Assignment #2

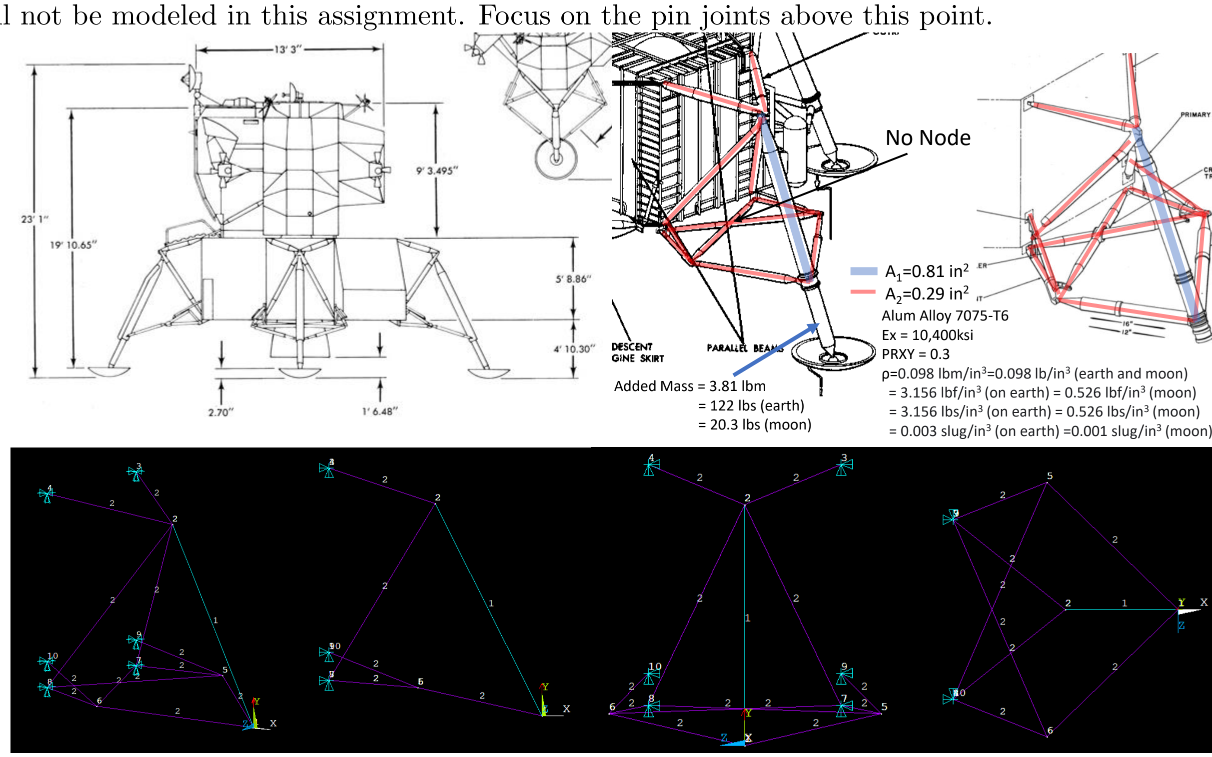
FEA of Apollo Lunar Module Landing Gear

Objective

The objective of this lab is to use ANSYS APDL to model the Apollo Lunar Module landing gear system. Using Link Element #180, you will simulate the two-force members with pinned joints that form the upper part of the landing gear. You will evaluate the structural response under varying gravitational loads and solve for displacements, stresses, and safety factors. A transient simulation will also be conducted to observe the dynamic response of the structure.

Problem Description

You are tasked with using ANSYS APDL to model the upper landing gear of the Apollo Lunar Module (LM). The model origin is at the intersection of the arms with the main strut, as shown in the figure below. The lower section acts as a gas-filled strut, which supports a shear force that will not be modeled in this assignment. Focus on the pin joints above this point.



Your task is to:

- Model the system using Link Element #180.
- Apply a working load of 16,500 lbs in the y-direction to the main strut.
- Perform simulations under three conditions: without gravity, with Earth's gravity, and with the Moon's gravity.

Questions

1. How many nodes and elements were used in your model?
2. What are the displacements of each node?
3. What are the reaction forces?

ANSYS APDL Setup and Simulation

Follow these step-by-step instructions to complete the analysis using ANSYS APDL:

Pre-Processing

1. Define the element type and material properties:

```
Preprocessor > Element Type > Add > Link #180
Preprocessor > Material Props > Material Models > Structural >
    Linear > Elastic > Isotropic > Specify Ex and PRXY
Preprocessor > Material Props > Structural > Linear > Density >
    Specify density in lbm
```

2. Define the real constants (mass properties) and sections:

```
Real Constants > Add > Select Element Type > Link 180
Sections > Link > Add > Section Name: Blue Link > Area: 0.81
Sections > Link > Add > Section Name: Red Link > Area: 0.29
```

3. Create nodes for the structure:

```
Preprocessor > Modeling > Create > Nodes > In Active CS >
Enter all 10 nodes (see node table below)
Menu Bar > List > Nodes (Check nodes)
```

4. Create elements connecting the nodes:

```
Preprocessor > Modeling > Create > Elements > Elem Attributes >
    Set section number to Blue
Preprocessor > Modeling > Create > Elements > Auto Numbered >
    Thru Nodes
Preprocessor > Modeling > Create > Elements > Elem Attributes >
    Set section number to Red
Preprocessor > Modeling > Create > Elements > Auto Numbered >
    Thru Nodes
```

Applying Boundary Conditions and Loads

5. Apply boundary conditions and forces:

```
Preprocessor > Loads > Define Loads > Apply > Structural >
    Displacements > On Nodes (Specify constraints for each node)
Preprocessor > Loads > Define Loads > Apply > Structural >
    Force/Moment > On Nodes
Apply a force of 16,500 lbs in the Y-direction on the main strut.
```

6. Apply gravitational loads:

```
Preprocessor > Loads > Define Loads > Apply > Structural >
    Inertia > Gravity > Global
    Y-Component: 1 for Earth, 0.6 for Moon
```

Solution Phase

7. Solve the static analysis:

```
Solution > Analysis Type > New Analysis > Static > OK
Solve > Current LS > OK
```

Post-Processing

8. Generate results and plots:

```
General Post Processor > Plot Results > Deformed Shape >
    Deformed + Undeformed Shape (/dscale,1,1)
General Post Processor > Plot Results > Vector Plot >
    Displacement Vector
General Post Processor > List Results > Nodal Solution >
    DOF Solution > Displacement Vector Sum
```

9. Plot Von Mises stress and element results:

```
/eshape,1,1
General Post Processor > Result Viewer > Stress > Von Mises Stress >
    Plot Results
General Post Processor > List Results > Element Solution >
    Stress > Von Mises Stress
```

Transient Simulation

10. Conduct a transient analysis:

```
Solution > Analysis Type > New Analysis > Transient > OK
Solution > Analysis Type > Sol'n Controls > End Time: 10 sec >
    Auto Time Stepping: Off > Substeps: 100
Solution > Solve > Current LS > OK
```

11. Generate time-history plots:

```
General TimeHist Post Processor > Click on (+) > DOF Solution >
    Select a node (e.g., Node at the origin)
Click on the graph button to view the Y-displacement vs. time.
Repeat for other nodes if needed.
```

Nodal and Element Data

Nodal List for Pin Joints (Units in Inches)

NODE	X	Y	Z
1	0.0000	0.0000	0.0000
2	-34.860	69.360	0.0000
3	-69.540	80.920	-27.750
4	-69.540	80.920	27.750
5	-40.500	9.2480	-39.300
6	-40.500	9.2480	39.300
7	-69.540	11.560	-27.750
8	-69.540	11.560	27.750
9	-69.540	20.808	-27.750
10	-69.540	20.808	27.750

Element List for Two-Force Members

ELEM	MAT	TYP	REL	ESY	SEC	NODES	
1	1	1	1	0	1	2	1
2	1	1	1	0	2	1	6
3	1	1	1	0	2	1	5
4	1	1	1	0	2	6	7
5	1	1	1	0	2	8	5
6	1	1	1	0	2	8	2
7	1	1	1	0	2	2	7
8	1	1	1	0	2	2	3
9	1	1	1	0	2	2	4
10	1	1	1	0	2	8	6
11	1	1	1	0	2	7	5
12	1	1	1	0	2	10	6
13	1	1	1	0	2	9	5