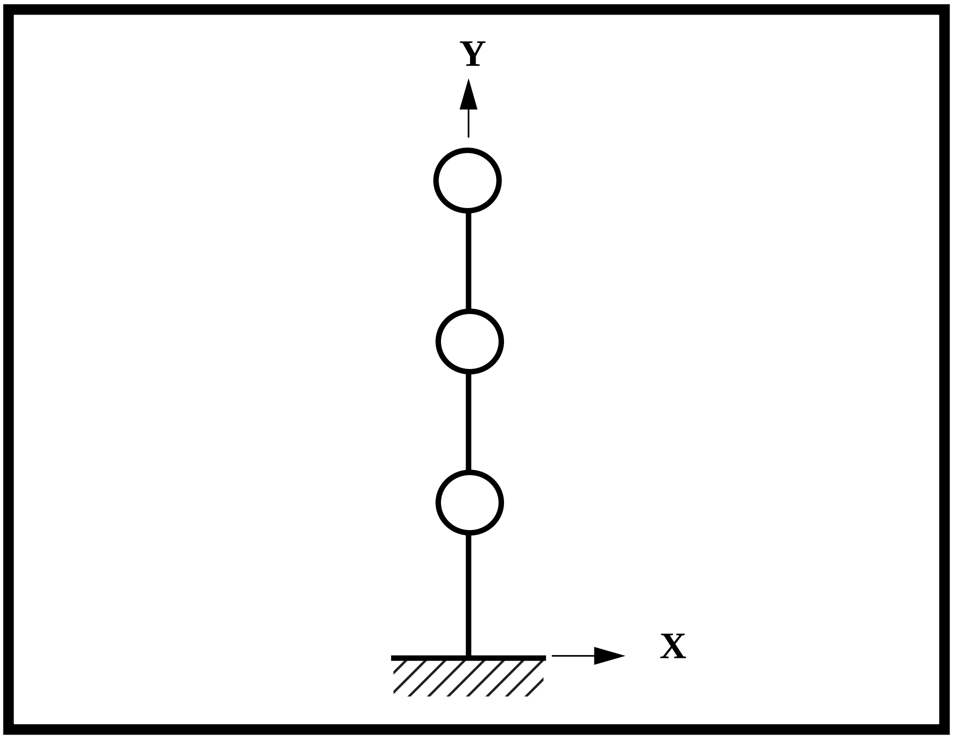
APPENDIX D

Simple Lumped Mass System

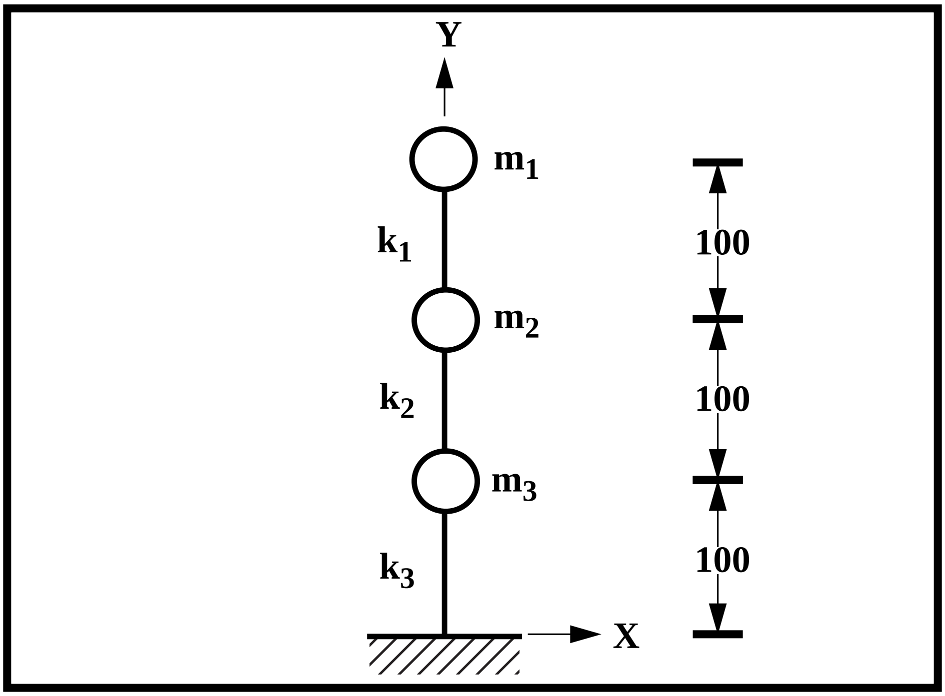


Objectives:

- Model a simple lumped mass system using beam elements and point elements.
- Apply the proper constraints to the model to ensure stability during analysis.
- Calculate the first two modes of vibration for the system.

Model Description:

Figure D.1 - Simple Lumped Mass System



Remember: for a beam, $k = \frac{12EI}{L^3}$

This system can be modeled using bar elements and concentrated masses.

In order to idealize the above lumped mass system, the following assumptions are made:

1) L1 = L2 = L3 = 100

2) E = 1.0E6

Thus, $I_1 = \frac{k_1 L^3}{12E}$, etc.

Purpose of Exercise:

Calculate the first two modes , where:

$$m_1 = 1.0 \frac{lb}{in}s^2 \qquad k_1 = 600.0 \frac{lb}{in}$$

$$m_2 = 1.5 \frac{lb}{in}s^2 \qquad k_2 = 1200.0 \frac{lb}{in}$$

$$m_3 = 2.0 \frac{lb}{in}s^2 \qquad k_3 = 1800.0 \frac{lb}{in}$$

Clough and Penzian:

$$\omega_1 = 14.5 \frac{rad}{sec} \phi_1 = \begin{Bmatrix} 1.0 \\ 0.646 \\ 0.301 \end{Bmatrix}$$

This example problem introduces normal modes analysis of a simple lumped mass system, represented by a beam with masses applied.

Exercise Procedure:

1. Start up MSC/NASTRAN for Windows V3.0 and begin to create a new model.

Double click on the icon labeled MSC/NASTRAN for Windows V3.0.

On the *Open Model File* form, select **New Model.**

Open Model File: **New Model**

2. Create a material called **mat_1**.

From the pulldown menu, select **Model/Material.**

Model/Material...

Title: **mat_1**

Young's Modulus: **1.0e6**

OK

Cancel

3. Create properties that will define the three bar elements and three lumped masses.

Since the model has three difference sections with different lumped masses and spring constants, six individual properties must be created. The three spring constant properties are created first.

Model/Property...

Title: **k1**

Elem/Property Type...

● **Bar**

OK

To select the material, click on the list icon next to the databox and select **mat_1.**

Material: **1..mat_1**

I1:	**50**
OK	
Title:	**k2**
I1:	**100**
OK	
Title:	**k3**
I1:	**150**
OK	

The next step is to create the lumped masses.

Elem/Property Type...	
Other Elements:	● **Mass**
OK	
Title:	**mass_1**
Mass, M or Mx:	**1**
OK	
Title:	**mass_2**
Mass, M or Mx:	**1.5**
OK	
Title:	**mass_3**
Mass, M or Mx:	**2**
OK	
Cancel	

4. Create the NASTRAN Finite Element Model.

The model of this beam is separated into three bar sections separated by lumped masses. Each section of the beam has its own spring constant.

First, create 4 nodes on the same axis but 100 unit length apart.

Model/Node...

CSys: **0 .. Basic Rectangular**

Locate-Enter Coordinates or Select with cursor.

X:	**0**	*Y:*	**0**	*Z:*	**0**	**OK**
X:	**0**	*Y:*	**100**	*Z:*	**0**	**OK**
X:	**0**	*Y:*	**200**	*Z:*	**0**	**OK**
X:	**0**	*Y:*	**300**	*Z:*	**0**	**OK**

Cancel

To fit the display onto the screen, use the Autoscale and Rotate feature.

View/Autoscale

View/Rotate... **Isometric**

OK

5. Create the finite elements and lumped masses for the model.

There are two step in creating the FEM. First is to create the bar elements having the spring constant properties. Then, the lumped mass will be placed at the nodes in between the bar elements.

Model/Element...

Property: **3..k3**

Type...

● **Bar**

OK

Nodes: **1** **2**

Orientation: **Vector...**

Base:	*X:*	**0**	*Y:*	**0**	*Z:*	**0**
Tip:	*X:*	**1**	*Y:*	**0**	*Z:*	**0**

OK

OK

Next, create the next two elements with the following set of data.

Property: **2..k2**

Nodes: **2** **3** **OK**

Property: **1..k1**

Nodes: **3** **4** **OK**

Now that the bar elements are created, the lumped masses can be placed in between the elements.

Type...

Other Elements: ● **Mass**

OK

Property: **4..mass_1**

Nodes: **4** **OK**

Property: **5..mass_2**

Nodes: **3** **OK**

Property: **6..mass_3**

Nodes: **2** **OK**

Cancel

6. Create the model constraints.

Before creating the appropriate constraints, a constraint set needs to be created. Do so by performing the following:

Model/Constraint/Set...

Title: **constraint**

OK

This constraint set will have 2 constraints. First, define the first relevant constraint for the model.

Model/Constraint/Nodal...

<select node 1>

OK

Fixed

OK

Next, define the second relevant constraint for the model.

<select nodes 2,3 and 4>

OK

On the *DOF* box, fix translational and rotational D.O.F except TX.

☒ **TY** ☒ **TZ**

☒ **RX** ☒ **RY** ☒ **RZ**

OK

Cancel

7. Create the input file for analysis.

File/Export/Analysis Model...

Analysis Format/Type: **2..Normal Modes/Eigenvalue**

OK

Change the directory to **C : \temp**.

File Name: **lump**

Write

☒ **Run Analysis**

OK

8. When asked if you wish to save the model, respond **Yes**.

Yes

File Name: **lump**

Save

When the MSC/NASTRAN manager is through running, MSC/NASTRAN will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran smoothly, we will not bother with the details this time.

Continue

9. View the results of the analysis.

To see the mode cases for this model, following these steps, and answer the questions.

View/Select...

Deform and Contour Data

Output Set

From the three mode sets, what is the frequency of each mode?

Case 1 Mode = _______________Hz

Case 2 Mode = _______________Hz

Case 3 Mode = _______________Hz

This concludes the exercise.

Mode 1	**2.311195**
Mode 2	**4.941397**
Mode 3	**7.336955**