

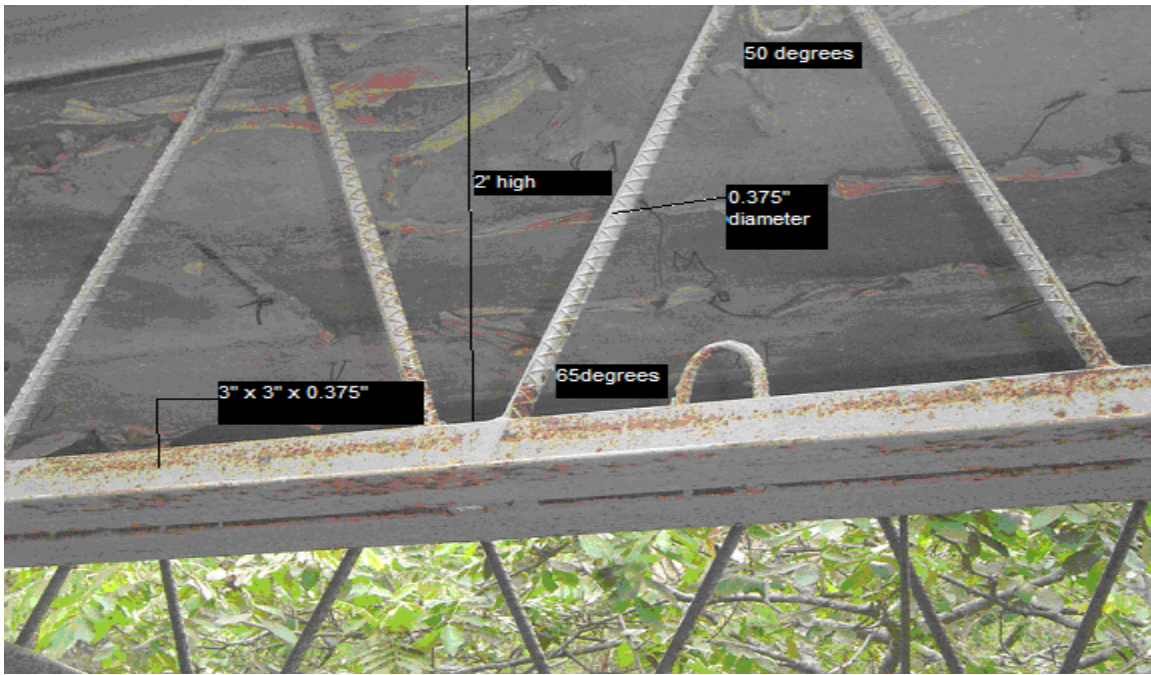
Truss Bridge Analysis Using SAP2000

SAP2000 is a structural analysis and design program that allows users to create and test a computer animated model of their figure. The program calculates dead loads, resulting forces, moments, deflection, and much more. SAP2000 can be used in various fields from transportations systems to stadiums. For the Rice CEVE 499 independent study course we will use SAP2000 to draft a model of a truss bridge that we intend to build in Nicaragua. These preliminary designs serve to help students visualize the forces that will effect the bridge, increase students ability to calculate forces, and ultimately allow a team of students to decide the best structural design to fabricate in Nicaragua.

SAP2000 integrates several templates, but the user must indicate the distances of the members and the type of material for each member. If the members are not assigned, SAP will use I-beams as a default in bridges. The following ~30ft. bridge in El Salvador was used as a guide for the dimensions of the Nicaragua bridge built in SAP2000.



The first measurements that we deduced from this figure is the height, width, and angles of the bays (sections) of the truss. Then we estimated the diameter of the rebar. The rebar is held in place by double angle iron. The double angle iron looks like to placed back to back with the width of the rebar separating them.



We also noticed that the bridge was separated into sections of groups of eight and five bays. These sections were held together by the double angle iron every eight bays running as columns, and by single angle iron every five bays running horizontal to connect the two sides of the truss. The angle iron measurements were estimated to be the same as the double angle iron used above.



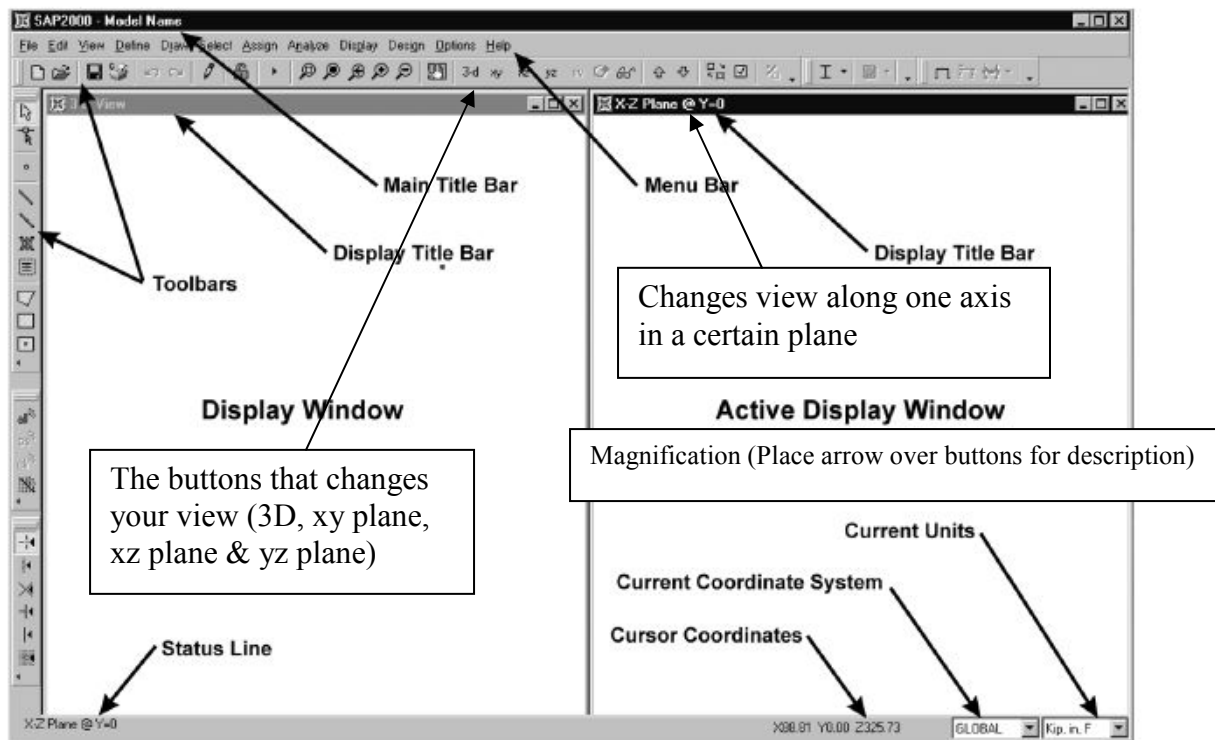
Another consideration was the deck. From the pictures of or model bridge, we estimated a deck that was 3" in thickness, built out of concrete.



Below, we have gone more in depth on how to construct a truss bridge based on this example.

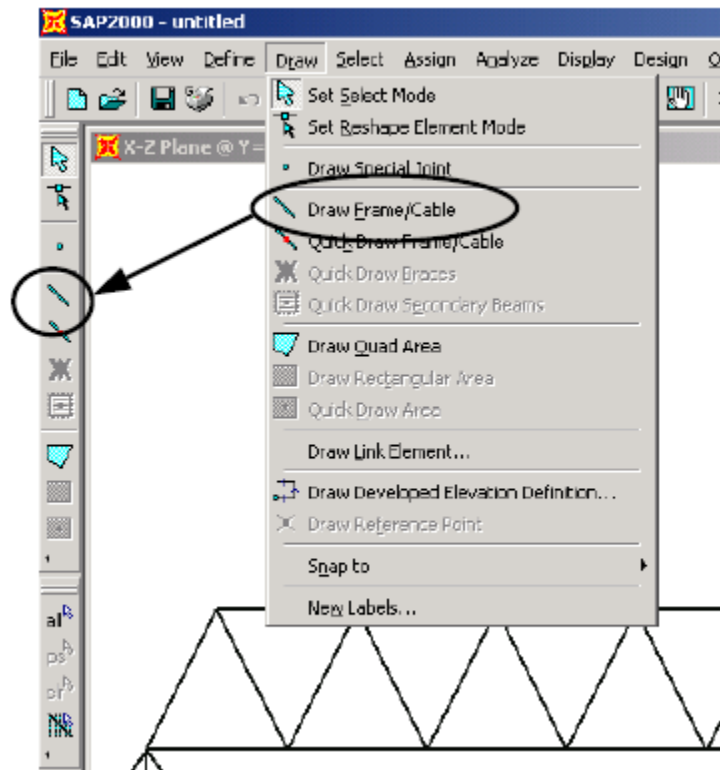
Introduction to SAP2000

The Main Screen (Courtesy of the SAP2000 Manual):



The Draw Buttons (Courtesy of the SAP2000 Manual):

Match the buttons with the description in the menu under ‘Draw.’



Using SAP 2000 to Create a Virtual Truss Bridge

1. Open SAP 2000.
2. File → New Model... → Sloped Truss
3. In the “Sloped Truss” menu, enter -
Number of Bays: 39
Height of Truss: 24
Truss Bay Length: 22.38
4. Click OK.
5. In the 3-D view or in the X-Z plane, select entire bridge by right clicking on the mouse and dragging it to the right to include the entire bridge.
6. Edit → Replicate...
7. In the “Linear” section of the “Replicate” menu, enter –
dy: 59.04
8. Click OK.
9. You should now see the two sides of the bridge in the 3-D view.
10. Click on the “xy” button in the toolbox at the top
11. Draw → Draw Quad Area

12. Click on the four corners of the deck clock-wise and double click on the last corner.
13. You should see a red rectangle that outlines the deck.
14. Select → Select → Pointer Window
15. Select everything by right clicking on the mouse and dragging it to the right to include everything.
16. Edit → Mesh Areas...
17. Click on “Mesh using selected Joints on edges”
18. Click OK.
19. You should see red vertical lines dividing the deck.
20. Draw → Frame cable
21. Draw in bottom supports every 5 bars by clicking on one end and then double clicking on the other end (so create 8 lines along the red lines every 5 red lines).
22. Repeat steps 15 – 17.
23. View → 2D View -> X-Z Plane
24. Magnify into the first triangular truss by clicking on the button with the magnifying glass with a square in it, and selecting the area to be magnified.
25. Draw → Frame cable
26. Click on the top vertex of the triangle and double click on the mid point of the base of the triangle.
27. Click on the mouse button in the menu to your left.
28. Select the line that you just drew in 26.
29. Edit -> Replicate...
30. Enter the following:
Increments
dx=179.04
dy=0
dz=0
Increment Data
Number: 4
31. Select everything in window.
32. Edit → Replicate
33. Enter:
Increments
dy=59.04
Increment Data
Number: 1
34. Click OK.
35. View X-Z Plane @ Y=0.
36. Select the top and the bottom lines of the truss.
37. View → Set 2-D View → X-Z Plane Y=59.04
38. Repeat 32.
39. Define → Frame/Cable Sections
40. Select the following:
Choose Property Type for Add
Import Double Angle

- Add Double Angle
41. Click to:
Add New Property...
 42. Enter the following:
Outside depth (t3): 3
Outside width (t2): 6.625
Horizontal leg thickness (tf): 0.375
Vertical leg thickness (tw): 0.375
Back to back distance (dis): 0.625
 43. Click OK.
 44. Click OK.
 45. Assign → Frame Cable → Sections...
 46. Choose FSEC1 from Properties.
 47. Click OK.
 48. Select the five vertical bars that you drew in.
 49. View → Set 2-D View → X-Z Plane Y=59.04
 50. Repeat 48.
 51. Assign → Frame Cable → Sections...
 52. Choose FSEC1 from Properties.
 53. Click OK.
 54. Select all the diagonal bars.
 55. View → Set 2-D View → X-Z Plane Y=59.04
 56. Repeat 46.
 57. Define → Frame/Cable Sections
 58. Select the following:
 59. Choose Property Type for Add
Import Circle
Add Circle
 60. Click to:
Add New Property...
 61. Enter the following:
Diameter (t3): 0.625
 62. Click OK.
 63. Click OK.
 64. Assign → Frame Cable → Sections...
 65. Choose FSEC2 from Properties.
 66. Click OK.
 67. View X-Y Plane as in step 10 or go to View → Set 2-D View → X-Y Plane.
 68. Repeat steps 15-17.
 69. Select the eight bottom supports that you drew in.
 70. Define → Frame/Cable Sections
 71. Select the following:
Choose Property Type for Add
Import Angle
Add Angle
 72. Click to:

- Add New Property...
73. Enter the following:
 - Outside vertical leg (t3): 3
 - Outside horizontal leg (t2): 3
 - Horizontal leg thickness (tf): 0.375
 - Vertical leg thickness (tw): 0.375
 74. Click OK.
 75. Click OK.
 76. Assign → Frame Cable → Sections...
 77. Choose FSEC3 from Properties.
 78. Click OK.

Loads and Analysis

1. Go to Define → Load Cases
2. Type in a name for the load (i.e. LIVE).
3. Click the arrow under type, and select from the pull down menu, the type of load you want to apply (i.e. LIVE).
4. Select Add New Load.
5. Click OK.
6. Highlight the area that will experience the load (i.e. the deck).
7. “Assign” → “Area Loads” → “Uniform(Shell)...”
8. Under the “Load Case Name” pull down menu select the load that you want to assign (i.e. LIVE).
9. Select the units of the Load from the pull down menu.
10. Insert the value of the load under the “Uniform Load” section.
(When assigning this load remember that it is per unit area, not per unit length.)
11. Click OK.
12. Go to Analyze → Run Analysis
13. Make sure that the Loads that you want to run are assigned “Run” under the Action column
14. To change the action condition, highlight the load and select Run/ Do Not Run Case (i.e. do not run the “Modal” option).
15. Click OK.
16. Another pop-up screen will come up, which is the analysis, select OK.
17. The deformed shape will show in your current window.
18. To change the scale factor, press F6 or the “show deformed shape” icon (2nd from the right).
19. Choose the load that you want to analyze (dead or alive)
20. Scaling, select Scale Factor, and insert 1
21. Select OK.
22. For a force analysis, click the arrow beside the Show Forces/Stresses icon (first from the right) → Frames/Cables...
23. A pop up window should show up.
24. Select the case load.
25. Select the component you want to analyze.

26. Select your scale factor (i.e. Scale Factor→1)
27. Click OK.
28. To view the tension of a certain member, right click on the member.
29. Click DONE to exit.