Analysis of Trusses using SAP2000 Introduction Manual





Prepared by: Abdellah Ait Elmouden Gabriella Sampaio This manual introduces you to SAP2000. The step-by-step instructions guide you through development of your first model. The intent is to demonstrate the fundamentals and to show how quickly and easily a model can be created using the program. Completing the tutorial will give you hands-on experience working with SAP2000, which for most people is the quickest way to become familiar with the program.

What is SAP2000?

SAP2000 is a general purpose finite element program which performs the static or dynamic, linear or nonlinear analysis of structural systems. It is also a powerful design tool to design structures following AASHTO specifications, ACI and AISC building codes. These features, and many more make SAP2000 the state-of-the-art in structural analysis program.

The SAP2000 graphic user interface (GUI) is used to model, analyze, design, and display the structure geometry, properties and analysis results. The analysis procedure can be divided into three parts:

- 1. Preprocessing.
- 2. Solving.
- 3. Postprocessing

Part I Preprocessing:

In preprocessing, the following information is needed by SAP2000.

- 1. Choosing the units for this project.
- 2. Setting up geometry.
- 3. Defining material and member section properties.
- 4. Assigning member section properties and element releases.
- 5. Defining load cases.
- 6. Assigning load magnitudes.
- 7. Assigning restraints.

PART II Solving In this part SAP2000 will assemble and solve the global matrix.

PART III Postprocessing.

The main options in postprocessing are:

- 1. Displaying the deformed shape.
- 2. Displaying the member forces.
- 3. Printing the results.
- 4. Designing the structural members and checking the safety of a design.

5. Modifying the structure.

Exercice 1: Analysis of Simple Truss Structure using SAP2000

The following exercise illustrates the joint equilibrium method for a simple truss.



Solve using SAP2000: Determine the force in members AB and AC shown in this figure. Given that the angle of member AC with the vertical is equal 60°, while the angle formed by member AB and the vertical direction is equal 90°. The weight applied is w=100lb. State whether is in tension or compression.

In order to solve the problem joint A may be isolated (method of joint). Observe that at this joint there is an externally applied force of 100-lb which is vertical and pulls down on the joint. On the other hand, member AC is pulling up on the joint trying to keep it from moving from its original position. Similarly, member AB also applies a force on the joint in order to keep it from moving from its original position.

The structure shown in the figure above is formed by truss members that can only carry forces along the longitudinal axes of the members, then the force in member AC will be directed from point A to point C. Similarly the force in member AB will be directed along the axis of AB.

The forces in these members can be calculated using the joint equilibrium method at joint A in that the horizontal components of the forces applied at the joint must cancel each other and the vertical components of the forces applied on the joint must also cancel each other. That is the condition of equilibrium of forces, $\Sigma FX = 0$ and $\Sigma Fy= 0$, must be satisfied at joint A.

Therefore,

$$\Sigma F_x = 0: \qquad AB + AC\sin 60^\circ = 0$$

and

 $\Sigma F_{v} = 0$: $AC\cos 60^{\circ} + W = 0$.

In order to maintain equilibrium in the structure, the second equation for vertical components gives the force in member AC as

$$AC = -W/\cos 60^\circ = -200$$
 lb.

Now using the results for AC in the first equation Joint A for horizontal components, the force in member AB is determined as

$$AB = -AC\sin 60^\circ = 173.21$$
 lb.



Solve the problem using SAP2000

About SAP2000 v17 : The Software is available *to all engineering students* during open lab *hours* (Room E344). You can also download an evaluation version from http://www.csiamerica.com

Step 1: Begin a New Model

Start SAP2000 from *File menu > New Model* command or the New Model button.

- Unit: The form shown in Figure will display. Verify that the default units are set to lb, in, F.

When a new model is started, SAP2000 will ask the user to specify a set of units. Those units become the "base units" for the model.

Grid Spacing - Determine the appropriate number of grid line and grid spacing to locate the joints of the truss. Click on Grid only from the set of templates.

New Model New Model Initializ Initialize Mod	ation lel from Defaults w lel from an Existin	iith Units D	in, F 🔽	Project Infor Modify.	mation /Show Info
Select Template					
Blank	Grid Only	<u>ہے ہ</u> ے۔ Beam	2D Trusses	3D Trusses	2D Frames
					T
3D Frames	Wall	Flat Slab	Shells	Staircases	Storage Structures
Underground Concrete	Solid Models	Pipes and Plates			

The Quick Grid Lines window is used to specify the grids and spacing in the X, Y and Z directions.

Number of Grid lines: Set the number of grid lines to 3 for the X direction, and to 3 for the Y and 3 for Z directions.

Grid Spacing: For the truss shown below, assume a length for the distance between joint B and C and then use a sine relationship from the trigonometry of the structure, yielding to a simple relation: If by instance you assume BC to be 24 ft, Then AB is 41.5692 ft. in our Grid Spacing section we will have X direction is 41.56ft, Y and Z 24ft.

First Grid Line location: make sure that these values are all set to zero for this tutorial. Click the **OK** button to accept the changes, and the program will appear as shown in *shown* in the *figure below*.

Note that the grids appear in two view windows tiled vertically. an X-Z "Plan" View on the left and a 3-D View on the right.



Quick Grid Lines	
Cartesian	Cylindrical
Coordinate System Name	e
GLDBAL	
Number of Grid Lines —	
×drection	3
Y direction	3
Z direction	3
Grid Spacing	
×drection	41.56
Y direction	24.
Z direction	24
First Grid Line Location	
imes direction	0.
Y direction	0.
Z direction	0.
OK	Dancel





X87.894 Y48.000 Z-46.313 GLOBAL

▼ b.t.F

Point and Grid intersection

a^k Pe^s 3

X-Z Plane @ Y=48

Note that the Global Axes are displayed as well, and that the Z positive is in the "up" direction. When SAP2000 refers to the direction of gravity, this is in the negative Z direction, or "down."

Step 2: Draw Frame Objects:

Make sure that the X-Z Plane view is active (Click inside the window to activate). This view should be in the left window. Also check that the Snap to Points and Grid Intersections command is active

This will assist in accurately positioning the frame objects. Alternatively, use the Draw menu > Snap to > Points and Grid Intersections command. By default, this command is active.

Now on the left bar of the window, there are several drawing tools. Click the Draw Frame/Cable button the Properties of Object pop-up form for frames will appear and Start drawing the frame as shown in Figure.



Once you have drawn the members Press the Esc key on the key-board to exit the Draw Frame/Cable/Tendon command.

Step 3 Assign joint restraints

Select both joints B and C. Once selected, we go to the bar menu on the top and select Assign ->Joint -> Restraints, and select the pin joint restraint. (Figure)

Make sure you have selected the second Fast Restraint which is the one representing a pinned connection, and then click OK.

Note: A joint restraint is the same as a joint support. It is a rigid connection of the joint to the ground. **Pinned:** All three translational degrees of freedom are restrained.





Once your joints are restrained, is time to assign the applied load. To apply the load, first select the node where it is to be applied, and go to **Assign -> Joint Loads -> Forces.**

The Joint Forces window shows Load Pattern Name and the different axes. Assign the type of load applied to the truss by entering the value of the load in the proper direction, taking in consideration the axes used in the model. On the Load Pattern Name drop down menu, click on the plus sign to add a new load type for load applied. On the Define Load Patterns window, click on the Load Pattern Name box and type a name for the load, for example LIVE, change the type on the next box to LIVE, and then click on Add New Load Pattern. Then click Ok. For this example, the load is applied in the negative direction of z, with a magnitude

Joint Forces	and Passes	
Load Pattern Name	•	Units
Force Global X	0.	GLOBAL
Force Global Z	0.	Options C Add to Existing Loads
Moment about Global X Moment about Global Y	0.	Replace Existing Loads Delete Existing Loads
Moment about Global Z	0.	Cancel

Load Patterns 1 oad Pattern Name Type IVe LIVE 2 DEAD DEAD LIVE	Self Weight A Multiplier L	Auto Lateral oad Pattern	Click To: Add New Load Pattern Modify Load Pattern Modify Lateral Load Pattern Delete Load Pattern Show Load Pattern Notes OK Cancel
---	-------------------------------	-----------------------------	---

Load Pattern Name	✓ Units ✓
Force Global X 0.	Coordinate System
Force Global Z -100 Moment about Global X 0. Moment about Global Y 0. Moment about Global Z 0.	Options O Add to Existing Loads Replace Existing Loads O Delete Existing Loads

The force then will be displayed in the node where it is applied for verification purposes. If more than one force is applied to the structure, then the model will be refreshed and only display the most recent force applied, so the previous applied forces will still be there, but just not visible.



Step 5: Run the analysis

Once all the forces are applied, you are ready to run the analysis of the model. Go to **Analyze -> Run Analysis**, or in the toolbar just look for the button that has the play shape to start the analysis. Once the Set Cases to Run window is displayed, you can select which cases to analyze and which ones to ignore. To accelerate analysis, the Modal case can be set to not run by highlighting it and clicking on the Run/Do not Run Case button.

Face Name	Turne	Cistus	Action	
DEAD	Linear Chatio	Miel Due	Due	Fun/Do Not Run Case
MODAL	Modal	Not Run	Do Not Bun	Show Case
Live1	Linear Static	Not Run	Run	Delete Besults for Case
				Bun/Do Not Run Al
				Didicte All Results
				Show Load Case Tree
alysis Monitor	Options			
Always Shov	y			Bunhlow
Never Show				
Show Alter	4 seconds			Dlí Cancel

Now click on the **Run Now** button. SAP2000 will prompt different results of the analysis according to what is requested. Reactions, member forces, moments, deformations and member displacements are available for display.

Step 6 Show Forces/Stresses - Frames/Cables/Tendons

Use the Display menu > Show Forces/Stresses > Frames/Cables/Tendons command to display column, beam, brace, cable, or tendon forces directly on the SAP2000 model.

1. Click the Display menu > Show Forces/Stress > Frames/Cables/Tendons command to display the Member Force Diagram for Frames form. Use the form to specify the parameters for the display. (See Figure)

- Case/Combo Name drop-down list. Choose the Load Case or Combination to be displayed.
- **Multivalued Options.** The type of load case/combination determines the option(s) available:
- \checkmark For multi-mode cases, choose the mode number for which results are to be shown.
- ✓ For multi-step cases, choose the step number, time step, or frequency step; choose Envelope to view the maximum and/or minimum results over all steps.
- Component options. Specify which component of force is to be displayed. Only one component can be displayed at a time.
- Scaling options. Select the Auto option to have SAP2000 automatically determine a scaling factor. Select the Scale Factor option to specify a scale factor.
- Fill diagram, Show Values on Diagram and Show Deformed Shape check boxes. Use these options to display the force diagrams filled with no text values, unfilled with no text values, unfilled with text values and on a deformed shape.
- ✓ To display force diagrams filled with no text values, check the Fill Diagram check box. Note that if the Show Values on Diagram check box is checked, uncheck it first before checking the Fill Diagram check box.
- ✓ To display force diagrams unfilled with text values, check the Show Values on Diagram check box. Note that if the Fill Diagram check box is checked, uncheck it first before checking the Show Values on Diagram check box.
- ✓ To display force diagrams unfilled with no text values, uncheck both the Fill Diagram and Show Values on Diagram check boxes.
- \checkmark Check the Show Deformed Shape check box to show the model as a deformed shape.

Case/Combo Case/Combo Name	DEAD 💌	
- Multivalued Options C Envelope (Range) C Step		Chose the load type/case
Type • Force	C Stress	
Component		
 Axial Force 	C Torsion	
C Shear 2-2	O Moment 2-2	
C Shear 3-3	C Moment 3-3	
		Choose the force
Scaling		component to display
Auto		
C Scale Factor		
Options		
Fill Diagram ————————————————————————————————————	OK	Choose the opti
C Show Values on Diag	ram and the second s	choose the opti

Check the force values in members AB and AC



Problem 1



1. Start New Model:

New Model Initializati Initialize Model Initialize Model 	on from Defaults with U from an Existing File	nits KN, m,	c 🗸	Project Information	v Information
Select Template	Grid Only	∬ Beam	2D Trusses	3D Trusses	2D Frames
3D Frames	Wall	Flat Slab	Shells	Staircases	Storage
Underground	Solid Models	Pipes and Plates			Structures

2. Click Grid Only

Cartesian Cylindrical	
Coordinate System Nam	e
GLOBAL	
Number of Grid Lines	
X direction	4
Y direction	4
Z direction	5
Grid Spacing	
X direction	4
Y direction	6.
Z direction	3.
First Grid Line Location	
X direction	0.
Y direction	0.
Z direction	0.
ОК	Cancel

3. Switch to XZ plan





4. Assign joint restraints : select joint 1 and 3 click on Assign menu Joints/Restraints

🔀 SAF	2000 v1	.7.3.0 Ult	timate - (l	Untitled)							1000		-
File	Edit	View	Define	Draw	Select	Assig	n Analyze	Display	Design	Options	Tools	s Help	
	V 🔚		201	/ 🔒	▶ ())	*	Joint			•	<u> </u>	Restraints	7
	📜 X-1	Z Plane	@ Y=18			1	Frame			F	47	Constraints	ev
7						3	Cable			Þ	*	Springs	

Select Pinned restraints: All three translational degrees of freedom are restrained.



Assign Load: Once your joints are restrained, is time to assign the applied load. To apply the load, first select the node where it is to be applied, and go to Assign -> Joint Loads - > Forces.



Joint Forces		
Load Pattern Name + DEAD	•	Units KN, m, C
Loads		Coordinate System
Force Global X	0.	GLOBAL
Force Global Y	0.	Ontions
Force Global Z	0.	Add to Existing Loads
Moment about Global X	0.	Replace Existing Loads
Moment about Global Y	0.	Delete Existing Loads
Moment about Global Z	0.	OK Cancel

The Joint Forces window shows Load Pattern Name and the different axes. Assign the type of load applied to the truss by entering the value of the load in the proper direction, taking in consideration the axes used in the model. On the Load Pattern Name drop down menu, click on the plus sign to add a new load type for load applied. On the Define Load Patterns window, click on the Load Pattern Name box and type a name for the load, for example LIVE, change the type on the next box to LIVE, and then click on Add New Load Pattern. Then click Ok. For this example, the load is applied in the negative direction of z, with a magnitude

Define Load Patterns	form .			
Load Patterns 1 Joad Pattern Name	Туре	Self Weight Multiplier	Auto Lateral Load Pattern	Click To: Add New Load Pattern 3
DEAD	LIVE 2	0 1 0		Modify Load Pattern
			▲	Delete Load Pattern
				Show Load Pattern Notes
				OK Cancel

oad Pattern Name + Live	•	Units KN, m, C
oads		Coordinate System
Force Global X	0.	GLOBAL
Force Global Y	0.	Options
Force Global Z	-75	 Add to Existing Loads
Moment about Global X	0.	Replace Existing Loads
Moment about Global Y	0.	Delete Existing Loads
Moment about Global Z	0.	OK Cancel

Click Ok the force then will be displayed in the node where it is applied for verification purposes.



Release Moments

Select all members-> Assign-> Frame-> Release/Partial fixidity -> select moment 22 and moment 33->ok

Frame Releases			Frame Partia	I Fixity Springs
	Start	End	Start	End
Axial Load				
Shear Force 2 (Major)				
Shear Force 3 (Minor)				
Torsion				
Moment 22 (Minor)	V	1	0	0
Moment 33 (Major)	\checkmark	V	0	0
No Releases			Units	KN, m, C
_				

Run the Analysis:

Once all the force are applied, you are ready to run the analysis of the Model. First save your Model then Go to Analyze \rightarrow Run \rightarrow Analysis. Set the Model case to not run by highlighting it and clicking on the Run/Do not run case Button

	_			Click to:
Case	Туре	Status	Action	Run/Do Not Run Case
DEAD	Linear Static	Not Run	Do Not Run	
MODAL	Modal Linear Static	Not Run	Do Not Run	Show Case
LIVE	Linear Static	NOLKUI	Null	Delete Results for Case
				Run/Do Not Run All
				Delete All Nesults
				Charul and Care Tree
				Show Load Case Tree
nahain Manitar Oni	liono			- Madal Albus
naiysis monitor Op	lions			Model-Alive
Always Show				Run Now
Never Show				
				OK Cancel

After running the Model You will see a deformed shape of your structure



Show Forces/Stresses - Frames/Cables/Tendons

Click the **Display menu > Show Forces/Stress > Frames/Cables/**Tendons command to display the Member Force Diagram. Make sure that axial force and show values on diagram are selected. Click **OK**

Case/Combo	
Case/Combo Name	live
Multivalued Options	
Envelope (Range)	
Step	1
Туре	
Force	Stress
Component	
Axial Force	Torsion
Shear 2-2	Moment 2-2
Shear 3-3	O Moment 3-3
Scalino	
Auto	
Scale Factor	
Options	
🔘 Fill Diagram	ок
Show Values on D	Diagram Cancel

Problem 2



1. Start New Model:

New Model Initializati Initialize Model Initialize Model 	ion from Defaults with Ur from an Existing File	nits KN, m,	C •	Project Information	v Information
Select Template	Grid Only	Beam	2D Trusses	3D Trusses	2D Frames
3D Frames	Wall	Flat Slab	Shells	Staircases	Storage Structures

2. Click Grid Only

X Quick Grid Lines	×
Cartesian Cylindrical	,
Coordinate System Name	e
GLOBAL	
Number of Grid Lines	
X direction	2
Y direction	1
Z direction	2
Grid Spacing	
X direction	5
Y direction	1
Z direction	5
First Grid Line Location	
X direction	0.
Y direction	0.
Z direction	0.
ОК	Cancel

3. Switch to XZ plan





4. Assign joint restraints : select joint 1 and 3 click on Assign menu Joints/Restraints

💢 SAP	2000 v1	7.3.0 Ult	timate - (l	Jntitled)								and provide a	
File	Edit	View	Define	Draw	Select	Assig	n Analyze	Display	Design	Options	Too	ls Help	
) 🗄	a	90	/ 🔒	▶ 🛞	*	Joint			•	5	Restraints	2
·····	📜 X-2	Z Plane	@ Y=18			1	Frame			×.	÷	Constraints	ev
7						3	Cable			►	-	Springs	

Select Pinned restraints: All three translational degrees of freedom are restrained.

Joint Restraints	1	
Restraints in Joint Local Directions		5
Translation 1 🔲 Rotation about 1		
Translation 2 Rotation about 2		
✓ Translation 3 C Rotation about 3		
Fast Restraints		
OK Cancel		



The Joint Forces window shows Load Pattern Name and the different axes. Assign the type of load applied to the truss by entering the value of the load in the proper direction, taking in consideration the axes used in the model. On the Load Pattern Name drop down menu, click on the plus sign to add a new load type for load applied. On the Define Load Patterns window, click on the Load Pattern Name box and type a name for the load, for example LIVE, change the type on the next box to LIVE, and then click on Add New Load Pattern. Then click Ok. For this example, the load is applied in the negative direction of z, with a magnitude

Define Load Patterns Load Patterns Load Pattern Name LIVI DEAD IVe LIVI DEAD LIVI	Type Self Weight Multiplier 2 0 10	Auto Lateral Load Pattern	Click To: Add New Load Pattern Modify Load Pattern Modify Lateral Load Pattern Delete Load Pattern
			Show Load Pattern Notes OK Cancel

Assign Load: Once your joints are restrained, is time to assign the applied load. To apply the load, first select the node where it is to be applied, and go to Assign -> Joint Loads - > Forces.

Load Pattern Name		Units		
+ live	•	KN, m, C 🗸		
Loads		Coordinate System	80.00	
Force Global X	80	GLOBAL		
Force Global Y	0.	Options		
Force Global Z	0.	 Add to Existing Loads 		
Moment about Global X	0.	Replace Existing Loads		
Moment about Global Y	0.	Delete Existing Loads		\mathbf{i}
Moment about Global 7	0.	OK Cancel	4	

Release Moments

rame Releases			Frame Partia	l Fixity Springs
	Start	End	Start	End
Axial Load				
Shear Force 2 (Major)				
Shear Force 3 (Minor)				
Torsion				
Moment 22 (Minor)	V	1	0	0
Moment 33 (Major)	V	V	0	0
No Releases			Units	KN, m, C
_				

Select all members-> Assign-> Frame-> Release/Partial fixidity -> select moment 22 and moment 33->ok

Run the Analysis:

Once all the force are applied, you are ready to run the analysis of the Model. First save your Model then Go to Analyze \rightarrow Run \rightarrow Analysis. Set the Model case to not run by highlighting it and clicking on the Run/Do not run case Button

				Click to:
ise	Туре	Status	Action	Run/Do Not Run Case
AD	Linear Static	Not Run	Do Not Run	Kalizbo Hori Kali Case
DAL	Modal	Not Run	Do Not Run	Show Case
e	Linear Static	Not Run	Run	Delete Results for Case
				Pun/Do Not Pun All
				Delete All Results
				Show Load Case Tree
		I		
sis Monitor Options				Model-Alive
Alwaye Show				
-iways onow				Run Now
Vever Show				

After running the Model You will see a deformed shape of your structure



Problem 3



1. Start New Model:



2. Click Grid Only

Cartesian Cylindrical	
Coordinate System Na	me
GLOBAL	
Number of Grid Lines	
X direction	3
Y direction	2
Z direction	2
Grid Spacing	
X direction	20
Y direction	1
Z direction	16
First Grid Line Location	n
X direction	0.
Y direction	0.
Z direction	0.
ОК	Cancel

5. Switch to XZ plan





4. Select Draw Frame/Cable tool 5. Draw the sample truss in Figure 3 2 3 2 3 2 4 2 3 2 4 2 4 2 3 4 4 5 5 6 7 6 7 7 8 1 1 2 2 4 4 4 4 4 5 5 6 6 7 6 7 7 8 7 9 9 1 <li1 1 1 1 1</l

6. Assign joint restraints : select joint click on Assign menu Joints/Restraints

×	SAP	2000 v1	7.3.0 Ult	imate - (l	Untitled)									
	File	Edit	View	Define	Draw	Select	Assig	n Analyze	Display	Design	Options	Тоо	ls Help	
) 🔚	a	201	/	▶ 🕑	*	Joint			•	5*	Restraints	7
		X-2	Z Plane	@ Y=18			1	Frame			F	ب	Constraints	ev
Ľ	2						C	Cable			F	**	Springs	

Select Pinned restraints: All three translational degrees of freedom are restrained.

ot 💢	int Restraints		×	_	_	_
l c	Restraints in Joint Local	Directions		2	2	(2
	✓ Translation 1	Rotation about 1			(A)	Ċ
	▼ Translation 2	Rotation about 2				
	▼ Translation 3	Rotation about 3	\sim	Δ		
	Fast Restraints					
		\Delta		4 /	\sim	
	ОК	Cancel	Ć.		\rightarrow	
-						

ROTATE THE SUPPORTS TO WORK ACCONDINTLY WITH THE MODEL

Select the supports -> Assign -> Joint-> Local Axis -> change the Y value to 90.



Assign Load: Once your joints are restrained, is time to assign the applied load. To apply the load, first select the node where it is to be applied, and go to Assign -> Joint Loads - > Forces.

Load Pattern Name + DEAD	•	Units KN, m, C
Loads		Coordinate System
Force Global X	-25	GLOBAL -
Force Global Y	0.	Ontions
Force Global Z	0.	 Add to Existing Loads
Moment about Global X	0.	Replace Existing Loads
Moment about Global Y	0.	Delete Existing Loads
Moment about Global Z	0.	OK Cancel

The Joint Forces window shows Load Pattern Name and the different axes. Assign the type of load applied to the truss by entering the value of the load in the proper direction, taking in consideration the axes used in the model. On the Load Pattern Name drop down menu, click on the plus sign to add a new

load type for load applied. On the Define Load Patterns window, click on the Load Pattern Name box and type a name for the load, for example LIVE, change the type on the next box to LIVE, and then click on Add New Load Pattern. Then click Ok. For this example, the load is applied in the negative direction of *z*, with a magnitude

1 Joad Patter	n Name	Self Weight Multiplier	Auto Lateral Load Pattern Click To: Add New Load Pattern Modify Load Pattern
DEAD live	DEAD		Modify Lateral Load Pattern Modify Lateral Load Pattern Show Load Pattern Notes OK
Joint	ad Pattern Name		Units KN, m, C
Lo	ads		Coordinate System
F	orce Global X	0	GLOBAL
F	orce Global Y	0.	Ontions
F	orce Global Z	-25	Add to Existing Loads
м	oment about Global X	0.	Replace Existing Loads
м	oment about Global Y	0.	Delete Existing Loads

Click Ok the force then will be displayed in the node where it is applied for verification purposes.



RELEASE MOMENTS

Select all members-> Assign-> Frame-> Release/Partial fixidity -> select moment 22 and moment 33->ok

Frame Releases			Frame Partia	I Fixity Springs
	Start	End	Start	End
Axial Load				
Shear Force 2 (Major)				
Shear Force 3 (Minor)				
Torsion				
Moment 22 (Minor)	V	V	0	0
Moment 33 (Major)	\checkmark	V	0	0
No Releases			Units	KN, m, C
	OK		Cancel	

Run the Analysis:

Once all the force are applied, you are ready to run the analysis of the Model. First save your Model then Go to Analyze \rightarrow Run \rightarrow Analysis. Set the Model case to not run by highlighting it and clicking on the Run/Do not run case Button

				Click to:
Case	Туре	Status	Action	Run/Do Not Run Case
DEAD	Linear Static	Not Run	Do Not Run	
MODAL	Modal Linear Static	Not Run	Do Not Run	Show Case
LIVG	Linear Static	Not Rull	Kuii	Delete Results for Case
				Run/Do Not Run All
				Delete All Results
				Boloto All Hoodito
				Show Load Case Tree
				Show Load Case free
alvsis Monitor Onti	005			Model-Alive
) Always Show				Run Now
Marrian Charter				

After running the Model You will see a deformed shape of your structure







At joint 2





$$EFy=0 - F_{23y} - 75=0$$
 eq II
 $F_{23y} = -75 \text{ kN}$ (which means the force is pointing up 1)

$$Fag Nin\theta = -75KN$$

 $l \theta = .6$

$$F_{23}(.6) = -75KN$$

 $F_{23} = -125KN$

Plug the value of F23 into equation I to find F21

$$-F_{21} - F_{23} cob = 0$$

 $-F_{21} - (-125)(.8) = 0$
 $F_{21} = +100 KN$

Member (1) has an internal force of 100 KN (tension) member (2) has an internal force of 125 KN (Compression) 0



01. FIND REACTIONS





FORCES 02, FIND INTERNAL

NODE 1:

. Fra	EFx=0	-80+F12=0	Fiz=80KN (tension)
$80 \leftarrow 0 \rightarrow F_{12}$	&Fy=0	-80 + F13=0	FI3 = 80KW (tension)

NODE 3:

$$EF_{x}=0 \quad 80 + F_{32}\cos\theta = 0 \quad eqI$$

$$F_{32} \quad EF_{x}=0 \quad -80 - F_{32}\sin\theta = 0 \quad eqI$$

$$F_{32} \quad F_{32} \quad F_{32}$$

Find angle from geometry

eqI -80 - $F_{32}\sqrt{\frac{1}{2}} = 0$ $F_{32} = -\frac{80(2)}{\sqrt{2}}$ (proved that any equilibrium equation give same results)

2

The internal forces are:

member (1): 80KN (tension) member (2): 80KN (tension) member (3): 113.13KN (compression)

Example

$$I = \frac{1}{164} \int_{F_{1}}^{F_{1}} \int_{F_{1}}^{F_{1}} \int_{F_{2}}^{F_{2}} \int_{F_$$

· NODE OI

62.5KN

$$\begin{aligned} & \mathcal{E}F_{X}=0 \\ & 625+F_{12}+F_{15}\cos\theta=0 \\ & \mathcal{E}F_{Y}=0 \\ & F_{15}\sin\theta=0 \implies F_{15}=0 \\ & F_{12}=-62.5\,\text{KN} \end{aligned}$$

$$\mathcal{E}Fy=0 \quad F_{42} \quad \text{Din}\theta - 25=0 \quad \theta''=\theta'=\theta$$

$$F_{42} = \frac{25}{.625} = 40 \text{ KN}$$

$$E+x=0$$
 $F+5=62.5+F+2.00.0=0$
 $F+5=62.5+(40)(.7B09)=0$
 $F+5=31.26$ KN

NODE 05 31.26 = 0 $F_{52} = 0$ $F_{52} = -40$ (.625) = -25KN

check equilibrium

$$31.26 - 40 (\cos \theta) = 0$$

 $31.26 - 31.24 \cong 0$ (approximation of results makes the difference)

Internal forces

$$F_{15} = 0$$

$$F_{1a} = 62.5 \text{W}(\text{compression})$$

$$F_{45} = 31.26 \text{W}(\text{-lension})$$

$$F_{4a} = +0 \text{KN}(\text{-lension})$$

$$F_{5a} = 25 \text{KN}(\text{compression})$$

$$F_{a3} = 31.24 \text{KN}(\text{compression})$$

$$F_{a3} = 40 \text{KN}(\text{-lension})$$

Problem 4 :

Solve using SAP2000: Determine the force in each member of the warren truss show. State whether is in tension or compression.



Solution:

 $F_{AB} = 7.50 \text{ kips } C$ $F_{AC}=4.5 \text{ kis } T$ $F_{BC}=7.5 \text{ kips } T$ $F_{BD}=9.00 \text{ kips } C$ $F_{CD} = 0 \text{ kips}$ $F_{CE} = 9.00 \text{ kips } T$