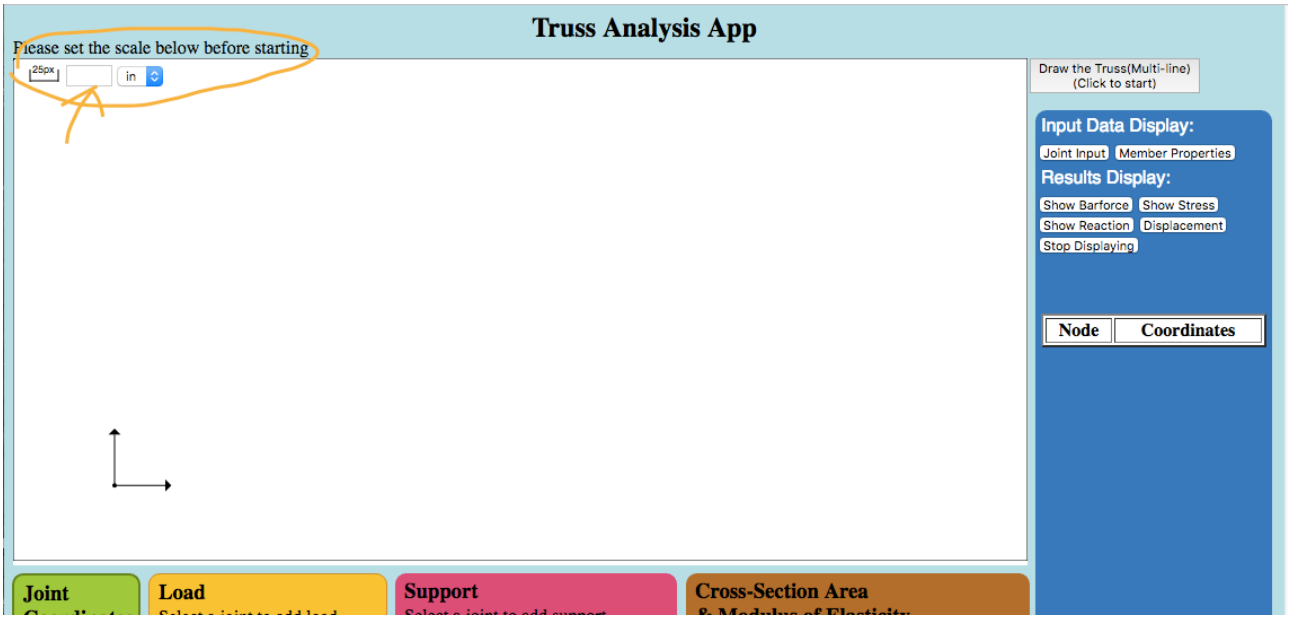
How-to-use Guide:

Step one: Go to biligeyang.github.io/truss-analysis

(Browser choice: Chrome or Firefox would perform well. IE is weird. Don't use IE.)

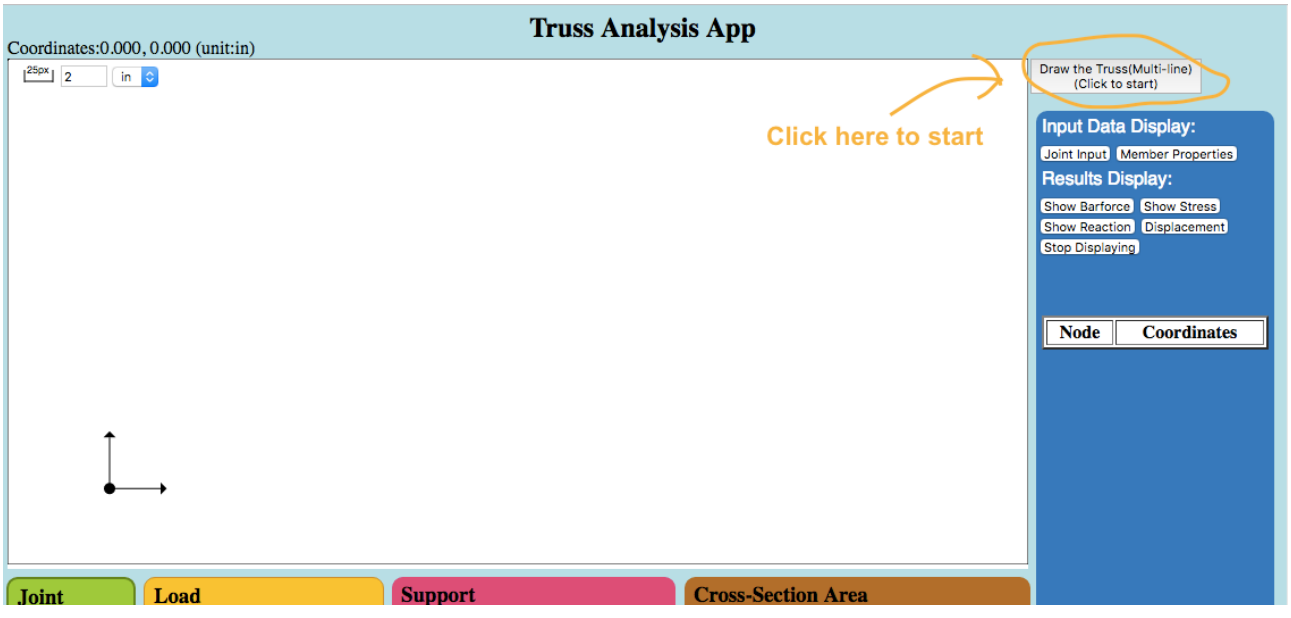
Step two: Set the scale on the upper left corner

The scale determines how many inches every 25 pixels(about half an inch) on the screen would represent. For E6 lab, I think 2 or 3 inches would do.



Step Three: Start drawing

Click the button on the upper right corner and the drawing starts. Every click onto the white canvas will create a dot. If you move your mouse onto an existing dot or an existing line, they would highlight and **if you click on them when they are highlighted**, the new dot you created would join the highlighted dot or line.
Press Escape Key if you want to stop drawing.

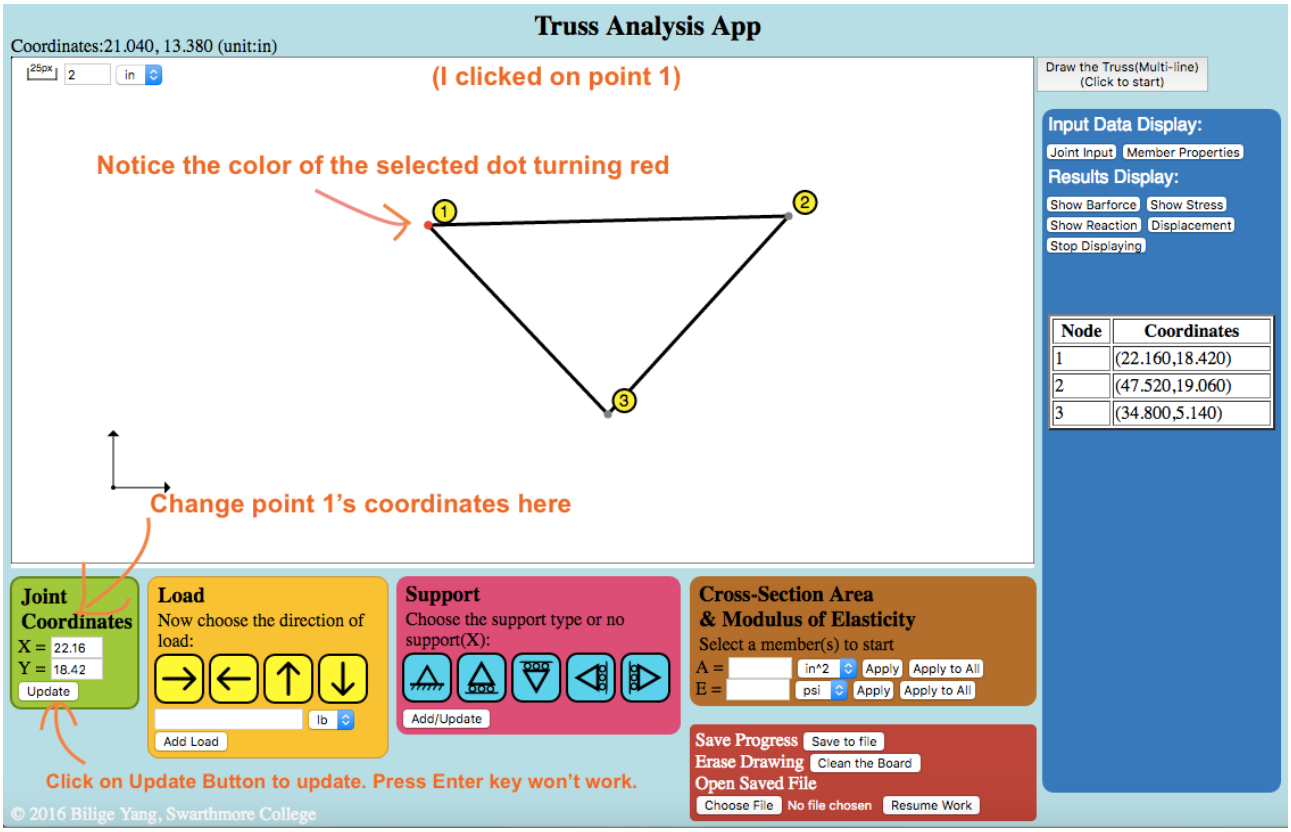


Step Four: Editing existing points (Green Box)

1. You can **drag the points** with your mouse. As a point is dragged, its **coordinates would show** on the upper left corner.

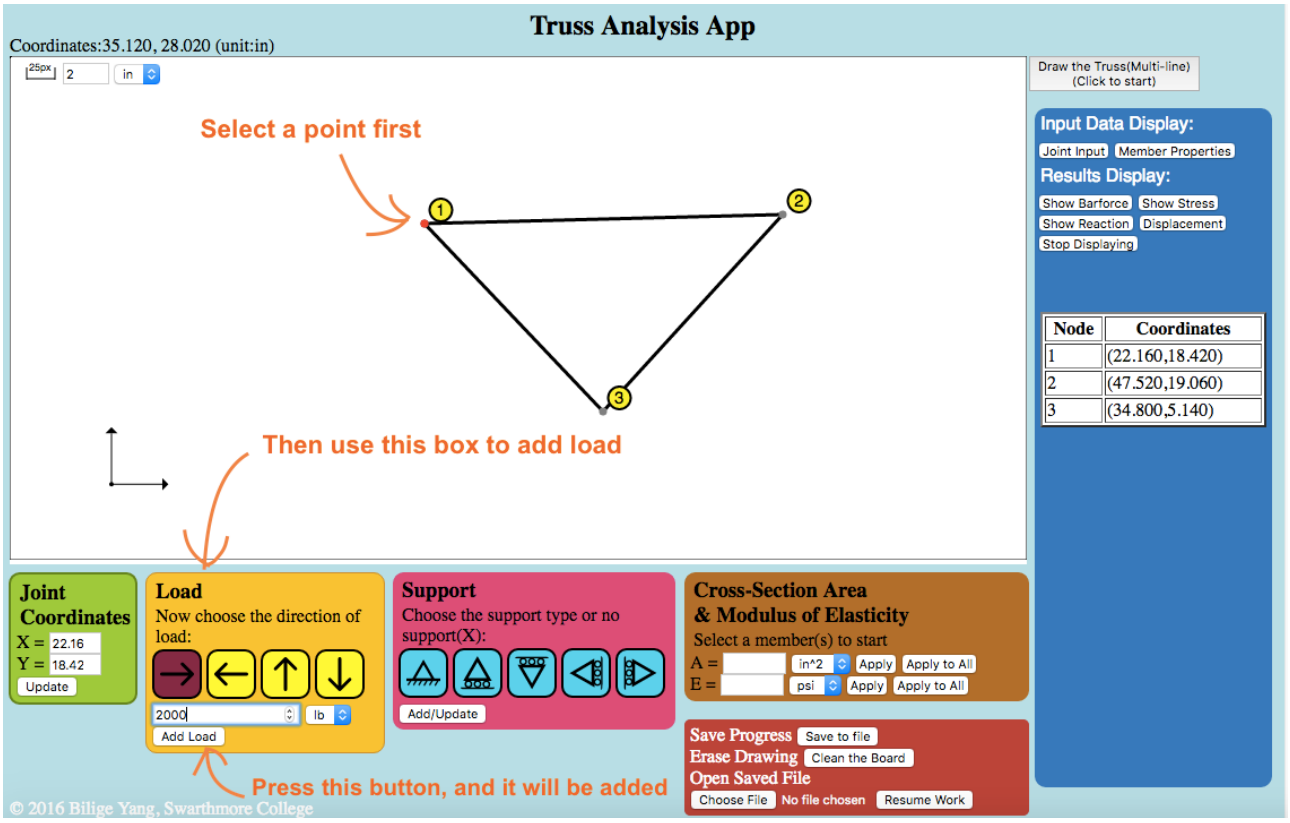
2. You can also **click on a point**, which would turn its color into red. If there is **only one dot clicked,** you can see its coordinates showing up in the **green box** on the lower left corner. You can then **change the numbers** and **click on the update button**. (Important Note: press the Enter key won't update the numbers. **Click** on the button instead.)

3. To **unselect a point,** simply click on the white space.



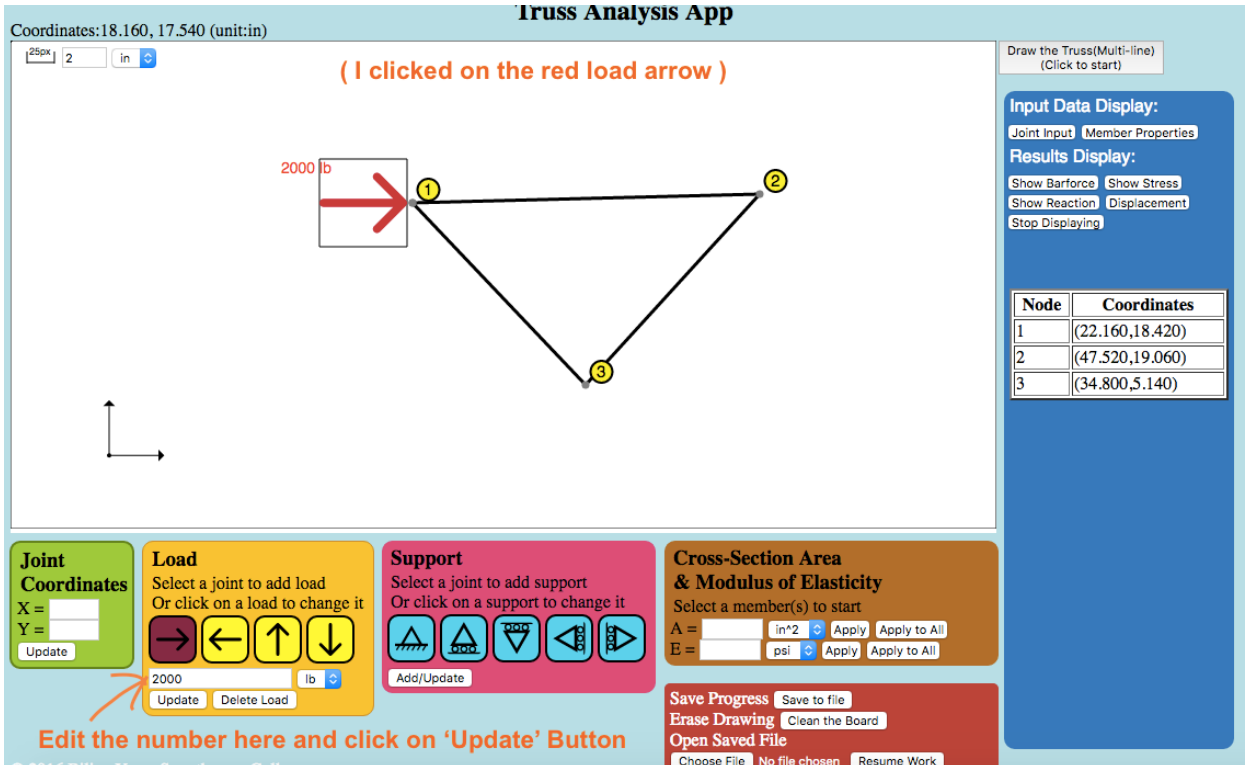
Step Five: Add load (Yellow Box)

1. Click on the point at which you want to add load on.
2. Then in the **yellow box**, select one of the four directions of load by clicking on it.
3. Enter the value of load in pound.
4. Click “Add Load” Button.



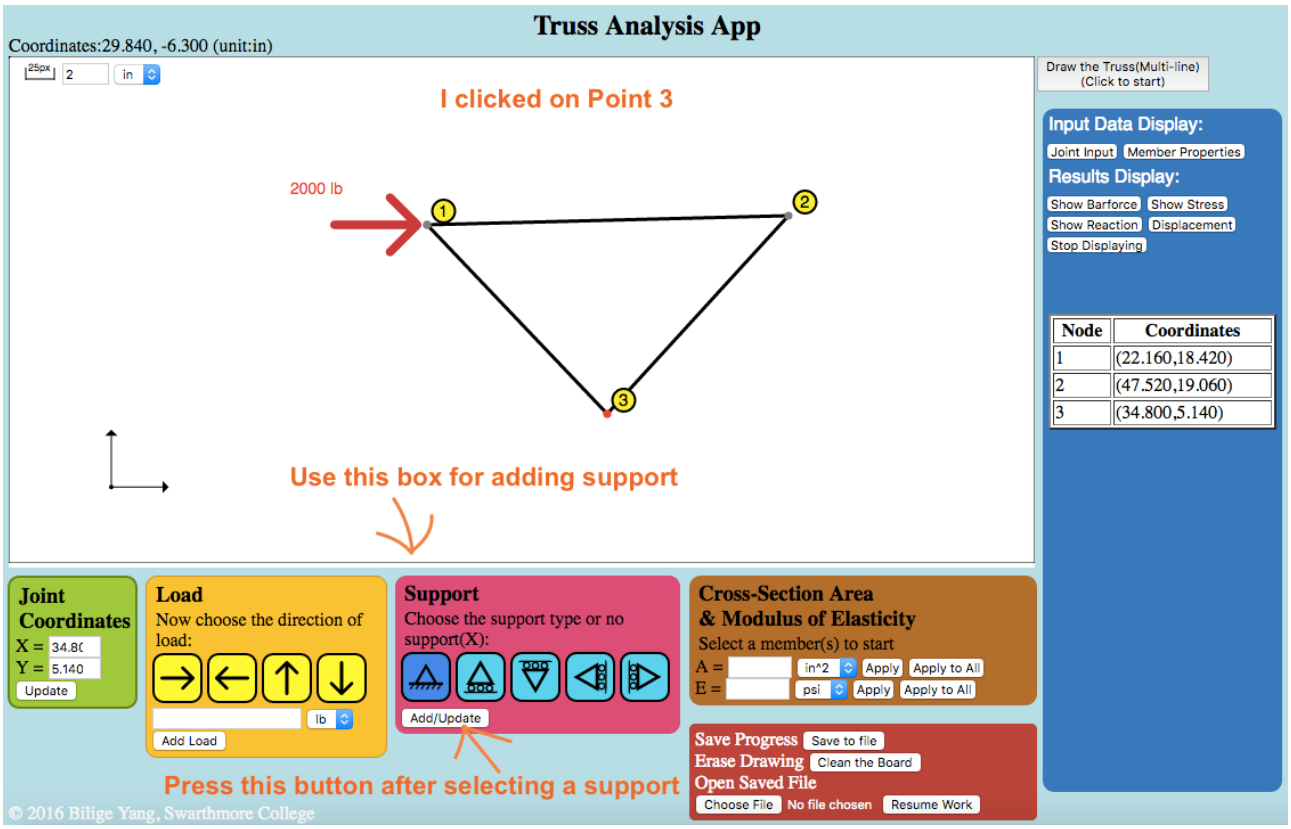
To edit existing load:

1. Click on the red load arrow
2. If you want to change magnitude of load, change the number and click on update.
3. If you want to change the direction of the load, you will have to delete the load first by clicking on “Delete Load” Button. And then add the desired load again. (Sorry for the complexity)



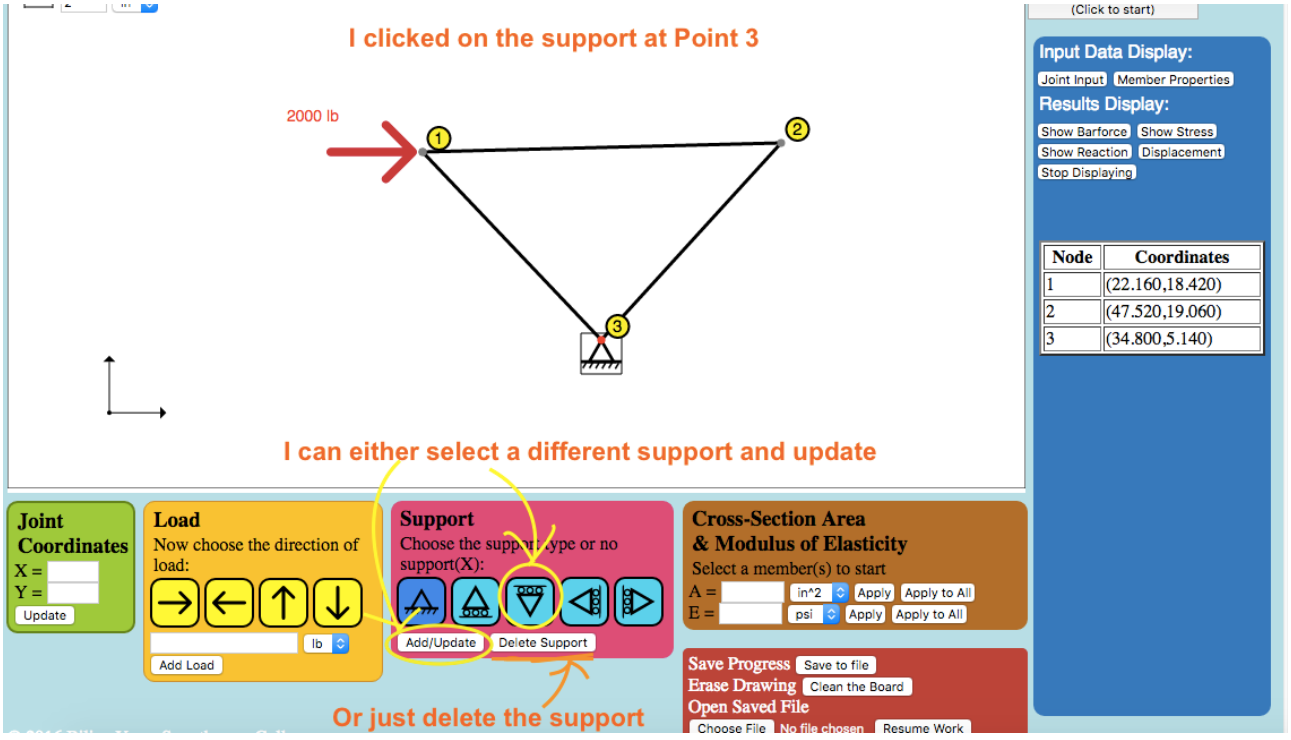
Step Six: Add Support (Pink Box)

1. Click on the point at which you want to add load on.
2. Then in the **pink box**, select one of the five supports by clicking on it.
   (The leftmost is pinned support. The rest four are roller supports to different directions.)
3. Click “Add/Update” Button



To edit existing support:

1. Click on the support you want to change
2. In the pink box, click on a different one and then click “Add/Update”
3. Or click “delete” if you want to delete it



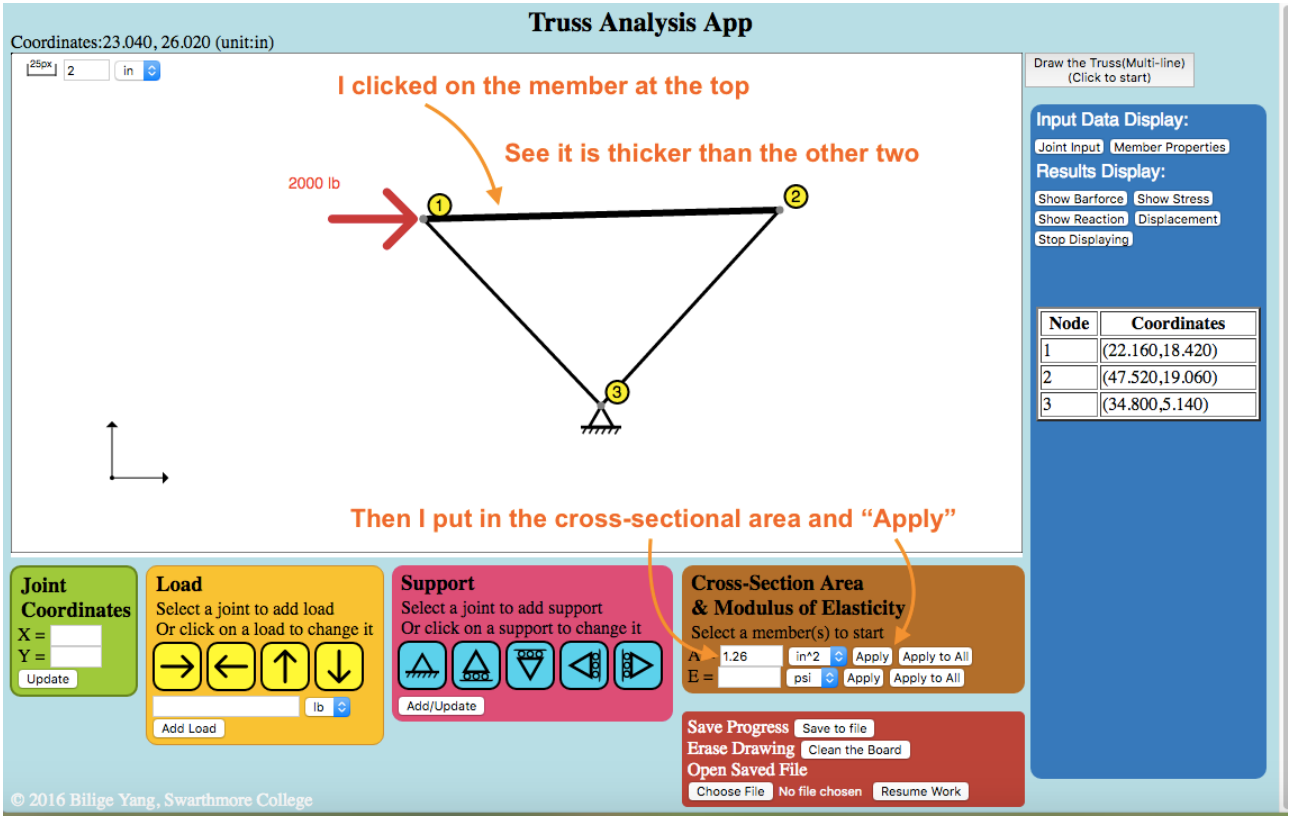
Step Seven: Add Member Properties (Brown Box)

Cross-Sectional Area:

1. If all members have the same cross-sectional area, type the number into A = ___ in the brown box and click “Apply to All.”
2. If not, follow the steps below:
    1) **Click on members** you want to add properties to (*Important Note*: Move the cursor close enough to the member **that the member is thickened and then click**. The click won’t work if the member is not thickened.)
    2) **To unselect all lines**, click on white space.
    3) **To unselect a specific line,** click on it again
    4) Type in cross-section area(A) for selected members and click “Apply.”
    5) Keep selecting and applying until all members’ cross-section area is defined.

Modulus of Elasticity:

1. If all members have the same modulus of elasticity, type the number into E = ___ in the brown box and click “Apply to All.”
2. If not, follow the same step as entering the cross-sectional area above.



For members of the same cross-sectional area, they will be colored in a same color.
For same modulus of elasticity, nothing will happen, since all members usually have same Es.

If you want to **see what you entered for member properties**, see Step Eight on the **next page**.

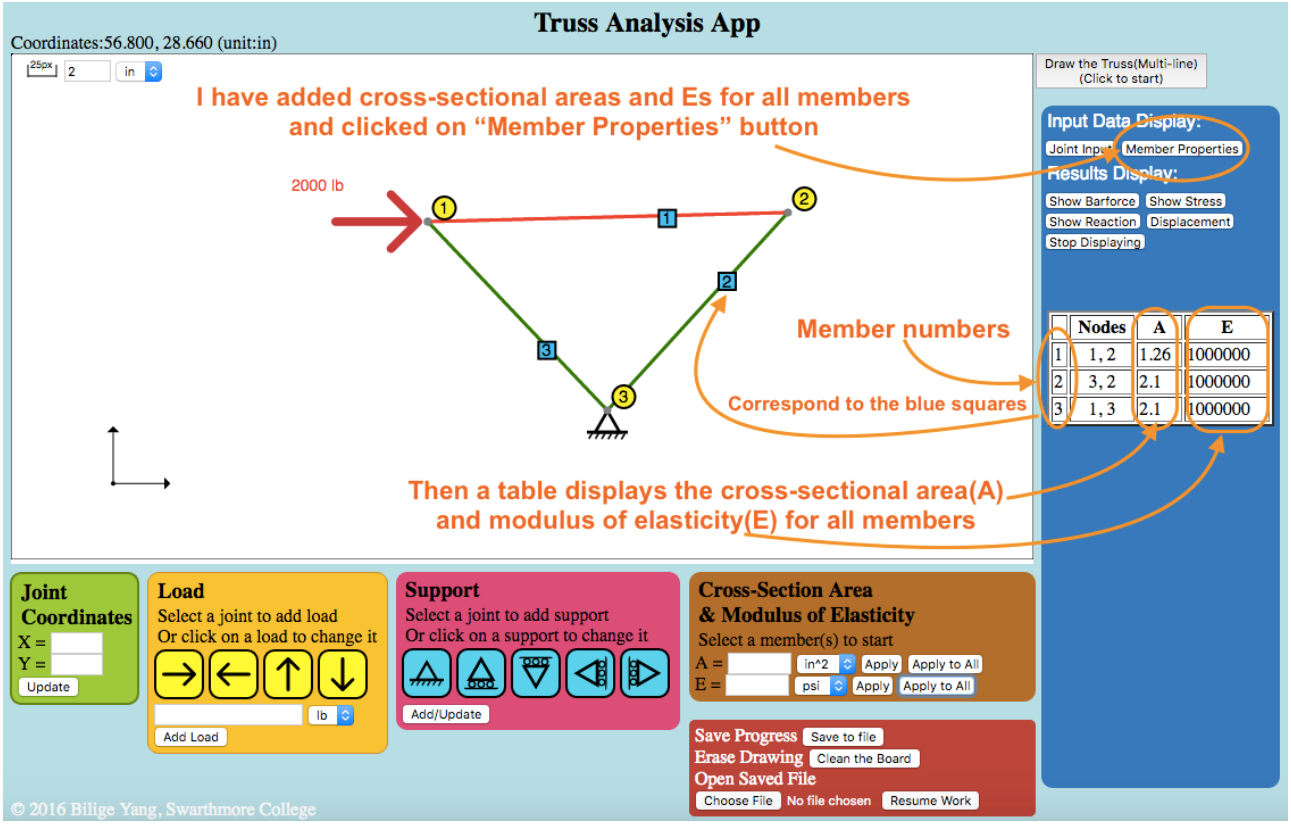
Step Eight: Edit Member Properties

To keep track:

1. To keep track of the cross-sectional areas and modulus of elasticity for the members, there is a great tool in the **Blue Box.**
2. In Input Data Display section **(the top of the box)**, click on "**Member Properties**."
3. Now, the blue box will keep displaying the properties of the members you entered.
4. To get the joint coordinates display back, click on "Joint Input" (the **first button** in the blue box).
5. To quit displaying, click on the "Stop Displaying" Button (the **last button** in the blue box).

To Edit:

Simply reenter the member properties in the **Brown Box.** The new values will cover the old ones.

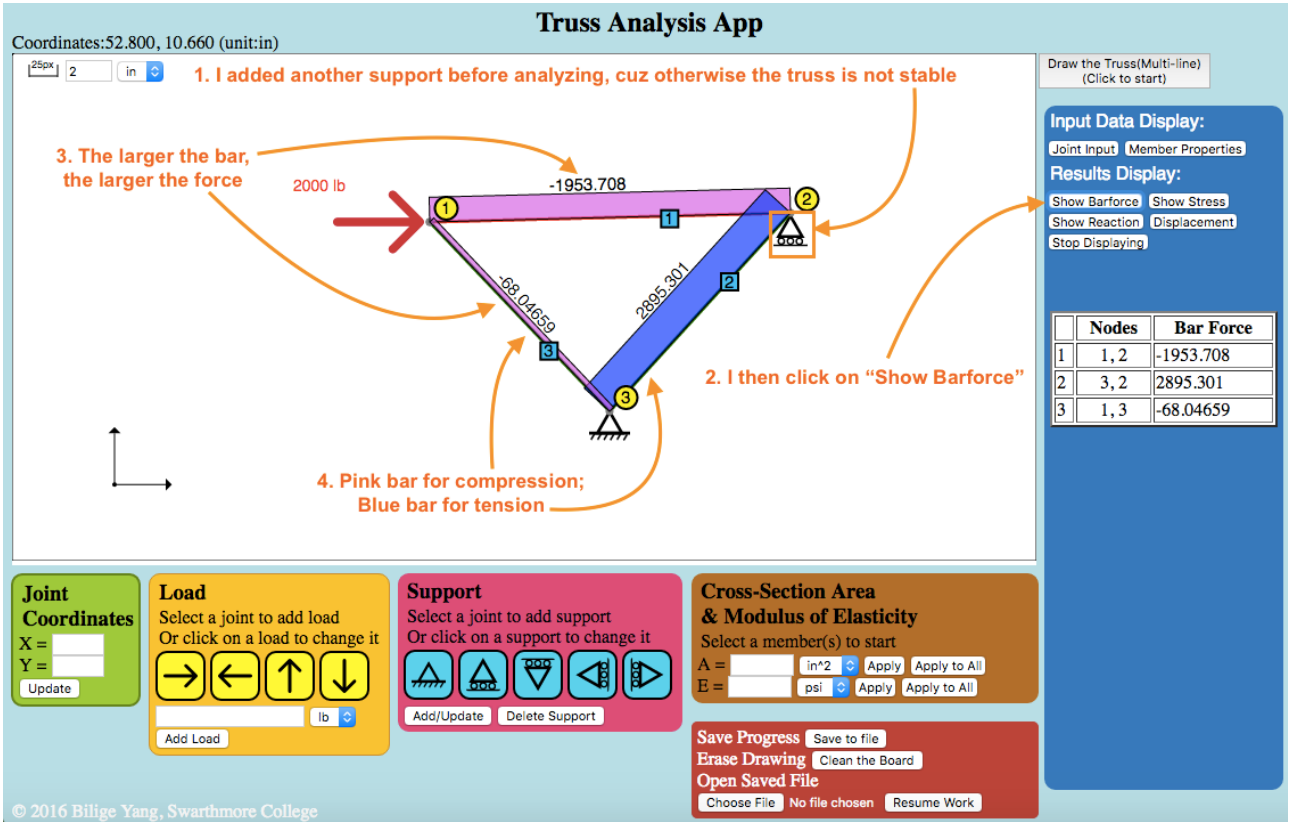


Step Nine: The Results!

After entering the truss geometry, load, support, and member properties, the results can now be calculated. Simply click on the buttons in the “Results Display” section in the **Blue box**. Results will be shown both visually on the truss and in a table in the blue box.

For bar force and bar stress:

1. The larger the bar is the larger the force/stress is.
2. **Blue bar** means the member is in **tension**; **pink bar** means in **compression**.
3. The number above the bar is the corresponding force/stress.
   (Negative number for compression; positive number for tension).



For Reaction:

1. Green arrows with number near the support shows the magnitude and direction of reaction.
2. Reaction results will not be shown in a table but only on the graph

For displacement:

1. The light blue line is the exaggerated deformed truss
2. The table in the blue box shows the displacement of each joint in inches.

Step Ten: Save and Retrieve Progress (Red Box)

You can save unfinished or finished truss design into a local file.

Save:

1. Click on “Save to file” Button in the **Red box**.
2. A window will pump out for you to select the folder and name for the file

Retrieve:

1. Click on “Choose File” in the Red box.
2. Choose the file you saved earlier
3. Click on “Resume Work”

Erase Current Drawing:

Click on “Clean the Board” in the Red box.