Lesson 19 – Structural Analysis with RISA-2D Educational

OBJECTIVES

1. Model a truss and frame using RISA.
2. Find axial forces, shear and moment diagrams, and displacements.

ANNOUNCEMENTS

NOTES

1. What is a truss?

2. What types of forces can be transmitted by a truss?

3. What is a frame?

4. What types of forces can be transmitted by a frame?
5. RISA-2D is a structural analysis program that can be used to analyze trusses and frames. There are many other structural analysis programs used by engineers—we will be using the educational version of RISA (pronounced rē’sa) in CEE 2120. Make sure to start up the educational version.*

6. Before you begin to set up a structure (truss or frame) in RISA, it is usually convenient to modify the drawing grid. Click the button shown above to modify the grid. Enter the grid increments. For example: To specify 8 grid increments of 4 feet each, enter “8@4”.

7. Once the grid is set, you can add members. After clicking the button, the program will ask for member properties. Maker sure that the “I End” and “J End” are “Pinned” when setting up a truss. For a frame, select “Fixed.”

8. Draw in the members. Make sure not to skip over any nodes. Hit “Esc” to stop drawing members and continue elsewhere. Hit “Esc” twice to stop drawing members or to change the member properties (by selecting the “add members” button again).

* Besides the educational version, UWP also has the full version of RISA-2D. This is a much more powerful (and complicated) that program you will be using in later courses.
9. **Add the boundary conditions.** You can click one of the standard reaction types shown below, or specify boundary conditions by selecting the appropriate “X Translation,” “Y Translation,” or “Rotation” condition.

![Boundary Conditions](image)

Click the “Apply” button and select the nodes that have the specified boundary conditions. You can also hold down the mouse button and draw a box around the desired nodes. Hit “Esc” to stop adding boundary conditions.

10. ![Point Load](image) Now we can **add point loads**. Specify the direction and magnitude. Click “Apply” and select the nodes where the point load will be applied. Hit “Esc” to stop adding point loads.

11. ![Distributed Load](image) Next, add any **distributed loads** just like adding point loads.* If you specify different start and end magnitudes, RISA will connect the two in a straight line.

12. With the members defined, the boundary conditions set, and the loads applied we are now ready to **run the analysis**. Click “Solve” on the menu or the solve button to run the analysis. The results of the analysis will immediately pop up. These results will be discussed in the next section.

**VIEWING RESULTS**

13. Once the analysis has been run, a structural engineer will want to know the reactions, forces in each member, deflections, etc. These results can be viewed using the “Results” panel shown at right. If you don’t see the Results Panel, click the Results Panel button:

![Results Panel](image)

If the Results Panel does not appear after clicking on the Results Panel button, it usually means you have not run the analysis or there was an error in solving the model.

---

* Would you add distributed loads to a truss?
14. Clicking on “Member Forces” in the Results Panel will show axial forces, shear forces and moments in each member:

```
<table>
<thead>
<tr>
<th>Member Label</th>
<th>Section</th>
<th>Axial[k]</th>
<th>Shear[k]</th>
<th>Moment[k-ft]</th>
</tr>
</thead>
<tbody>
<tr>
<td>M1</td>
<td>1</td>
<td>-7.989</td>
<td>10.008</td>
<td>-92.182</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>-7.989</td>
<td>10.008</td>
<td>-52.148</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>-7.989</td>
<td>10.008</td>
<td>-12.114</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>-7.989</td>
<td>10.008</td>
<td>27.92</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>-7.989</td>
<td>10.008</td>
<td>67.953</td>
</tr>
<tr>
<td>M2</td>
<td>1</td>
<td>9.992</td>
<td>-7.989</td>
<td>67.953</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>9.992</td>
<td>-7.909</td>
<td>33.999</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>9.992</td>
<td>-7.989</td>
<td>0.044</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>9.992</td>
<td>-7.989</td>
<td>-33.91</td>
</tr>
</tbody>
</table>
```

The values are given at 5 equidistant points along the member, the ends, quarter points, and midpoint. The number of points shown can be set by changing the “Number of Sections” in the global parameters (click “Global” on the menu to set global parameters).

15. **Labeling the members** is helpful when interpreting the results.

   ⇒ View ⇒ Member Labels  (Or click the Member Labels Button:  

   There are also options to view node numbers, member properties (area, modulus of elasticity, moment of inertia), boundary conditions, etc.

16. The **deflected shape** of the member can be shown by clicking the deflected shape button.

17. **Axial Load, Shear and Moment Diagrams** can be displayed directly on the model by selecting the appropriate button (A for axial, V for shear, M for moment).

18. When you select the Details button your cursor changes and you can select a member to view the axial (A), shear (V), moment (M), and deflection (D) plotted. Click any of the plots to get a more detailed view or to print a plot for the individual member.

---

* What should the moment and shear be for a truss?
Printing in RISA-2D

19. When you select ⇒ File ⇒ Print... in RISA the default is to print the view of the model displayed on the screen. To print the model graphically, you can set the desired options* and then select “Continue…” to go to the standard windows printer dialog.

20. To print a text report, select the “Print a Report Instead…” button. Select the report sections you wish to print.

* The plot will generally be more readable if you increase the scale factors from the defaults of 0.5.
Common mistakes when using RISA.