

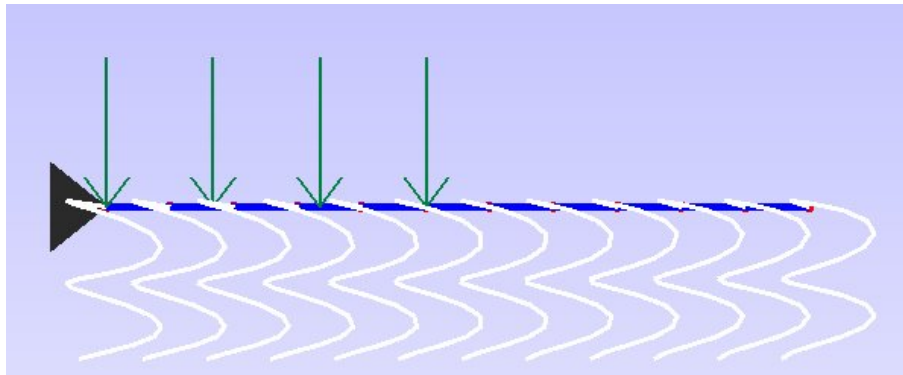
Elastic Foundation

Project Description:

This example shows how you might model an elastic foundation or a "beam on grade." Foundations can be modeled using spring supports to represent a soil's subgrade coefficient. Springs may be added to VA models that provide resistance in compression, tension, or both. This tutorial will explore the differences of modeling a foundation as springs providing "compression only" support and a conventional "two-way spring".

FEATURES:

Split Member
Generate Spring Support
Generate Copies



Setting Up the Project

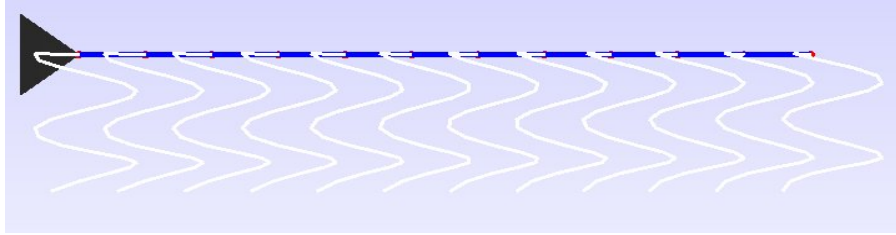
When you start up VisualAnalysis, you are presented with the VisualAnalysis Startup Wizard. The wizard presents you with a number of options. We want to start fresh with a new project, so **choose** the "Start a New project" radio button and **click** Start!. **Select** this structure type as Plane Frame (**Modify Tab**), set the unit style to Kips & Feet (**Edit | project Units**), and name the project "Elastic Foundation" (**Modify Tab**).

Basic Level Users: The Basic Level of VA does not support compression-only spring elements. This example will suggest an iterative method that could be used if necessary so you may wish to complete this tutorial.

Modeling the Structure

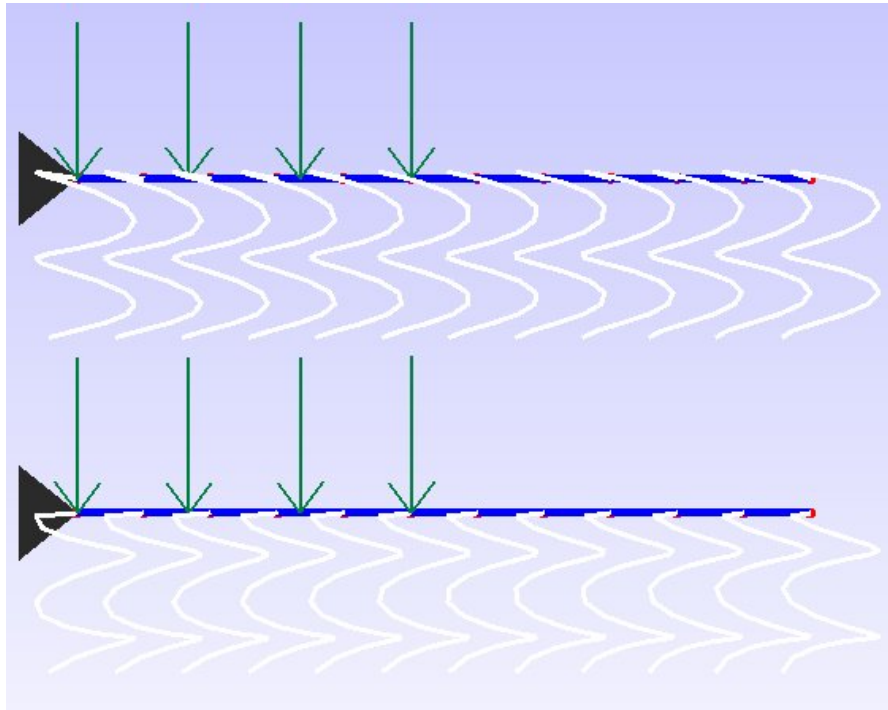
Draw an 11 ft beam and specify it as a rectangular standard shape 12 inches wide and 18 inches deep. **Choose** a material of 4 ksi concrete. Use the **Model | Split Member** command to split the beam into 11 equal sections. The more sections you use, the more accurate the results of the model will be. The trade off is a more complex model, more results, and potentially slower analysis time. To view the nodes of the split member, turn off the high quality drawing in the Filter Tab.

Now we will set the boundary conditions of the model. **Fix** the left most node of the model in the X Direction. This ensures the model will not have infinite displacement off to one side. For the subgrade conditions **select** all the nodes, **right-click**, and **choose** Generate Spring Support. Specify the spring stiffness as 150 K/ft, acting in the -Y direction, linear, and elastic.



Loading the Structure

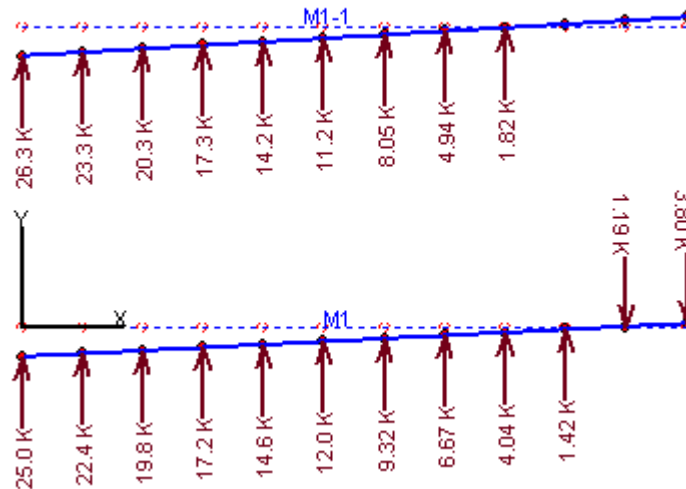
Apply a -25 K/ft load to the first five feet of the member, starting from the left. Then use the **Model | Generate Copies** command to create a copy of the entire model and loads. Change the springs on the second model to compression-only.



Top model has compression-only spring supports, bottom model has normal two-way.

Analysis and Results

Now that we have our two models created and loaded, we should check them for errors before setting the analysis options and actually analyzing the structure. Select **Analyze | Check Model for Errors** to make sure there are no problems. Now select **Analyze | Analyze Now** or *click* the lightning bolt button on the toolbar.



Top model is compression-only springs; the bottom model is two-way springs.

The conventional two-way spring model would have to be altered by deleting the springs in tension and re-analyzing. This process would then be repeated until you only have results with springs in compression. This is the procedure that may be followed by those using the Basic Level of VisualAnalysis. VisualAnalysis Standard and Advanced automate the iterative process with their compression only spring models.