HW 09 - turn in one week from today in Canvas

Turn in the 2 questions as a single .py file onto canvas. Use comments to clearly indicate which question you are working on. Your filename should end as _py2.py if you use Python2 and _py3.py if you use Python3.

Please read through the TenBar_FullyStressed.pdf file which was HW08 for a Finite Element Method class (EML5526 Prof. N. H. Kim `http://www.springer.com/us/book/9781441917454`).

Notice how this problem can be expressed as a optimization problem, where you are minimizing the overall weight of the structure. You have 10 design variables, which are the cross section areas of each individual truss member. You have 10 constraints, which are the allowable stress in each truss member. You are given a sample Abaqus (Finite Element Analysis) input and output files for the problem. This HW will be prepare you for setting up a practical optimization problem in Python.

HW 09 - turn in one week from today in Canvas

1. You are given a file *TenBarModArea.inp* which is an Abaqus input file. The input file has parameters area01, area02, $\cdots$, area10 which are the cross sectional areas of the truss members. You're job is to create a function called write_areas, which reads *TenBarModArea.inp*, replaces area01, area02, $\cdots$, area10 with user specified cross sectional areas, and saves the modified input file as *TenBarArea.inp*. Run the function with the specific 10 areas as write_areas((0.5, 0.5, 0.5, 0.6, 0.7, 0.25, 0.25, 0.4, 0.3, 0.7)). In this case 0.5 = area01, 0.5 = area02, $\cdots$ 0.7 = area10.

HW 09 - turn in one week from today in Canvas

2. You are given a file *TenBarArea.dat* which is an example Abaqus output file for 10 truss problem. You are to create a function which reads the stress values from the output file. Then print the stress values.

Hint: There are many ways to do this. The stress values are the S11 values between lines 752-763. As it happens with this problem (running Abaqus on the same machine), these line locations will never change! So it's okay to hard code the specific line locations. The stress for element 1 is 1.2872E+05, the stress for element 2 is 1.0838E+05, $\cdots$, element 10 is -1.7681E+04.