Assignment #1

FEA of Rigid Plate and Springs System

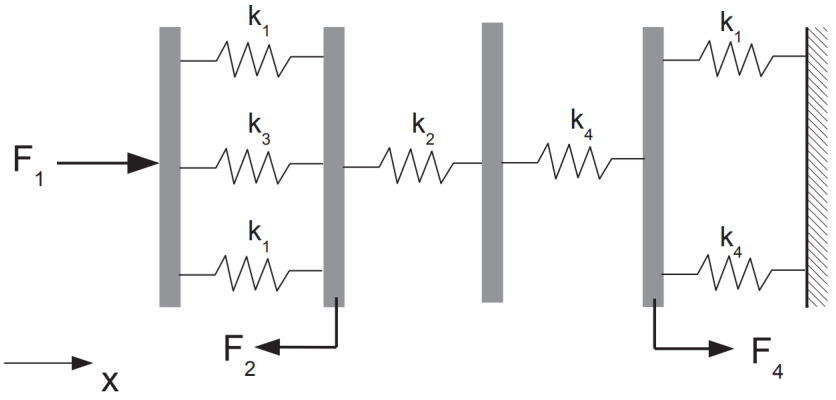
Objective

The objective of this lab is to use ANSYS APDL to perform a finite element analysis (FEA) of a system of rigid plates connected by springs. You will use `Combination Element #14` in ANSYS to simulate the system and analyze the resulting displacements and reaction forces. This assignment will guide you through the steps to set up and solve the problem in ANSYS APDL.

Problem Description

Consider the spring system illustrated below, with rigid plates connected by springs. The plates are spaced 1 km apart. Apply external forces on the nodes as shown in the diagram. Your task is to:

- Determine the number of nodes and elements required for the system.
- Solve for the unknown displacements of the rigid plates and the reactions at the walls.



$k_1 = 250\text{N}/m$
$k_2 = 320\text{N}/m$
$k_3 = 520\text{N}/m$
$k_4 = 200\text{N}/m$
$F_1 = 10\text{kN}$
$F_2 = -2\text{kN}$
$F_4 = 5\text{kN}$

Questions

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

ANSYS APDL Setup and Simulation

Follow these step-by-step instructions to perform the analysis in ANSYS APDL:

Pre-Processing

1. Define the element type:

```
Preprocessor > Element Type > Add > Combination > Link 14
```

2. Define the real constants (spring constants):

```
Preprocessor > Real Constants > Add > Select Element Type >
Specify K1, K2, K3, K4 (Spring Constants)
```

3. Create nodes for the rigid plates spaced 1 km apart:

```
Modeling > Create > Nodes > In Active CS >
Node 1 (0,0,0) > Apply
Node 2 (1000,0,0) > Apply
Node 3 (2000,0,0) > Apply
Node 4 (3000,0,0) > Apply
Node 5 (4000,0,0) > OK

Menu Bar > List > Nodes (Check nodes)
```

4. Create elements connecting the nodes:

```
Modeling > Create > Elements > Auto Numbered > Thru Nodes
Real Constant Set 1 > (Nodes 1,2) > Apply
Real Constant Set 2 > (Nodes 2,3) > Apply
Real Constant Set 3 > (Nodes 3,4) > Apply
Real Constant Set 4 > (Nodes 4,5) > Apply

Menu Bar > List > Elements (Check elements)
```

Applying Boundary Conditions and Loads

5. Apply boundary conditions:

```
Loads > Define Loads > Structural > Displacement >
Select Node 5 > All DOF > Displacement Value = 0 > OK

Select Nodes 1, 2, 3, 4 > UY and UZ > Displacement Value = 0 > OK
```

6. Apply forces:

```
Loads > Apply > Structural > Force/Moment > On Nodes
Select Node 1 > FX Direction > Load = 10000 N > OK
Select Node 2 > FX Direction > Load = -2000 N > OK
Select Node 4 > FX Direction > Load = 5000 N > OK
```

Solution Phase

7. Solve the static analysis:

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

Post-Processing

8. View displacement and reaction results:

```
General Post Processor > List Results > Nodal Solution >
DOF Solution > X Displacement > OK

General Post Processor > List Results > Reaction Solution > OK
```

9. Plot results:

```
General Post Processor > Plot Results > Vector Plot > Predefined >
Translational > OK
```