Guitar Materials Testing

Overview

The goal of this project was to create a roadmap whereby we can 1) prepare samples of different materials to some spec that can be 2) used to obtain properties/values through physical tests that can then be 3) used inside of an ANSYS simulation model to help us make informed recommendations about how a particular material will behave when sized/shaped as a guitar neck wherein string tension would be applied.

Project Sponsor

Rachel Rosenkrantz sponsored this research, worked with the project team regularly, and, among other things, provided the following preamble: https://youtu.be/qKdXiBAvHL4

Workflow

The following sections describe what future teams would need to do to **1)** prepare a sample (all the constraints, variables, boundaries, ideal parameters, etc.) physically and, **2)** memorialize that information into an accessible understandable document (this report). A section addresses specifically **3)** the different tests that *could* be done to a neck and their strengths and weaknesses, pros and cons for guitar-building related applications, and **4)** how and what information needs to be obtained from the physical tests in order to make the ANSYS model more robust. Another section addresses **5)** how we navigate the ANSYS model developed by our team, input these values, and draw conclusions.

How the ANSYS Model Works With Known Materials (like wood)

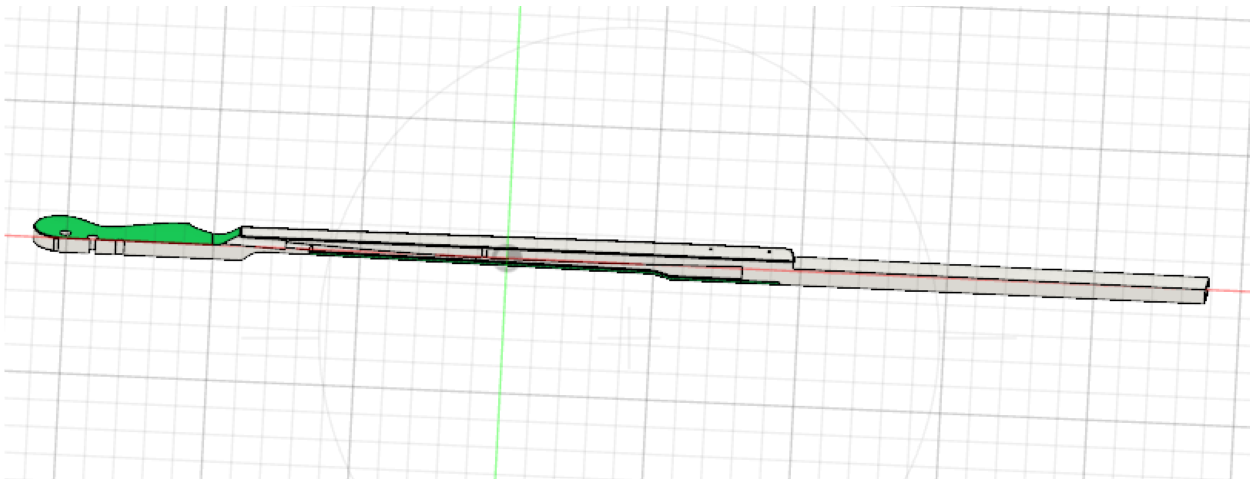
As a baseline for comparison, we will begin by simulating the stress of the strings on a maple neck.

Pre-Simulation Preparation

3D Model of the Guitar Neck

Before simulation in ANSYS, we must have a 2D or 3D model of the neck shape we are looking to evaluate. In the following example, we used Rachel's fusion 360 files and took a 2D cross section of the neck for use in ANSYS.

To get this cross section, use the extrude cut feature on a sketch to slice the neck in half, similar to what a section analysis view would show. Use the “Project/Include” tool in sketch settings and select the outer lines of the cross section to make a new drawing. Save this drawing as a DXF file by right clicking on the cross section sketch in the sketches folder. Import this into a new fusion 360 file, and using the appropriate surface tools, create a surface from the drawing. For more details on how to use surface tools, refer to the Autodesk Fusion 360 help forums. Save this file as igs or an iges file type.

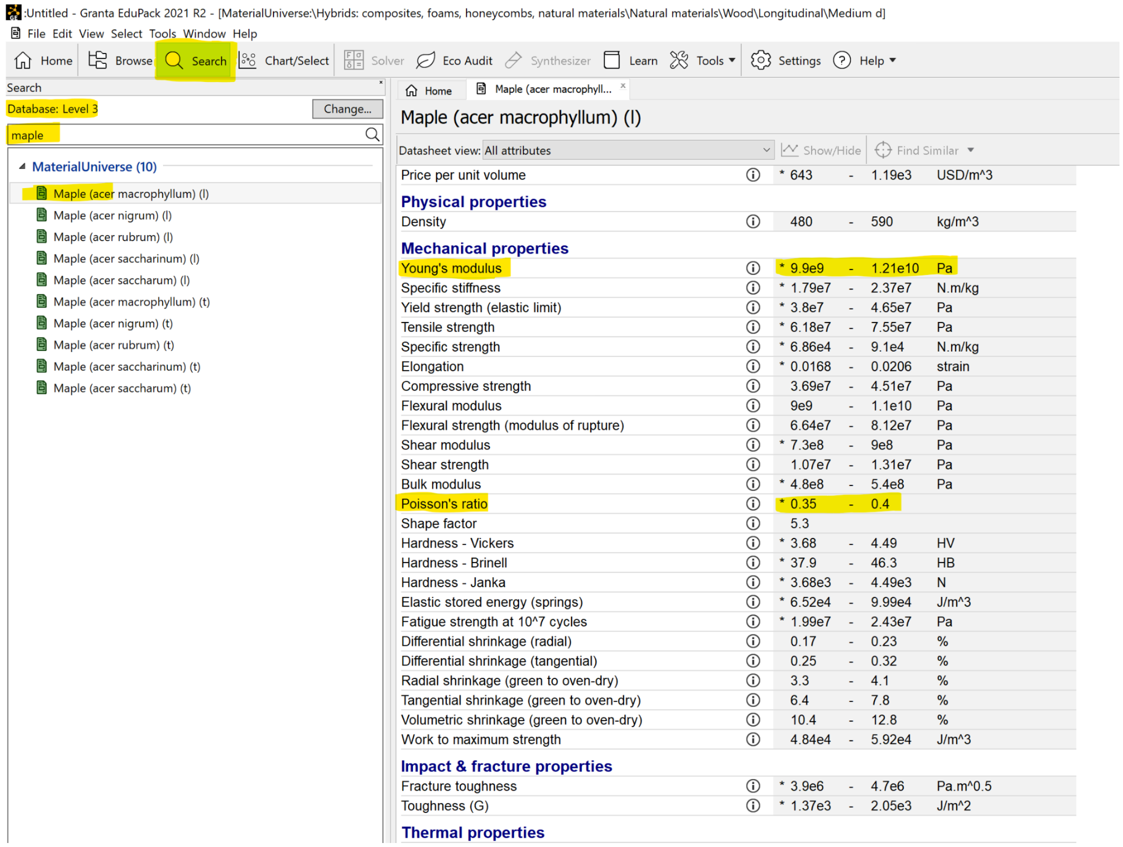


String Tension on the Guitar Neck

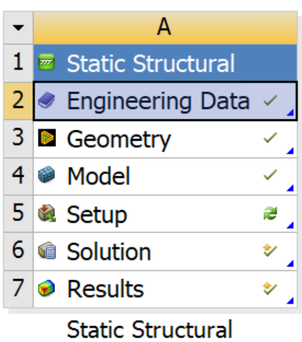
There are several different online resources that can calculate the tension for you depending on the gauge and scale length of the neck. (See [Stringjoy Guitar String Tension Calculator]) This is the force that we will apply in the simulation. For 9 gauge nickel wound strings, the total tension on the neck was about 90 lbs.

Finding the Material Properties of Wood (or another common material)

The best resource for finding the material properties of a specific type of wood is Granta EduPack 2021, which can be found on the ARC-Lab 1 server (connect via remote desktop, Hostname: arc-lab1.wpi.edu). Navigate to the Level 3 database on the startup menu, and use the search option to look up the desired material. Click on the material to bring up the material summary, which includes a long list of material properties. Find the Young’s Modulus and Poisson's Ratio of the material: we will use these properties in the ANSYS simulation.



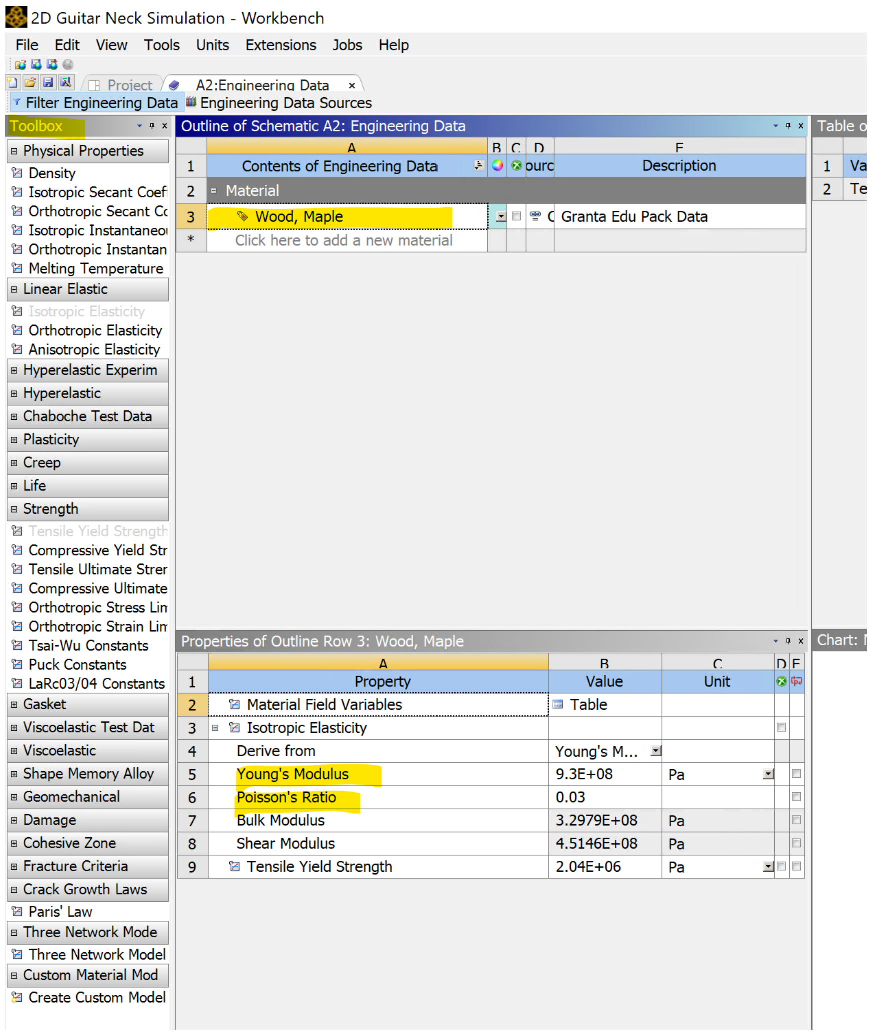
Making the ANSYS Simulation



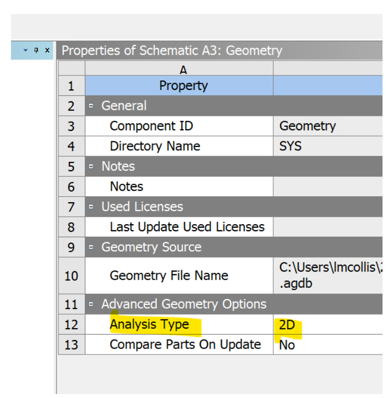
ANSYS Workbench can be found on [these]() WPI servers. On the startup menu, drag the Static Structural Analysis System from the Toolbox onto the Project Schematic. There should be a project with 6 sections listed underneath Static Structural: Engineering Data, Geometry, Model, Setup, Solution, and Results. Work through these sections in order to create the simulation. I've included some important notes relating to each step below. For more information on how to use ANSYS, refer to the ARC's canvas site for projects, as well as through the ANSYS training course held by the ARC. For specific questions, email Dr. Hera from the ARC, who consulted on the creation of this simulation.

Engineering Data

In the Engineering Data workspace, we can add the materials of our model and the properties of those materials. To add a new material, click on an empty row and input the name of the material. The menu below will show an empty table of properties. To add properties of our material to this table, click and drag the appropriate property from the toolbox on the left. In the case of this simulation, we called our material “Wood, Maple” and dragged Isotropic Elasticity and Tensile Yield Strength from the toolbox into the properties table. We then added Young's Modulus and Poisson's Ratio (tensile yield strength is optional) in the drop down menu.



Geometry

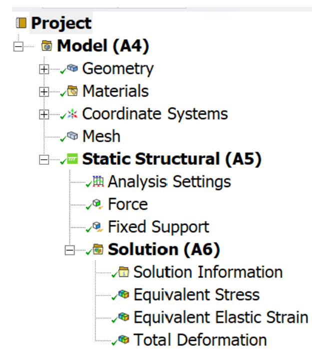


Before opening the geometry workspace, ensure that the analysis type is correct. Change the analysis type in the right hand properties of the schematic table, as shown to the right, under advanced geometry properties. In the case of a 3D simulation this will not need to be changed.

There are several different geometry workspaces built into ANSYS, all of which work similar to most CAD softwares. For this project we used Design Modeler, but choose the workspace you feel most comfortable working with. Open the workspace of your choice and import the Fusion 360 file of the 2D guitar neck cross section.

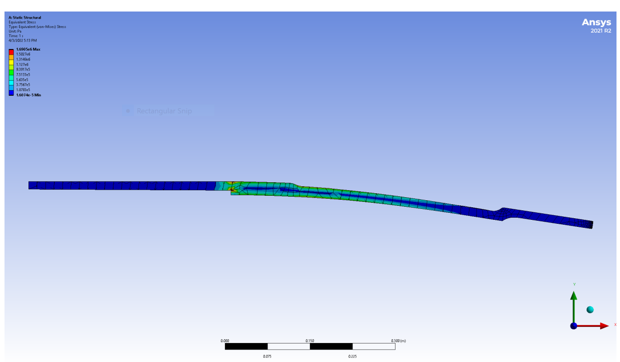
Model and Results

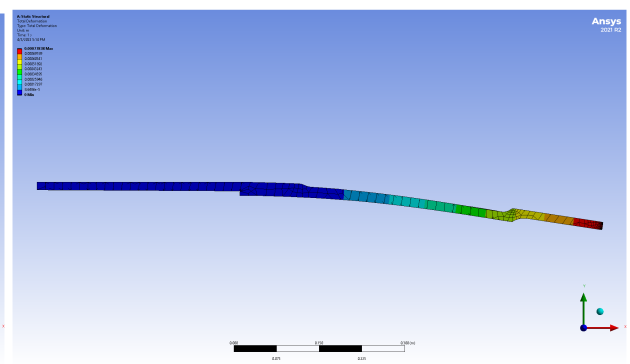
In the left hand menu of the model workspace there will be a project tree with several settings we will need to adjust. Some important steps to be aware of, without going into too much detail:

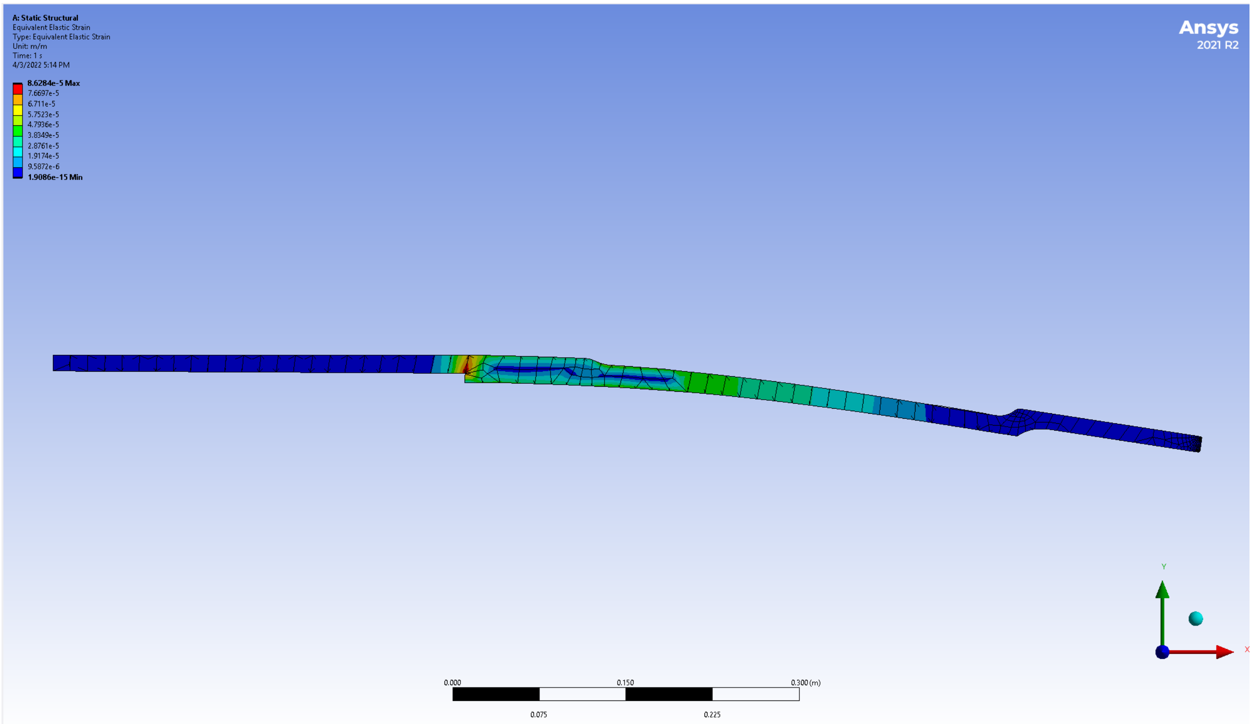


- <u>Materials</u>: assign the materials you added in engineering data to parts of the model.
- <u>Mesh:</u> generate the mesh and adjust mesh size as needed
- <u>Adding a force:</u> right click static structural, then click insert force. In the force details table, you can select the placement, direction, and value of the force.
- <u>Adding a fixed support:</u> right click static structural, then click insert fixed support. In the fixed support details table, you can select the type of support and placement on the model.
- <u>Solution:</u> right click and add any analysis types needed. In this case we added equivalent stress, strain, and total deformation. Click solve in the top menu. If there is an error, or you have skipped a step, ANSYS will show a pop up message describing where the issue is.

Below are the initial simulation results:







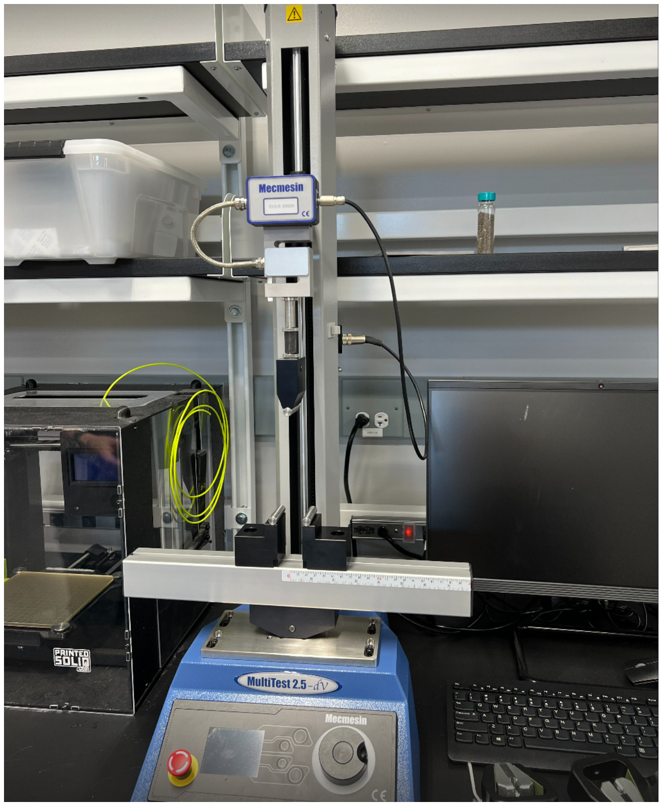
In order to simulate how an *unknown* material will perform under similar tension as a similarly sized neck, we must obtain data from that unknown material to use in our ANSYS model.

Preparing a Sample from Unknown Materials

In order to get an accurate ANSYS model for unknown materials, different constants must be determined. The main properties needed are Young's Modulus and Poisson's Ratio. In order to find these properties, different tests need to be performed on samples of the unknown material. The process to prepare the sample for ANSYS testing is as follows:

- Prepare 9 samples of the material, 3 for each test. 6 of the samples must be 1cm thick, 2cm wide, and around 5cm high. 3 of the samples must be cylinders with a diameter of 2cm and a height of 5cm
- The 6 pieces of material with the rectangular geometry are used for the three-point bending test and uniaxial tensile testing. The 3 cylinders are used for the uniaxial compression testing.

Jack Grubbs, a Ph.D candidate and research assistant at WPI, has offered uses of his equipment in the Cote Research Lab for this project. The three tests, three-point bending test, uniaxial tensile test, and uniaxial compression test, will be performed on a Mecmesin Multi-Test dV(u) system with a 2500N load cell. The tests can be whenever the sample material is ready, likely at some point this summer. The test takes approximately one hour to complete, but will need to be done for each new material.



Available Tests

- Three-point bending test
    - Data produced: Flexural strength
    - The main advantage of this test is that it is similar to the stress that would occur due to strings on a guitar neck.
    - The main disadvantage would be that it doesn't produce as much data to plug into ANSYS as the tensile test
- Uniaxial tensile test
    - Data produced: Young's modulus, offset yield strength, ultimate tensile strength, strain at fracture, and toughness
    - The main advantage of this test would be the amount of data produced
    - The main disadvantage is that it won't provide data about different strain rates of the material. This means it would be unable to determine the strain of material with respect to time.
- Uniaxial compression test
    - Data produced: Ultimate compressive strength
    - The main advantage of this test is it shows the stability of the material

- The main disadvantage of this test is it's not as relevant of the needs of the project as the other tests

Obtaining the Data

From these tests, the values that will be determined are Flexural strength, ultimate compressive strength, Young's modulus, offset yield strength, ultimate tensile strength, strain at fracture, and toughness. The most useful values for the ANSYS simulation are Young's modulus and Poisson's Ratio. Young's modulus can be determined directly through uniaxial tensile testing, and an approximation of Poisson's Ratio can be determined through data from uniaxial tensile testing. With the data from these three tests, the ANSYS simulation would be able to accurately represent the stress a guitar neck made with this unknown material, without actually having to build it.

Implementing the Data

To implement the data into the ANSYS model, change the material through the steps listed under the Engineering Data section of this report. The placement of the force and fixed support should all remain constant, so that the simulation of the contending material can be compared with the simulation of maple, as provided in this report.

The ANSYS simulation makes several assumptions that are not true to the neck in its implementation. Our simulation does not include a truss rod to counteract the force of tension, it assumes isotropic properties of the material (it is difficult to simulate the anisotropic properties of wood) and we have not investigated how the attachment of the neck to the guitar body could affect its performance. Due to these assumptions, the ANSYS model can only tell us so much about the neck's strength.

The best use of the ANSYS results would be to compare the visual deformation, stress, and strain of the contending material neck simulation with a standard maple neck simulation. If the simulation is comparable to wood, we can make the conclusion that the material would behave similarly as a guitar neck. It is difficult to conclude the accuracy of the numerical values provided on the simulation results due to these assumptions.

For future improvements to this workflow, we should investigate ways of reducing these simplifications to our model, and methods of getting more numerically accurate results to draw conclusions from.