CE 342

RISA-2D

Background

- RISA-2D is a structural analysis program, which can model:
  - Beams, frames, trusses and plates.
  - Any linear elastic structural material.
  - Typical supports, such as pins, rollers and fixed supports.
  - A wide variety of loads:
    - Static loads such as dead, live, and snow.
    - Dynamic loads such as earthquakes.
    - Thermal and settlement-induced loads.

Why RISA-2D?

- There are many structural analysis packages available.
- RISA-2D has an excellent user interface;
  - It's much easier to learn and to use than many of its competitors.
- It is intended to model 2D structures.
  - We only use 2D models in this course.
  - 3D modeling is significantly more complicated (take CE 445)

Availability

- The professional version of RISA-2D is available in BEL 117.
- A demonstration version is available for free download from:
  - http://www.risatech.com/demo_request.asp
  - You can use this version for homework; even though the graphics have "demonstration version" written across the page.

Tutorials

- Tutorials for RISA-2D are also available for download from Risa Technologies (at the same URL).
- The tutorials will complement the information given in this demonstration.
  - There are many ways of doing things in RISA.

Example Frame Problem

- The easiest way to explain how to use RISA-2D is to work and example problem.
- The example problem will be a simple frame:
  - Kassimali, homework problem 13.23.

Beginning

- When you begin a model in RISA-2D, the first dialog box asks if you want to begin drawing members in the model.
  - For now, I will skip this option in order to clarify some of the details in the process.

Global Parameters

- If the program doesn't begin by asking for global parameters, click on the **Global** menu item.
  - Specify **Model Title,** and **"Designer"**.
  - Ignore the **Codes, Concrete**, and **Footing** tabs.

Solutions Tab

- The default number of sections (5) will provide reasonably smooth shear and moment diagrams; more sections provide smoother plots.
- I usually uncheck the shear deformations.
  - Risa will match our answers more closely without them, since our hand calculations don't include shear deformations.
- Ignore **P-delta Tolerance**, **Gravity Accel-eration, Internal Sections**, and **Eigensolution**.

Units

- Select the **Units** menu item to specify units for the model.
  - For this problem we will use "Standard Metric" for SI units.
  - Use "Standard Imperial" for U.S. units.

Data Entry

- If the **Data Entry** menu is not exposed, click on the "D" icon.
- The model description can be entered in the order listed on the **Data Entry** menu.
  - Help for any data entry screen can be had by typing F1 when the screen is active.
  - F4 will delete any row from a data entry screen.

General Materials

- The **Materials** menu item allows you to select the type of material.
  - Tabs refer to various material types.
    - Hot-rolled and cold-formed are types of steel.
  - In the General tab, we can specify the material type by providing its Label, Young's Modulus E, Shear Modulus G or Poisson's Ratio ν, Coefficient of Thermal Expansion α, and Density.

Section Sets

- The **Section Sets** menu allows us to create a list or catalog of shapes for the model.
  - Tabs refer to the same material sets as before.
    - Since our material was “General” our Section Set will also be “General.”
    - Properties for standard steel shapes are included in the Hot Rolled tab.
  - Our problem doesn’t specify numbers, so we assume realistic values for *A* and *I*.
    - We *must* provide both.
    - Note that *I* for the columns is twice *I* for the beams.

University of Idaho 13

Section Sets (cont.)

- Under the General tab, we can specify “BEAM” and “COLUMN” for the section labels.
- **Shape** allows you to calculate the cross-section properties from cross section dimensions.
- **Type** is used in design. We will ignore it.
- **Material** is selected from the pull-down list of materials, which should include the material we defined earlier.

University of Idaho 14

Section (concl.)

- **A** is the cross-section area (watch the units.)
- **I(90,270)** is the moment of inertia of the section about an axis in the plane of the page. This is ignored in analysis.
- **I(0,180)** is the moment of inertia about an axis perpendicular to the page.
  - If the problem doesn’t specify a weak or strong axis moment of inertia, you can enter the same value in both.

University of Idaho 15

Design Rules

- Since we’re not designing, we will skip this menu item.
- We will also skip
  - **Plates**, and
  - **Moving Loads**

University of Idaho 16

Joint Coordinates

- We could specify joint coordinates in a spreadsheet-style list, but graphical entry can be easier.
- In the Model View window, the icon representing a grid with a pencil allows us to modify the drawing grid.
  - We will specify two spaces at 4 m for the *x*-axis and 1 space at 10 m for the *y*-axis.

Members

- The icon representing a pencil and a line with endpoints allows us to draw members.
  - **The program will automatically add joints or nodes at the end of each member.**
- Under the Draw Members tab:
  - Select General from the Material Type and Shape.
  - Select “Assign a Section Set” and choose “Beam” from the pull-down menu.
  - Use “Fully Fixed at Both Ends” for frame members.

Members (cont.)

- Under the Draw Tab (cont.):
  - Select “Both Ways” from the pull-down list
    - Tension-only for cables and slender members
    - Compression only for foundations members.
  - Click “OK” and draw the beam members on the grid.
- Reselect the Draw Members icon, select “Column” from the Assign Section Set, etc. and draw the columns

Local Axis

- The program automatically creates a local axis for each member you draw.
  - The local *x*-axis goes from the first node to the second node you select as you draw the member.
  - The local *y*-axis is perpendicular to the *x*-axis and obeys the right-hand rule with the *z*-axis coming out of the page.
- The local axes are useful for specifying loads and reactions.

Boundary Conditions

- The green triangle icon on the Model View window allows us to specify boundary conditions.
- The boundary conditions provide fixed, free, pinned or roller connections.
  - Reactions will be calculated at all supports, unless otherwise specified.
- If you don't specify enough boundary conditions to prevent rigid body motions, the program will generate errors.

Loads

- Loads are specified in a two-step process.
  - The loads are first assigned to a Basic Load Case,
  - The Basic Load Cases are combined in the Load Combination screen to create cases for analysis.

Basic Load Cases

- Select **Basic Load Cases** from the **Data Entry** menu.
- Enter appropriate names for the various load components.
- Member self-weights can be included in either the *x* or *y*-directions.

Basic Load Cases (cont.)

- Loads can be graphically assigned to a **Basic Load Case** using one of the three icons to the right of the green triangle:
  - Joint loads.
  - Distributed loads.
  - Point loads.

Joint Loads

- Joint loads can be
  - Forces – in global coordinate directions.
  - Displacements – also in global coordinate directions
  - Mass – in global coordinate directions (used for dynamic analyses).
- Joint loads should also be assigned to one of the Basic Load Cases.

UniversityofIdaho 25

Distributed Loads

- Distributed loads can be:
  - Forces in global (X & Y) directions
  - Forces in local (x & y) coordinates
  - Forces projected on lateral and vertical (L & V axes).
  - A temperature load, T.
- Again, the distributed loads should be assigned to one of the Basic Load Cases.

UniversityofIdaho 26

Load Combinations

- **Load Combinations** can be selected from the **Data Entry** menu.
  - An appropriate name can be assigned to the load combination.
  - Loads can be included in an envelope solution by checking **Solve**.
  - **P/Delta** includes the load in a **P/Delta** analysis.

UniversityofIdaho 27

Load Combinations (cont.)

- **SRSS** is for seismic analysis.
- **Basic Load Cases** (BLC) are included by listing:
  - The number of each Basic Load Case you wish to include and
  - The factor by which the load case is scaled in the analysis.
- All the basic load cases included in the load combination must be listed and scaled.

UniversityofIdaho 28

Solve

- You can solve for displacements, reactions, and member forces by clicking on the equal sign icon and selecting a load combination to analyze.
  - If rigid-body motions are allowed in your system – i.e., if the system is unstable - the program will try to fix the problem for you by locking displacements or rotations.

University of Idaho 29

Output

- RISA-2D generates tables of output for
  - Joint Reaction forces and moments,
  - Joint Deflections, and
  - Member Forces.
- These results are viewed by clicking on the **Results** menu or icon and selecting the desired output table.

University of Idaho 30

Graphical Output

- A variety of plots can be generated by clicking on the **Plot Options** icon or by pressing F2.
- The option "tabs" are:
  - Joints
  - Members
  - Plates
  - Loads
  - Deflection Diagrams
  - Miscellaneous

University of Idaho 31

Joints

- The **Joints** tab allows you to:
  - Show/not-show joint labels, and
  - Show/not-show boundary conditions (support conditions).
  - Show/not-show reactions.

University of Idaho 32

Members

- The **Members** tab allows you to:
  - Select the way members are rendered and labeled.
  - Draw shear, moment, and axial force diagrams.
    - And scale and label the diagrams.
- The sign convention for moment diagrams is opposite that used in class.
  - It uses the sign convention in the AISC *Steel Construction Manual*.

University of Idaho 33

Loads

- The **Loads** tab allows you to:
  - Display/not-display loads
  - Select which basic load case or load combination to display.
- We won't use **Plates** in this class.

University of Idaho 34

Deflection Diagrams

- The **Deflection Diagrams** tab allow you to:
  - Display deflection diagrams
  - Select which load case generates the displacement diagram.
  - Scale the displacements.
  - Display the undeflected shadow.
  - Animate the deflection diagram.

University of Idaho 35

Trusses

- Trusses are two-force members.
- RISA can create truss members when drawing members.
  - After clicking on the **Draw New Members** icon, the **Properties of New Members** dialog box is displayed.
  - Under **Release Codes**, select either:
    - Pinned at both ends, or
    - Select the **Advanced Options** and release the *i* or *j* end.

University of Idaho 36

Joint Instabilities

- If *all* the members connected to a joint have pinned ends, the rotation at the joint is unrestrained, and a rigid body motion is possible (rotation of the joint).
  - During the solution phase, RISA-2D detects this as an instability and will offer to lock the joint.

Fixing Joint Instabilities

- If this problem occurs you can either:
  - Add a boundary condition at the joint.
    - Select **Boundary Conditions** from the **Data Entry** menu and FIX the rotation at that joint.
    - RISA won’t calculate the fictitious reaction for FIXED boundary conditions.
  - Or you could have one of the members connected to the joint *not* have a pinned end at the joint.

Summary

- RISA-2D analyzes a variety of structures and loads.
- The process requires that we define the structure geometry, material, and loads.
- RISA-2D allows us to do this in a variety of ways, including:
  - Entering data in spreadsheet-style lists.
  - Entering information graphically.