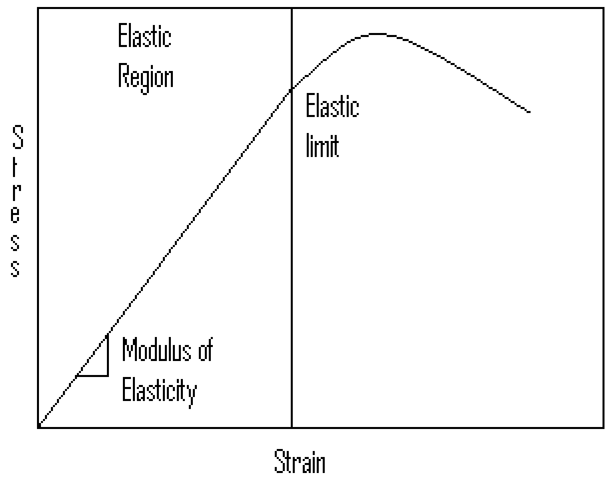

Figure 6

Stress strain curve

There are two materials, sweetgum and wood glue, described in this thesis. Since there were no failures in the glue, it will not be analyzed and its mechanical properties will not be needed. However, the mechanical properties of sweetgum will be essential to the stress analysis. Like most wood, sweetgum is orthotropic. This means that it has different properties depending on how the fibers of the wood are oriented when a force is applied to the wood. Therefore, the general practice is to define the properties with respect to the orientation of the grain as shown in Figure 7.

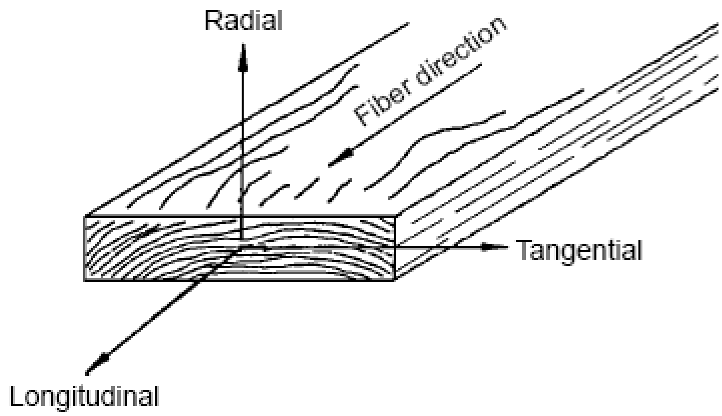

Figure 7

A board with fiber orientation shown

To define the elastic behavior of an orthotropic material like sweetgum, nine constants are needed: three moduli of elasticity, three shear moduli, and three Poisson's ratios. The three moduli of elasticity are defined as E_L, E_T, E_R. The subscripts L, T, and R refer to the longitudinal, tangential, and radial axes as defined by the fiber direction depicted in Figure 7. E_L refers to the modulus of elasticity in the longitudinal axis. It can be thought of as a measure of how difficult it is to deform the material in the longitudinal axis. Likewise, E_T and E_R can be thought of as a measure of how hard it is to deform the material in their respective axes. The shear moduli are defined as G_{LR}, G_{LT} and G_{RT}. The subscripts L, R, and T refer to the principal axes. The first subscript refers to the direction of the outward normal to the material surface, and the second refers to the direction in which stress is applied. The Poisson's ratios are μ_{LR}, μ_{LT} and μ_{RT}. The subscripts here are the same as those that appear in the shear moduli. The nine material properties of sweetgum, as defined in Chapter 4 of the Wood Handbook, are given in Table 1 [2].

Table 1

The material properties of sweetgum lumber

E_L psi	E_T psi	E_R psi	G_{LR} psi	G_{LT} psi	G_{RT} psi	μ_{LR}	μ_{LT}	μ_{RT}
1.20e6	6.00e4	1.38e5	7.32e4	25200	1.07e5	0.403	0.309	0.325

The main type of load acting on the beam will be bending. There will also be a vertical shear force between the layers of the beam. During normal use this shear force will not be large compared to the bending force because, as depicted in Figure 8, the width of the tires will distribute the vehicle's weight across several layers of the beam. The forces acting on the beam have dictated how the Forestry Department glued the beam together. Figure 9 shows two boards in the same layer when the beam is being bent. The tops of the two boards are being pushed together while the bottoms are being pulled apart. Since glue does not perform well when the two faces it is holding are being pulled apart, boards within a single beam layer are not glued end to end because bending tends to pull these boards away from one another. However, glue does perform well when the two faces it holds together are in shear, so boards in adjoining layers are glued together because they are in shear.

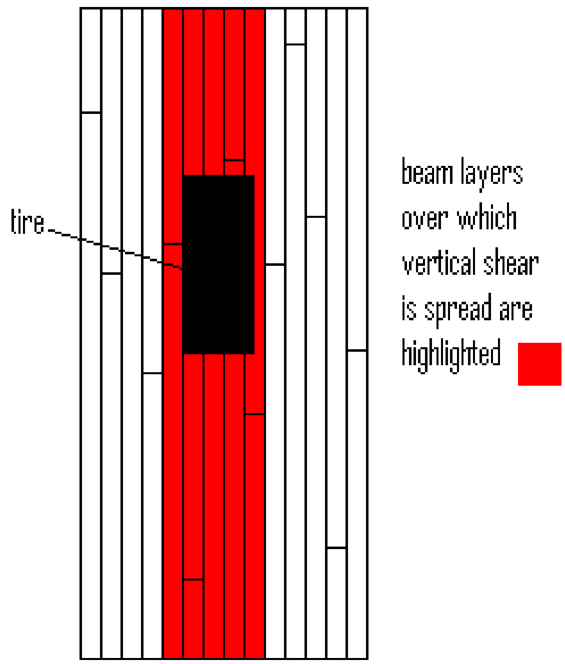

Figure 8

Tire distributing shear stress over several layers of a beam

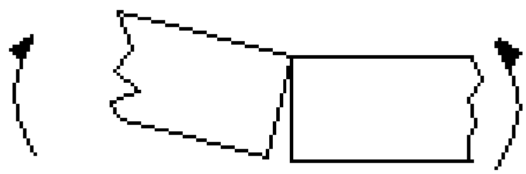

Figure 9

Two boards in a single layer of a beam under bending

The American Society for Testing and Materials covers the standard for evaluation of structural composite lumber products in ASTM D5456-01. The ASTM standard covers how to find several properties associated with laminated timber beams including maximum bending stress and modulus of elasticity. Finding these properties requires a 4-point bend test to be conducted.

Due to the usage of the Forestry Department's beams as crossing platforms, the property that they were most interested in was the bending stress. ASTM D5456-01 states that the allowable bending stress shall be determined by using Equation 1, found in Chapter 1.

The 4-point bend test is commonly used in the testing industry to determine bending stress. In a 4-point bend test, the beam is supported by a pin support at one end and placed on a roller at the other end. Also, two load heads press on the beam. With a support at both ends and the two load heads pressing in the middle of the beam, it is being loaded at four points as the name of the test, 4-point bend, suggests. ASTM D5456-01 dictates that the 4-point bend is the standard test used to determine the maximum bending stress of laminated structural timber. The Forestry Department followed the regulations in this ASTM standard to conduct their 4-point bend test and then determine the maximum bending stress.

The Forestry Department located their load heads 40.25 inches and 80.50 inches measured from one end of the beam. This puts the load heads 40.25 inches apart, which is greater than the minimum of six inches. They are also 40.25 inches from either end. The minimum distance for this beam would be two times 5.75 inches (11.50 inches) from the end. In this manner, 30 beams were tested.

Using computer software in conjunction with testing hardware, the force of the two load heads pushing against the beams when failure occurs is recorded as Pmax (Appendix A). With this information and the dimensions of the beam, Equation 1 then treats the beam as an isotropic material rather than the orthotropic material that it actually is. An isotropic material has the same material properties in all three principal directions, so an isotropic material will only have one elastic modulus, one shear modulus, and one Poisson's ratio. Making this assumption greatly simplifies the stress analysis by bringing the number of material properties from nine down to three. They also made the assumption that the beam was homogeneous. In other words, they assumed that the beam was a single block material rather than pieces of material glued together. When the material is assumed to be homogeneous and isotropic, the expression of the bending stress in the material simplifies to Equation 1, repeated here for clarity.

$$S = \frac{PL}{bd^2} \tag{1}$$

where:

S	=	Maximum bending stress in psi
P	=	Maximum load in lbf
L	=	Length of the beam in inches
b	=	Width of the beam in inches
d	=	Depth of the beam in inches, as oriented for the 4-point bend test

Using this equation, the maximum bending stress of all 30 beams, along with their average bending stress, was calculated and compiled. This information can be found in Appendix A.

Chapter III will focus on creating finite element models that incorporate orthotropic material properties and non-continuous geometry. The information from Appendix A will be used to verify the accuracy of the finite element models. After the accuracy of the models is verified, future uses for the models will be discussed in Chapter IV.

CHAPTER III

FINITE ELEMENT ANALYSIS

Using the results of the physical testing and classical analysis performed by the Forestry Department, the results of an alternative method of analysis will be verified. If this different method produces results that corroborate the results of the method used by the Forestry Department, there will be a greater sense of confidence in the results, and the alternative method can be used to perform further analysis without the need for further physical testing. Numerical analysis is a different method that might offer some advantages when analyzing the bridge beams.

Numerical stress analysis is useful in situations like this where an exact solution is not available due to the complexity of the governing differential equations which represent the balance of mass, force, or energy. The numerical stress analysis will not yield an exact solution at every point in a body. Only at discrete points called nodes will the exact solution be approximated. The two most common classes of numerical stress analysis are the finite difference method and finite element method. The finite difference method uses a differential equation at each node. The differential equations are then replaced by difference equations resulting in a set of simultaneous linear equations. This works well for simple problems, but the linear equations become complicated with complex geometries and nonisotropic material properties like the bridge beams. The finite element method uses integral formulations in order to create a system of algebraic equations. A system of algebraic equations will be easier to solve in complex geometries than the linear equations that result from the finite difference method [3]. This makes the finite element method a better choice for analyzing the bridge beams.

While the finite element method can be traced back to the early 1900's, the modern movement is usually credited to Richard Courant who, in the 1940's, used it to investigate torsion problems. During the 1950's, Boeing began to use the finite element method to model airplane wings, and, in the 1960's, the first book dedicated entirely to finite elements was written [3].

In 1971, ANSYS, a finite element solver, was released for the first time. ANSYS has since become a leading finite element analysis program, and is known to be a powerful and impressive engineering tool [3]. It is important to remember that ANSYS is only a tool. No matter how powerful the tool may seem, it must be used properly in order to achieve worthwhile results. In order to use ANSYS properly, one must know the basics of the finite element method.

There are seven basic steps in the finite element method, divided into three phases [3]. They are:

Preprocessing Phase

1. Create a model of the problem and subdivide it into nodes and elements.
2. Assume a continuous function that represents the physical behavior of the elements.
3. Develop equations for each element
4. Assemble the elemental equations into a global stiffness matrix that represents the entire model.
5. Apply the boundary and initial conditions.

Solution Phase

6. Solve the set of algebraic equations simultaneously to obtain nodal solutions.

Postprocessing Phase

7. Obtain any other important information using the nodal solutions.

In ANSYS, the three phases are mimicked with three processors: the preprocessor, the processor, and the general postprocessor. The preprocessor contains commands similar to the steps in the preprocessing phase that allow the user to create a model:

- Define element types
- Define material properties
- Create model geometry
- Define the boundary and initial conditions
- Mesh the model

Likewise, the solution processor solves for the nodal solutions as in step 6 in the solution phase. Also, the general postprocessor allows the user to view the results of any of several analyses [3].

While knowing the basic steps of finite element analysis and how they are applied in ANSYS does increase the user's ability to perform an analysis with realistic results, it does not guarantee it. Engineers specializing in the field of finite element analysis usually use a building block approach to ensure that results can be trusted. In other words, simple models that are closely related to the complex model are solved first. The exact solutions to these simple models are well-documented in literature, so, after solving the model in ANSYS, the finite element results and the literature results can be compared. If the comparison shows a close match, the engineer will then modify the simple model to look more and more like the complex model. After each step of modification, the results are compared to the results of the previous step. Any large differences in the results of two steps, indicate there might have been human error, and the model should be checked closely.

The beam models here will be constructed in the same step-by-step modification manner. The first model will be an isotropic, homogeneous beam in 4-point bending. The results of this model should match the results of the Forestry Departments analysis which used Equation 1. Next, the model will be modified to have the orthotropic material properties of sweetgum. The results of this model will be compared to the results of the first model to determine how the modifications affect the results. Then the model will be modified yet again by changing it from a homogeneous beam to 14 separate boards glued together. Again the results of this model will be compared to the previous results to determine the effect of the modification. Finally, models of two actual beams that were previously tested will be created. Their results will be compared not only to the previous models but also to the results of the Forestry Department testing. The comparison of the finite element results and the physical testing results will give a sense of how precise the two methods are, and how accurately the finite element model predicts the stresses in the beams.

In creating the simple model of the homogeneous isotropic beam, the basic steps of the finite element method were followed in ANSYS. First, the element type is defined, the material properties are defined, the geometry is created, and the model is meshed. Next, the nodal solutions are obtained. Finally, a stress analysis is performed using the nodal solutions.

To begin, an element type must be selected for the model. There are several to choose from. For example, there are two-dimensional elements like three-node and six-node triangles, and four-node and eight-node squares. There are also three-dimensional elements like eight-node bricks and ten-node tetrahedrons. The bridge beam is three-dimensional, so a three-dimensional element is required. Also, the large length to span ratio of the beam means that large deflections could result in some curvature. Curvature is a problem in finite elements because, as stated earlier, the solution only approximates the exact solution at nodes and interpolates the solution between nodes. This means that, between the nodes, the finite element solution will have some error, and the more curvature, the greater the error as shown in Figure 10. To minimize this error in ANSYS, an element with more nodes must be chosen, so for this model the ten-node tetrahedron is an appropriate element type and is represented in Figure 11 [3].

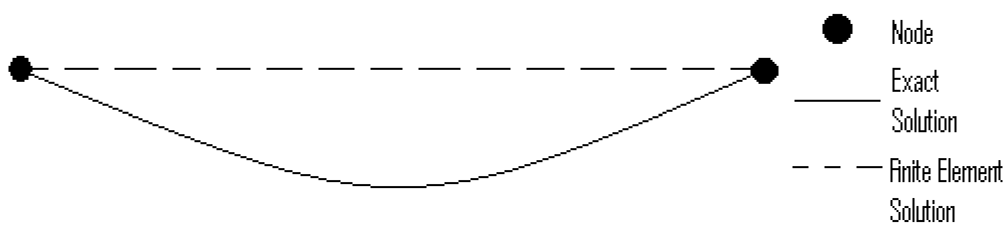

Figure 10

Error due to curvature between nodes

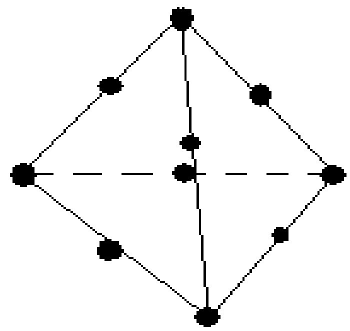

Figure 11

Ten-node tetrahedron

Next, the material properties are defined. Since this will be an isotropic model, it only requires one modulus of elasticity and one Poisson's ratio to define the material properties. The Forestry Department found the modulus of elasticity of their beams as a whole to be $1.60x10^6$ psi. This is used to define the modulus of elasticity for the finite element model. Because the Poisson's ratio of the beams as a whole is unknown, the Poisson's ratio will be defined as 0.33 as that is the general practice for defining unknown Poisson's ratios. After defining the element type and the material properties, a geometric model is created. As stated previously, the geometry for this model will be a single block as shown in Figure 12.

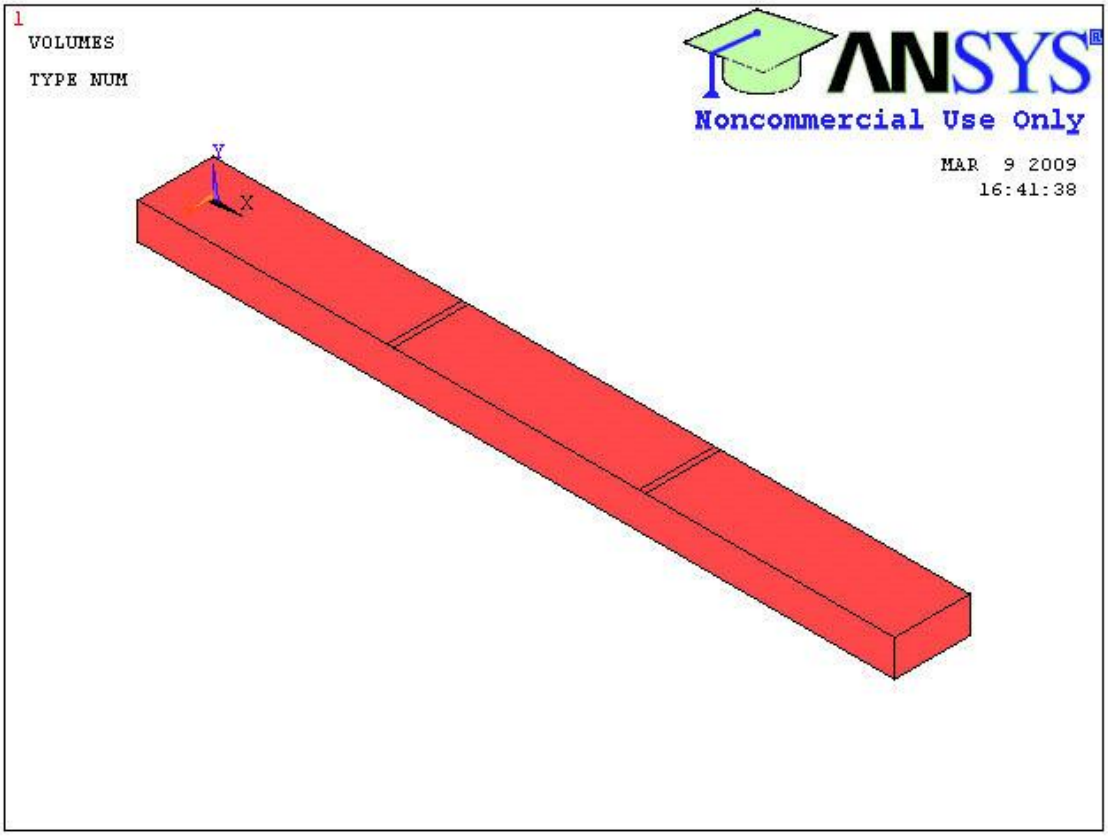

Figure 12

Isotropic solid beam

Next, the boundary and initial conditions are applied. One end of the beam will be fixed in both the x and y directions. The other end will only be fixed in the y direction. The average load that the tested beams held was 23,350 lbf. This load was divided between the two load heads that delivered the force to the beam. In this model, it is assumed that the load heads distribute the load across the top of the beam in two strips. Each strip is one inch wide and is centered about the locations of the load heads during the physical tests (40.25 in. and 80.50 in.). The model with its boundary and initial conditions is shown in Figure 13.

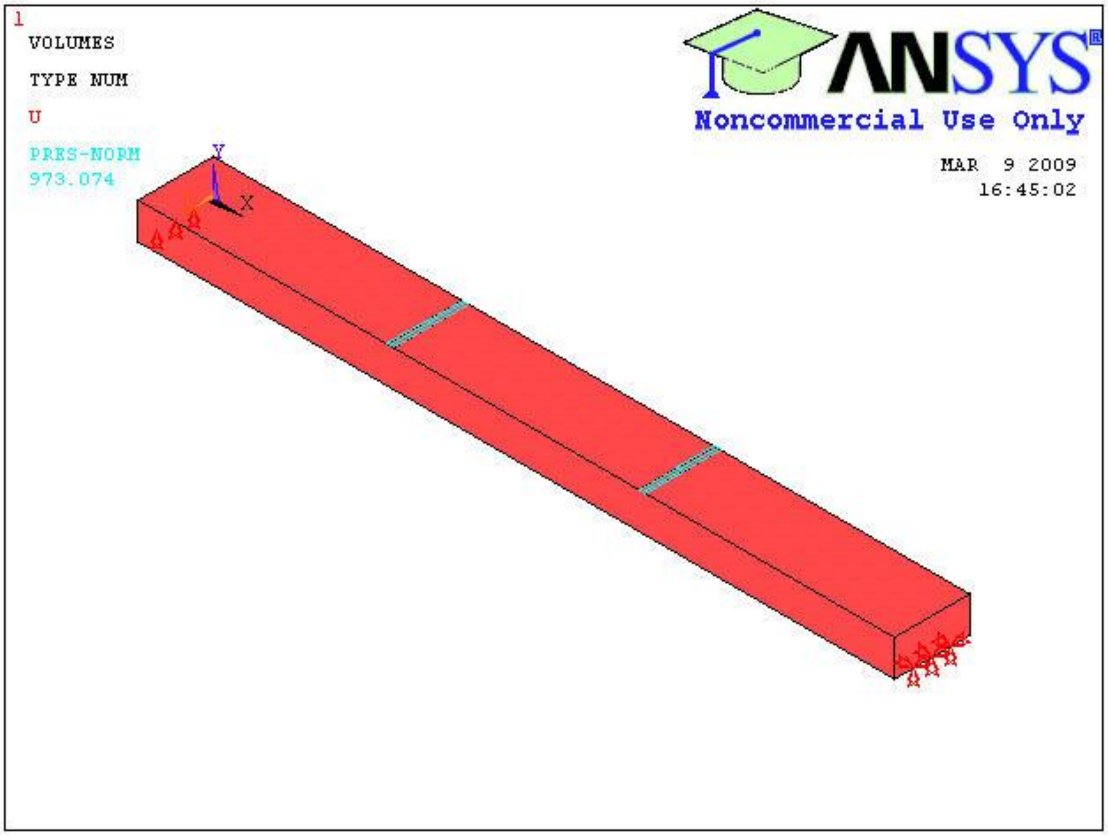

Figure 13

Isotropic solid beam with boundary and initial conditions

The final step in the preprocessor is defining the mesh. Due to the eventual complexity of the beam geometry, it is best to free mesh the beam rather than map a user mesh. The free mesh uses tetrahedrons to divide the model into elements of a defaulted size. While the defaulted size still produces results, the deformation of the beam causes the elements to become extremely deformed, and ANSYS issues a warning that the results may not be accurate. When elements experience extreme deformation, the finite element analysis tends to break down and produces less accurate results. Using trial and error, the element size limit should be adjusted until there are no shape warnings produced. The final mesh is shown in Figure 14.

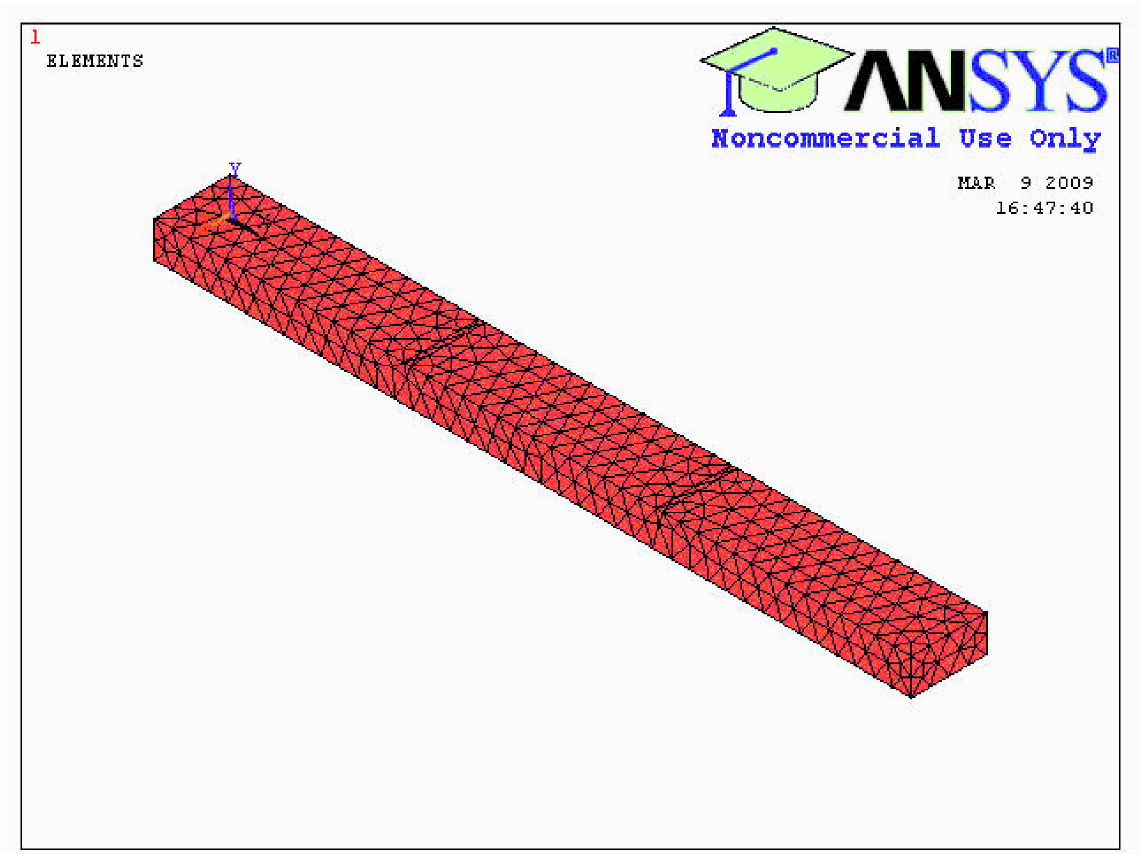

Figure 14

Meshed isotropic solid beam model

After the preprocessing phase, the model is solved using the processor in ANSYS. Then the postprocessing phase begins. For this model, a contour plot of the stress as shown in Figure 15 will give a better sense of what is happening along the beam. Also in Figure 15, it can be seen that the maximum stresses occur on the top and bottom surface of the beam in between the two applied loads. The maximum stress is calculated to be 7,145 psi.

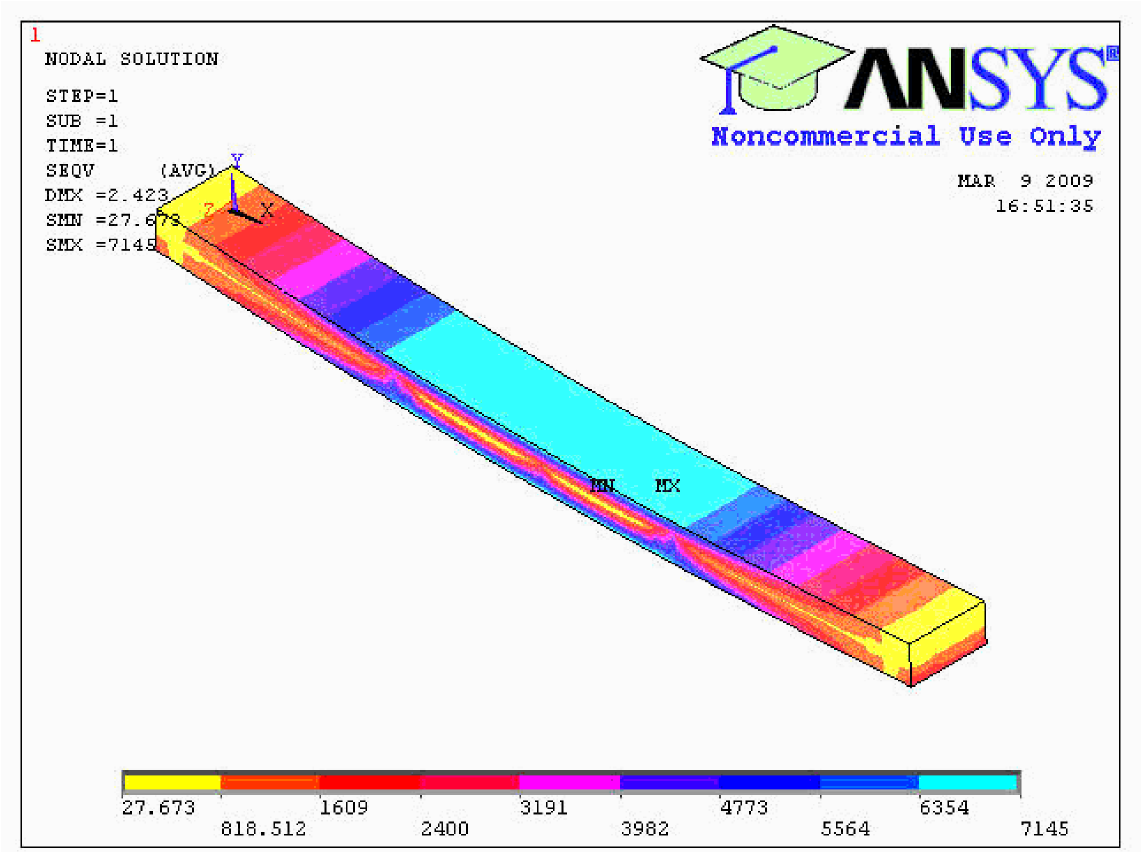

Figure 15

Contour plot of the stress in the isotropic solid beam model

When the average load of 23,350 lbf is substituted into Equation 1, the maximum stress is 7,203 psi. This is within 1% of the stress calculated using the finite element method, so it can be said that the results are precise. Having gained confidence in the finite element model by matching the results of Equation 1, the model could then be modified to better represent an actual beam.

The first modification will be to change the material model. In the preprocessor of ANSYS, the material model was changed from isotropic to orthotropic. While making sure to match the material property to the orientation of the beam, the material properties of sweetgum are applied to the model. Figure 16 presents the input screen with the appropriate values. Then the model with new material properties was solved, and a contour plot of the stress was created (Figure 17). The plot is similar to the previous model's with the maximum stresses lying on the top and bottom surface of the beam between the two load heads.

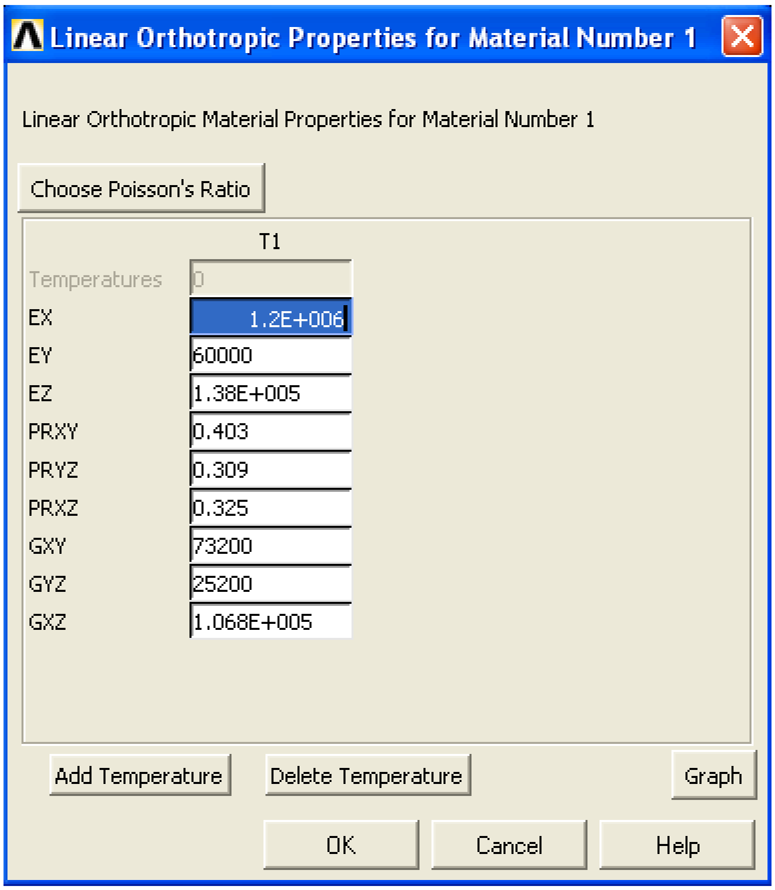

Figure 16

The orthotropic material properties as defined in ANSYS

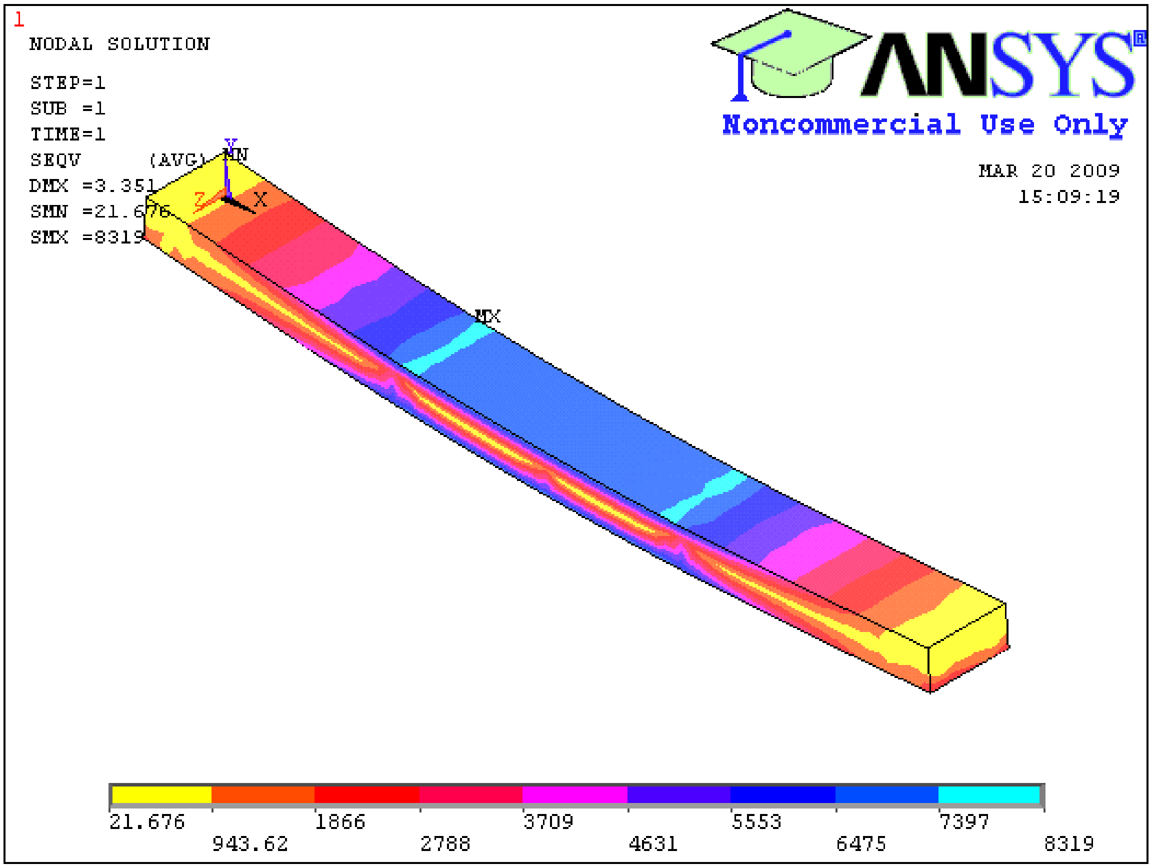

Figure 17

Contour plot of the stresses in the orthotropic solid beam model

The value of the maximum stress for this model was calculated to be 8,319 psi. When compared to the previous model's stress of 7,145 psi, there has been a 14.1% increase in the maximum stress. This is a large jump that is cause for concern. Because the only thing that was modified in this model was the material properties, it is suspected that the change is due only to the addition of the orthotropic material properties.

The next modification changes the geometry from a solid beam to 14 layers glued together to form a beam, which is depicted in Figure 18. It is important to remember that the glue is not modeled in this thesis. The layers are glued together using the ANSYS glue function in the preprocessor. The glue function mathematically forces nodes that share the same initial position, yet belong to separate layers of the beam, to deflect in unison. This makes the boards act as if they are glued together, however, the mathematical bond between the two nodes cannot be broken. In addition, the new geometry must be meshed again before it is solved.

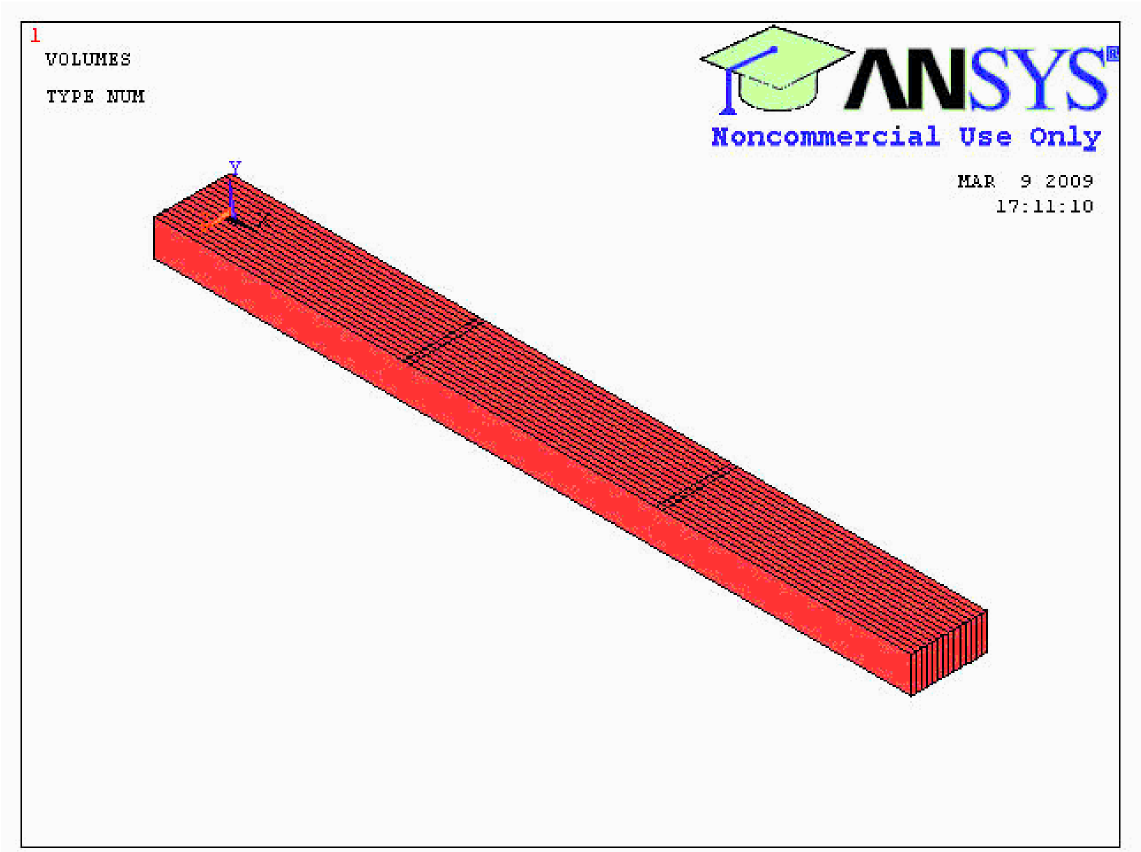

Figure 18

The 14 layer orthotropic beam model

As one would expect, given the results of the previous models, the maximum stresses again fall on the upper and lower surface of the beam in between the two load heads as shown in Figure 19. The maximum stress is calculated to be 8,117 psi. This is a decrease of 2.4% from the 8,319 psi found in the orthotropic solid beam model and is 11.7% above the 7,145 psi found in the isotropic solid beam model. In this model, the difference in maximum stress is attributed to the change in geometry. A 2.4% change in maximum stress is not large enough to suspect that human error may have been involved.

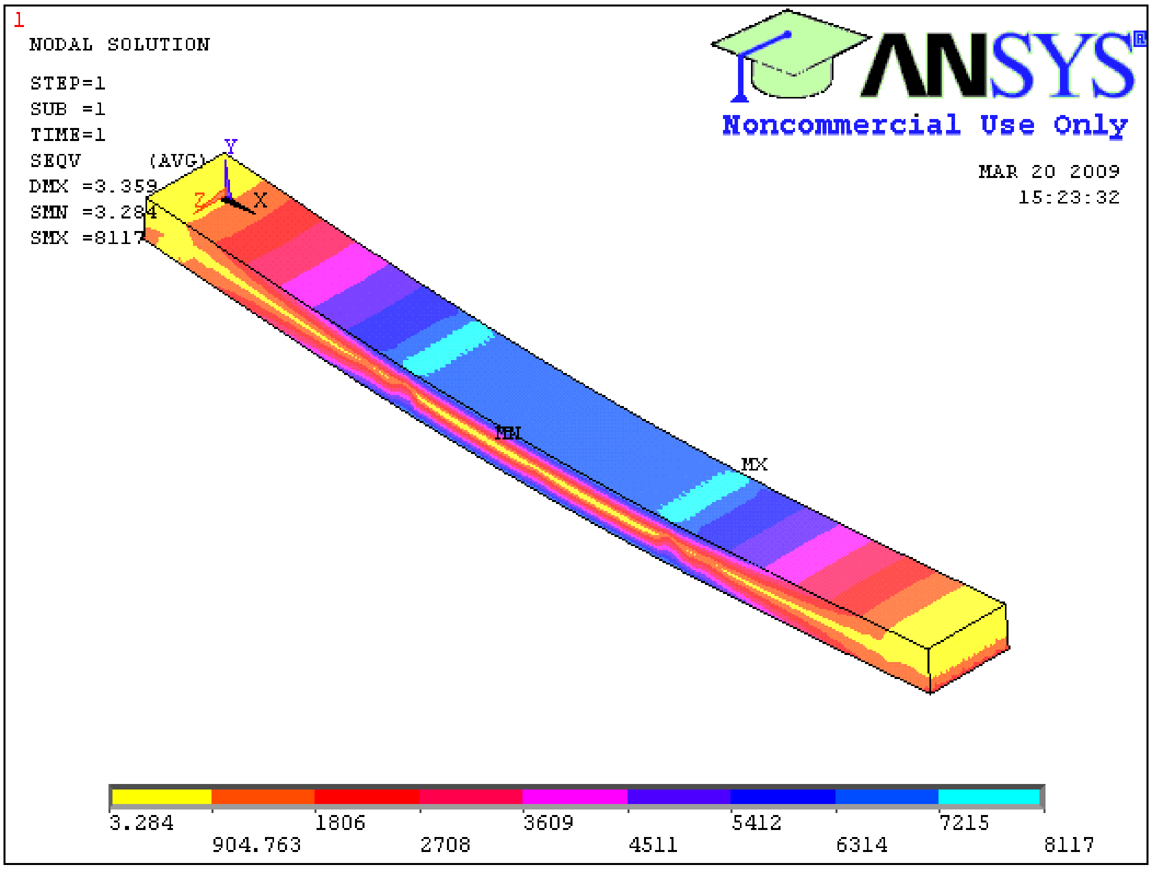

Figure 19

Contour Plot of the stresses in the 14 layer orthotropic beam model

To produce models of tested beams, the geometry is all that will need to be changed. The beam models that are reproduced herein are referred to as beam 1 and beam 2. Both beams consist of fourteen layers, and within each layer there are multiple boards. The location where two boards in the same layer meet is referred to as a joint, and the joint locations for each beam are given in Table 2. Using these joint locations and the average beam dimensions, beam 1 and beam 2 are modeled. Figure 20 shows beam 1 with its boundary and initial conditions. After the beams are modeled, they are meshed and solved.

Table 2

Joint locations measured from the end of the beam

Beam # 1	**Joint 1 (in.)**	**Joint 2 (in.)**
Layer 1	9.25	
Layer 2	21.50	
Layer 3	119.30	
Layer 4	53.75	
Layer 5	91.75	
Layer 6	45.00	
Layer 7	106.80	
Layer 8	29.50	71.00
Layer 9	79.75	
Layer 10	96.50	
Layer 11	59.25	
Layer 12	38.50	
Layer 13	15.25	
Layer 14	117.00	
Beam # 2	**Joint 1 (in.)**	**Joint 2 (in.)**
Layer 1	68.25	
Layer 2	52.50	
Layer 3	6.50	113.00
Layer 4	23.00	
Layer 5	36.00	
Layer 6	13.00	
Layer 7	76.25	
Layer 8	106.25	
Layer 9	83.50	
Layer 10	118.50	
Layer 11	96.00	
Layer 12	69.75	
Layer 13	45.75	
Layer 14	7.50	

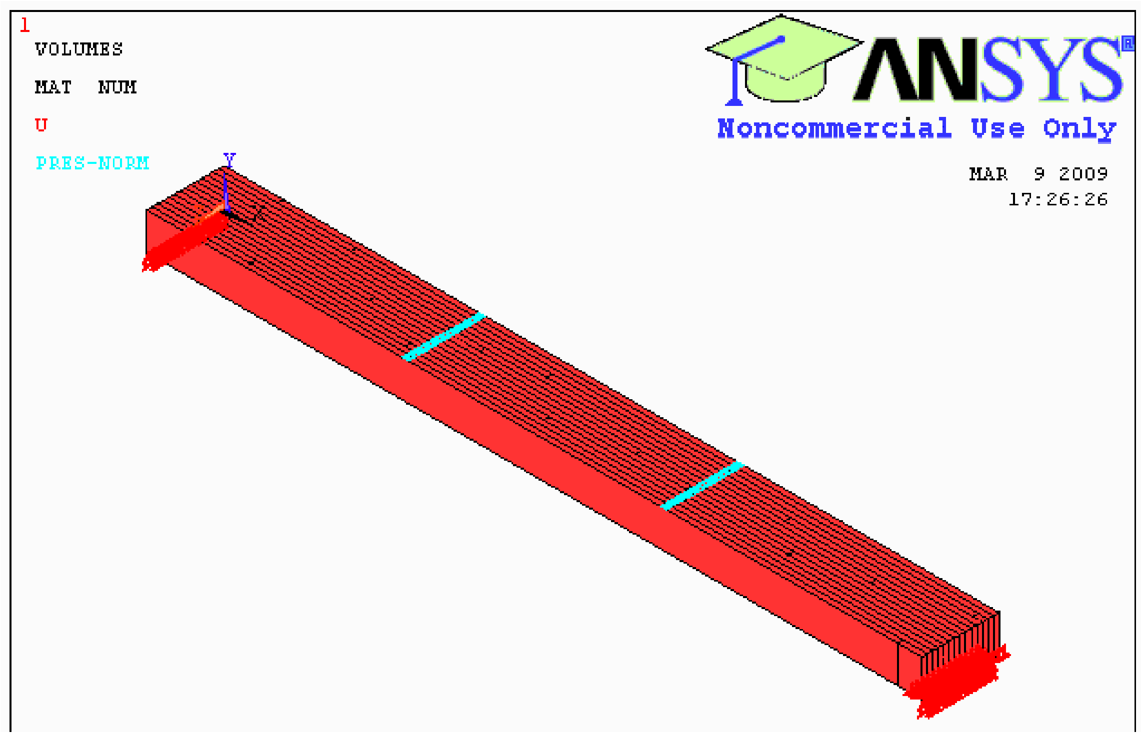

Figure 20

Beam 1 with boundary and initial conditions

After solving the two models, contour plots of the stress in both models are created and can be seen in Figures 21 and 22, respectively, for beam 1 and beam 2. Looking at these figures, it can be seen that the larger stresses can still be found on the upper and lower surfaces of the beam and lie roughly between the two load heads. The maximum stresses for beam 1 and beam 2 are 14,150 psi and 12,800 psi, respectively. When compared to the previous model of the beam with single board layers, which had a maximum stress of 8,117 psi, there has been an increase of over 30%. This is a large jump in stress and suggests that there may be a problem with the model, so closer investigation is required.

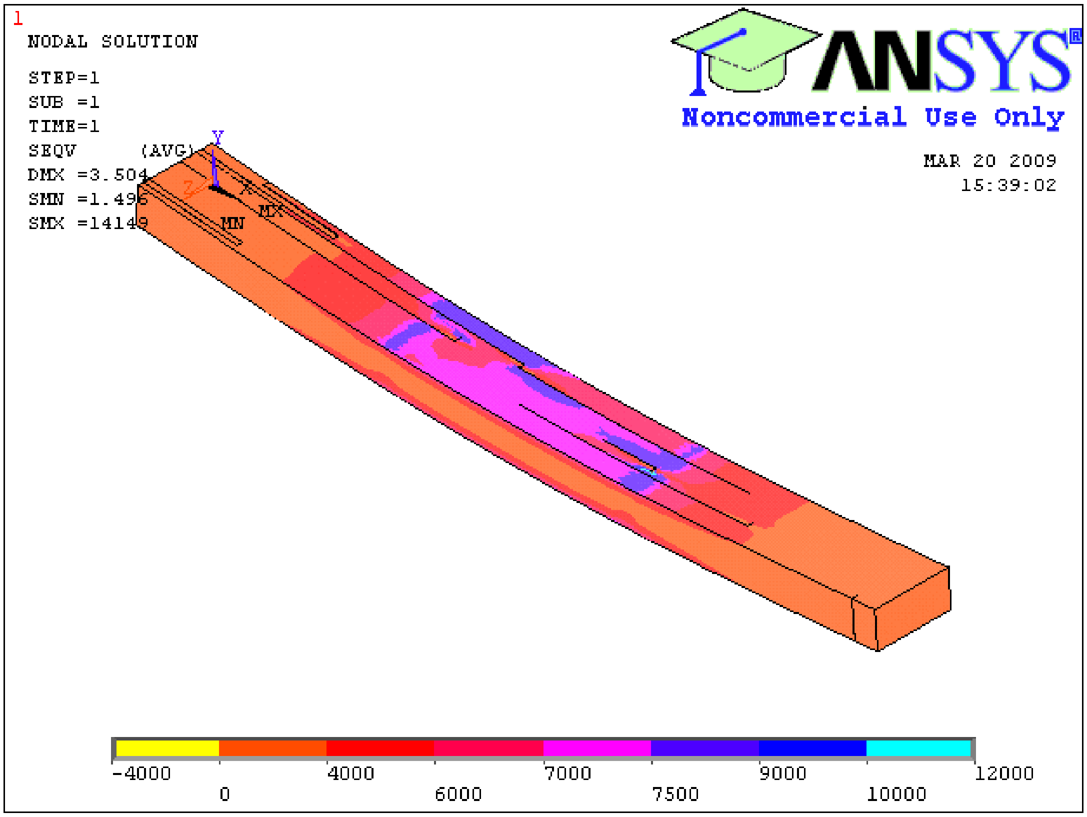

Figure 21

Contour plot of the stress in beam 1

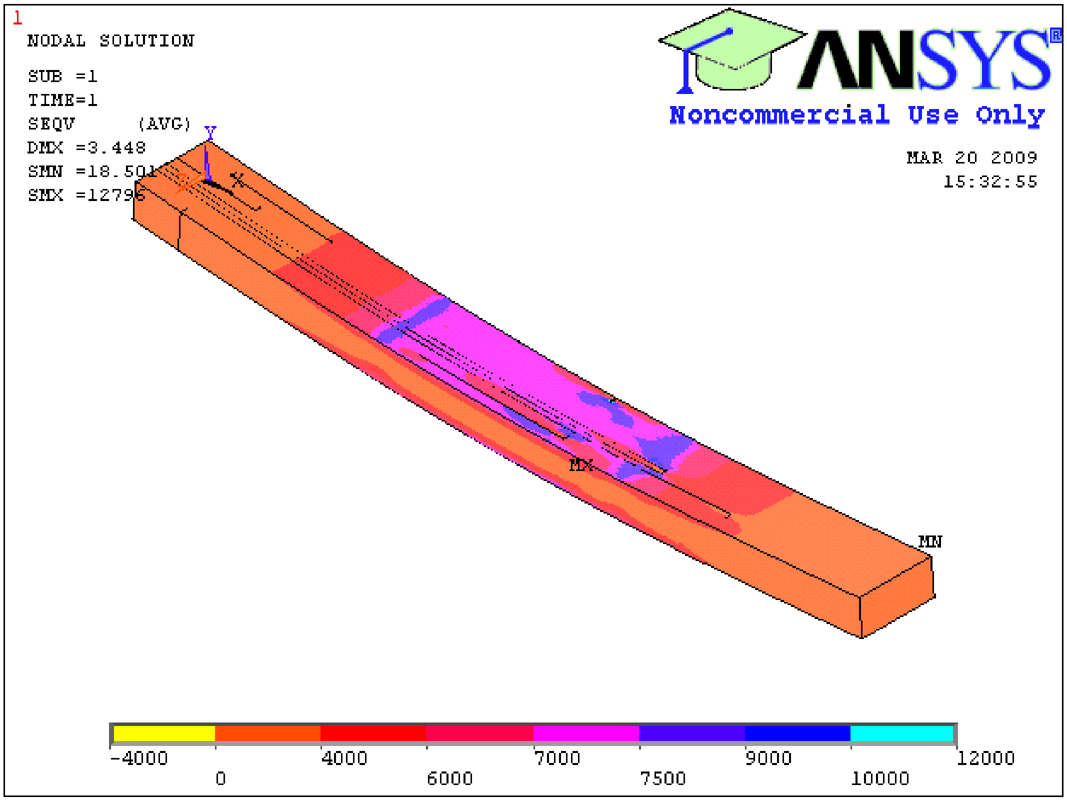

Figure 22

Contour plot of the stresses in beam 2

In the case of beam 1, the maximum stress does not occur between the two load heads as all of the previous model's maximum stresses did. Figure 21 shows that it occurs at the corner of a board where a joint is located. Likewise, the maximum stress in beam 2 occurs at the corner of board where a joint is located. This suggests that there may be a singularity present at those corners in the model.

In stress analysis, singularities occur at points where the stress cannot be defined. In Equation 4, as the area approaches zero, stress becomes infinite.

$$\sigma = \frac{F}{A} \tag{4}$$

where:

σ	=	Stress in psi
F	=	Force in lbf
A	=	Area in in^2

At points like the corners where the maximum stresses occur in beams 1 and 2, the area that forces are acting on approaches zero, thus creating a singularity. Because the finite element method divides geometries into elements whose stress is always finite, stress will never be undefined, but stress will approach infinity as the model is refined with smaller and smaller elements. Therefore, it is theorized that in beams 1 and 2 there are singularities at the corners of the boards where joints are located. This in turn is what is causing the large jump in maximum stress. While in reality there will be stress spikes at the sharp corners, the large stresses will be confined to the corners and will not affect the overall stress state of the beam. Therefore, the maximum stresses will be ignored, and instead the range of stress lying between the two load heads will be used.

The stress contour plot for beam 1 shows that the majority of stress ranges from 7,000 psi and 7,500 psi. This range encompasses the 7,144 psi stress found in the isotropic solid beam. Also, 7,000 psi to 7,500 psi is a range of 6.7%, so the majority of stresses are falling within 6.7% of each other. Beam 2's contour plot shows that the majority of stress between the two load heads also ranges from 7,000 psi to 7,500 psi again encompassing the maximum stress found in the isotropic solid beam. Since the range of stresses in both beams agrees well with the previous models' maximum stresses, there is no reason to suspect an error in the results, and the beams are assumed to be accurately modeled.

Having created finite element models in ANSYS that accurately describe the stresses in the bridge beams, several conclusions can be made regarding future analyses and the construction of the beams. The next chapter will elaborate on these conclusions.

CHAPTER IV

CONCLUSION

The first goal of this thesis was to match the results of a finite element model of a solid isotropic beam with the results of Equation 1, repeated here for clarity.

$$S = \frac{PL}{bd^2} \tag{1}$$

where:

S	=	Maximum bending stress in psi
P	=	Maximum load in lbf
L	=	Length of the beam in inches
b	=	Width of the beam in inches
d	=	Depth of the beam in inches, as oriented for the 4-point bend test

In Chapter III, the solid beam model with isotropic material properties duplicates the assumptions made in Equation 1. The results of the isotropic model and Equation 1 fall within 1% of each other, so it can be said that these results match.

The second goal of this thesis was to create two models which systematically eliminated the assumptions of Equation 1 in order to better understand the effect of each assumption. The finite element model of a solid beam with orthotropic properties eliminated the assumption that the beams are isotropic. From Table 3, it is evident that subtracting the isotropic assumption increased the stress by 14.1%. Next, the finite element model of the orthotropic 14 layer beam with single board layers eliminated the assumption that the beam was homogenous. Table 2 shows the maximum stress in this beam was 2.4% less than that of the orthotropic solid, and 11.7% higher than the isotropic solid. So, while adding orthotropic material properties changed the maximum stress by 11.9%, changing the geometry of the beam to account for multiple layers only changed the maximum stress by 2%. From these results, it appears that the beam is more sensitive to changes in material properties than geometry.

Table 3

Summary of results

Model	Maximum Stress	Percent Difference From Isotropic
Isotropic	7145	
Orthotropic	8319	14.1
14 layer	8117	11.7
Beam 1	7000-7500	Encompassing
Beam 2	7000-7500	Encompassing

The third and final goal of this thesis was to accurately model the stress state of two beams that had already been tested. The results of these models are verified with the results of the physical testing and analysis performed by the Forestry Department. In Chapter III, the majority of stresses between the two load heads for beams 1 and 2 fell between 7,000 psi and 7,500 psi, a window of 6.7%. This range encompasses the maximum stresses found in the isotropic model and is within 7.5% of the orthotropic solid and the 14 layer model. Because the range of stresses matches the similar models built before it, there is no reason to doubt the validity of the results. The results are also precise since the stresses fall within 6.7% of each other. Also, the Forestry Department found the average maximum stress of the tested beams to be 7,108 psi using Equation 1. This stress is also encompassed by the 7,000 psi to 7,500 psi range, so the results of the finite element models corroborate the results of Equation 1.

The actual average maximum stress of the beam likely lies somewhere within the 7,000 psi to 7,500 psi range. However, due to the fact that this is a small window of stresses, all within 6.7% of each other, and the fact that the results of Equation 1 lie within this range, it should suffice to say that these models are accurate.

In the future, there are several more studies that can be made using this method for constructing accurate beam models. The Forestry Department has expressed an interest in examining the effect of joint placement on the stress in the beams. Ultimately, the beams will be constructed by unskilled labor in a factory setting, and the Forestry Department would like to relax the construction requirements so that workers will find the beams easier to construct. The effect of joint placement can easily be studied using the modeling methodology in this thesis by progressively constructing models with more and more joints spaced closely together.

The Forestry Department would also like to know the effect of eliminating layers from the beam. This will reduce the amount of material used, but could weaken the beam to the point that it would not hold the designed load. The effect of eliminating layers cannot be done without more physical testing. Because all of the beams in this thesis have the same dimensions, the average load found during physical testing was used as an initial condition for the finite element models. Eliminating layers will change the dimensions of the beam, and require a different load to be used. To find this new load would require more testing of the beams with few layers. Then, of course, it would be simpler to use Equation 1 to calculate the stress in the beam.

REFERENCES

[1] Shmulsky, Rubin and Sheldon Shi. "Development of Novel Industrial Laminated Planks from Sweetgum Lumber." Journal of Bridge Engineering. January/February. 2008. 64-66.

[2] Green, David W. et al. "Chapter 4: Mechanical Properties of Wood." The Wood Handbook: Wood as an Engineering Material. Madison, WI: U.S. Department of Agriculture Forest Sevice, 1999. 4.1-4.45.

[3] Moaveni, Saeed. Finite Element Analysis: Theory and Application with ANSYS. Third Edition. Upper Saddle River, NJ: Pearson Prentice Hall, 2008.

APPENDIX A

PHYSICAL TEST DATA COLLECTED BY THE MISSISSIPPI STATE UNIVERSITY FORESTRY DEPARTMENT

Table 4

Results of the Forestry Department's Physical Testing

Specimen	**Depth** (in.)	**Width** (in.)	**Pmax** (lbf)	**S** (psi)
1	5.746	12	22,338	6,808
2	5.717	12	24,806	7,638
3	5.703	11.938	21,425	6,664
4	5.74	11.938	20,081	6,165
5	5.776	12	23,180	6,990
6	5.728	12	22,799	6,992
7	5.755	12	22,979	6,982
8	5.723	12	24,639	7,571
9	5.715	11.938	24,008	7,436
10	5.716	11.938	22,206	6,875
11	5.763	12	25,293	7,664
12	5.744	12	19,779	6,033
13	5.702	12	22,314	6,907
14	5.726	12	21,561	6,616
15	5.7	11.938	22,433	6,984
16	5.74	11.75	22,743	7,094
17	5.77	12	24,905	7,527
18	5.66	12	23,491	7,379
19	5.7	12	23,556	7,296
20	5.7	12	21,622	6,697
21	5.66	12	23,687	7,440
22	5.64	12	25,458	8,053
23	5.68	12	26,938	8,402
24	5.72	12	23,892	7,348
25	5.72	12	24,135	7,423
26	5.64	12	24,932	7,887
27	5.71	12	20,467	6,317
28	5.7	12	24,906	7,714
29	5.79	12	24,998	7,503
30	5.76	11.875	25,042	7,675
Count	30	30	30	30
Max.	5.79	12	26,938	8,402
Min.	5.64	11.75	19,779	6,033
Mean	5.718	11.977	23,353	7,203