HOMEWORK 7

This problem is in addition to problems assigned on the Course Calendar.

**Problem 1**: In this problem, you will build an Arcade model of a (very) simple structure from scratch and check that its response corresponds to hand calculation. The structure is shown below:

![Structure Diagram]

a) Given that the cross section area of the bar is 4 square inches, and the material is steel (E modulus = 29,000 ksi). Calculate the elongation of the bar when the load is applied.

b) Verify your answer in part a) by creating an Arcade model of the structure and examining its behavior. Detailed steps are described below

- **Create a new model, defining nodes, a support, and a single element.**
  - **Turn on the snap grid.** Click *Settings > Graphic > Grid* in the Settings Menu. In the window above the Build View, click the box *Snap grid on*.
  - **Add two nodes.**
    - Click *Build > Nodes*.
    - Move the mouse to the Build View and add two nodes where one is 10 feet directly above the other (the default grid using a 10’ module).
  - **Add a Truss-1 element.**
    - Click *Build > Elements > Truss-1*.
    - In the palette to the left of the Build View, set the following values
      - *Section Area (in^2):* 4
      - Leave the material set to the default *steel*, which has an E-modulus of 29,000 ksi already defined.
      - You can look at the material properties by clicking on the *Truss-1 Materials* tab.
    - In the build view, click from one node to the other. An element will appear.
- **Add damping**
  - Click **Settings > Damping > Truss-1**.
  - In the area labeled **Truss-1 mass-proportional damping**, click the button labeled **Lite**.
    When the load is applied to the bar in the Arcade model, it makes the bar vibrate. Damping makes those vibrations die out, like shock absorbers on a car. "Mass proportional viscous" means that the amount of damping is proportional to the amount of mass in the bar.

- **Add a support**
  - Click **Build > Supports**.
  - In the palette to the left of the Build View, make sure the following values are set:
    - In the Physical group:
      - **X Motion**: reaction
      - **Y Motion**: reaction
      - **Rotation**: free
    - In the Graphic group:
      - **Draw**: above
        After adding a support, you can change where it is drawn by right clicking on it.

- **Add a load**
  - Click **Build > Loads**
  - In the palette to the left of the Build View, set the following values:
    - In the Physical group:
      - **X (k)**: 0
      - **Y (k)**: -100
      - **Moment (k-ft)**: 0
    - In the Graphic group:
      - **Draw tail at node**
        All loads belong to a load pattern, which is a set of loads that can be scaled up and down together. The default load pattern is called **pat1**. You can change the name of the default pattern or add a new pattern by clicking on the **Load Patterns** tab and using the **Load Patterns** table.

    You can change where a force is drawn relative to its node by right clicking on the force.

- **Adjust graphic properties of the load**
  - Click **Settings > Graphic > Forces**
  - In the **Graphic Scales** group, click the <<, <, >, and >> buttons to adjust the graphic scales until the load looks in scale with the diagram.
• Run the Simulation
  o Start
    ▪ Click Simulation > Start.
  o Check the displacement of the lower node
    ▪ Click General > Show Info
    ▪ In the palette to the left of the Simulation View, in the Node Info group, check the box labeled Displacement.
    ▪ Click on the lower node.
      A pop-up window will appear, listing the X and Y displacement for the node, meaning the distance that the node has moved from its original position.
    ▪ Verify that the computer results agree with your hand calculation in part a).
      The value for Y displacement for the lower node should match your answer for part a), except it will have a negative sign, which indicates that the node moved downward.

• Submit your work.
  o Submitting calculations
    ▪ Include your calculations with the other paper-based homework submissions, including the hard copy of the screen.
  o Submitting the file.
    ▪ Copy the file to the following directory:
      Arch324-Martini-SP04\submit\hw_07\YOURNAME_EMAILID