1) Upon starting, Dr. Frame displays an existing model
   Remove this model.
   EDIT -> CLEAR ALL

2) Set units (in this case, U.S. engineering system – notice different units)
   OPTIONS -> UNITS..-> U.S. -> OK
3) Set up grid parameters.
   OPTIONS -> GRID...-> GRID PARAMETERS
   Change the grid extent to
   Horizontal Divisions 20
   Vertical Divisions 10

4) Enlarge grid display
   Select the lens icon
   Mark a rectangle around the area you wish to enlarge

   Note: To reduce scale, hold shift key while selecting the lens icon and
   click on the window.
5) Create truss geometry
   Select the bar element creation icon (on the left, with joints on both ends)
   Click on grid point to position first joint
   Click on second grid point to position second joint
   Repeat for each new bar

6) Apply constraints
   Select the appropriate constraint icon (on the left, either hinge or roller)
   Click on joint to apply
7) Remove auxiliary diagram and show dimensions and joint labels

OPTIONS -> AUX DIAGRAM -> NONE
OPTIONS -> MEMBER DISPLAY -> SHOW LENGTHS
OPTIONS -> JOINT & SUPPORT DISPLAY -> SHOW JOINT LABELS

8) Member properties
Select entire truss (or individual element)
MODELING -> MEMBER PROPERTIES
Area = 64in$^2$; $E = 2 \times 10^6$ psi ($ksi = 10^3$ psi); all other properties should be left unchanged, since they will not affect the solution
9) Apply load
Select the load icon (P with arrow), and then click on the joint where load needs to be applied.

10) Adjust the magnitude of the load
Select (click) on the red load vector you have drawn.
Enter the exact x y load components in the load info window.
11) Display the computed deformed shape (or remove the deformed shape)
MODELING -> SHOW DISPLACED SHAPE
To increase the displacement, click over the 4th icon from the bottom, on the right column

12) Display the computed element forces and reactions
Remove the deformed shape
MODELING -> SHOW DISPLACED SHAPE
Select the entire truss
OPTIONS -> MEMBER DISPLAY -> SHOW AXIAL FORCE VALUE
OPTIONS -> JOINT & SUPPORT DISPLAY -> SHOW SUPPORT REACTIONS
13) Display tension and compression in color (compression in red)  
Select the entire truss  
OPTIONS -> MEMBER DISPLAY -> TENSION AND COMPRESSION COLORING

14) Display x y displacement values at joint  
De-select the entire truss (click anywhere outside of the truss)  
Select a joint (for example, joint B) by double clicking on joint  
Joint Info panel shows joint co-ordinates and computed displacement values.
15) Model dead load only (due to weight of bars)
   Compute weight of each bar ($\text{Area} = 64\text{in}^2$, specific weight (douglas fir) = 0.017lb/in$^3$)
   Compute force acting at each joint due to all bars connected with joint
   Apply force to joint (note units!)

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>D-B</td>
<td>64 x 0.017</td>
<td>96</td>
<td>104.4</td>
<td>52.2</td>
</tr>
<tr>
<td>2</td>
<td>B-C</td>
<td>64 x 0.017</td>
<td>135.6</td>
<td>147.5</td>
<td>73.8</td>
</tr>
<tr>
<td>3</td>
<td>C-A</td>
<td>64 x 0.017</td>
<td>192</td>
<td>208.9</td>
<td>104.5</td>
</tr>
<tr>
<td>4</td>
<td>A-B</td>
<td>64 x 0.017</td>
<td>135.6</td>
<td>147.5</td>
<td>73.8</td>
</tr>
<tr>
<td></td>
<td>TOTAL WEIGHT (lb)</td>
<td></td>
<td></td>
<td>608.3</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Contributions from elements [lb]</th>
<th>Total weight acting at joint [lb]</th>
</tr>
</thead>
<tbody>
<tr>
<td>D</td>
<td>52.2</td>
<td>52.2</td>
</tr>
<tr>
<td>B</td>
<td>52.2 + 73.8 + 73.8</td>
<td>199.8</td>
</tr>
<tr>
<td>C</td>
<td>73.8 + 104.5</td>
<td>178.3</td>
</tr>
<tr>
<td>A</td>
<td>73.8 + 104.5</td>
<td>178.3</td>
</tr>
<tr>
<td>TOTAL WEIGHT (lb)</td>
<td></td>
<td>608.6</td>
</tr>
</tbody>
</table>