



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

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

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 $\mathbf{E} = 200$ GPa and v = 0.3. Also, define a frame section having moment of inertia $\mathbf{I}_3=100.10^6 mm^4$. 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 ...

	Joint Masses	
Select Mass	Specify Joint Mass	
	As Mass	
	C As Weight	
	C As Volume and Material Property	
	Material +	
	Mass Direction	
	Coordinate System Joint Local 💌	
	Mass	Mass in kg
	Local 1 Axis Direction 20000.	in local 1 axis
	Local 2 Axis Direction 0.	
	Local 3 Axis Direction 0.	
	Mass Moment of Inertia	
	Rotation About Local 1 Axis 0.	
	Rotation About Local 2 Axis 0.	
	Rotation About Local 3 Axis 0.	
	_ Options	N, m, sec
	C Add to Existing Masses N, m, C	lead to kg
	Replace Existing Masses	
	C Delete Existing Masses	
	Cancel	

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.

Case Name	Туре	Status	Action	Click to:
DEAD MODAL	Linear Static Modal	Not Run Not Run	Do not Run Run	Run/Do Not Run Case Show Case Delete Results for Case
				Run/Do Not Run All Delete All Results
				Show Load Case Tree
nalysis Monitor (Options			Model-Alive
Always Show	i			Run Now
Never Show				nuri NUW
Show After	4 seconds			OK Cancel

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 ...

Choose Tables for Display	
Edit	
MODEL DEFINITION (0 of 48 tables selected) System Data Property Definitions Other Definitions <th>Load Patterns (Model Def.) Select Load Patterns 1 of 1 Selected Load Cases (Results) Select Load Cases 1 of 1 Selected Modify/Show Options Set Output Selections Options Selection Only Show Unformatted Named Sets Save Named Set Delete Named Set Delete Named Set</th>	Load Patterns (Model Def.) Select Load Patterns 1 of 1 Selected Load Cases (Results) Select Load Cases 1 of 1 Selected Modify/Show Options Set Output Selections Options Selection Only Show Unformatted Named Sets Save Named Set Delete Named Set Delete Named Set
Table Formats File Current Table Formats File: Program Default	

Units: As Noted Modal Periods And Frequencies							
(DutputCase Text	StepType Text	StepNum Unitless	Period Sec	Frequency Cyc/sec	CircFreq rad/sec	Eigenvalue rad2/sec2
	MODAL	Mode	1	0.596075	1.6776	10.541	111.11

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:

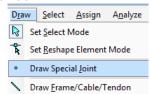
- Structure works in XZ plane.
- All members are made of steel, *E*=200*GPa*.
- All members' self-weight is neglected.
- The only existing mass is concentrated in the roof.
- Structure is fixed at its base.

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

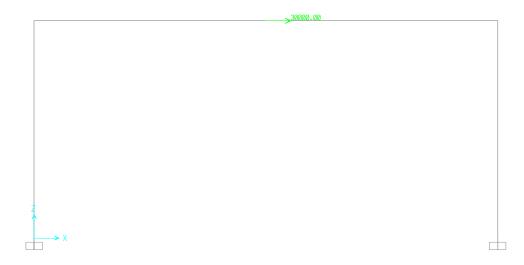
Roof will be modeled in **four** different ways.

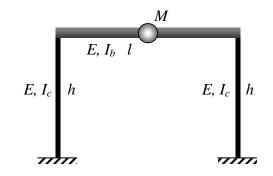
Quick steps:

- 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 1^{st} 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.

$$T = 0.284162 \text{ sec}$$

 $f = 3.5191 \text{ Hz}$



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.

Assign/Define Constraints	
Constraints	Choose Constraint Type to Add
NULL	Diaphragm 🗸
	Click to:
	Add New Constraint
	Modify/Show Constraint
	Delete Constraint
	OK Cancel
-	

phragm Constraint	
Constraint Name	DIAPH1
Coordinate System	GLOBAL
Constraint Axis	
⊂ × Axis	C Auto
C Y Axis	
C Z Axis	
	nt diaphragm constraint t selected Z level
ŪK	Cancel

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 secf = 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 secf = 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

Fra	me Property/Stiffness Modificatio	n Factors
Γ	Property/Stiffness Modifiers for Analy	sis
	Cross-section (axial) Area	1
	Shear Area in 2 direction	1
	Shear Area in 3 direction	1
	Torsional Constant	1
	Moment of Inertia about 2 axis	1
	Moment of Inertia about 3 axis	10
	Mass	1
	Weight	1
	(OK	Cancel

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 secf = 4.2569 Hz

d. No rotation in the top beam

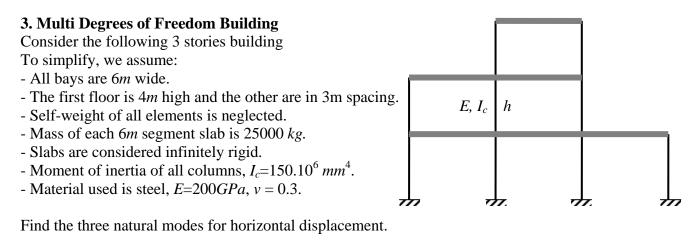
Remember to set the property modifiers to 1 again.

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

Joint Restraints					
Restraints in Joint Local Directions					
🔲 Translation 1 📄 Rotation about 1					
Translation 2 🔽 Rotation about 2					
🗖 Translation 3 🔲 Rotation about 3					
Fast Restraints					
OK Cancel					

This is the closest "realistic" condition to the use $12EI$	-	<u> </u>	
of the formula $k = \frac{12EI}{h^3}$ for column stiffness.			
T = 0.219775 sec f = 4.5501 Hz			
Why do you thik the above result is different from yours?	Z A → X		

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



Quick steps:

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

2D Frame Model	2D Frames 3 S	tories @4m high Bays @6m width
2D Frames		
2D Frame Type Portal	Portal Frame Dimensions Number of Stories 3 Number of Bays 3 Use Custom Grid Spacing and Locate Origin Section Properties Beams Default Columns Default	Story Height 4 Bay Width 6. Edit Grid
🔽 Restraints	OK Cancel	

- 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, v = 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

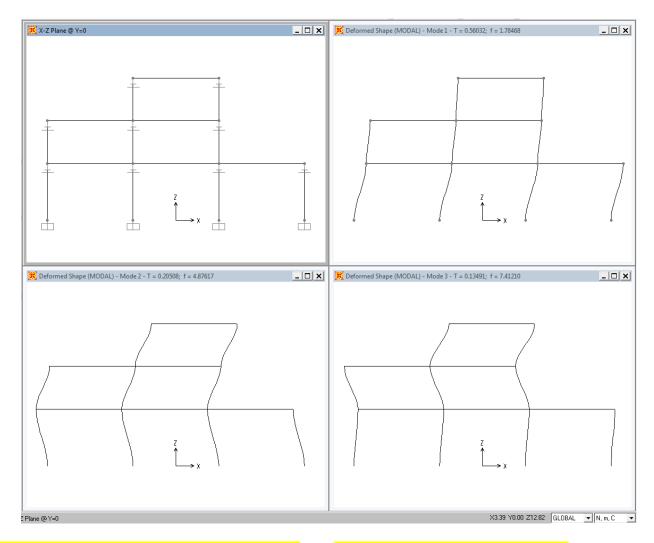
- Assign a resultant concentrated mass to each level: 25000 to the roof, 50000 to the second floor and 75000 to the first floor (my choice was to prescribe them along the second column)



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

	Diaphragm Constraint		
	Constraint Name DIAPH1		
	Coordinate System GLOBAL 💌		
	Constraint Axis		
Horizontal Diaphragm	C X Axis C Auto C Y Axis		
Each level	© Z Axis		
	Assign a different diaphragm constraint to each different selected Z level		
	Cancel		

- 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



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

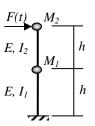
4. Two Degrees of Freedom System with Time History Analysis

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 $k = \frac{12EI}{k^3}$ 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 2^{nd} axis is blocked.

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

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 \ldots

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.

e History Sine Function Defini	tion			
Fund	tion Name		SIN1	
Parameters		Define Functi	on	
Period	1.	Time	Value	_
Number of Steps per Cycle	20	0.	▲ 0. ·	Add
Number of Cycles	5	0.05	0.309	Modify
Amplitude	1.	0.1 0.15	0.5878 0.809	Delete
	,	0.2	0.9511	
		0.3	0.9511	
Convert to User D	efined	0.4	▼ 0.5878	-
Function Graph				
				-
	Display Graph	0.0),0.0	
	OK	Cancel		

Functions Choose Function Type to Add FramPTH UNIFTH Click to: Add New Function Modify/Show Function		Define Time History Functions
Delete Function	ne K to: Add New Function Modify/Show Function	BAMPTH

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

Choose a	Load Case Data - Linear Modal History		First choose
name for the	Load Case Name Notes	Load Case Type	Time History
load case	TH Set Def Name Modify/Show	Time History 📃 🔽 Design	
	□ Initial Conditions	Analysis Type Time History Type	
	 Zero Initial Conditions - Start from Unstressed State 	🕫 Linear 🕞 Modal	
	Continue from State at End of Modal History	C Nonlinear C Direct Integration	For Steady
	Important Note: Loads from this previous case are included in the	Time History Motion Type	state choose
	current case	O Transient	- T enouic
	Modal Load Case	Periodic	
	Use Modes from Case MODAL		
Change Function to SIN1 and Scale	Loads Applied		
factor to 1e7 then	Load Type Load Name Function Scale Factor		
click on Add	Load Patterr V DEAD V SIN1 V 1e7		
	Load Pattern DEAD SIN1 1e7 🔺	Add	
		Modify	
		Delete	
Change to	Show Advanced Load Parameters		
finer time	└─ Time Step Data		
step	Number of Output Time Steps		
Neglect the			
Neglect the damping effect	Output Time Step Size		
	Other Parameters		
	Modal Damping None Modif	y/Show	
		Cancel	

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).

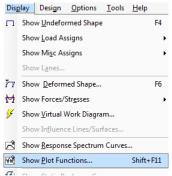
If you want to see the transient solution (starting from time t = 0 sec) click on Transient.

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...



Load Case (Multi-stepped Cases) TH 💌				
Choose Plot Functions List of Functions Input Energy Add - C-Rem Show	Vertical Functions	Time Range From 0. Reset Defau To 0. Axis Range Override Min Max Portical Axis Labels Horizontal		
Horizontal Plot Function	n TIME 💌	Vertical		
Selected Plot Function Line Option C Solid Line C Dashed		Grid Overlay		
Vertical Scale Factor		Save Named Set Dis	olay	

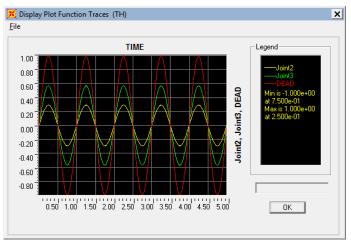
We need first to define our probing stations. I have chosen the two horizontal displacements of the concentrated masses and the unit harmonic load.

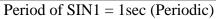
Click on Define Plot Functions...

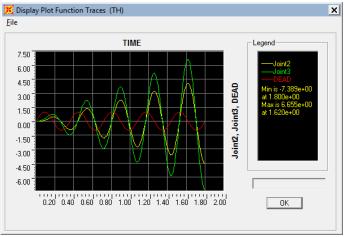
	Joint Plot Function	
ot Functions	Plot Function Name	Define once for 2 then for 3
Plot Functions Choose Function Type to Add Add Joint Disps/Forces Click to: Add Plot Function Modify/Show Plot Function Delete Plot Function OK Cancel	Joint ID Vector Type © Displ C Abs Displ C Vel C Abs Vel C Accel C Abs Accel C Reaction Component © UX C RX C UY C RY C UZ C RZ Cancel	
Plot Functions Plot Functions Choose Function Type to Add Add Load Functions Click to: Add Plot Function Modify/Show Plot Function Delete Plot Function	Time History Load Functions	
OK Cancel	OK Cancel	

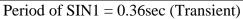
The final step is to add these three Plots to the vertical Functions side and click on Display

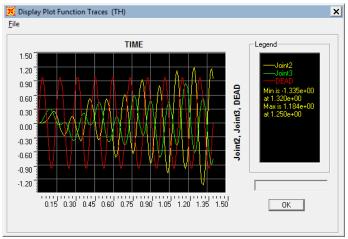
t Function Trace Display Definition				
Load Case (Multi-stepped Cases) TH 💌				
Choose Plot Functions		Time Range		
Define Plot Func List of Functions Input Energy OEAD Add -> C Remove Show Horizontal Plot Function	Vertical Functions Joint2 Joint3	From 0. Reset Defaults To 0. Axis Range Override Min Max Horizontal Vertical Vertical Vertical		
Selected Plot Function Line Options	,			
C Solid Line C Dashed Li		🔽 Grid Overlay		
Vertical Scale Factor		Save Named Set Display		
Line Color		Show Named Set Done		

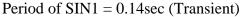


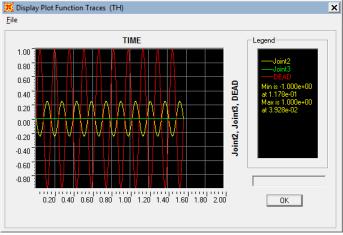












Period of SIN1 = 0.157sec (Periodic)

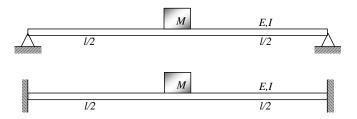
Note that in the last run, Joint3 is still the node where the load is applied. But in this case it is not moving at all. Can you explain why?

6. Additional Examples

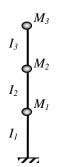
6.1. Find the fundamental period and frequency of the following beams with a lumped mass at midspan.

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.