UNIVERSITY OF SASKATCHEWAN

COLLEGE OF ENGINEERING

Department of Civil & Geological Engineering

CE 463.3 – Advanced Structural Analysis

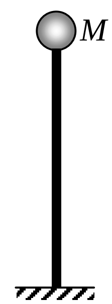
Lab 5 –SAP2000 Dynamic Analysis

T.A: Ouafi Saha **Professor: M. Boulfiza**

1. Natural Mode for a Single Degree of Freedom system

http://www.youtube.com/watch?v=HTOt2uJgdRg

Let's start with a simple single degree of freedom system composed of a column fixed at its base and a concentrated mass at its top. We need to know the natural frequency and period of this structure.



M = 20 000 kg
E = 200 GPa
I = 100 10^6 mm^4
h = 3m

$$T = 2\pi\sqrt{\frac{m}{k}} = 2\pi\sqrt{\frac{m}{3EI/h^3}} = 2\pi\sqrt{\frac{20000\left(kg\right)\times 3^3\left(m^3\right)}{3\times 200.10^9\left(N/m^2\right)\times 100.10^{-6}\left(m^4\right)}} = 0.596\sec$$

i. General Definitions

It is HIGHLY recommended to choose **(*N*, *m*)** as principal units so mass will be in ***kg***. Otherwise, conversion will not be obvious. Version 14 of SAP2000 allows you to enter mass as weight, this may simplify data input, but you must be careful on the meaning of each possibility.
A simple grid system may be X(0), Y(0), Z(0,3)

ii. Material and Section Definition

Define a Material having **E = 200 GPa** and ***v* = 0.3**. Also, define a frame section having moment of inertia **I_3=100.10^6*mm*^{4}**. Make sure to choose the appropriate Material for this section.

iii. Drawing the Model

Draw a frame from point p_1(0,0,0) to p_2(0,0,3).

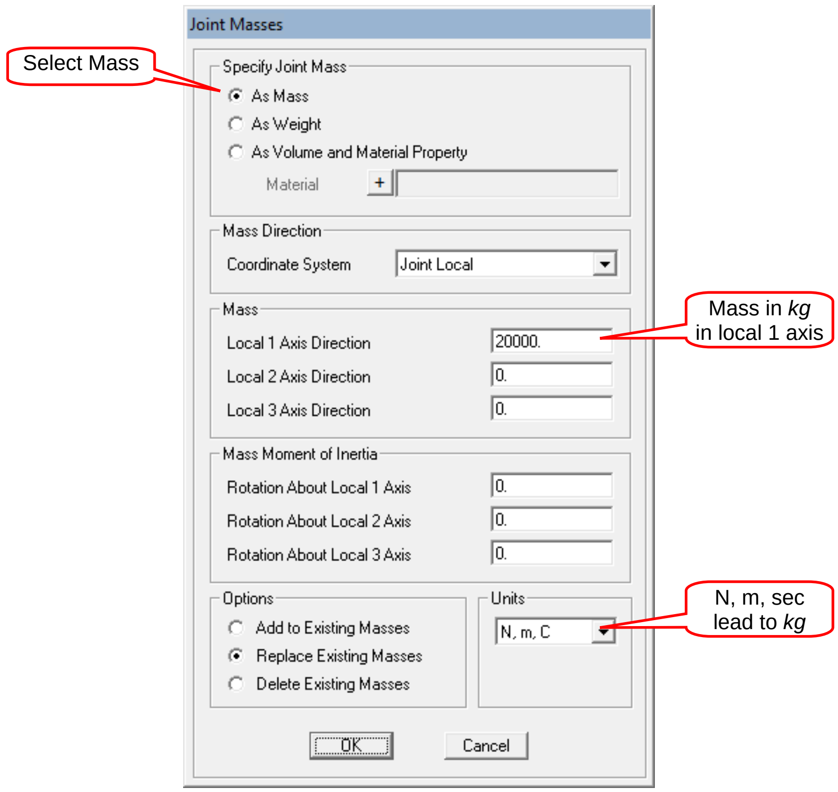
iv. Boundary Displacement Conditions

Assign a fixed restraint to the base of our element.

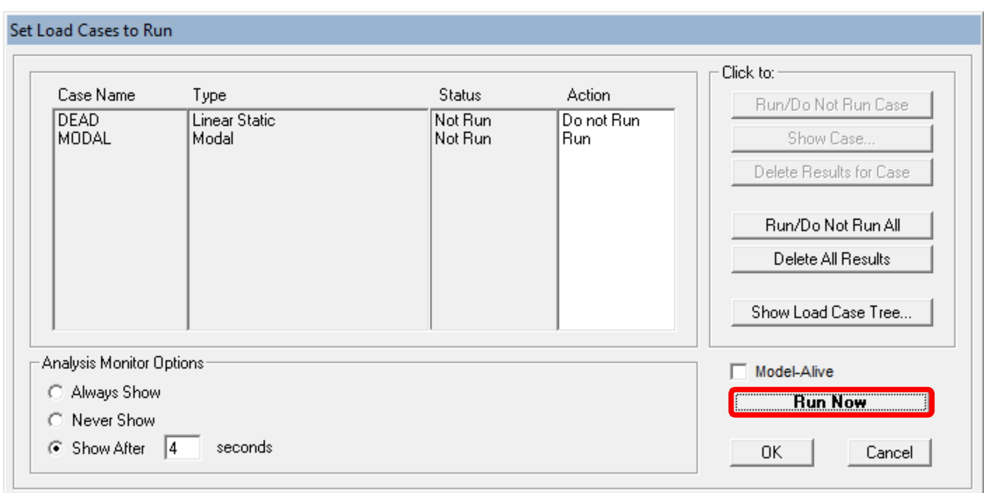
v. Loading Condition

Since we are just looking for the dynamic properties of our structure we don't need a loading condition, but we need to assign a concentrated mass to the top of our column.

Select the top node (joint)
Assign > Joints > masses …

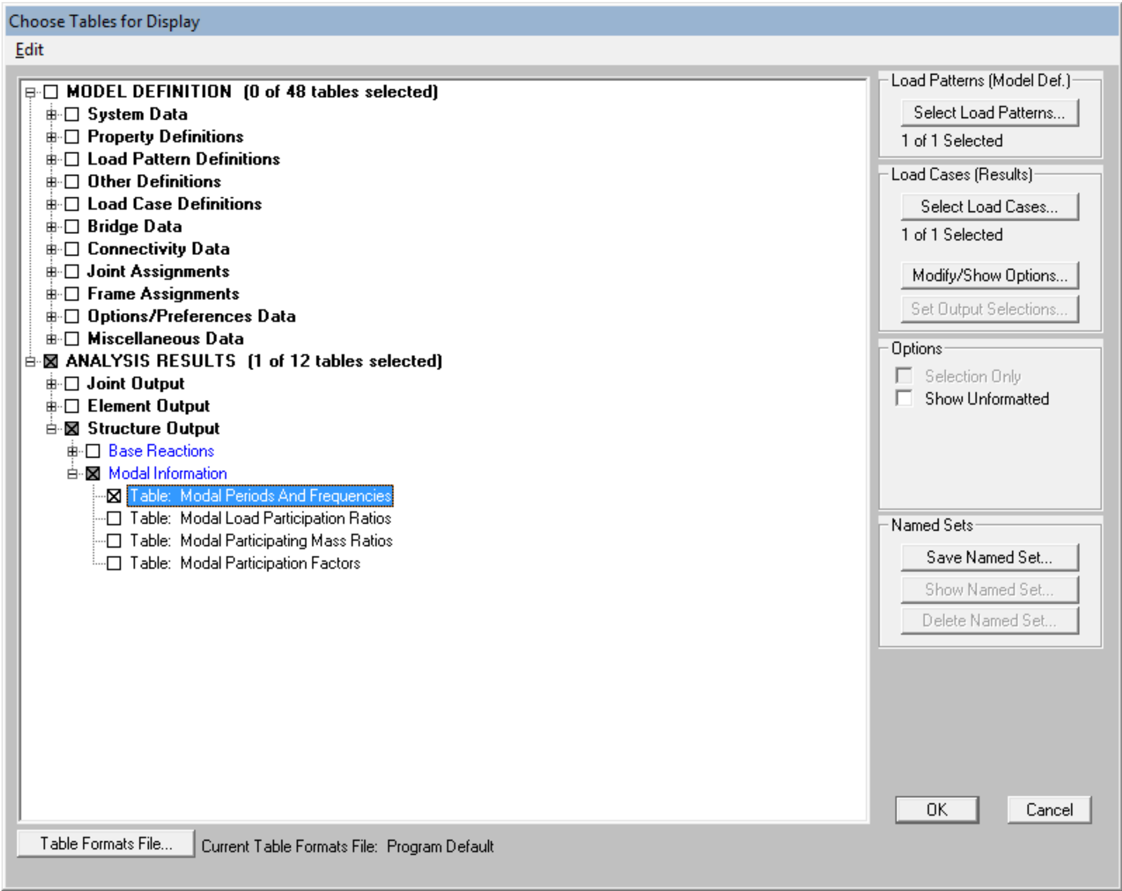


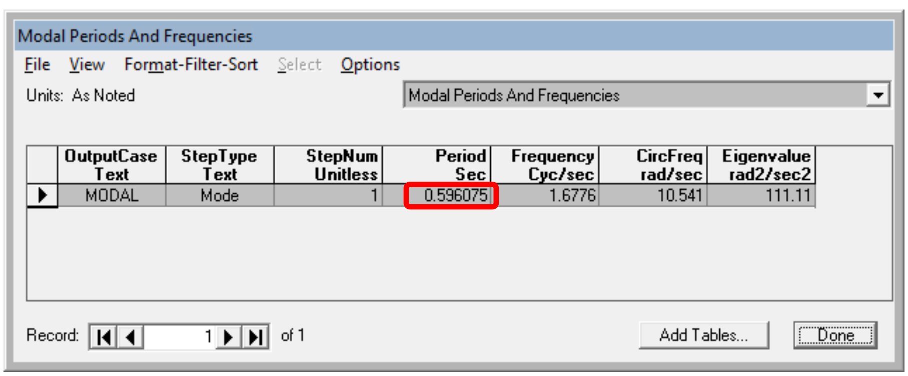
vi. Analyse the System
Simplify analysis by choosing XZ Plane in "Set Analysis Options" menu
Make sure to set MODAL to run in the "Run Analysis" dialogue box, no need to run Static analysis.



Then **Run** the analysis.

vii. Display Output
An easy way to see the dynamic characteristics of the system is to use the tabular form output.
Select Menu Display > Show Tables …





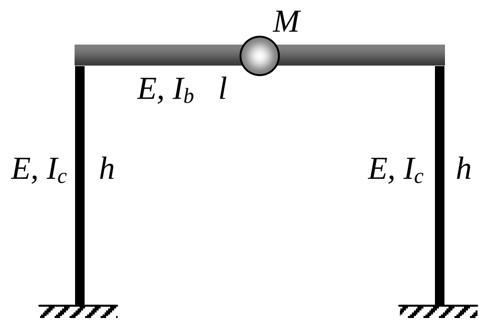
viii. Discussing Results

As we can see, the same period T = 0.596075sec as the hand calculated one is obtained. The natural frequency is $f = 1.6776\ Hz$ and the circular frequency is $\omega = 10.541$ rad/sec

2. Natural Mode for a Single Degree of Freedom one storey Building

http://www.youtube.com/watch?v=O1rZRojOf4c

Let's study the structure shown in the next figure.

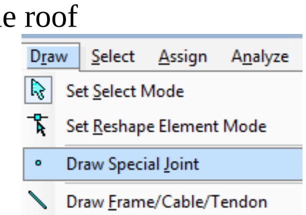


Assumptions:
- Structure works in XZ plane.
- All members are made of steel, E=200GPa.
- All members' self-weight is neglected.
- The only existing mass is concentrated in the roof.
- Structure is fixed at its base.

Columns W310x74 (from CISC data base) $I_c = 165.10^6 mm^4$
$h = 3m$, $l = 6m$, $M = 30\ 000kg$.

Roof will be modeled in **four** different ways.

Quick steps:
- Choose ***N, m, C*** units
- Define grids X(0,3,6) Y(0) Z(0,3)
- Add new material Mat (E=200E9, v = 0.3, Self-weight=0)
- Add section W310x74 by importing I/Wide flange from CISC database
- Draw the frame using the **same** section for all parts
- Fix foundation
- Draw Special Joint at the middle of the roof



- Assign concentrated mass to that joint = 30000 in local 1st direction as mass. (§ expl. 1)
- Only planar XZ degrees of Freedom are needed for this problem
- No need for static analysis

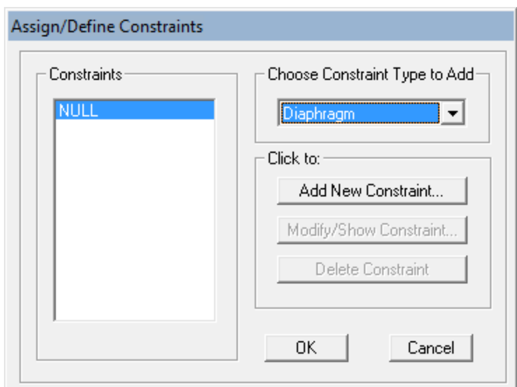
a. Roof as Normal beam

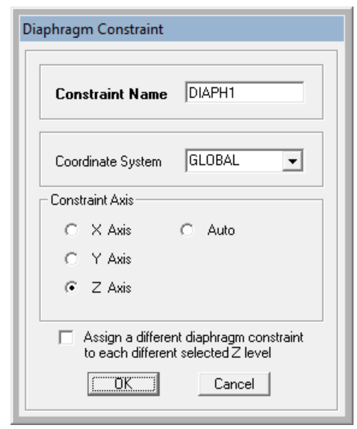
Just like we have already defined our structure
In this case $I_b = I_c$
The roof is very flexible.

T = 0.284162 sec
f = 3.5191 Hz

b. Use of Diaphragm

Select all three top nodes then go to Menu Assign > Joints > Constraints …
Select diaphragm in the first dialogue box and keep Z axis selected in the second dialogue box.
This enforces the selected joints to maintain exactly the same distance from each other while moving in the XY plane.





This constraint is usually used to model concrete slabs or decks.

This does not lead to a big change in the example under consideration. The reason is that only the compression in the roof beam has been constrained.

T = 0.28251 sec
f = 3.5397 Hz

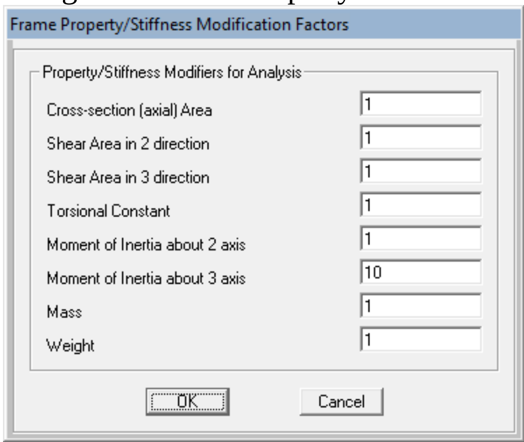
Alternatively, we can use the Equal Constraint.
In this case choose all DOF to be equal. As we see from the figure, the structure is stiffer but this condition is not realistic.

T = 0.243478 sec
f = 4.1071 Hz

c. Increase the stiffness of the roof beam

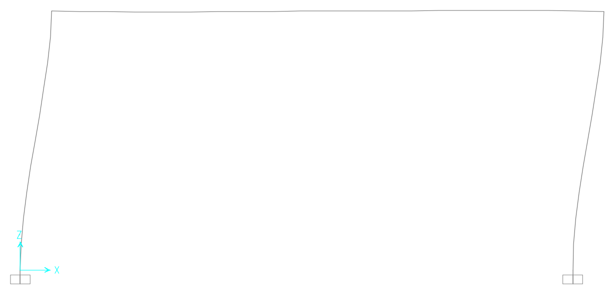
Remember to remove constraints before doing this step.

Select the top beam then Menu Assign > Frame > Property Modifiers …



In this case we are multiplying the flexural stiffness (Moment of Inertia I_3) of the top beam by a factor of 10.

It's clear that the top slab is almost horizontal.



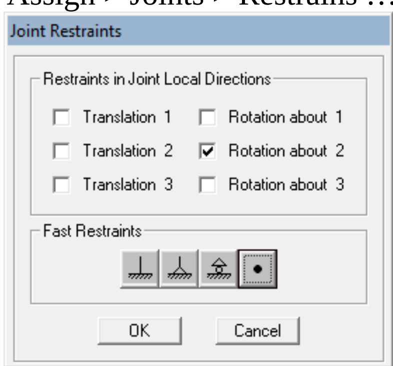
T = 0.234911 sec

f = 4.2569 Hz

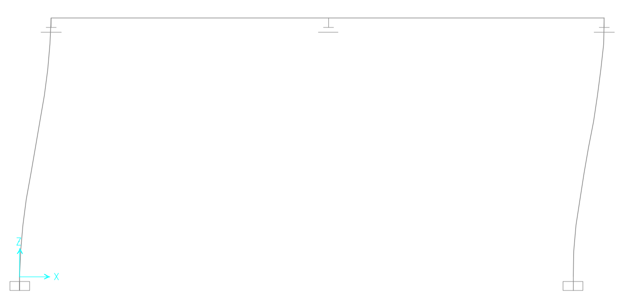
d. No rotation in the top beam

Remember to set the property modifiers to 1 again.

Select the three nodes of the top and Assign > Joints > Restrains …



This is the closest "realistic" condition to the use of the formula $k = \dfrac{12EI}{h^3}$ for column stiffness.



T = 0.219775 sec

f = 4.5501 Hz

Why do you thik the above result is different from yours?

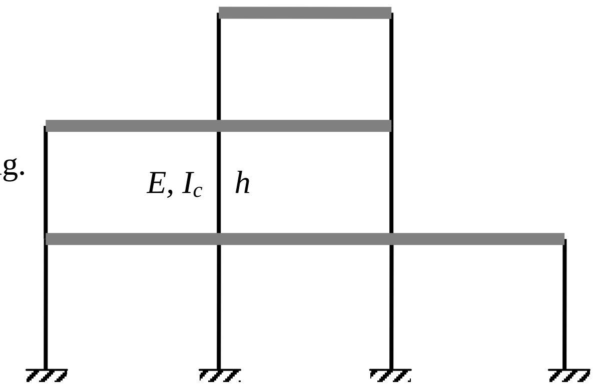
Theoretical period calculated with formula above is T=0.2009sec. How can you find it with SAP2000?

3. Multi Degrees of Freedom Building

Consider the following 3 stories building

To simplify, we assume:

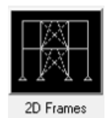
- All bays are 6*m* wide.
- The first floor is 4*m* high and the other are in 3m spacing.
- Self-weight of all elements is neglected.
- Mass of each 6*m* segment slab is 25000 *kg*.
- Slabs are considered infinitely rigid.
- Moment of inertia of all columns, I_c=150.10^6 mm^4.
- Material used is steel, *E*=200*GPa*, ν = 0.3.

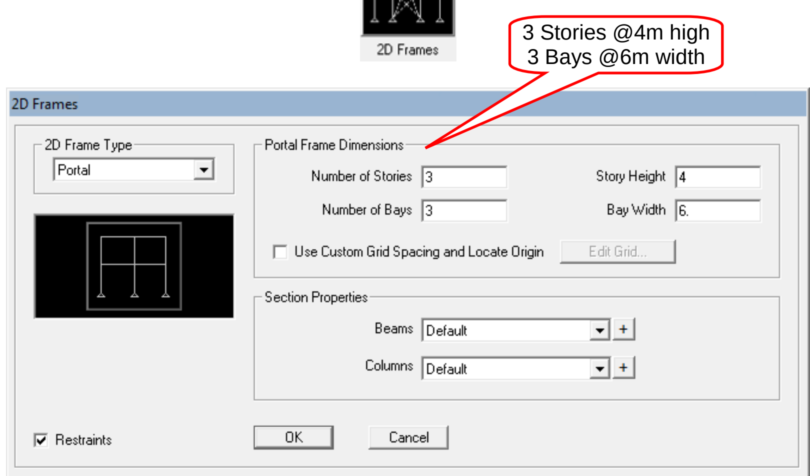


Find the three natural modes for horizontal displacement.

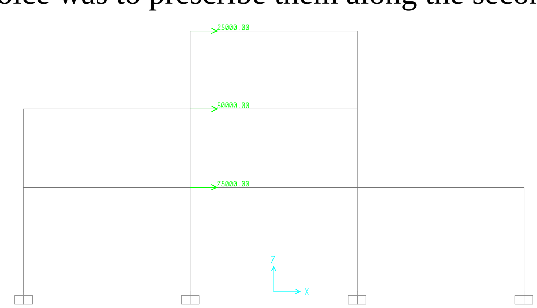
Quick steps:

- Choose ***N, m, C*** units (so we can use *kg* as unit for mass)
- Use the predefined 2D Frame Model

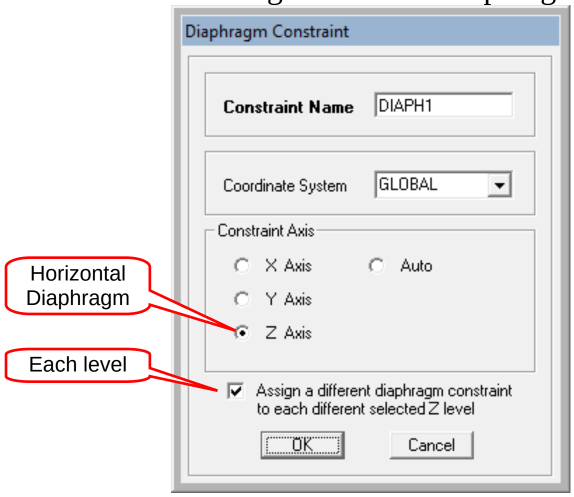




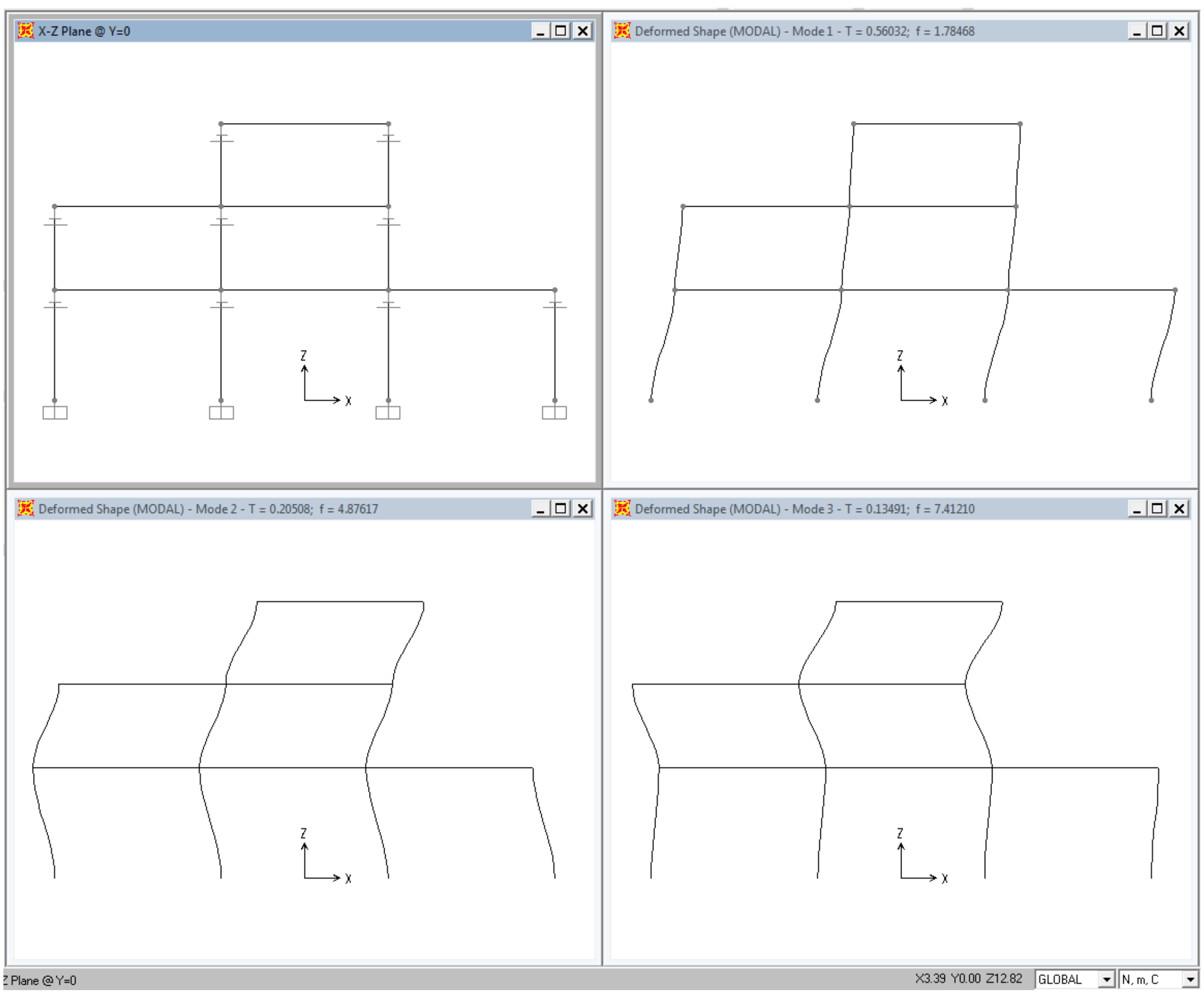
- Change grid lines Z (8, 12) to (7, 10) respectively, check on “glue joint to gridlines” before validating
- Delete unwanted parts from the drawn model
- Add new material MAT (E=200E9, ν = 0.3, Self-weight=0)
- Add section SEC as General where you put only I_3=15E7mm, don’t forget to choose MAT as material
- Select all structure elements and assign “SEC” to them
- Fix foundation
- Since we are restricting our structure to move only along the horizontal direction, position of the concentrated mass along X axis does not matter, as long as it is at the slab levels
- Assign a resultant concentrated mass to each level: 25000 to the roof, 50000 to the second floor and 75000 to the first floor (my choice was to prescribe them along the second column)



- Select all nodes above foundation and assign horizontal diaphragms to each level



- Select all nodes above foundation and assign restrained rotation about local 2nd axis
- Only planar XZ degrees of Freedom are needed for this problem
- Run the analysis with No static analysis



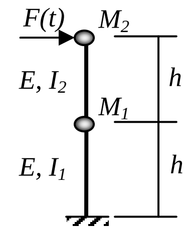
T_1=0.56032sec, T_2=0.20508sec, T_3=0.13491sec and $f_1=1.785Hz$, $f_2=4.876Hz$, $f_3=7.412Hz$

4. Two Degrees of Freedom System with Time History Analysis

http://www.youtube.com/watch?v=njWwO4hOwmI

Let's assume the simplified 2 DOF system shown below:

- Self-weight of all elements is neglected.
- Material used is steel $E = 200\ GPa$, $v = 0.3$.
- $h = 3\ m$, $M_1 = M_2 = 50000\ kg$, $I_1 = I_2 = 450.10^6\ mm^4$
- No rotations are allowed at the two levels (so we can use $k = \frac{12EI}{h^3}$ formula)

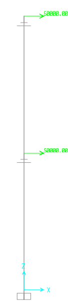


Use modal analysis to find the two fundamental frequencies.
In addition there is a harmonic concentrated load at the top level $F(t)$.

Solution:

Modal Analysis:

The first part will be done just like the first example. Two differences are however worth mentioning; we have two stories and rotation about local 2nd axis is blocked.



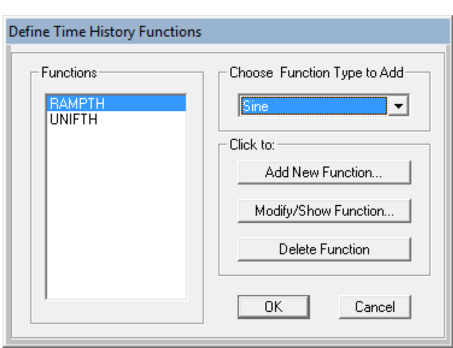
A first run will result in: T_1=0.35944sec, f_1=2.78213*Hz*, T_2=0.13729sec, f_2=7.28371*Hz*

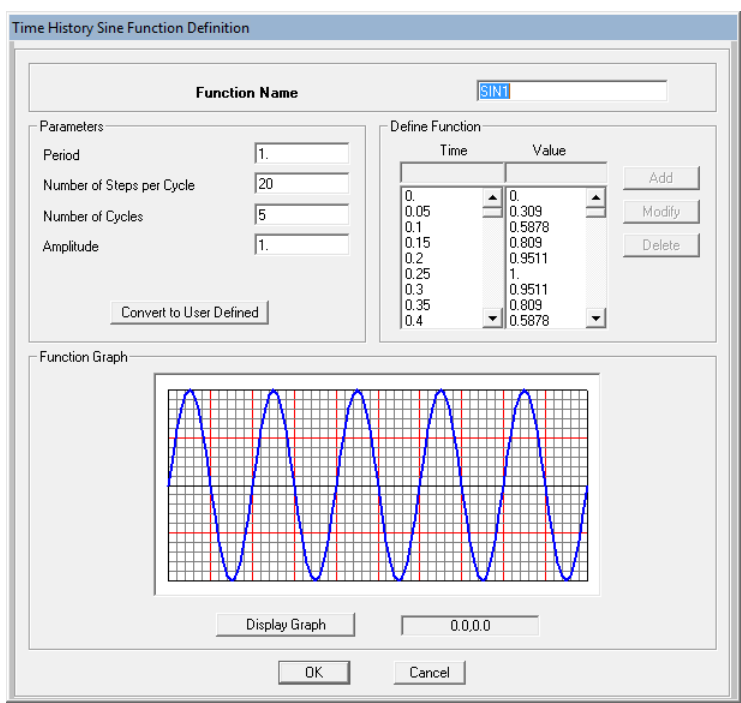
Time History Analysis:

The second part needs more concentration!

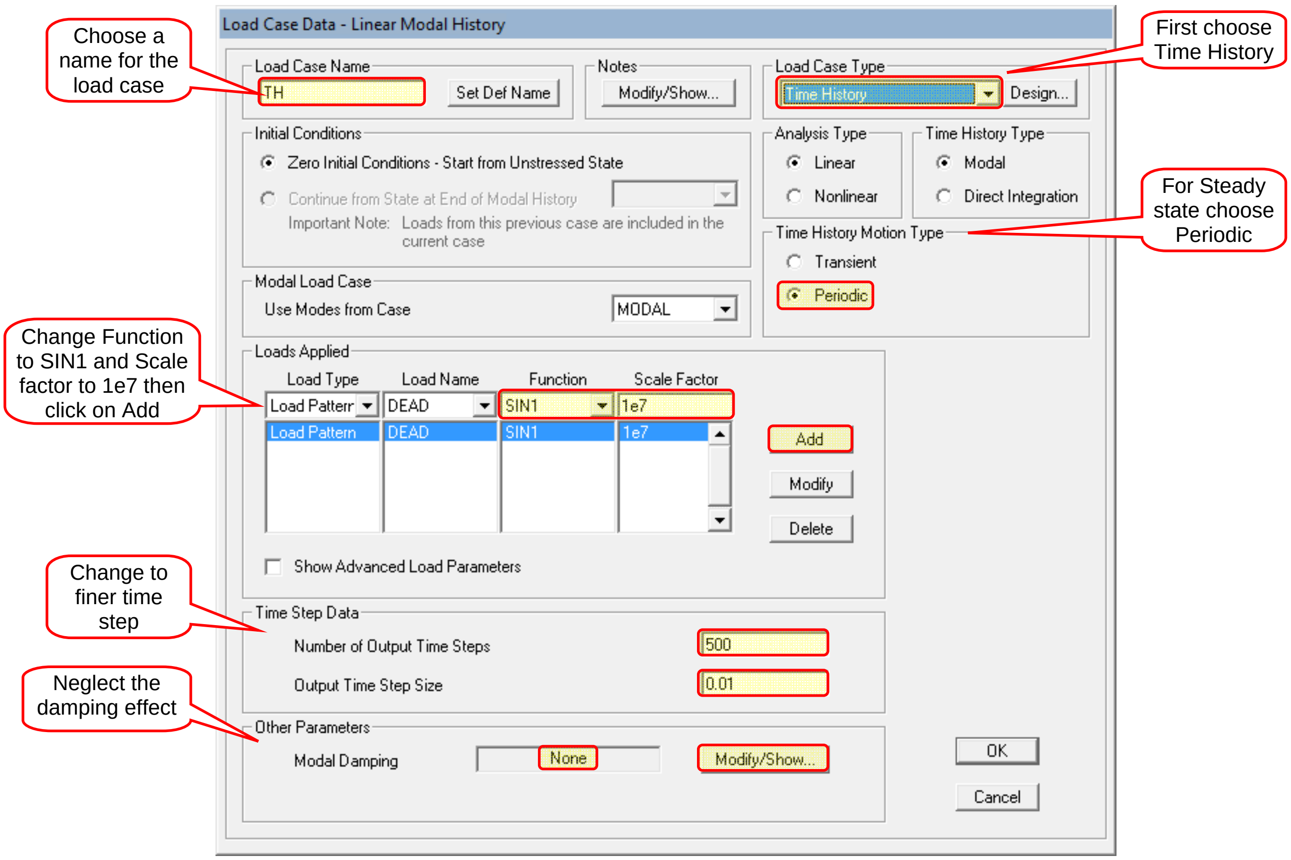
- Since we have the dynamic Concentrated Load at the top level, we need to add a concentrated static unit load in the Dead Load Case, even if we don't need to run the static load Analysis.
- Define the Harmonic function, Menu Define > Functions > Time History …

In the first dialogue box choose Sine and click on Add New Function, in the second dialogue box change just the name of the function to SIN1, further functions can be generated later.





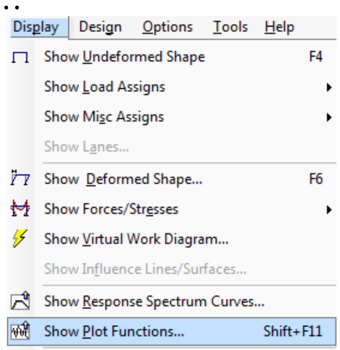
- Define a new Load case for the Time History Analysis, Menu Define > Load Cases …
Click on Add New Load Case … button

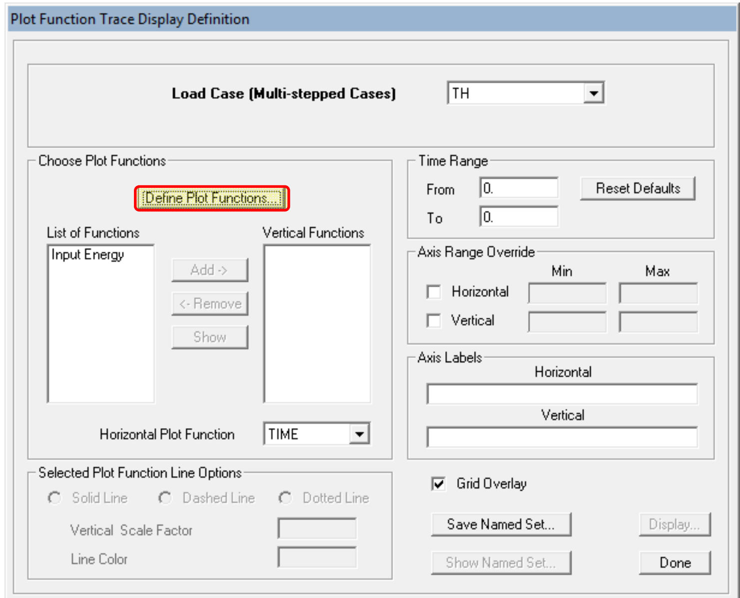


Scale Factor has been used because the unit dead load introduced previously (1N) is not big enough to move the system. (1E7 is a bit exaggerated).
If you want to see the transient solution (starting from time t = 0 sec) click on Transient.
It is highly recommended to use Time Step Data in accordance with Time History Function Definition, in this case SIN1, to avoid direct integration numerical perturbation.
To neglect the effect of damping, click on Modify/Show… button under other parameter and put 0 for constant damping for all modes.

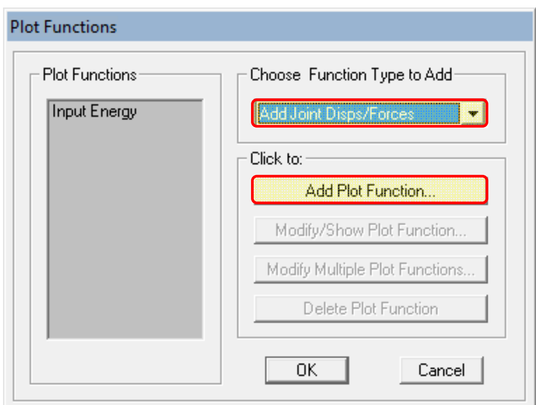
Now **Run**! The analysis

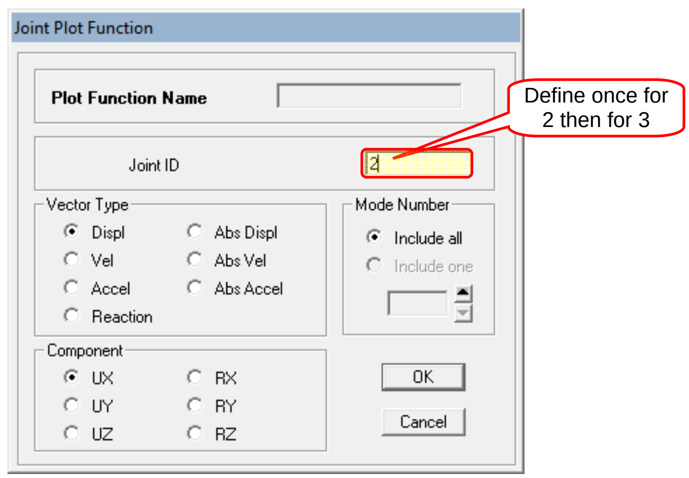
A good way to display the results for time history analysis is to use the built-in plot engine
Menu Display > Show Plot Functions…

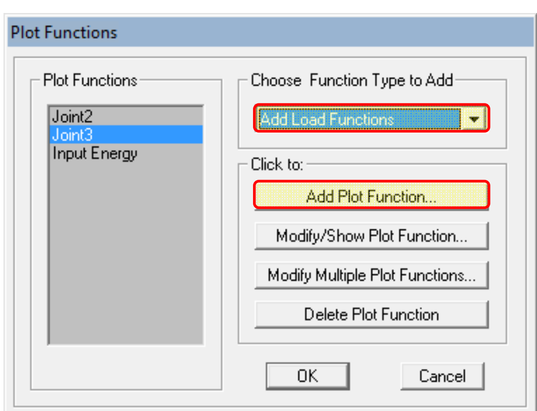


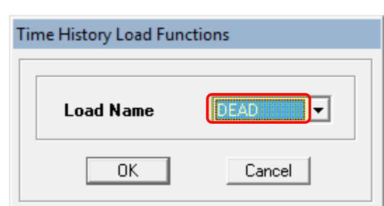


We need first to define our probing stations. I have chosen the two horizontal displacements of the concentrated masses and the unit harmonic load.
Click on Define Plot Functions…

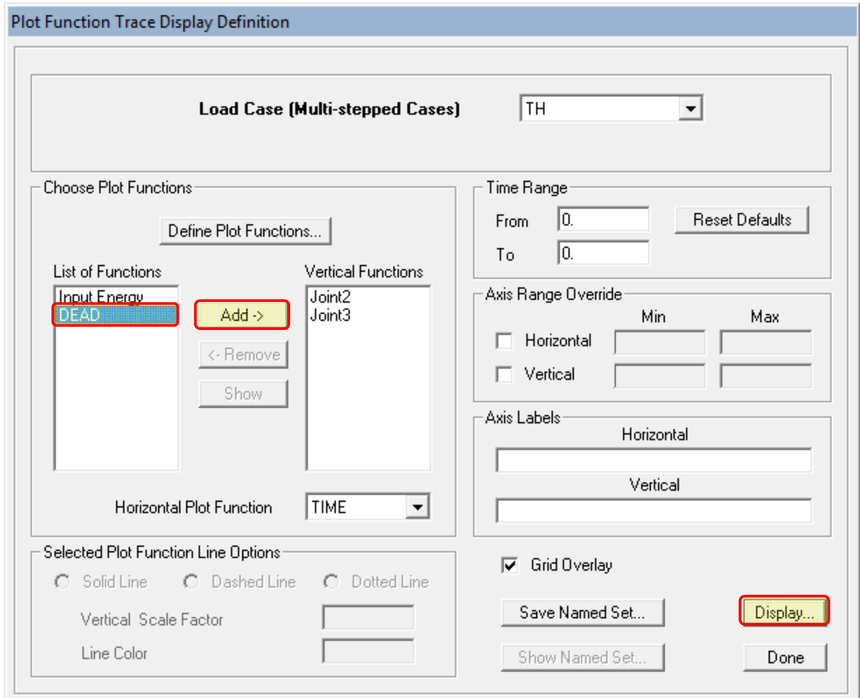


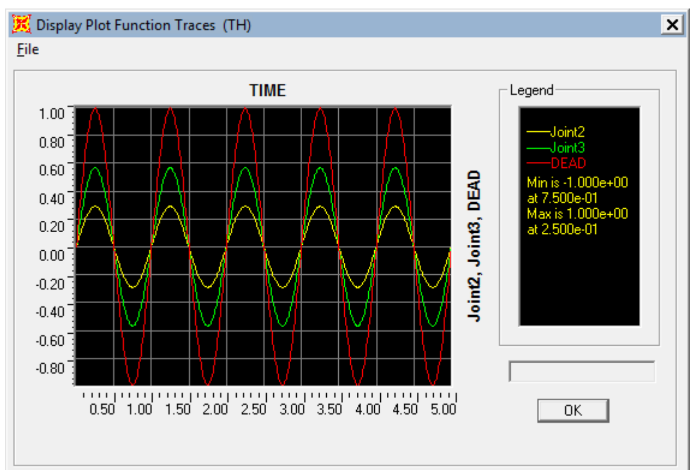




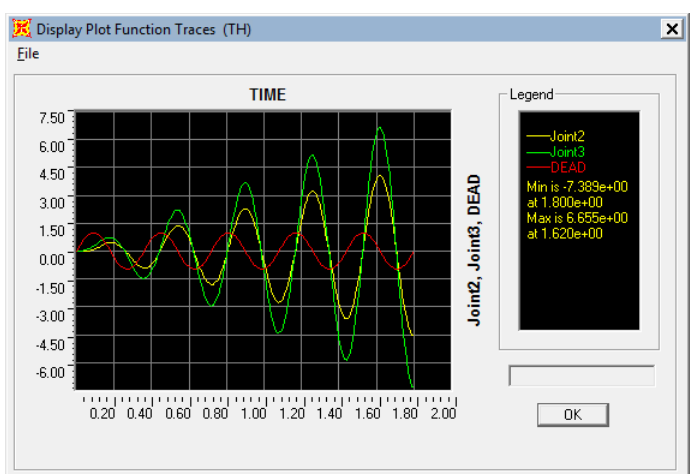


The final step is to add these three Plots to the vertical Functions side and click on Display

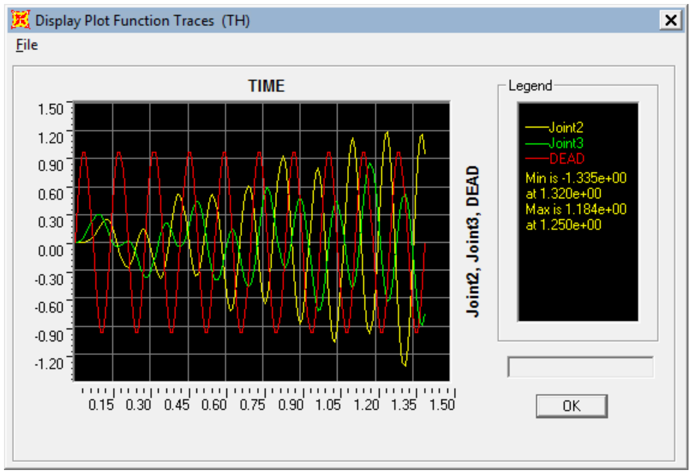




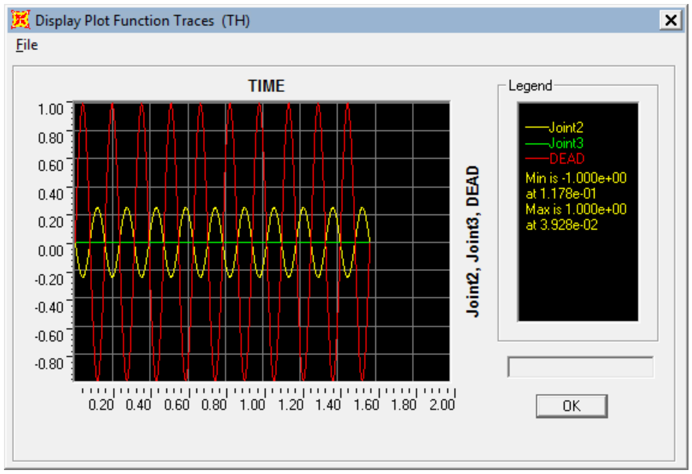
Period of SIN1 = 1sec (Periodic)



Period of SIN1 = 0.36sec (Transient)



Period of SIN1 = 0.14sec (Transient)



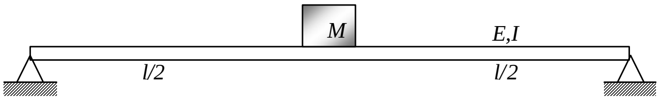
Period of SIN1 = 0.157sec (Periodic)

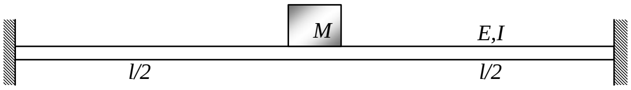
Note that in the last run, Joint3 is still the node where the load is applied. But in this case it is not moving at all. Can you explain why?

6. Additional Examples

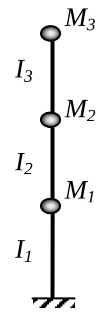
6.1. Find the fundamental period and frequency of the following beams with a lumped mass at midspan.
Compare the results with hand calculated formula.

$M = 10\,000\ kg,\ E = 200\ GPa,\ I = 150\ 10^6\ mm^4,\ l = 6m$





6.2. Repeat example 3 using the simplified model shown below, made of one vertical column, and three concentrated masses.



6.3. Try to use the weight as masses and compare the results.