CE 160 SAP 2000 Notes for 2D Problems

SAP 2000 Main Screen Highlights



Global and Local Reference Coordinate Systems for 2D Problems



Default sign convention for a horizontal element

View in the element 1-2 Plane



Note: in SAP 2000 positive shear is opposite the "usual" Civil Engineering sign convention for positive shear

Default sign convention for a vertical element

(We will do a problem with vertical elements in the next lab)



Positive Joint Displacements



Positive Support Reactions



CE 160 SAP 2000 Lab Problem I

Use SAP 2000 to analyze the concrete beam subjected to dead load and live load below:



Beam Cross Section

Set up and define geometry for the problem

- Start → Programs → SAP2000 20 → SAP2000 (click OK on tip of the day dialog box if it appears);
- From File choose New Model...;
- Choose **Kip**, **ft**, **F** units;
- From the available templates, click on the **Beam** icon;

	Beam Dimensions
<u>~~~</u>)	Number of Spans 3 Span Length 12.
	Use Custom Grid Spacing and Locate Origin Edit Grid
	Section Properties
	Beams Default ~ +
Restraints	OK Cancel

- Type in **Number of Spans** = 3
- **Span Length** = 12 ft (we will modify this below);
- Keep Beam Section Properties as **Default**;
- Leave **Restraints** check box checked (this adds a pin support at the left and roller supports between each span);
- Check Use Custom Grid Spacing box and click on Edit Grid...;

- A dialog box showing the global coordinates of the defined system (4 grid lines in the global X direction separated by 12 ft). Note that the default position for the origin of the global coordinate system is in the middle of the beam;
- Note also that the SAP 2000 default is that 2D models are contained in the XZ Plane;
- Make sure to toggle to Primary on the Line Type when changing gridline coordinates;

System Nam	e	CSY	S1				Grid Lines Quick Start
Grid Data							
Grid ID	Ordinate (ft)	Line Type	Visible	Bubble Loc	Grid Color		
A	-17	Primary	Yes	End		Add	
в	-5	Primary	Yes	End			
С	5	Primary	Yes	End		Delete	0
D	15	Primary ~	Yes	End			-
		Primary Secondary					
Grid Data							Display Grids as
Grid ID	Ordinate (ft)	Line Type	Visible	Bubble Loc	Grid Color		Ordinates O Spacing
1	0	Primary	Yes	Start		Add	
						Delete	Hide All Grid Lines Glue to Grid Lines Bubble Size 3.
Grid Data							Reset to Default Color
Grid ID	Ordinate	(ft) Lin	е Туре	Visible	Bubble Loc		Reorder Ordinates
Z1	0	Pr	imary	Yes	End	Add	
						Delete	Locate System Origin.

- In order to set up the beam with the dimensions given in the problem, change the first X gridline from -18 to -17;
- Change the second X gridline from -6 to -5;
- Change the third X gridline from 6 to 5;
- Change the last X gridline from 18 to 15;
- Check Glue to Grid Lines check box and then click OK to leave Define Grid System Data dialog box;

💢 Beam		×
	Beam Dimensions Number of Spans 3 Span Length 12.	
	Use Custom Grid Spacing and Locate Origin Edit Grid	
	Beams Default v +	
Restraints	OK Cancel	

• Click OK to exit the **Beam** dialog box;

• Click "X" to delete 3-D view so X-Z plane view is only window open (note where the origin of the global coordinate system is located);



Display options in the view window

- Many useful items can be turned on and off from the Display Options menu. To get to this menu from the View menu choose Set Display Options...;
- As an example, to see the local element axes check the Local Axes box under the Frames heading and click OK and the local axes will be displayed. The local 1 axis is red, the local 2 axis is green, and the local 3 axis is blue (in the 3-d view you can see all three axes clearly). To remove the local axes, go back to View, Set Display Options... and uncheck the Local Axes box under the Frames and uncheck the Invisible box under Joints and then click OK. You should see the joints now as visible dots when you look at the model.

bject Options General	Options	
Joints Labels Restraints Springs Local Axes Invisible Not in View Frames Labels Sections Releases Local Axes Not in View	Cables Labels Sections Not in View Tendons Labels Sections Local Axes Not in View Areas Labels Sections Local Axes Not in View	Solids Labels Sections Local Axes Not in View Links Labels Local Axes Not in View
Apply to All Windo	ws Reset Form to Default Value set Form to Current Window S DK Close	es iettings Apply

Assign joint supports (restraints)

- Note that we have an extra roller support that we need to remove so click on the joint where we want to remove the roller support (the joint should be highlighted with a dotted "X");
- Note: to deselect a joint just click on the highlighted joint and the "X" will be removed;
- From Assign move to Joint and choose Restraints... choose the free end (the button with a black dot) under Fast Restraints;
- Click OK to leave joint restraints dialog box (the roller support icon should now be gone from the joint).

💢 Assign Joint Restraints	×											
Restraints in Joint Local Directions												
Translation 1	Rotation about 1											
Translation 2	Rotation about 2											
Translation 3	Rotation about 3											
Fast Restraints												
	<u>→</u>											
OK Clos	e Apply											

Define frame sections

- Change to **Kip**, **in**, **F** units in the lower right corner of the main view window;
- From the **Define** menu choose **Section Properties** and then **Frame Sections...**;
- Click Add New Property... and choose Concrete as the Frame Section Property Type [note that the SAP 2000 default section (FSEC1) is a steel section];
- Choose a **Rectangular** Concrete Section;
- Change **Depth** to 30 in and **Width** to 12 in and leave **Material** as 4000 psi and the **Section Name** as FSEC2;
- Click OK to go back to Frame Properties dialog box and click OK again to return to the main window.

Section Name	FSEC2		Display Color
Section Notes	М	odify/Show Notes	
Dimensions			Section
Depth (t3)	;	30	2
Width (t2)		12	
			3
			•••
			Properties
Material	Prope	ty Modifiers	Section Properties
+ 4000Psi	~	Set Modifiers	Time Dependent Properties

Assign frame sections (FSEC2) to all frame members

- Click on all 3 of the frame members so that they are highlighted (highlighted members have dotted lines) From Assign choose Frame and then Frame Sections... highlight FSEC2 in the dialog box and then click OK in the dialog box to assign FSEC2 to all of the highlighted frame members;
- FSEC2 should appear next to each frame member in the view window.



Define Load Patterns

- From **Define** choose **Load Patterns...**;
- For Dead load Pattern (DEAD) change self weight multiplier to 0 and click on Modify Load Pattern;
- Under Load Pattern Name type LIVE and select LIVE as the pattern type, keep self weight multiplier at 0 and click Add New Load Pattern (two load patterns should now be listed);
- Click **OK** to leave the **Define Load Patterns** dialog box.

oad Patterne					- Click To:
Load Pattern Name	Туре		Self Weight Multiplier	Auto Lateral Load Pattern	Add New Load Pattern
LIVE	Live	~	0		Modify Load Pattern
DEAD LIVE	Dead Live		0 0		Modify Lateral Load Pattern
					Delete Load Pattern
					Show Load Pattern Notes
					ОК

Assign loads

Assign uniform dead load (DEAD)

- Change to **Kip**, **ft**, **F** units;
- Click on each member that will receive dead load (the 3 beam members) to highlight each member;
- From Assign go to Frame Loads and choose Distributed...;
- Check that Load Pattern Name is DEAD and type in a Uniform Load of 0.5 k/ft;
- Leave the default direction as Gravity and check Replace Existing Loads;
- Click **OK** to leave the dialog box (the uniform dead load should appear on the view window).

🗙 s	AP2000 v20	0.0.0 Ulti	mate 64	-bit - (Un	titled)														\times
File	Edit	View	Define	Draw	Select	Assign	Analyze	Displa	y Design Options	Тоо	ls Help								
B	N		9		• •			2	a 3-d xy xz y	z nv .	361	-	I	•		1717	- nd -	I	•
	📜 Fran	ne Sectio	on Prope	rties															• ×
	_								🂐 Assign Frame Distrib	outed Lo	ads							×	
*	General Options																		
Ċ.			$\begin{pmatrix} 1 \end{pmatrix}$					Load Pattern		DEAD ~				O Add to Existing Loads					
R=1			X					Coordinate System	System GLOBAL ~		Ÿ		Replace Existing Loads						
52				ΓÀ.					Load Direction		Gravity		J		O Delete	e Existing l	Loads		
				\searrow					Load Type		Force		ý	Ur	niform Loa	ad			
									Load Type						0.5	kip	/ft		
						F	SEC2		Trapezoidal Loads										
											1.	2.		3.		4.			
Ļ									Relative Distance		0 0.25 0.75		75	1					
******									Loads	0		0	0		0		kip/ft		
-0-4									Relative Distance	e from E	nd-l	O Ab	solute Dista	nce from Er	nd-l				
John a											_								
\times												Reset Form	to Default V	/alues					
											OK		Close	Apply					
2																			
3 Fr	ames Select	ed											X-2	24.294 YO. Z	Z9.549	GLOBAL	~	Kip, ft, F	- ~



Assign uniform live load (LIVE)

- Click on the member that will receive distributed live load (the beam member at the left) to highlight the member;
- From Assign go to Frame Loads and choose Distributed...;
- Check that Load Pattern Name is LIVE and type in a Uniform Load of 1.0 k/ft;
- Leave the default direction as Gravity and check Replace Existing Loads;
- Click **OK** to leave the dialog box (the uniform live load should appear on the view window).



Assign point live load (LIVE)

- Click on the joint that will receive the 8 k point live load and highlight that joint;
- From Assign go to Joint Loads and choose Forces...;
- Select LIVE for Load Pattern Name in dialog box;
- Type in a force in the global Z direction of -8.0 (force is in negative Z direction);
- Leave the default coordinate system as Global and check Replace Existing Loads;
- Click **OK** (the 8 k point live load should appear on the view window);





Show assigned loads

- To confirm all loads assigned to a particular load pattern one can use the Display menu;
- From Display go to Show Load Assigns and choose Frame/Cable/Tendon...;
- To look at live loads, select LIVE for the Load Pattern Name and checking the boxes Show Joint Loads with Span Loads will show the point live loads together with the distributed live loads;
- Click **OK** to leave the dialog box (the live loads should appear in the view window).

Analyze the model

- From Analyze choose Set Analysis Options... and click on Plane Frame/XZ Plane button then click OK;
- From Analyze choose Run Analysis and frame analysis will start after you click Run Now in the dialog box;
- Choose a name to save your frame problem and choose the **desktop** as the destination;
- Note that if you choose a name (e.g. *filename*) SAP 2000 creates 19 files with various extensions (*filename.ext*) that contain the data from your problem. To save your problem, copy all of these files to portable media. To open saved files, after opening SAP 2000, go to the **File** pulldown menu and select **Open...** and navigate to find the file;
- SAP 2000 will now solve the system of equations for the problem and a deformed view of the DEAD load case will be displayed in the view window;
- In case of problems or errors, from the pulldown menu **Analyze** selecting **Show Last Run Details...** will show a dialog box with a summary of the solution process. If there were any stability problems or other irregularities in the solution process, a warning message will be displayed. It is good practice to check the run details.

Fast DOFs]
Space Frame	Plane Frame	Plane Grid	Space Truss	ОК
				Cancel
	XZ Plane	XY Plane	K X	Solver Options
Tabular File	lly save XML, Exc	el or Microsoft Ac	cess tabular file aft	er analysis
Filo namo				

After a successful run, usually the deformed shape of one load pattern (usually the dead load pattern) is displayed in the view window as shown below.



Making changes to the model

- If you want to make changes to the model after you have run the analysis, you must first "unlock" the model;
- You can "unlock" the model by clicking the **lock icon** on the upper task bar on the main view window;
- After making changes to the unlocked model, don't forget that you must then re-analyze the model for the changes to be included in the analysis.

Viewing output

Display frame internal forces

• Go to **Display** then go to **Show Forces/Stresses** and then choose **Frames/Cables...**;

💢 SAP2000 v20.0.0 Ultimate 64-bit - lab 6		– 🗆 ×
File Edit View Define Draw Select Assign Analyze	Display Design Options Tools Help	
□ ♦ 🗏 🚔 🕫 🛯 🖌 🔒 🕨 💽 🔍 🤤 🔍	□ Show Undeformed Shape F4	nht-
Deformed Shape (DEAD)	Show Misc Object Assigns	- ×
-0-	Show Misc Element Assigns	
	Show Object Load Assigns	
	🗞 Show Element Load Assigns 🕨	
L	Show Paths	
	Show Load Case Tree	
	Show Deformed Shape F6	
国	Show Forces/Stresses	F7
	Show Virtual Work Diagram	
	Show Influence Lines Frames/Cables/Tendon	F8
	6 Show Response Spectrum Curves Shells	F9
	Show Response Spectrum envesion Planes Planes	
	Asolids	
8	A Show Static Pushover Curve	F10
×	Show Hinge Results Links	F11
	Show Tables Ctrl+T	
1°.	Save Named Display	
Right Click on any joint for displacement values	nd Show Named Display 👉 📥 GLOBAL	✓ Kaf, mm, C
	Show Manad View	2.1

💢 Display Frame Forces/	Stresses X
Case/Combo	
Case/Combo Name	DEAD ~
Multivalued Options	
Envelope (Max or M	Ain)
Step	1
Display Type	
Force	○ Stress
Component	
 Axial Force 	 Torsion
O Shear 2-2	O Moment 2-2
O Shear 3-3	Moment 3-3
Scaling for Diagram	
 Automatic 	
○ User Defined	
Options for Diagram	
Fill Diagram	 Show Values
	Report Form to Default Valuer
	Reset form to Deladit values
R	eset Form to Current Window Settings
	OK Close Apply

- Choose the load case that you want to see and then choose:
 - **Moment 3-3** and then OK to see the moment diagram of the beam (Note that the default is to draw moment diagram on the tension side, this may be changed using the **Options** pull down menu;
 - Shear 2-2 and then OK to see the shear diagram of the frame;
 - Axial Force and then OK to see the axial force diagram of frame (no axial load in this example);
 - Check Fill Diagram or Show Values on Diagram to see values;
 - Click **OK** to see internal forces in the view window;



- To see internal force diagrams of individual members, right click on the frame member to open the individual member force diagram box. You can even choose different load cases and units for this particular element;
- Find values of the internal forces at different positions along the member by moving the cursor across the frame element. Check the box **Show Max** to see the maximum values and location.
- Note that placing the cursor on the frame shows internal force values at that point on the frame.



Display frame reactions

- Go to **Display** then go to **Show Forces/Stresses** and then choose **Joints...**;
- Choose the load case that you would like to see;
- Checking **Display Type** as **Arrows** shows directions as vectors, otherwise the reaction values are listed;
- Click OK to see the reactions in the view window;
- Right clicking on the support displays a dialog box with the reaction values.

💢 s	AP2000 v	20.0.0 UI	timate 64-	bit - lab 6	j												_		×
File	Edit	View	Define	Draw	Select	Assign	Analyze	Disp	ay Design	Options	Tools	Help							
	\		201		 () 	Q Q	€, ⊕,	П	Show Undefo	ormed Shape	F	4	•		1.	•	1	הר	₩-
	📜 De	formed	Shape (DE	AD)				2	Show Misc O	bject Assigns		•							• ×
-0-								2	Show Misc E	lement Assign	ns	•							
A								2	Show Object	t Load Assigns	;	•							
Ľ								2	Show Elemer	nt Load Assigr	ns	•							
5-1								HI.	Show Paths										
								语	Show Load C	Case Tree									
								5	Show Deform	ned Shape	F	6							
								11	Show Forces	/Stresses		•	4	Joints			F7		
								5	Show Virtual	Work Diagran	n		Ť`	Soil Press	ure				
			Z	Δ				4	Show Influen	nce Lines			1	Frames/0	Cables/	Tendons	F8		
								In 6	Show Respor	nse Spectrum	Curves			Shells			F9		
									Show Plot Fu	inctions	F1	2		Planes					
-0-4								10				-		Asolids					
Å								<u>(ri</u>	Show Static F	Pushover Curv	/e		a	Solids			F10		
X								<u>r</u> L	Show Hinge	Kesults			3.	Links			F11		
									Show Tables.		Ctrl+	Т	E.U.						
54								nd	Save Named	Display									
Rig	nt Click on	any joint	for displace	ement value	es			nd	Show Named	d Display			tion	4	⇒ (GLOBAL	~	(ip, in, f	~
								nv	Show Named	d View									

💢 Display Joint Rea	actions				×
Case/Combo					
Case/Combo Na	me D	DEAD		~	
Multivalued Optio	ns				
Envelope (Ma	ax or Min)				
Step			1	▲ ▼	
Display Types					
Arrows					
 Tabulated 					
	Reset For	m to Default \	/alues		
	Reset Form to 0	Current Windo	ow Settings		
	ОК	Close	Apply		



Display deformed shape

- Go to **Display** and then go to **Show Deformed Shape...**;
- Choose the load case that you want to see and then;
- Check Wire Shadow and check Cubic Curve;
- Click OK to see the deformed shape in the view window;
- To see the individual joint displacements, right click on the joint to open the individual joint displacement dialog box where joint rotations and translations are listed. You can also see the values by placing the cursor on the joint.



Case/Combo	
Case/Combo Name DEAI	D ~
Multivalued Options	
Envelope (Max or Min)	
 Step 	1
Scaling	
Automatic	
O User Defined	
Contour Options	
Draw Contours on Objects	
Contour Component	
Show Continuous Contours	
Automatic	User Defined
Minimum Value for User Contour Range	e
Maximum Value for User Contour Rang	e
Options	
Wire Shadow	
Cubic Curve	
Reset Form to	o Default Values
Reset Form to Curr	rent Window Settings
OK	lose Apply

