

Problem S

Finite Element Model of Steel Beam With Web Openings

Steel

$E = 29000$ ksi

Poissons Ratio = 0.3

Built-Up Beam: $d = 40$ in

$t_w = 0.75$ in

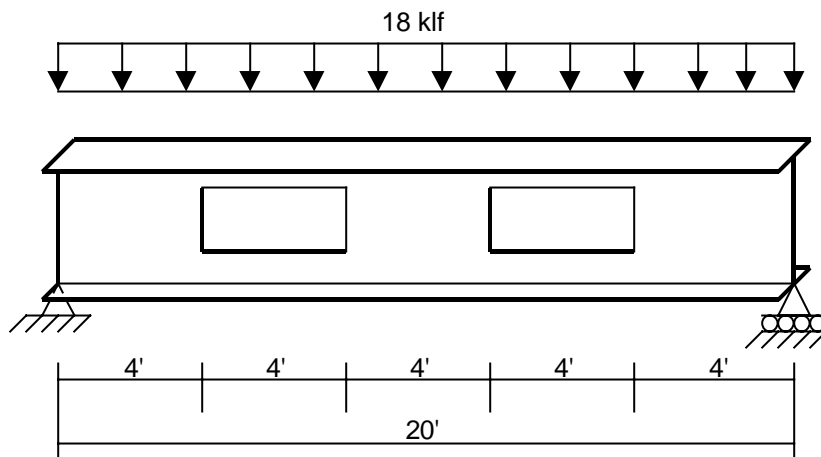
$b_f = 16$ in

$t_f = 2$ in

Beam openings are 20" high by 48" wide centered over the depth of the beam





To Do

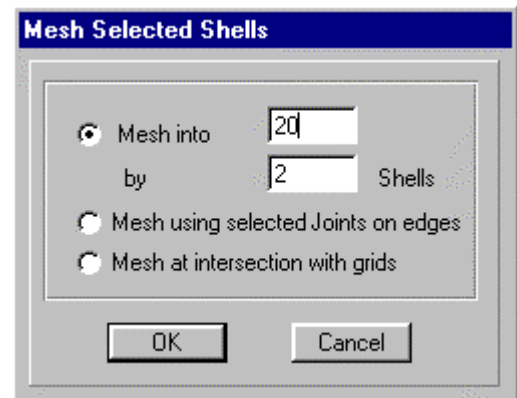
Construct a finite element model of this simply supported beam using shell elements. Determine moments and shears at center of beam and at center of left opening. Plot shear stresses (S12) in beam web.



Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem S Solution

1. Click the drop down box in the status bar to change the units to kip-in. 
2. From the **File** menu select **New Model...** This displays the Coordinate System Definition dialog box.
3. In this dialog box
 - Select the Cartesian Tab.
 - In the Number of Grid Spaces area type **1** in the X direction edit box.
 - In the Number of Grid Spaces area type **2** in the Y direction edit box.
 - In the Number of Grid Spaces area type **1** in the Z direction edit box.
 - In the Grid Spacing area type **240** in the X direction edit box.
 - In the Grid Spacing area type **8** in the Y direction edit box.
 - In the Grid Spacing area type **40** in the Z direction edit box.
 - Click the **OK** button.
4. Click in the window titled X-Y Plane @ Z=40 to make sure it is active. The window is highlighted when it is active. The screen appears as shown in Figure S-1.
5. Click the **Draw Rectangular Shell Element** button  on the side toolbar or select **Draw Rectangular Shell Element** from the **Draw** menu.
6. Click on the point labeled “A” and then the point labeled “B” in Figure S-1 to enter a shell element.
7. Click the **Pointer** button  to exit Draw Mode and enter Select Mode.
8. Click on the shell element to select it.
9. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.
10. Fill in this dialog box as shown in the adjacent figure and click the **OK** button.
11. Click the **Select All** button  on the side toolbar to select all elements.



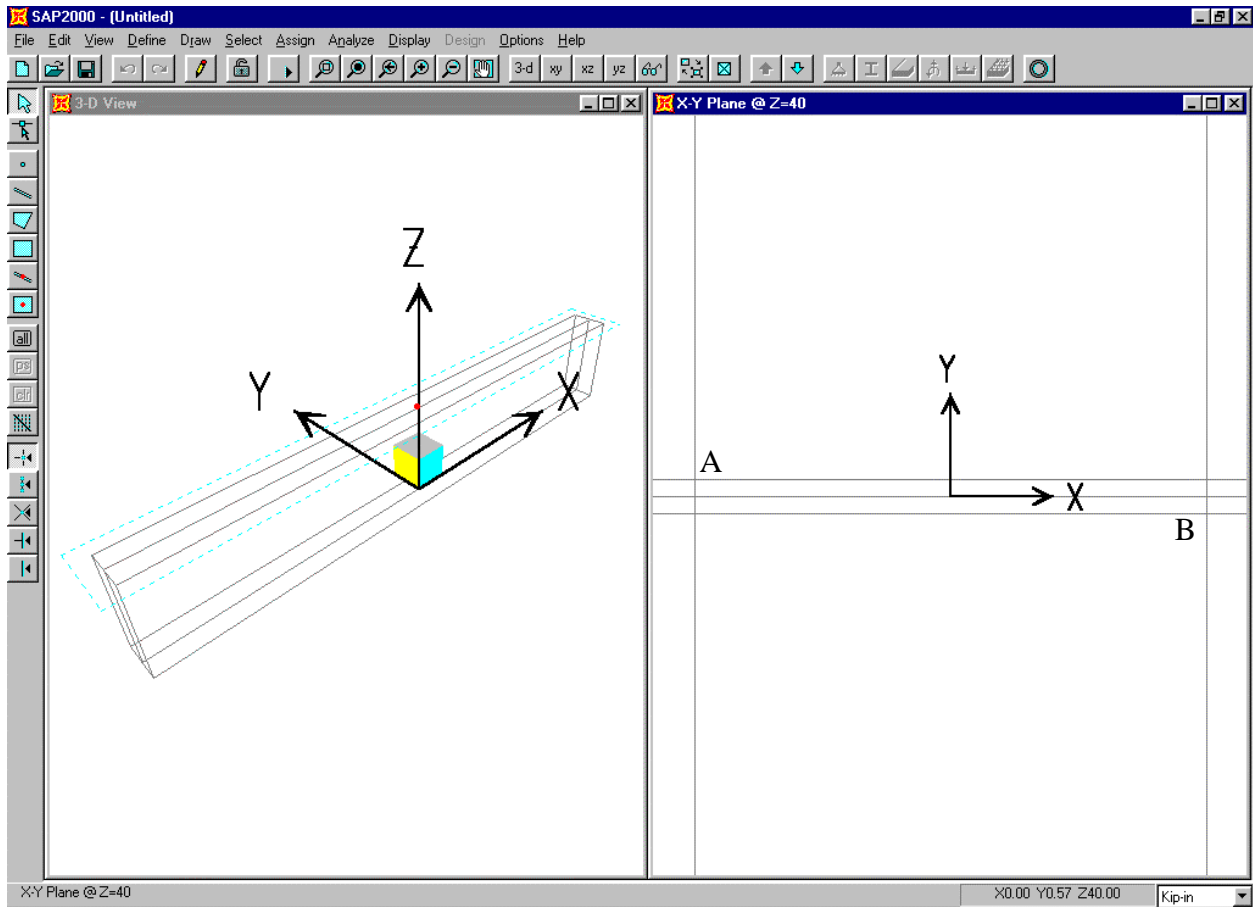


Figure S-1: Screen After Step 4

12. From the **Edit** menu select **Copy**.
13. From the **Edit** menu select **Paste...** to display the Paste Coordinates dialog box.
14. In this dialog box:
 - Type **-40** in the Delta Z edit box.
 - Click the **OK** button.
15. Click in the window titled X-Y Plane @ Z=40 to make sure it is active. Do not accidentally select any elements while doing this.
16. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.
17. In this dialog box:
 - Select the X-Z Plane option.
 - Type **0** in the Y= edit box.

- Click the **OK** button. The screen appears as shown in Figure S-2.

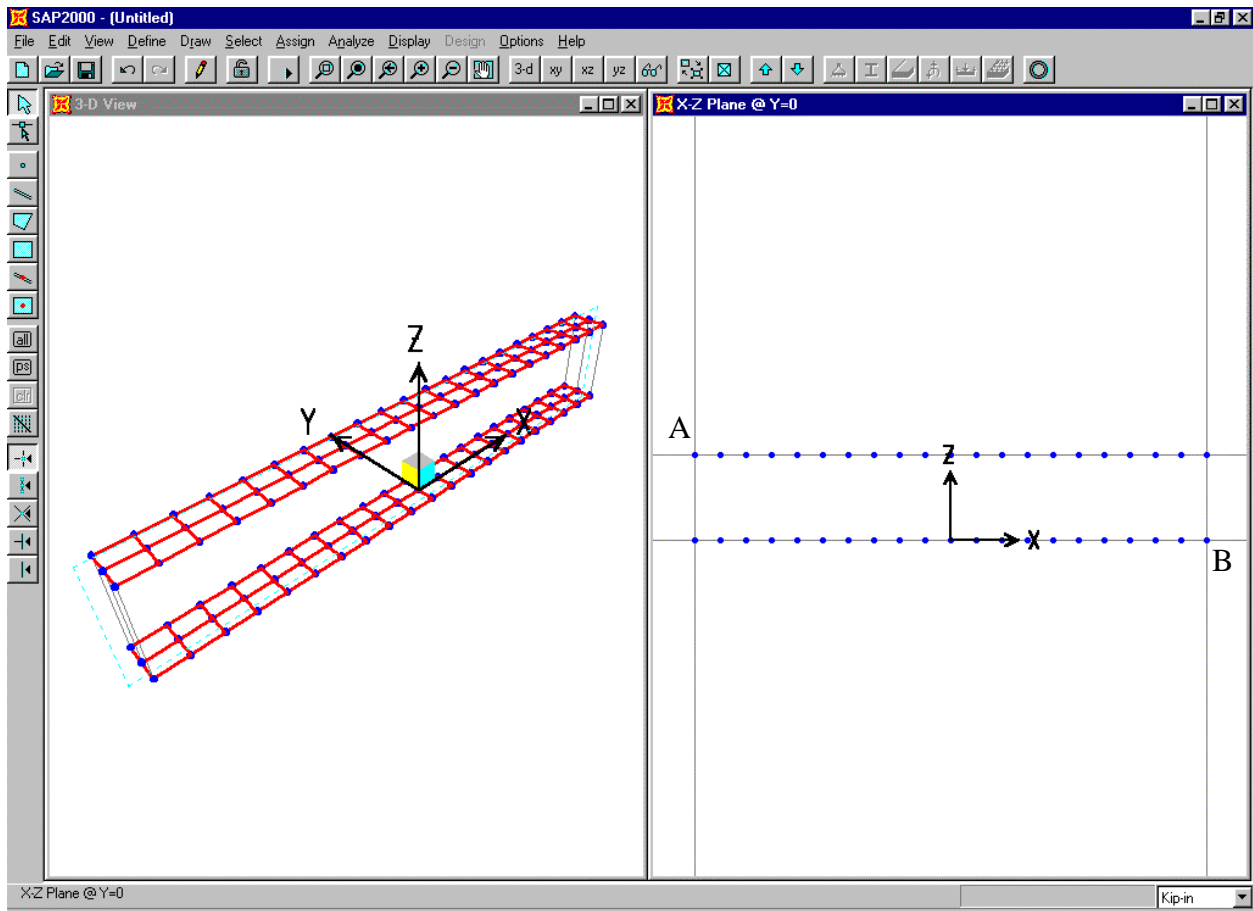


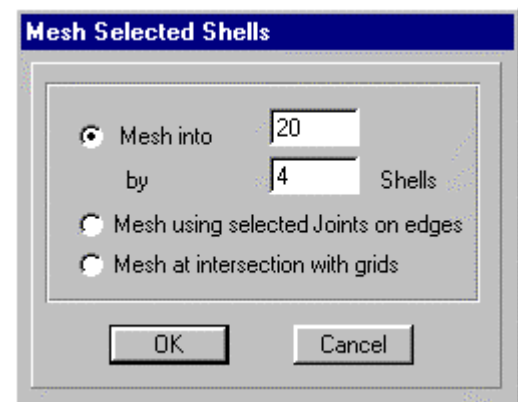









Figure S-2: Screen After Step 17



18. Click the **Draw Rectangular Shell Element** button  on the side toolbar or select **Draw Rectangular Shell Element** from the **Draw** menu.
19. Click on the point labeled “A” and then the point labeled “B” in Figure S-2 to enter a shell element.
20. Click the **Pointer** button  to exit Draw Mode and enter Select Mode.
21. Click on the shell element to select it.
22. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.
23. Fill in this dialog box as shown in the adjacent figure and click the **OK** button.
24. Click in the window titled 3-D View to make sure it is active.











25. Click the **Select All** button  on the side toolbar to select all elements.
 26. From the **Edit** menu select **Change Labels...** to display the Relabel Selected items dialog box.
 27. In this dialog box:
 - Type **1** in the Joint Next Number edit box.
 - Type **1** in the Shell Next Number edit box.
 - Click the **OK** button.
- Note: It is not necessary to change relabel the elements. We are doing it here to assure that later on in the solution when we refer to element labels that everybody has the same labeling system.*
28. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
 29. In this dialog box:
 - Check the Fill Elements box.
 - Click the **OK** button.
 30. Click in the window titled X-Z Plane @ Y=0 to make sure it is active.
 31. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
 32. In this dialog box:
 - Check the Labels box in the Shells area.
 - Check the Fill Elements box.
 - Click the **OK** button.
 33. Select shell elements 65, 66, 67, 68, 85, 86, 87 and 88 by “windowing”.
 34. Press the Delete key on the keyboard to delete these shell elements.
 35. Select shell elements 73, 74, 75, 76, 93, 94, 95 and 96 by “windowing”.
 36. Press the Delete key on the keyboard to delete these shell elements.
 37. Click the **Refresh Window** button  on the main toolbar to refresh the view.




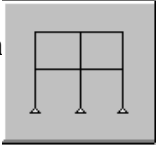

38. Click in the window titled 3-D View to make sure it is active.
39. Click the **Refresh Window** button  on the main toolbar to refresh the view.
40. Click in the window titled X-Z Plane @ Y=0 to make sure it is active.
41. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
42. In this dialog box:
 - Check the Labels box in the Joints area.
 - Uncheck the Labels box in the Shells area.
 - Click the **OK** button.
43. Select joint 22 (lower left-hand corner) by clicking on it.
44. From the **Assign** menu, choose **Joint**, and then **Restraints...** from the submenu. This will display the Joint Restraints dialog box.
45. In this dialog box:
 - Verify that the Translation 1 and Translation 3 boxes are checked.
 - Uncheck the Translation 2 box.
 - Verify that the Rotation About 1, Rotation About 2 and Rotation About 3 boxes are *not* checked.
 - Click the **OK** button.
46. Select joint 42 (lower right-hand corner) by clicking on it.
47. From the **Assign** menu, choose **Joint**, and then **Restraints...** from the submenu. This will display the Joint Restraints dialog box.
48. In this dialog box:
 - Uncheck the Translation 1 box.
 - Verify that the Translation 2 box is *not* checked.
 - Verify that the Translation 3 box is checked.
 - Verify that the Rotation About 1, Rotation About 2 and Rotation About 3 boxes are *not* checked.

- Click the **OK** button.
49. Select joints 148 and 168 (upper left-hand and right-hand corners) by clicking on them.
 50. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
 51. In this dialog box:
 - Type **-9** in the Force Global Z edit box in the Loads area.
 - Click the **OK** button.
 52. Select joints 149 through 167 by “windowing”.
 53. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
 54. In this dialog box:
 - Type **-18** in the Force Global Z edit box in the Loads area.
 - Click the **OK** button.
 55. Click the **Show Undeformed Shape** button  to remove the display of joint force assignments.
 56. From the **Define** menu select **Materials...** to display the Define Materials dialog box.
 57. Highlight the STEEL material and click the **Modify/Show Material** button to display the Material Property Data dialog box.
 58. In this dialog box:
 - Verify that the Modulus of Elasticity is 29000.
 - Verify that Poisson’s ratio is 0.3.
 - Click the **OK** button twice to exit the dialog boxes.
 59. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.
 60. In this dialog box:
 - Click the **Add New Section** button to display the Shell Sections dialog box.
 - In this dialog box:

- Type **WEB** in the Section Name edit box.
 - Select **STEEL** from the Material drop-down box.
 - In the Thickness area type **.75** in both the Membrane and Bending edit boxes.
 - Verify that the Shell option is chosen in the Type area.
 - Click the **OK** button to return to the Define Shell Sections dialog box.
- Click the **Add New Section** button to display the Shell Sections dialog box.
 - In this dialog box:
 - Type **FLANGE** in the Section Name edit box.
 - Select **STEEL** from the Material drop-down box.
 - In the Thickness area type **2** in both the Membrane and Bending edit boxes.
 - Verify that the Shell option is chosen in the Type area.
 - Click the **OK** button twice to exit all dialog boxes.
61. Click the **Select All** button  on the side toolbar to select all elements.
 62. From the **Assign** menu select **Shell** and then **Sections...** to display the Define Shell Sections dialog box.
 63. In this dialog box:
 - Highlight **FLANGE** in the Shell Sections area by clicking on it.
 - Click the **OK** button.
 64. Select all of the elements in the X-Z Plane @ Y=0 window by “windowing”.
 65. From the **Assign** menu select **Shell** and then **Sections...** to display the Define Shell Sections dialog box.
 66. In this dialog box:
 - Highlight **WEB** in the Shell Sections area by clicking on it.
 - Click the **OK** button.
 67. Click the **Show Undeformed Shape** button  to remove the display of shell section assignments.

68. Click in the window titled X-Z Plane @ Y=0 to make sure it is active.
69. From the **View** menu select **Set Limits...** to display the Set Limits dialog box.
70. In this dialog box:
 - In the Set X Axis Limits area type **-12** in the Min edit box.
 - In the Set X Axis Limits area type **0** in the Max edit box.
 - Click the **OK** button.
71. Click the **Perspective Toggle** button  on the main toolbar. A perspective view of the X-Z elevation is displayed.
72. Click the **Rubber Band Zoom** button  on the main toolbar and zoom in on the view by “drawing” a window tightly around it.
73. Select all of the joints on the right side of the section by “windowing”.
74. Click the **Set Intersecting Line Select Mode** button  and select all of the shell elements by “drawing” a line through them. There should now be 9 joints and 8 shells selected. You can verify this by looking at the left-hand side of the status bar at the bottom of the SAP2000 window.
75. From the **Assign** menu select **Group Name...** to display the Assign Group dialog box.
76. In this dialog box:
 - Type **CENTER** in the Groups edit box.
 - Click the **Add New Group Name** button.
 - Click the **OK** button.
77. Click the **Show Undeformed Shape** button  to reset the limits of the display.
78. From the **View** menu select **Refresh View** to rescale the view.
79. From the **View** menu select **Set Limits...** to display the Set Limits dialog box.
80. In this dialog box:
 - In the Set X Axis Limits area type **-60** in the Min edit box.
 - In the Set X Axis Limits area type **-48** in the Max edit box.
 - Click the **OK** button.

81. Click the **Rubber Band Zoom** button  on the main toolbar and zoom in on the view by “drawing” a window tightly around it.
82. Select all of the joints on the right side of the section by “windowing”.
83. Click the **Set Intersecting Line Select Mode** button  and select all of the shell elements by “drawing” a line through them. There should now be 8 joints and 6 shells selected. You can verify this by looking at the left-hand side of the status bar at the bottom of the SAP2000 window.
84. From the **Assign** menu select **Group Name...** to display the Assign Group dialog box.
85. In this dialog box:
 - Type **LEFT** in the Groups edit box.
 - Click the **Add New Group Name** button.
 - Click the **OK** button.
86. Select all of the joints on the right side of the top half of the section by “windowing”.
87. Click the **Set Intersecting Line Select Mode** button  and select all of the shell elements in the top half by “drawing” a line through them. There should now be 4 joints and 3 shells selected. You can verify this by looking at the left-hand side of the status bar at the bottom of the SAP2000 window.
88. From the **Assign** menu select **Group Name...** to display the Assign Group dialog box.
89. In this dialog box:
 - Type **LEFTTOP** in the Groups edit box.
 - Click the **Add New Group Name** button.
 - Click the **OK** button.
90. Select all of the joints on the right side of the bottom half of the section by “windowing”.
91. Click the **Set Intersecting Line Select Mode** button  and select all of the shell elements in the bottom half by “drawing” a line through them. There should now be 4 joints and 3 shells selected. You can verify this by looking at the left-hand side of the status bar at the bottom of the SAP2000 window.
92. From the **Assign** menu select **Group Name...** to display the Assign Group dialog box.
93. In this dialog box:
 - Type **LEFTBOT** in the Groups edit box.

- Click the **Add New Group Name** button.
 - Click the **OK** button.
94. Click the **Show Undeformed Shape** button  to reset the limits of the display.
95. From the **View** menu select **Refresh View** to rescale the view.
96. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
97. In this dialog box:
- Uncheck the Labels box in the Joints area.
 - Click the **OK** button.
98. Click on the **xz 2D View** button  on the main toolbar to return from the perspective view to a 2-D view. Note the title of the window is X-Z Plane @ Y=0.
99. Click the “X” in the upper right-hand corner of the 3-D View window to close it.
100. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.
- In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.
 - Click the **OK** button.
101. Click the **Run Analysis** button  to run the analysis.
102. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.
103. From the **Display** menu select **Show Group Joint Force Sums...** to display the Select Groups dialog box.
104. In this dialog box:
- Click on CENTER to highlight it.
 - Holding down the Shift key on the keyboard click on LEFTTOP to highlight it. All of the groups except the default ALL should now be highlighted.
 - Click the **OK** button to display the Group Joint Force Summation window and view the group joint force sums. When finished viewing the group joint force sums, click the “X” in the top right-hand corner of the Group Joint Force Summation window to close it.

105. From the **Display** menu select **Show Element Forces/Stresses** and then **Shells...** from the submenu to display the Element Force/Stress Contours For Shells dialog box.

106. In this dialog box:

- Click the Stresses option button to select it.
- In the Component area click the S12 option button to select it.
- In the Stress Averaging area verify that the At All Joints option is selected.
- Check the Display on Deformed Shape check box.
- Click the **OK** button to display the shell stresses. The screen should look similar to that in Figure S-3.

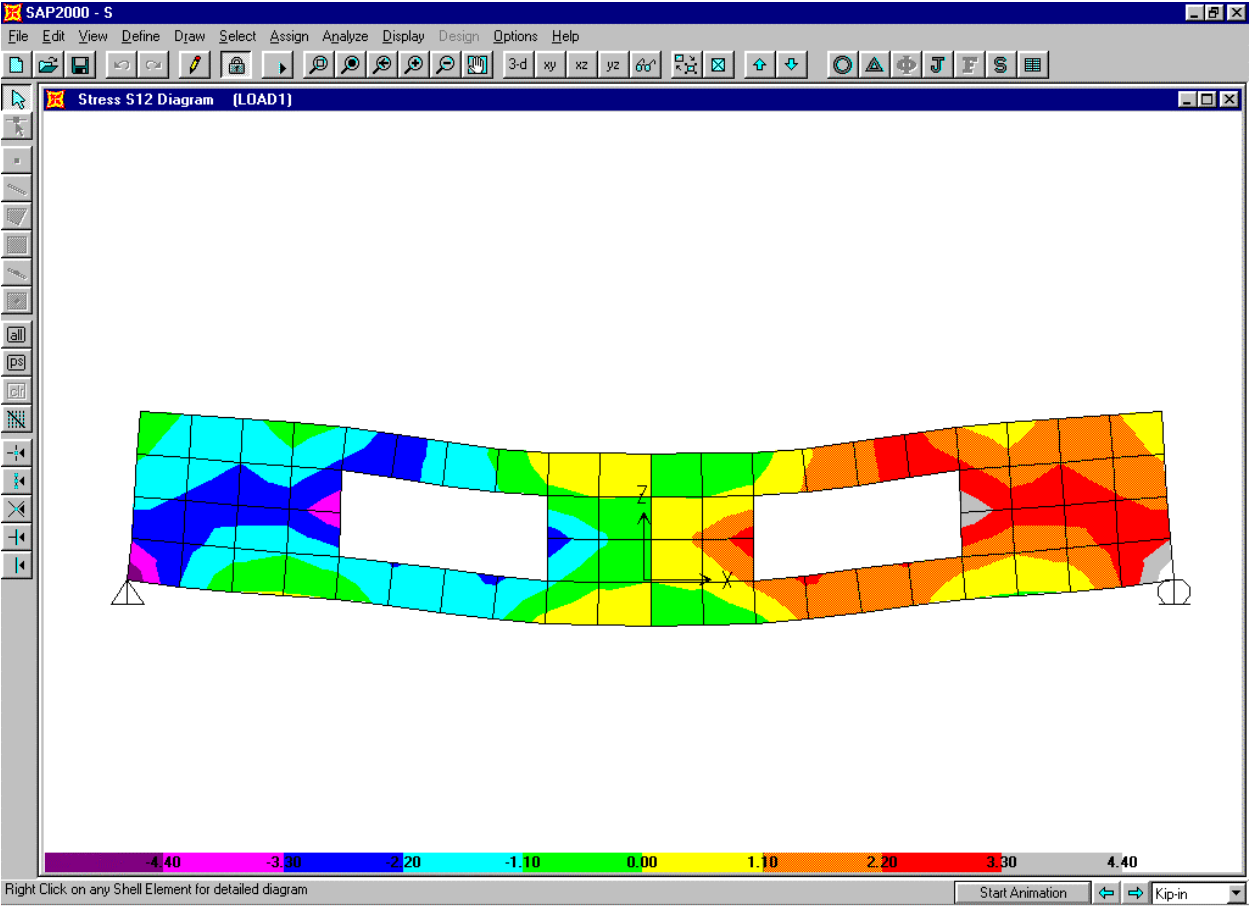


Figure S-3: Shell Stresses