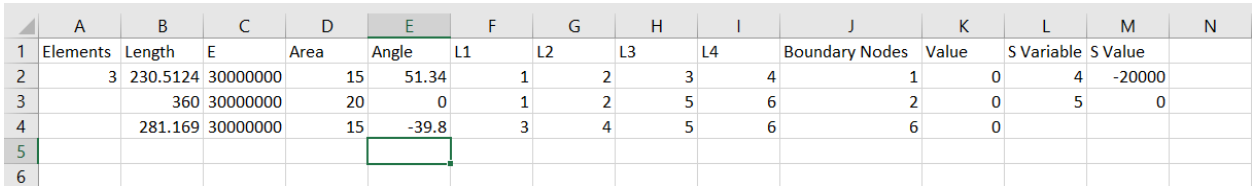
Follow the set of instructions to execute the code successfully.

- All the essential inputs are to be saved in an excel file and the file should be in the same working directory of the code.
- Enter no of elements same as that of provided in excel file.



	A	B	C	D	E	F	G	H	I	J	K	L	M	N
1	Elements	Length	E	Area	Angle	L1	L2	L3	L4	Boundary Nodes	Value	S Variable	S Value	
2	3	230.5124	30000000	15	51.34	1	2	3	4	1	0	4	-20000	
3		360	30000000	20	0	1	2	5	6	2	0	5	0	
4		281.169	30000000	15	-39.8	3	4	5	6	6	0			
5														
6														

For example in the above image each line represents information about an element.

- Enter no of nodes and the code takes the variables of each node accordingly.
- For new file creation follow the same template given for each question.
- Physical units are not taken care of in the code. If it is Metric units give all the values in the metric system only. Same for imperial units.
- For each instance 2 problems were solved and verified with hand solved solutions.
    1. 1D Bar
        Q425.xlsx --- Question no 4.25 of JN Reddy textbook.
        Q434.xlsx --- Question no 4.34 of JN Reddy textbook.
    2. Truss
        Q439.xlsx --- Question no 4.39 of JN Reddy textbook
        Q440.xlsx --- Question no 4.40 of JN Reddy textbook
    3. Beam
        Q507.xlsx --- Question no 5.07 of JN Reddy textbook
        Q511.xlsx --- Question no 5.11 of JN Reddy textbook
    4. Frames
        QA410.xlsx --- Example problem 4.10 of Asghar Bhatti textbook
- Modules used for coding are:
    1. Numpy
    2. Matplotlib
    3. Sympy
    4. Pandas
- 1D bar and truss can be solved with the same code, so there won't be any difference of input file between 1D bar and truss.
- For quick extraction of variables
    1. Assembled stiffness matrix python variable is “K”
    2. Primary variable vector is “U”
    3. Secondary variable vector is “Q”
    4. Internal force term is “IF”
    5. Connectivity matrix is “B”

Follow the set of instructions to understand the input file format.

Truss:

	A	B	C	D	E	F	G	H	I	J	K	L	M	N
1	Elements	Length	E	Area	Angle	L1	L2	L3	L4	Boundary Nodes	Value	S Variable	S Value	
2	3	230.5124	30000000	15	51.34	1	2	3	4	1	0	4	-20000	
3		360	30000000	20	0	1	2	5	6	2	0	5	0	
4		281.169	30000000	15	-39.8	3	4	5	6	6	0			
5														
6														

1. **Angle**: Angle of each truss member is taken from the horizontal.

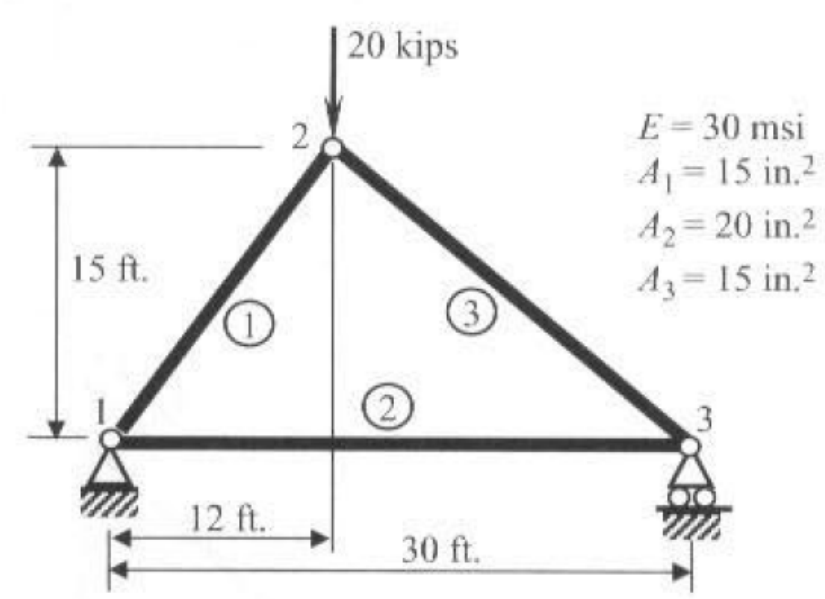


Figure P4.39

2. **L1,L2,L3,L4:** These are the columns of the connectivity matrix. Here in the above example no of nodes is 3, and each node has 2 variables. So node 1(1,2) similarly for node 2(3,4) etc.
3. **Boundary Nodes:** Here we need to enter the Dirichlet boundary Nodes.
4. **Value:** Here we need to enter the corresponding values of dirichlet boundary nodes.
5. **S Variable:** Enter the Neumann boundary nodes.
6. **S Value:** Enter the corresponding values of Neumann boundary nodes.

Note: Same code is used for 1D bar problems so the input file format won't change. Give all angles as 0 and connectivity matrix similar to that of truss.

Uniform Load: This is for beams. Here give the uniformly distributed load value for the corresponding element.

EA: Product of young's modulus and area of the cross section.
EI: Product of young's modulus and area moment of inertia.

THE END

##########