University of Central Florida
Department of Civil and Environmental Engineering

A Brief Overview and Tutorial

SAP2000 (version 9)

Prepared for Structural Engineering Track Course

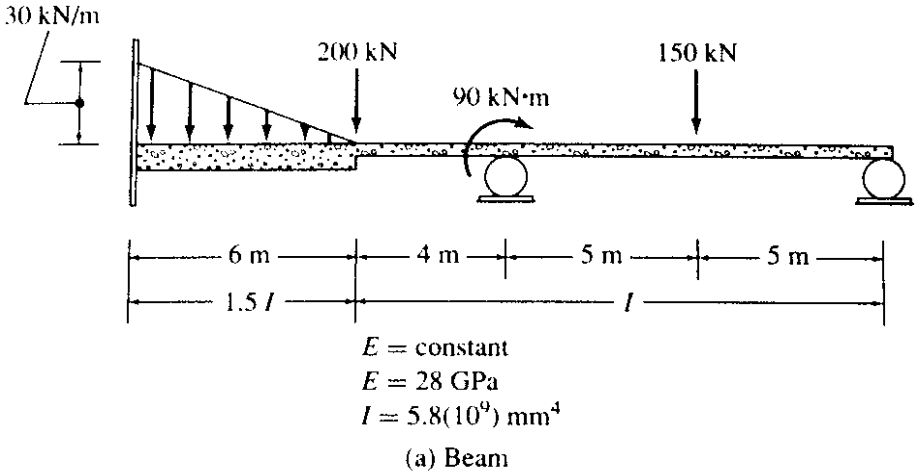
Instructor:
Dr. F. Necati Catbas
Assistant Professor

Prepared By:
Jason Burkett
Graduate Assistant

Spring 2005

Problem # 1 (Example 5.7, pg 219 of Textbook)

Determine the joint displacements, member end forces, and support reactions for the beam shown in fig 5.18 (a), using the matrix stiffness method.



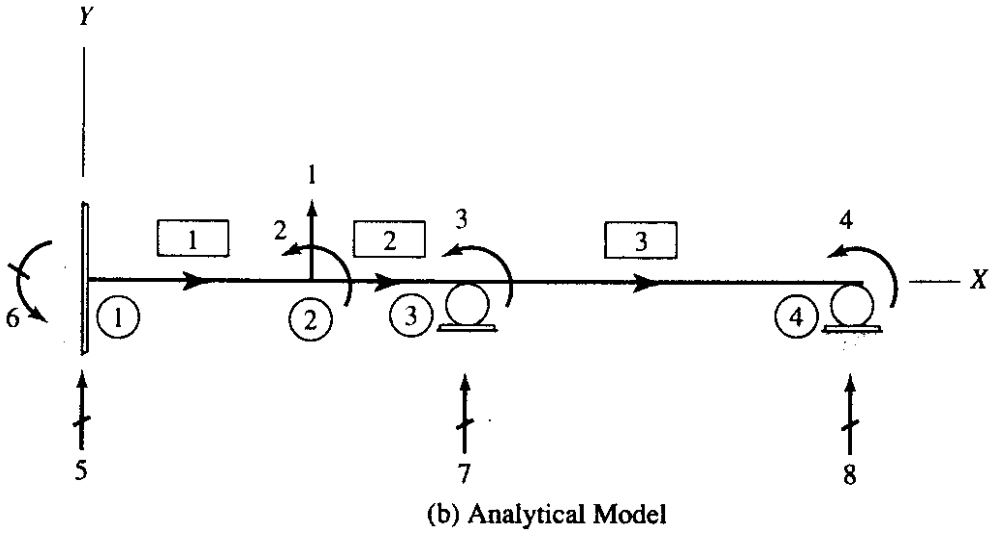


Fig 5.18

Solution:

General Procedure for using SAP2000 Nonlinear

Phase I) Pre-processing

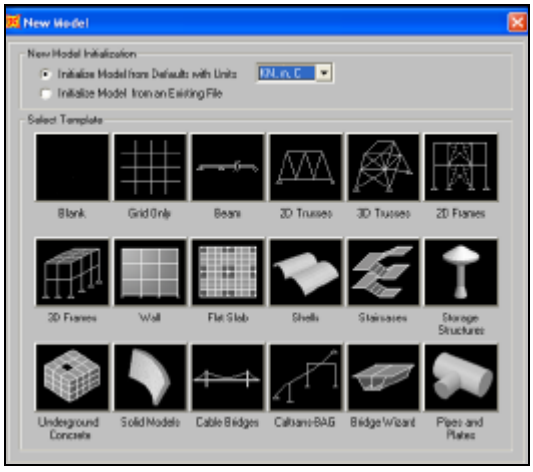
Phase II) Analysis

Phase III) Post-processing

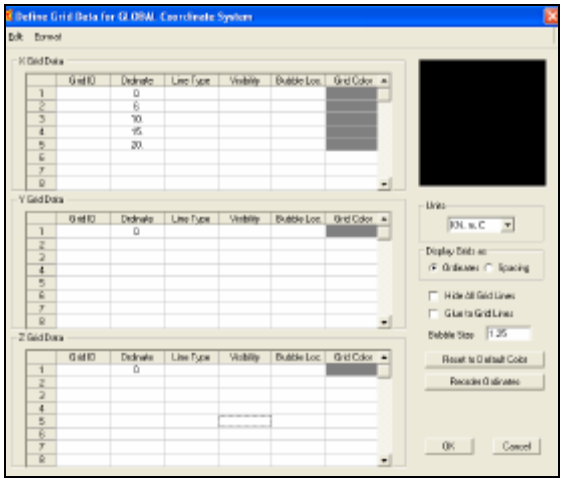
Phase I) Pre-processing

- Opening the Program
  - From Start menu > All Programs > SAP2000 9 > SAP2000

- Setting up the model Geometry
  - File > New Model > Select Units: KN, m, C > Grid Only



  - New Coord/Grid System > Edit Grid
  - Delete all cell data for X,Y, and Z Data > Define Ordinates for each axis respectively [X Grid Data: 0,6,10,15,20], [Y Grid Data: 0], [Z Grid Data: 0]
    - You may also select the option to define Grid Spacing
    - **Note: Z-axis is in the upward direction by default in SAP2000**



  - Work from a 3D view to start so that you do not draw members out of plane
  - Draw > Draw frame/cable/tendon (*Select blue line on left toolbar*)
    - Move cursor over predefined grid and left click to draw elements

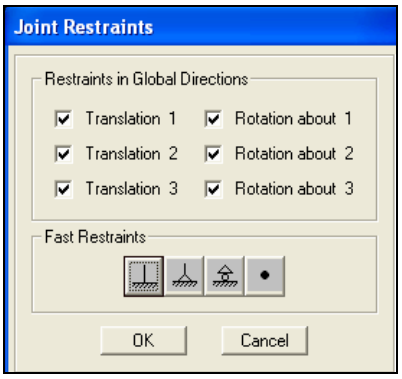
        - Ignore the material/section properties for now
        - Right clicking stops the current element being drawn
    - Renaming Members can be two ways
        - Use cursor to window(select) around all members > Edit > Change Labels
        - Right click on individual members or joints and change name

    - Other views may be selected from the top tool bar according to axis orientation
    - Use the *3D Rotate* button to rotate model and look from different angles
- Defining Material Properties
    - Choose units from bottom right corner of screen: N-mm
    - Define > Materials > Concrete > Modify/Show Material
    - Type of Material > Isotropic
    - Specify Analysis Property data > E = 28,000 N/mm^2
        - **Note: Textbook problems will usually neglect mass and weight properties (i.e. they are ZERO)**

- Defining Member Sections
    - Choose units from bottom right corner of screen: kN-mm
    - Define > Frame Sections > Add Rectangular Section > Click Add New Property
        - Section Name: 1.5I > Depth: 676mm > Width: 338mm
            - Make sure the appropriate material is selected
            - Click on section properties to verify the inertia, area, …
    - Repeat the above step for Section I
        - Depth: 610.8 mm, Width: 305.4 mm

- Assigning Member Sections
    - Select the member > Assign > Frame/Cable/Tendon > Frame Section
    - Select the appropriate section from the list > OK
    - Repeat for other elements
        - Window around several elements to define multiple members at once
    - *Assign > Frame/Cable/Tendon* is also where you can define “releases(hinges)”
    - View > Set Display Options
        - Select which options you want displayed

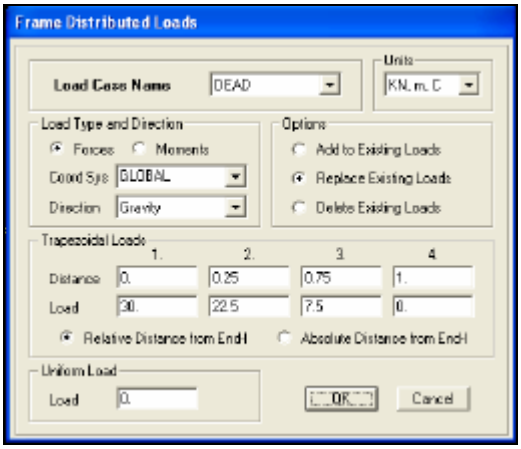
- Assign Joint Restraints (support conditions)
    - Select J1
        - Assign > Joint > Restraints
        - Fast Restraints > Fixed > OK
        -



- Restraints of other joints
  - J2, J4 > U2, R1, R3 (Fast Restraint > Free)
  - J3, J5 > U2, U3, R1, R3 (Fast Restraint > Roller)
- **Notes**
  - **Select several joints to define multiple restraints at once**
  - **Be careful to define lateral restraint to plane frame and truss analyses that are started from a grid**
    - **This is an assumption made for you in the new model wizard if you select a plane structure**

- Assigning Load Cases
  - Define > Load cases
    - **No new load cases needed since all loads act at once**
    - **Select *Add New Load* for alternative load cases to the same structure**

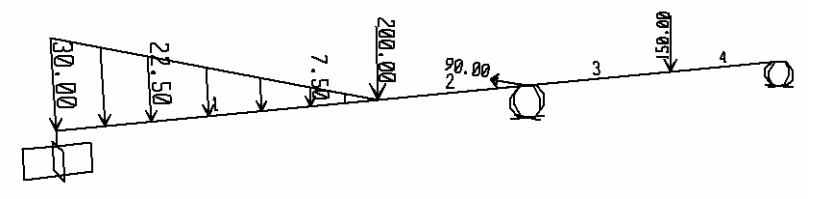
- Assign Loads
  - First select individual members or joints to which you want to assign loads.
  - Select M1
    - Assign > Frame/Cable/Tendon Loads > Distributed
    - Load Case Name: DEAD > Units: [kN, m, C] > Forces > Replace Existing Loads **(Select *Add Existing Loads* if multiple loads are needed on the same member or joint)**
    - Trapezoidal Loads: [0, 30] [0.25, 22.5] [0.75, 7.5] [1, 0] > OK



  - Select J2

- Assign > Joint Loads > Forces
- Load Case Name: DEAD > Units: [kN, m, C] > Replace Existing Loads
- Loads: Force Global Z [-200]

o Repeat the above step for 150 KN concentrated force at Joint J4.
o Select J3
- Assign > Joint Loads > Forces
- Load Case Name: DEAD > Units: [kN, m, C] > Replace Existing Loads
- Loads: Moment about Global Y [90] (+ Moment Clockwise) > OK

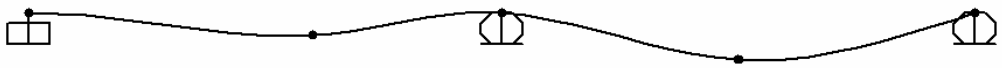


Phase II) Analysis

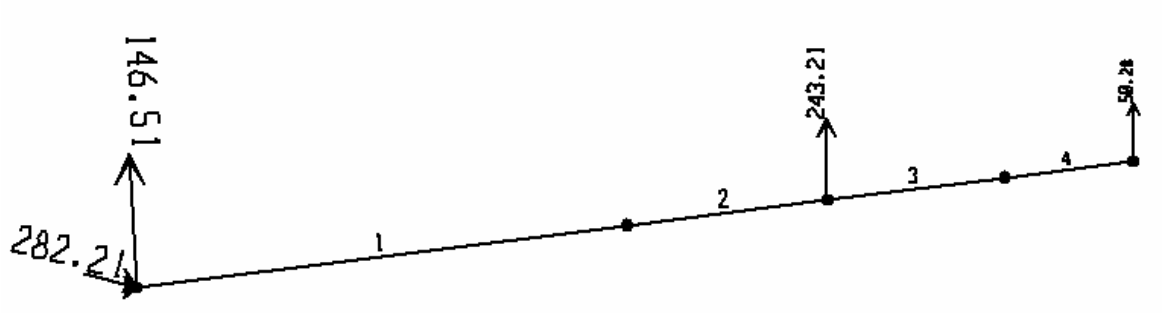
- **SAVE** project if not already done
  - **DO NOT SAVE TO *MY DOCUMENTS* OR *DESKTOP*** (Analysis will not run)
  - Preferably save project in its own separate folder b/c **MANY** output files will be created
- Analyze > Run Analysis (Play Button on top toolbar)
  - Select which Load Cases to Run/Do Not Run
    - Select MODAL > Do Not Run Case > Run Now
  - Analysis Complete > Check for Errors and Warnings > OK

Phase III) Post-Processing

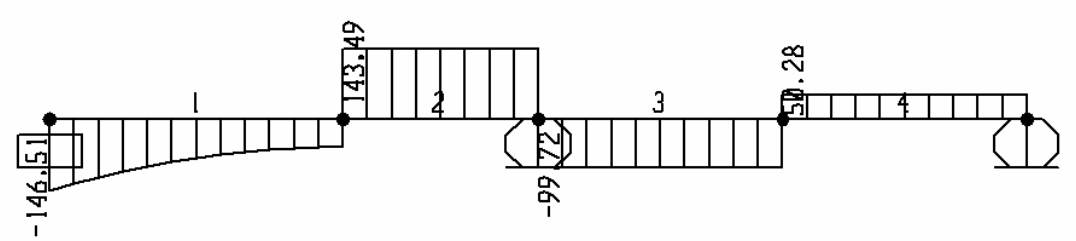
- Edit View for Printing and Display Purposes
  - Options > Colors > Display > Background > White > …
  - Options > Preferences > Dimensions/Tolerances > Screen Line Thickness > …
- Check deflected shape
  - Display > Show Deformed Shape > Choose Load Case > Auto Scaling > Wire Shadow > Cubic Curve



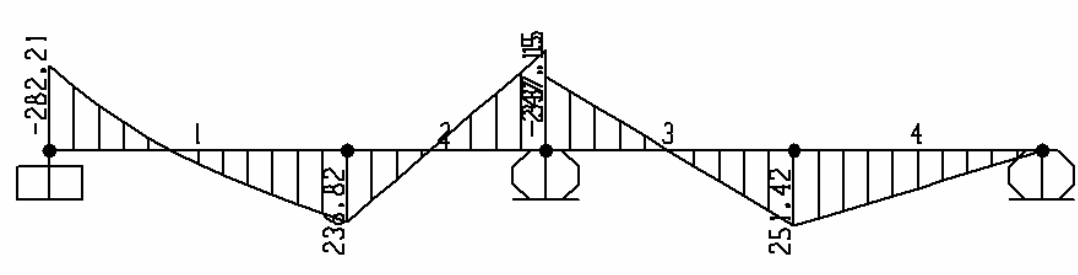
- Check Reactions
  - Display > Show Forces/Stresses > Joints
  - Case/Combo Name: DEAD > Type: Reactions > Show as Arrows > OK
  - **Select 3D View from toolbar or *View* menu (to view the moments at supports)**



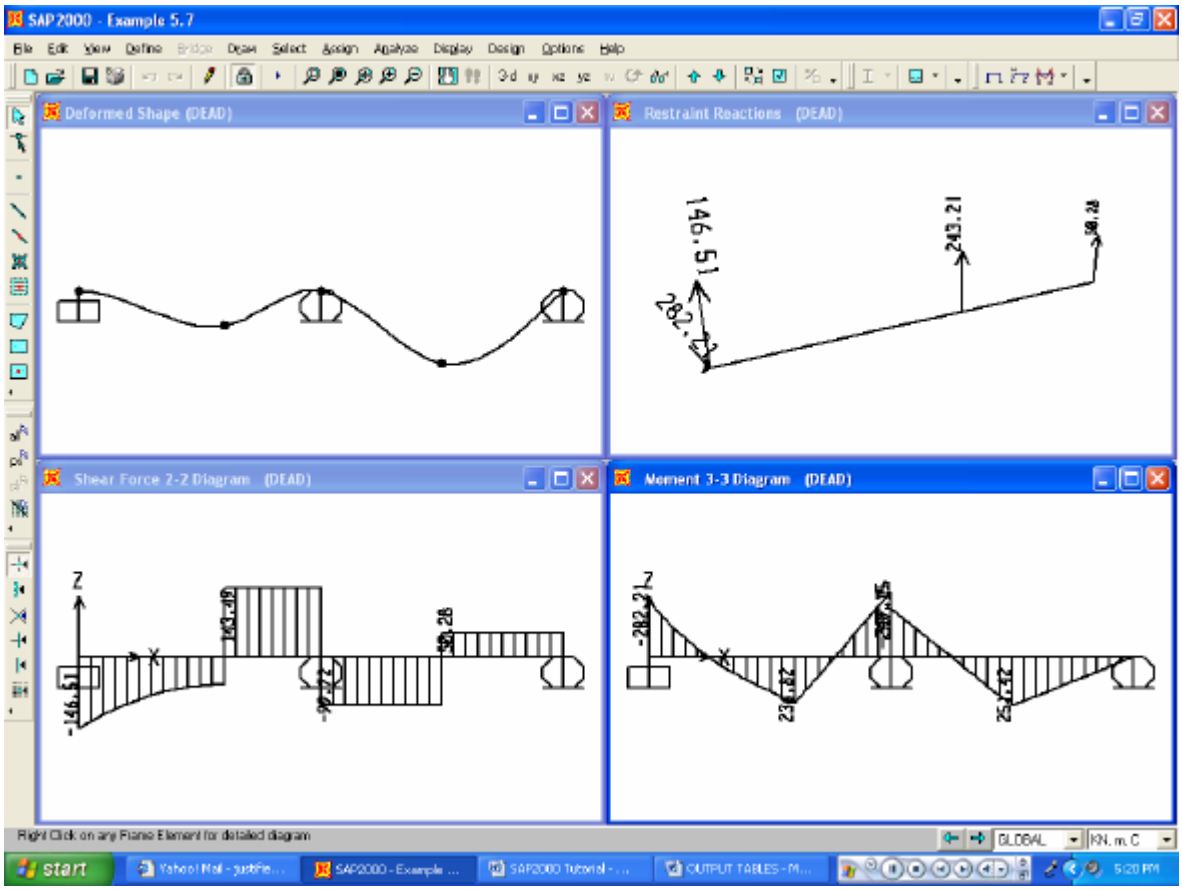
- Check Internal Forces and Diagrams
  - Shear
    - Display > Show Forces/Stresses > Frames/Cables
    - Case/Combo Name: DEAD > Shear 2-2 > Auto Scaling > Show Values on Diagram > OK



  - Bending Moment
    - Display > Show Forces/Stresses > Frames/Cables
    - Case/Combo Name: DEAD > Shear 3-3 > Auto Scaling > Show Values on Diagram > OK



  - Multiple Windows
    - Options > Windows > Four
    - Select window and modify view as desired



- View and Print Output Tables/Results
  - File > Print Tables
  - Analysis Results: Joint Displacement, Joint Reactions, Frame Output
  - Load Cases: DEAD > Analysis Case: Dead > RTF > Print to File > OK
  - File Name: OUTPUT TABLES > Open
  - **Outputs will be a text file located in the folder you save to**

SAP2000 Output Tables

Element Joint Forces

Frame	Joint	OutputCase	F1	F2	F3	M1	M2	M3
Text	Text	Text	KN	KN	KN	KN-m	KN-m	KN-m
1	1	DEAD	0	0	146.506	0	-282.2139	0
1	2	DEAD	0	0	-56.506	0	-236.8242	0
2	2	DEAD	0	0	-143.494	0	236.8242	0
2	3	DEAD	0	0	143.494	0	337.1504	0
3	3	DEAD	0	0	99.715	0	-247.1504	0
3	4	DEAD	0	0	-99.715	0	-251.4248	0
4	4	DEAD	0	0	-50.285	0	251.4248	0
4	5	DEAD	0	0	50.285	0	-3.52E-14	0

Joint Displacements

Joint	OutputCase	U1	U2	U3	R1	R2	R3
Text	Text	m	m	m	Radians	Radians	Radians
1	DEAD	0.000000	0.000000	0.000000	0.000000	0.000000	0.000000
2	DEAD	0.000000	0.000000	-0.004755	0.000000	-0.000549	0.000000
3	DEAD	0.000000	0.000000	0.000000	0.000000	0.000686	0.000000
4	DEAD	0.000000	0.000000	-0.009939	0.000000	0.000621	0.000000
5	DEAD	0.000000	0.000000	0.000000	0.000000	-0.003250	0.000000

Joint Reactions

Joint	OutputCase	U1	U2	U3	R1	R2	R3
Text	Text	KN	KN	KN	KN-m	KN-m	KN-m
1	DEAD	0.000	0.000	146.506	0.0000	-282.2139	0.0000
3	DEAD	0.000	0.000	243.209	0.0000	0.0000	0.0000
5	DEAD	0.000	0.000	50.285	0.0000	0.0000	0.0000

Comparison of Textbook and SAP2000

Joint	Reactions			Displacements		
	Kassimali	SAP2000	% Difference	Kassimali	SAP2000	% Difference
1	146.33	146.51	0.12%	-	-	-
	281.19	282.21	0.36%	-	-	-
2	-	-	-	-0.004473	-0.004755	6.30%
	-	-	-	0.000561	0.000549	2.14%
3	243.46	243.21	0.10%	0.000684	0.000686	0.29%
4	50.21	50.3	0.18%	0.00323	0.00325	0.62%