

Visual Analysis Instructions:

Getting Started

File -> New Project

Select -> My structure lies in a 2-D plane

Select -> Frames, beams, shear walls, tanks, etc. (as opposed to trusses)

Select -> Don't generate any load cases

Enter Title and Name

Use Default Units (kips, feet, degrees, fahrenheit)

Finish -> This sets up a Model View window in which you define your structure

What does VA need to create your structure?

For trusses or beams/frames, the following fundamental information is needed for a 2-D structure.

- Nodal Location: define nodes in cartesian coordinates
- Nodal Connectivity: how are the nodes connected?
- Nodal Boundaries: which nodes are free to move, which are constrained? (roller, pin, wall)
- Element Definition: what kind of elements are connecting nodes? (truss, beam?)
- Element Properties: Material properties of the elements (A, E, I)
- Load location (at node, along beam)
- Load type (distributed, concentrated)
- Load value and direction (as X- Y- components)

VA needs additional information if structure is in 3-D

- Beams require two moments of inertia (weak axis bending and strong axis bending)
- Beam orientation (strong plane of bending for beams)
- Additional Nodal boundaries (x- y- and z- directions to account for)

How to define structure: There are two modes

option 1) Use when nodes are not spaced in convenient round numbers apart (in feet)

Define one node at a time by clicking the Create Node icon

define each element by connecting defined nodes

define loads

analyze by clicking lightning bolt

option 2) Use when the geometry is simple and distances between nodes are in even feet (or meters)

First be sure the grid is consistent with your chosen units. For example, if you gave feet as the default unit when you set up the problem, double click the coordinates in the bottom right corner of VA and be sure the 'spacings:' for X and Y say '1 ft'. If not correct, type it in.

Define elements by point and click after clicking the Draw Members icon

(coordinates are given in bottom right hand corner to help out)

nodes are automatically defined at the ends of each element

After drawing the FIRST element, double click and establish its properties (I, E, A)

These will be carried over to all the others automatically.

double click nodes to establish boundary conditions

apply loads

analyze by clicking lightning bolt

Hint: Use View -> Zoom -> Zoom Full model to get a close up look

Creating Load Cases:

VA let's you create different sets of loads to be applied separately, called service cases
Any one service case can have many different loads

Click the Load Icon

General Tab : leave it

Source tab : You can choose to include structural weight here, leave it out for now

Loads tab: defines loads one at a time

Select load type (nodal or member - trusses are nodal only)

New -> Placement tab - select node at which point load is applied

Load tab -> pick type, direction and magnitude

Okay

back to service case window, pick 'New' to create next load

repeat until all desired loads in that service case are defined

How to Analyze:

After drawing the structure and applying loads, we can get some answers

Click the lightning bolt to get the analysis wizard.

Analysis Type: Static

Performance: Minimize bandwidth

Next

Static Analysis Type: First order

Next

Load cases to analyze: there is a Service Case per load applied

highlight the ones you want. If you want more than one, hold the 'ctrl' key down to pick more than one

Finish

View Results:

View -> Report Wizard

select static results

OR

double click any node or member in the deformed shape views
to get a quick report of results at that location.

Changing the visual display of results:

By default, VA will display a displaced view of the structure. You can alter what else is reported in this figure (add reactions, moment diagrams, etc.) by going to

View -> View Manager

here you can turn on or off several features, such as nodal labeling, shear, moment diagrams, legend, etc.

Example:

An excellent way to practice is to go back to HW#1 and #2 and solve these problems using VA. You already have the answers, so you will know if you're on the right track.

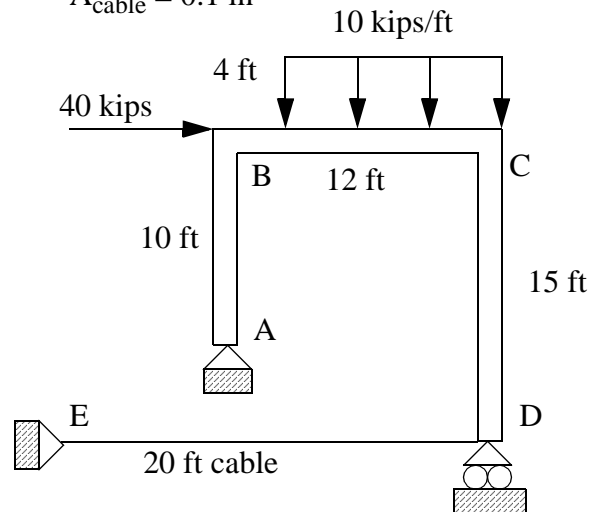
For this example, we'll start with a new problem. The three member frame is restrained at A with a pin and at D with a roller. D is additionally restrained with a 20 ft cable made of the same material as the frame. Find the horizontal deflection at D, the force in the cable, and the moment diagrams for the frame members.

$$E_{\text{all}} = 29000 \text{ ksi}$$

$$I_{\text{beams}} = 500 \text{ in}^4$$

$$A_{\text{beams}} = 2 \text{ in}^2$$

$$A_{\text{cable}} = 0.1 \text{ in}^2$$



1) Set up a new VA window for a 2-D frame by following the directions on page one under the **Getting Started** heading.

2) We will create the nodes and members by drawing directly on the window. This option 2 under **how to define structure** on page one.

- Draw member AB first and see that it creates nodes at each end as well as a member. Double click that member to get an edit menu. Under the 'shape' tab select 'user defined' for 'Category'. Note though that you have the option of selecting from many pre-stored standard shapes if you select a category other than 'user defined'. Select the I,A properties tab. Change the Area, A_x and the Inertia, I_z values to those in the example problem. Now click the Material tab and select 'Steel'. Note that the E value is set automatically to 29000 ksi. Note that you could also select 'User Defined' if you want to customize. Now click 'okay' to get back to the model. From this point on, all new members drawn will carry these same assigned properties.

- Draw members BC, CD and DE with the mouse.

- Member DE has different properties than the beams. It is a cable, and has a different area, so double click member DE and change the area. The cable won't support a moment where it connects at D, so we need to add a hinge at this location to represent the cable. Again double click the member DE and select the 'Connections' tab. Node D should be node N4, so you need to 'Release the strong moment M_z ' and click OK. You should see a small green circle appear at that node signifying a hinge.

3) Boundary conditions are now needed at A, D and E. Double click node 1 (A) and click the 'Support' tab. Click 'support type' and select 'pinned from the pull down menu. Take note of your other options. Now do the same at D and E, selecting the appropriate support at each.

4) Apply loading to the structure: Look along the lower tool bar at the top of VA for the 'Loads' button and click it. Leave the General tab window set to Service Case 1. Click the 'Loads' tab and notice that the 'type' window offers nodal loads and member loads.

- We will first apply the 40 kip point load at B (node 2), so select nodal load and click 'New'. You will get a new window on top of the current window that lists the nodes defined in your structure under the 'Placement' tab. Select node 2, and click the 'Load' tab. Select 'Force X' for type and direction, and 40 kips for the magnitude (note that if the force were to the left we would enter -40 kips). Click OK and the 'New Service Case: Service Case 1' window will re-appear.

- This time select 'member loads' as the type and click new. Again a new window comes up over the previous, with a list of members in your structure. Select member 2, then click the 'type' tab. Select 'Uniformly distributed', 'Global Direction', and 'Force Y for direction. Click the 'Magnitude' tab and enter -10 Kips/ft for the downward load. Unselect the 'full length member' box, and enter 4 and 12 as the start and stop. Click ok, then ok again to finish the load case.

You now have a model with nodes, boundary conditions, elements, and loads. its time to get results.

5) Run the analysis: click the lightning bolt in the top tool bar. Select static, minimize bandwidth, next. Select First order, next. Select Service case 1, finish. The analysis will run, (click okay at the 1% balance warning), and a picture with the deformed shape will show up. The displacements have all been scaled to visualize the motions.

Viewing results: Try the following:

- double click node 4 (D) to see the reactions and displacement at that node
- double the top horizontal member to see a table of member internal forces at evenly spaced increments along the beam.
- bring the deformed shape window back up, unselect the member, and click view -> view manager. Note the various things you can check and uncheck. Play with a few, clicking 'apply' after each. Try selecting 'members -> forces -> z moment' and apply. Back in the view manager, scroll down and unselect 'window -> displaced shape' to get a cleaner view of the moment diagram. Go to edit -> copy and then paste the figure into a Word document.
- select 'view -> quick report' for a summary of the results in one table. Sections of this could be copied and pasted into a Word document along with moment and shear diagrams to produce a homework write-up...