# *Virtual* Work

Dynamic Analysis Primer

VisualAnalysis 5.5 Mode Shapes and Response Spectra

VisualAnalysis offers sophisticated dynamic analysis capabilities. Often times, engineers make it through school without ever seeing or using this kind of structural analysis, and so when the building code requires it, or when you think it might give you a competitive advantage on a certain job, you must be "self taught". While the theory may be complicated beyond a *VirtualW*ork newsletter, we can still provide some practical tips on how to make the most of these features.

What can VisualAnalysis Do?

The answer to this question depends upon which level of VisualAnalysis you have purchased. The basic level does not offer any dynamic capabilities, only the static analysis. The standard level will calculate modes of vibration, as well as perform a modal-superposition based on a design spectrum (an idealized earthquake definition). The latter is called a "Response Spectrum" analysis in VisualAnalysis. In the advanced level, VisualAnalysis can also perform time-history analyses for seismic events or for vibrating machinery, but this topic will be left to a future newsletter.
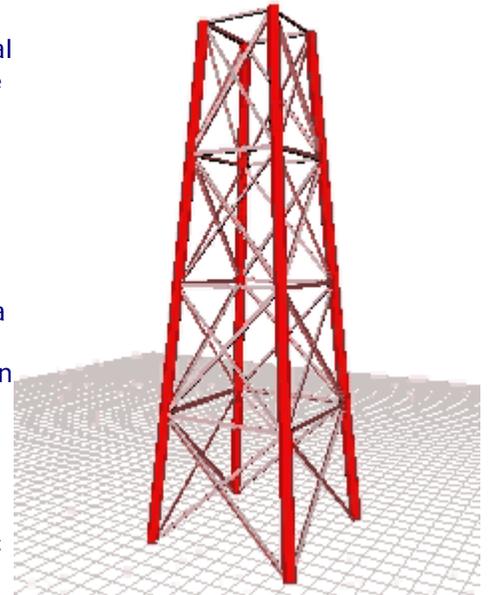
Seismic Analysis per IBC 2003 and ASCE 7-02

Current US building codes require careful and complicated consideration of seismic forces. If you are fortunate enough to be working in a low-seismic region, or are designing buildings of low "importance", you might be able to get away with a simplified equivalent lateral force procedure [IBC 2003 1617.4; ASCE 7-02 9.5.5] or something similar.

In all cases, however, you are permitted to perform more sophisticated investigation, such as the Model Response Spectrum Analysis [IBC 2003 1618; ASCE 7-02 9.5.6]. With VisualAnalysis, such an investigation might be easier and more accurate.

Running a Dynamic Response Analysis

Our advanced system specialist, Jeff Lutzenberger, Ph.D., has put together a white paper on Response Spectrum Analysis using VisualAnalysis, with more details that we can cover here, and you will also find more information available in the VisualAnalysis Help file and the references cited in both these resources.

1. The basic steps are these:
2. Correctly model the mass of your structure.
3. Create a response spectrum load case.
4. Determine which Design Spectra to use (or generate a new one)
5. Run your analysis to get base shear forces
6. Either use Result Superposition (advanced level only) or a manual procedure to include the seismic forces within your load combinations.

Rigid Link Rescue!

Modeling Tips & Tricks

It's called the Rigid-Link, but it's really just a member element that lets you model a "geometric separation" between to elements in your model. Rigid links enforce a relationship between the displacements at two nodes in your model.

Modeling is an Approximation

As you are aware, FEA modeling is an approximation: we model real structures using mathematical simplifications one of which is the linear element known as a member in VisualAnalysis. Member elements represent a true physical member,

such as a reinforced concrete column with a line passing through the centroid of the column, and given stiffness properties to capture the behavior of the true structural entity.

Mathematical Simplifications
This leads to further simplifying assumptions, such as all members are connected at the intersections of the member's centroid. If you are modeling a 10 ft by 10 ft concrete column and a W8x10 steel beam is framing into this column, connected at the face. Our "little assumption" is that the connection is made 5 ft from the actual location of the connection! Now VisualAnalysis advanced happens to have a feature to account for this very easily: End Zones. You can specify either a rigid or flexible (panel) near the end of the member to simulate various types of connections. However, there are other situations where VisualAnalysis doesn't help you with a built-in feature.

Rigid Links to the Rescue!
A Rigid-Link could be used to solve the above connection problem. Split your W8x10 member at 5 ft from the centerline of the column. Define the last 5 ft of the beam as a very stiff member (say 1000 times stiffer than a W8x10), with a weightless material. Now the behavior of the beam in the model will be more like the real world. You can just ignore the rigid link, as if it were not there, when looking at results, or designing members!

Other Uses of Rigid Links
There are actually many uses for rigid link members, some of which we have discussed in the past.

- Scissors Joints (http://www.iesweb.com/selfhelp/)
- Stacked Beams (3rd Qtr 2005)
- Moment Released Plates
- Top & Bottom Lateral Bracing

There are many other situations where a little rigid link can go a long way toward simplifying your life or improving the quality of your models! Let's take a closer look at each of these last three items, as they have not been discussed here before.


Moment Released Plates
Plate elements in VisualAnalysis do not offer "moment releases" like member elements do, so all connections to and between plate element in a frame model are "rigid" by default. If you are connecting a beam to a plate, you can use a member end release directly. However if you want to simulate a "hinged" connection between two plate meshes, there is no built-in feature to accomplish this. The solution is to separate the two meshes by a small distance, and insert short rigid link members between the two. Use a moment release at one end of each link. Here is a sample project (version 5.5) to demonstrate a plate mesh with moment releases.
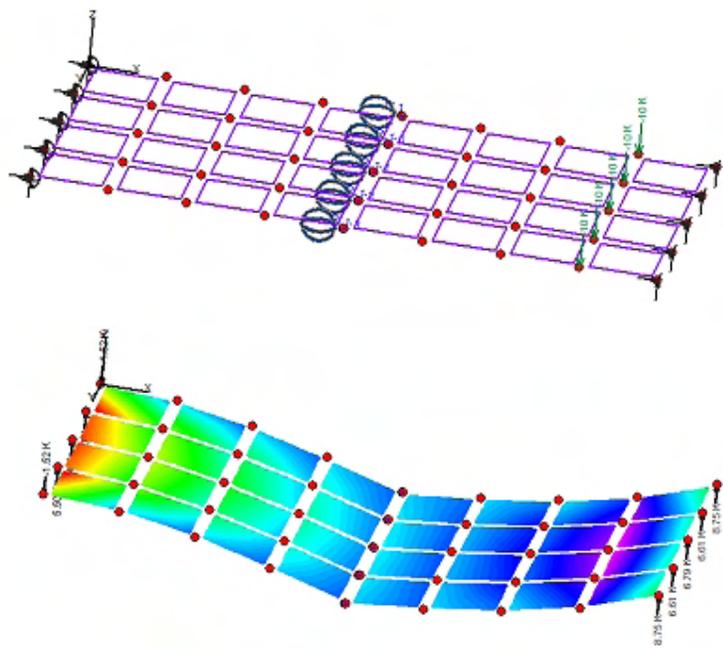
Top & Bottom Lateral Bracing
Bridge girders are often connected by horizontal trusses at the top flange. Sometimes you will also have horizontal bracing attaching to the bottom flange, or diagonal bridging between girders. Rigid links make this easy. Here is a sample project file (version 5.5) that demonstrates all three kinds of bracing and also shows how you might use the Name Filter feature to hide the rigid link members in Picture View, so that things don't look so cluttered.
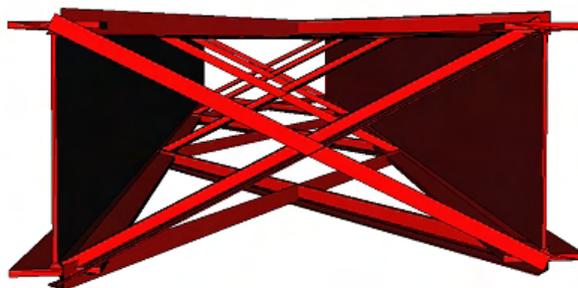
Now its Your Turn
Let the Customer Speak!

VisualAnalysis 6.0 has been under development since just before the release of version 5.5 and we are working hard to build a system that "***blows the doors off***" any tool we ever created before. Much of our work is based on feedback over the years from you, and is also based on ideas we have.

Who Wants What?
Today, we have reached a milestone in our development cycle and it is time to ask you some questions before we move on. If you are running <u>VisualAnalysis 5.5</u>, please take a minute or two to complete our short survey. If you are not running VisualAnalysis 5.5, please send us an email telling us why not!
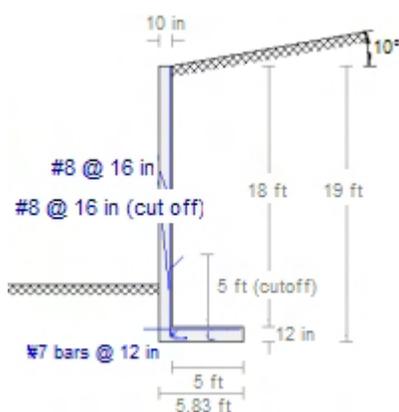
<div align="center"><u>Customer Survey VA6.0</u></div>

Please, Pretty Please!
Click the above survey link and help us, help you by answering four easy questions about VisualAnalysis 6.0.

Property Line Walls
"How To" in IES QuickRWall

It is not uncommon to have a retaining wall or basement wall that sits directly adjacent to a property line. In this situation your footing will be constrained to have just a toe or a heel, but not the other. You can quickly overcome this restriction in the design of your wall using IES QuickRWall for retaining wall design or checking, here's how:

On the **Footing** tab of the **Input Criteria** windows, simply enter a zero length for either the **Heel length** or the **Toe length**. Of course, the hard part will be dealing with all those little red x's:

If you have a basement wall, you will typically have a lateral support somewhere near the top of the wall. To do this in QuickRWall, just check the **Has Lateral Support (Restrained Wall)** option under the **Stem** tab of the **Input Criteria** windows to define the location of this support.

Latest Updates
Fixes in the Olympic Spirit

Hey, we all fall down once in a while! Sometimes we slip, sometimes we crash, and sometimes we just get caught "showboating" a little bit!

IES tools have had their share of bobbles lately, but we pick ourselves up and dust ourselves off and keep on going for Gold! Our latest releases are available <u>on the web site</u> along with "<u>Change Logs</u>" that will help you decide if you need to install the updates yet.

Update Service Changes
You may have noticed that the **Check for Updates** feature does not work in some IES tools, and in others the command simply takes you to our web site to download the latest release manually.

We are in the process of launching a newer, simpler update service that will notify you automatically when updates are available.

We appreciate your patience as we roll out these changes, with the goal of making your life easier, at least in the long run.

In the mean time, you will always find our latest releases available on our web site under **Download | Updates and Patches**.