

## **ANSYS Absolute Beginner's Guide**

This is a walkthrough on how to open and use ANSYS to complete a static structural analysis on a solid body. Before beginning the analysis, make sure you have the body you would like to model already designed in Solidworks or another CAD program.

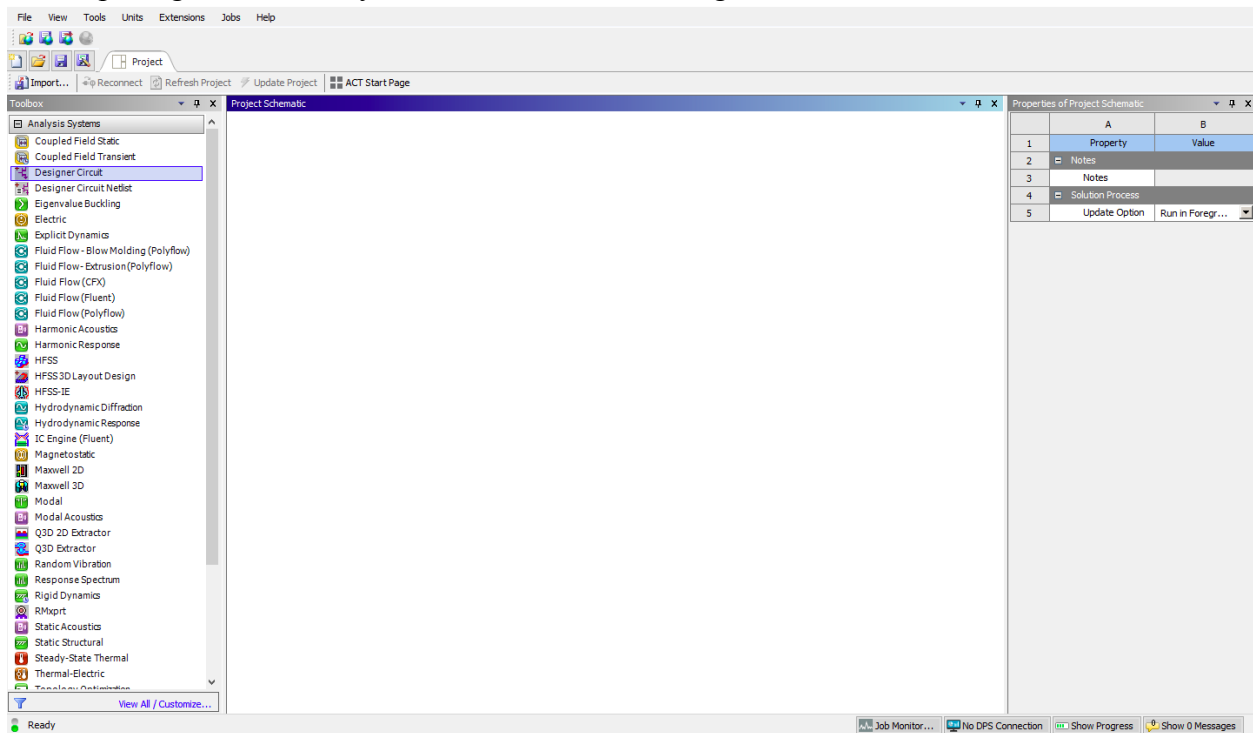
It is helpful to create a sketch or other individually defined entity in Solidworks at any location you wish to apply a load/support/etc. within ANSYS, unless you wish to apply the force/support/etc. over an entire face. This is because in ANSYS, you will need to select the point or area for the load to be applied, and you will need to have defined this area beforehand.

## Preliminary

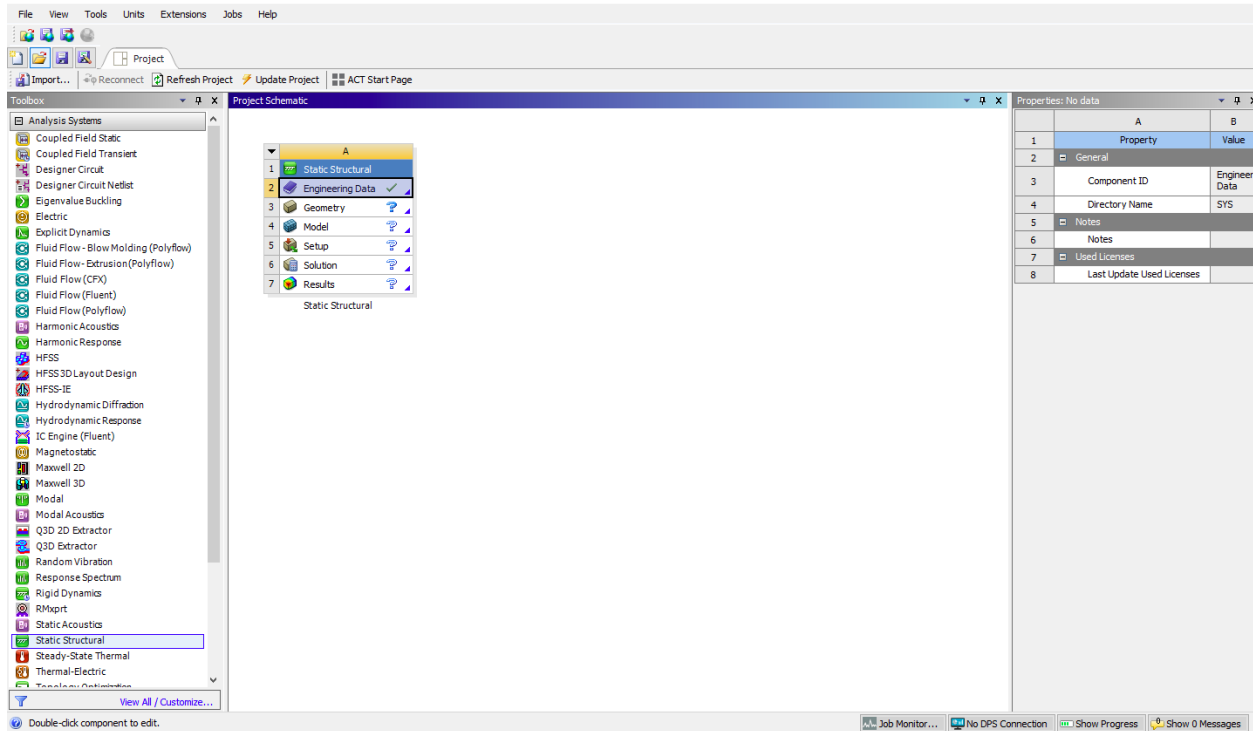
Start by opening ANSYS Workbench 2020. This is available for download, or on WPI computers (elabs.wpi.edu can be used for remote desktop access).

- The application won't always show up in the search bar, so go to Windows in the bottom left > Ansys 2020 R2 > scroll all the way down to find Workbench 2020 R2

After opening Workbench, you should see the following:



In the left navigation tab, find “Static Structural” (under the “Analysis Systems” section) and double click to bring it into the main “Project Schematic” area. It is possible to run multiple analyses at once, in which case you could bring multiple analysis systems into the project schematic area. The figure below shows how the project tab should look.



A brief overview of the sections under your Static Structural analysis shown above:

**Engineering Data:** choose materials for your analysis

- ANSYS automatically includes a material database, but you can also make up your own material or insert one that is not already there by specifying the properties

**Geometry:** details about the physical model you are working with (the model we will bring in from CAD)

**Model:** make changes to the system being modeled, such as the mesh

**Setup:** set up boundary conditions and other necessary components for the analysis

**Solution:** “solving” the given setup

**Results:** results of the solution; these will depend on what you are solving for (deformation, stresses, etc.)

Note that you do not need to worry about these individual parts from here, the steps below walk you through everything you need to do for the setup and solution. This is just to give an idea of what they each mean.

It is important to specify if a 2-D or 3-D analysis is being completed. For this example, it will be a 3-D analysis. To specify this within ANSYS, right click on “Geometry” under your static structural analysis and select “Properties.” From here, a list of selections will show up in the right hand tab. Under “Advanced Geometry Options,” select either 2-D or 3-D. In general, the program default is 3-D analysis.

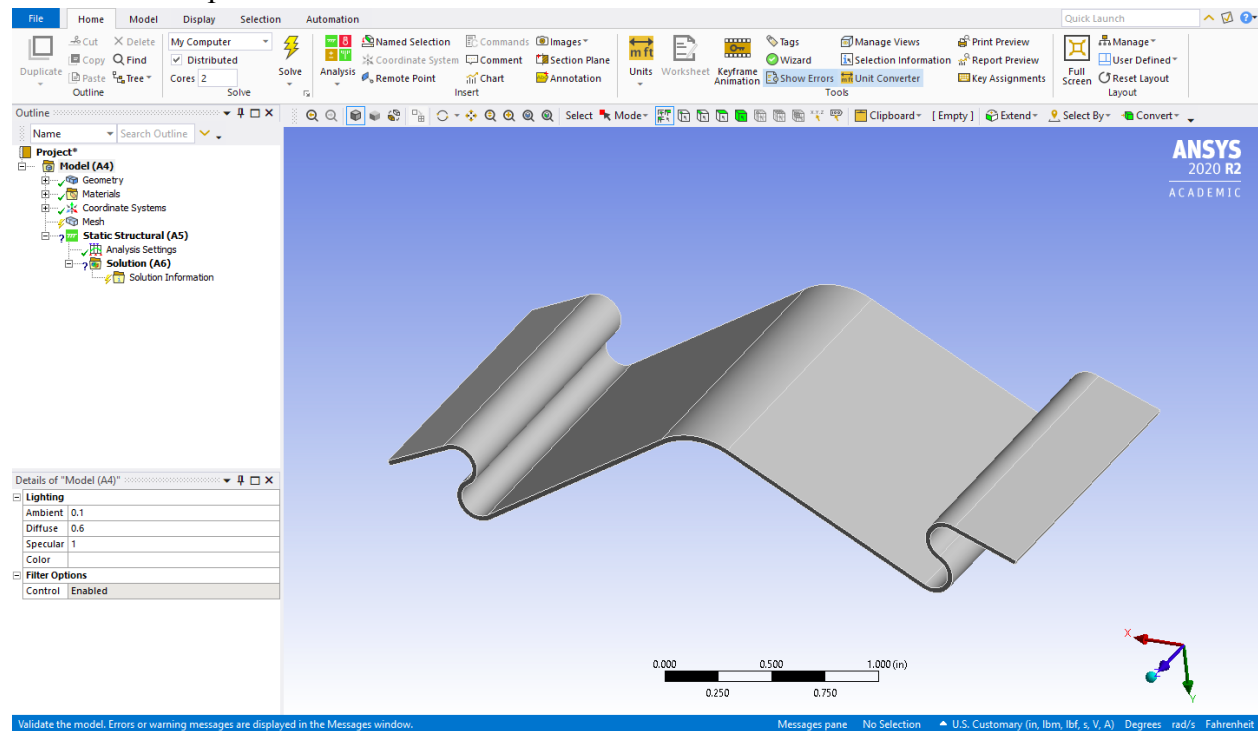
You will see that most of the sections under the Static Structural analysis have question marks next to them. This is because nothing has been specified for the analysis yet. Engineering Data automatically has a green check mark next to it. This is because ANSYS automatically includes structural steel as a default material. If you want to use structural steel, do nothing. If you would like to add more materials, you can double click on Engineering Data and add materials from there. It is also easy to add these materials later, once you have imported your model (I prefer doing it later). If you have opened Engineering Data, you can navigate back to the project by clicking the “Project” tab in the upper toolbar, or just closing the Engineering Data tab (your selections will be saved, you don’t need to leave it open).

## **Geometry**

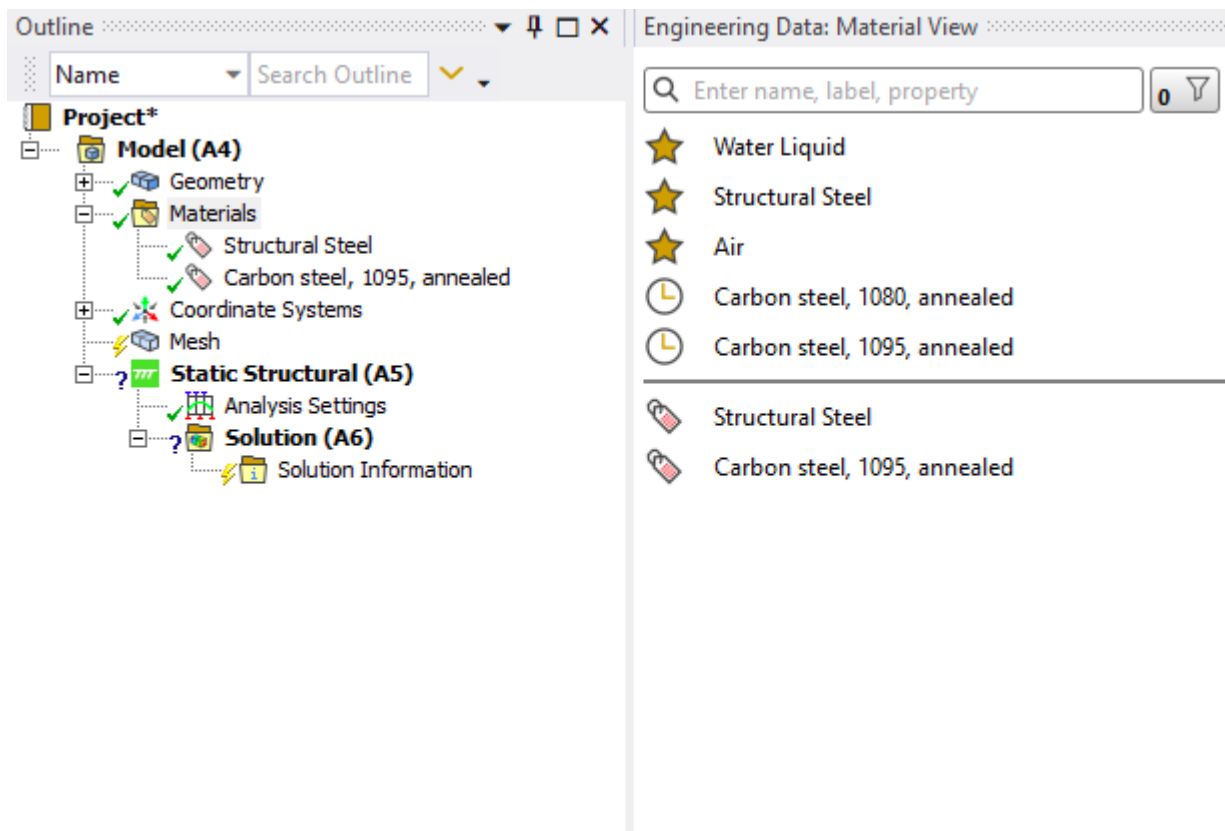
Next, double click Geometry. This will open SpaceClaim, which is a CAD system within ANSYS (if you wanted to, you could create your model here directly, but if you know another CAD system better, that is recommended). To import your model in SpaceClaim, click Open in the top left, and find your model (make sure to look for your type of file, not just .scdoc). Verify that the model is correctly imported into SpaceClaim, and rotate/edit the part from here if desired. Once this is complete, you can close SpaceClaim and navigate back to the workbench (changes will be imported to your project without any additional steps).

## Materials

In the Project Schematic, double click Model. Something similar to the figure shown below should open:



This is where the analysis will be completed. First, add any materials you would like to use if you did not do so earlier. You can do this by clicking “Materials” on the left hand side, then searching for whatever material you want. Click on the plus sign next to the material to add it to your project. It should then appear in your project. In the figure below, an example of this is shown (structural steel was the default, and I added annealed 1095 carbon steel). As you can see, the carbon steel now appears on the left under structural steel.



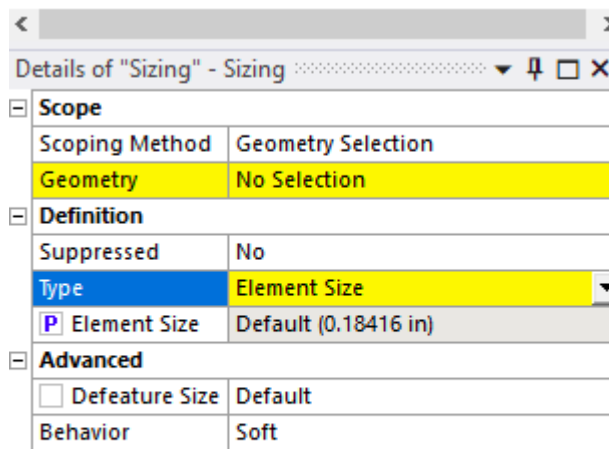
To navigate back to your model, click the lower x on the right side (right above the ANSYS 2020 logo).

If you added a material, additional steps need to be taken to apply it to your solid model. Right click on your material in the left navigation bar and click “Create Material Assignment.” A “new” material will appear for this assignment (for example, I would see “Carbon steel, 1095, annealed assignment” directly under “Carbon steel, 1095, annealed”). To apply the material to your model, first single click on the material assignment in the left tab. Then triple click on the model to select the entire body. The body will be highlighted and an icon with a set of yellow arrows will appear by your mouse; if you hover over the icon you should see “apply selection.” This applies whatever you selected on the left to whatever body or face you selected in the center. So, if you are satisfied with the selection, click the arrows to apply the material selection to the body.

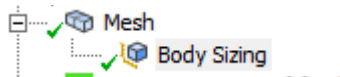
## Mesh

The next step is to generate a mesh. The mesh breaks the body into smaller elements for the analysis. The mesh size should be sufficiently small so that the simulation is accurate. A good way to know if your mesh size was small enough is to run the analysis again with a smaller mesh (more elements). If the simulation yields different results, the initial mesh was insufficient. If you get the same results, that means the mesh size is sufficient. To generate a mesh, single click “Mesh” in the left navigation bar. Then, find the “mesh” tab near the top of the screen, after “Home” and before “Display,” “Selection,” and “Automation.” Once you are in the Mesh tab you will see “Generate” as one of the first options below the tab. Click it to automatically generate the mesh.

If you are unhappy with the size, in the same area you found “Generate,” click “sizing.” The figure below shows what should appear in the bottom left of your screen:

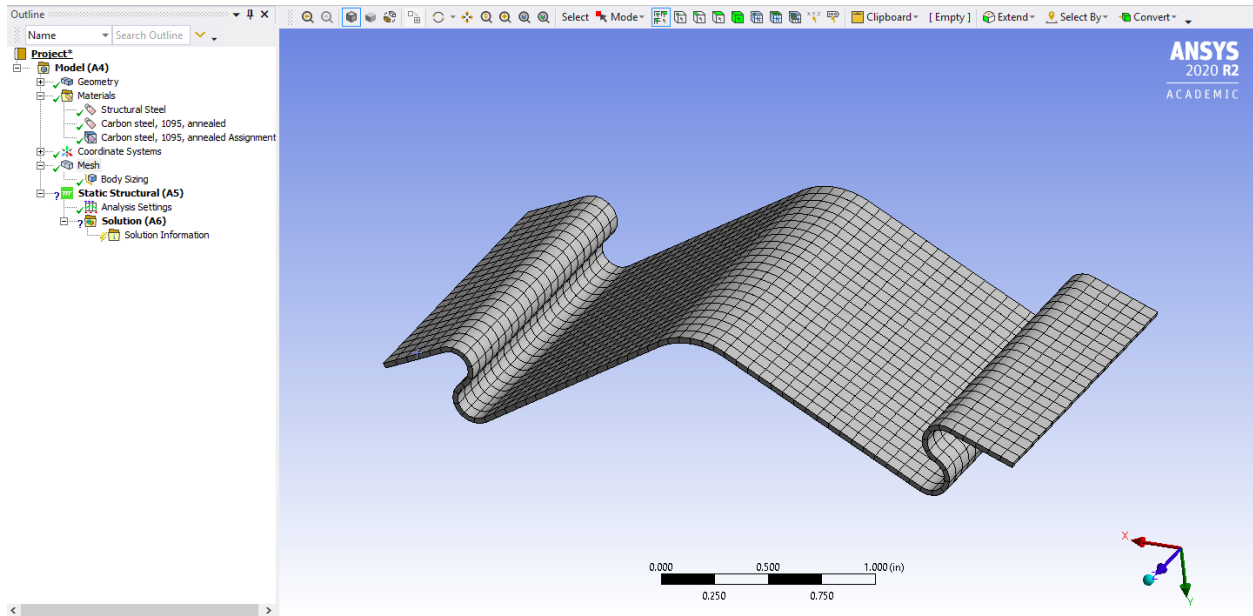


First, make sure to select your geometry by clicking where it says “No Selection” and choosing your entire model (again by triple clicking and hitting apply to selection). Then click in the box next to “Element Size” (above, it currently says “Default (0.18416 in)”). You can change the size by typing in a new number. For example, I will change the above to 0.07 inches for a smaller mesh. Then, right click in the navigation tab above on your sizing feature and click generate mesh.



If nothing happens, you may have chosen too small of an element size and the program is slowed down trying to generate the mesh. You can view the progress of the mesh generation (and any other process in the program) in a loading bar in the bottom left of the screen.



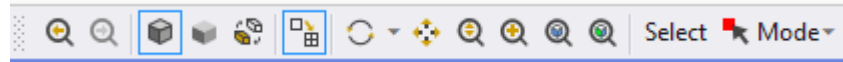


The mesh I generated is shown above, which is of an appropriate element size for my analysis. Once you are happy with your mesh, you can move on to the next part of the analysis setup.

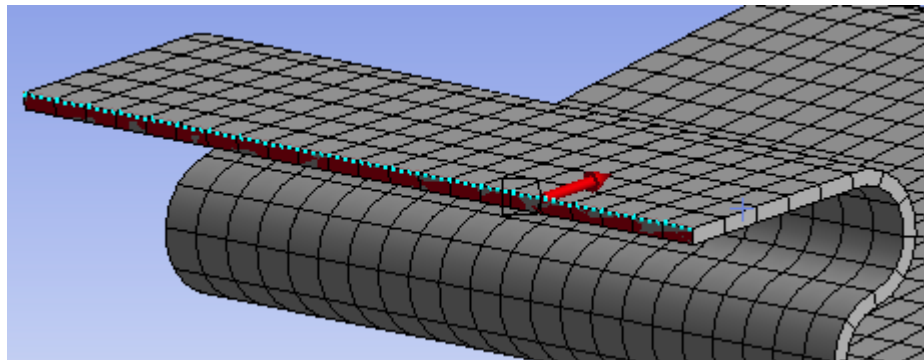
## Static Structural Setup

For the next section, we will apply boundary conditions to the body. These are any forces, supports, displacements, etc. that are applied to your body. Insert these by right clicking “Static Structural” beneath “Mesh,” going to “Insert,” and choosing the appropriate selection. For the spring in its “resting” state in the back of the guitar (when it is not being played or when the vibrato arm is not in use), we add a force, a fixed support, and a frictionless support.

A note before adding these boundary conditions: it will help to toggle between the “rotate” and “select” options in the toolbar when selecting various faces. For selecting, go to select mode (shown at the right below) and choose “single select.”



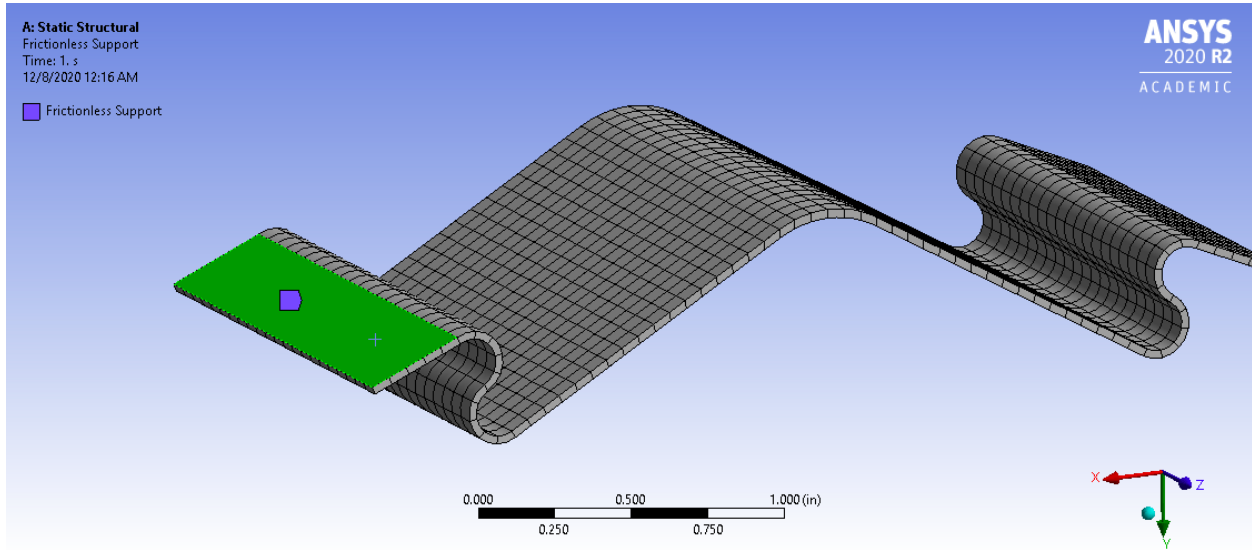
For the force: you need to specify the geometry which the force is applied to, the magnitude of the force, and the direction. For the geometry, select the thin edge (technically a face) on one side of the spring as shown below. Unlike selecting the entire body, which requires a triple click, you only need to single click. However, make sure you select this whole face and not just an edge (it helps to zoom in). Then, as you have done previously, apply the selection to the current scope. For now, set the magnitude as -84.4 lbf. This is the number that should be adjusted for different string gauges. For the direction, select an edge perpendicular to the face to tell the program that the force is along that direction (i.e. normal to the face it is applied to).



Details of "Force"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Force
Define By	Vector
Applied By	Surface Effect
<input type="checkbox"/> Magnitude	-84.4 lbf (ramped)
Direction	Click to Change
Suppressed	No

For the fixed support: the procedure is very similar to the one for the force, but you only need to select the face, not input direction or magnitude. Apply the fixed support to the identical face as the one you chose for the force on the opposite side of the spring.

For the frictionless support: the support is applied to the face shown below. Make sure it is on the same side as the applied force, not the fixed support.

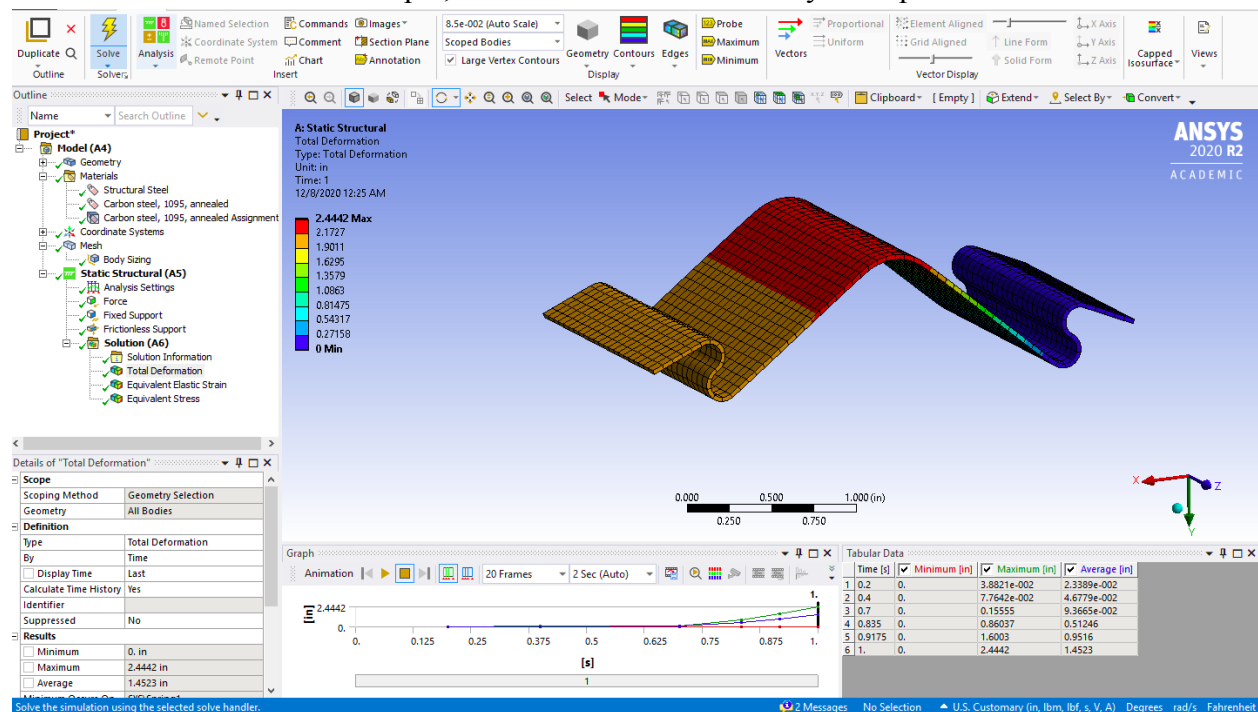


With the boundary conditions applied, you are ready to move on to the solution.

## Solution

Finally, it is time to obtain results of the analysis. Single click “solution” in the left navigation bar, then in the same way you navigated to the Mesh tab, navigate to the Solution tab to the left of “Home.” Here, you can select what results you would like to see. For this project, we are interested in stress, strain, and deformation. You can add these in the Solution tab by simply clicking on what you are interested in seeing. Choose Deformation > Total, Stress > Equivalent (von Mises), and Strain > Equivalent (von Mises). These should all then be added to your left navigation bar under Solution. Then, in the solution tab, click “Solve” (the yellow lightning bolt).

The analysis is then complete! You can view the various results by clicking the relevant item under Solution. For example, the total deformation for my example is shown below:



The shape is abnormal for a guitar flat spring because an experimental shape was used in this example, but this should give an idea of how to set up and use the program to conduct further experiments.

If desired, the project can be saved. It is recommended to save a basic setup, which you can then go back to and re-import the geometry into if it is changed, update material, etc. You will need to check that everything remains the same if you go back to import a new geometric model; this may involve reselecting the faces to apply the boundary conditions to if these have moved, or reassigning the material to the new updated body. The green check marks in the left navigation tab should indicate what needs to be updated.