



Member to Plate Modeling

Is Finite Element Modeling Art or Science?



Modeling typical frames or trusses with members is usually a very straightforward process following standard practices. When plate elements are required, things get more interesting and you have to start thinking, planning, and making multiple models. If you decide you want to create a detail model of some connection, then you cross a threshold into the finite element world, becoming part artist and part engineer. This month we explore some of

Contents

[Member to Plate Modeling](#)

[Shear Diagrams and](#)

[Numerical Analysis](#)

[ASCE 7-05 Seismic](#)

[QuickMasonry 2.0 Beta](#)

[Needle in a Haystack?](#)

the issues involved in modeling a beam to plate connection, such as the one shown at right.

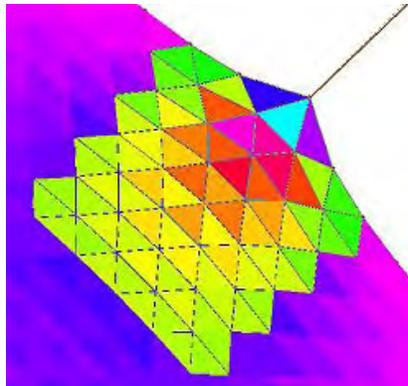
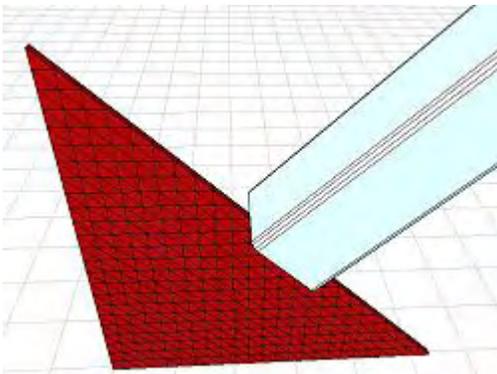
Getting Started

The most important issues to keep in mind when modeling this connection are the following:

- What are you interested in learning?
- Is your model accurate, yet as simple as possible?
- Have your results converged on the "true" solution?

If you are only interested in the member forces for the braces, beams, columns, etc., then you do not need to model the connection at all! On the other hand, if you are doing forensic work to understand a failure or if you are trying to determine if an undersized plate, already fabricated and installed, will perform adequately, a detailed connection model may be warranted, and your first model attempt may be to create the plate and attach a beam element to one of the edge nodes, something like the **Picture View** below. (By the way, more detailed pictures may be seen by

clicking on the image to pop up a 'full size' version.)

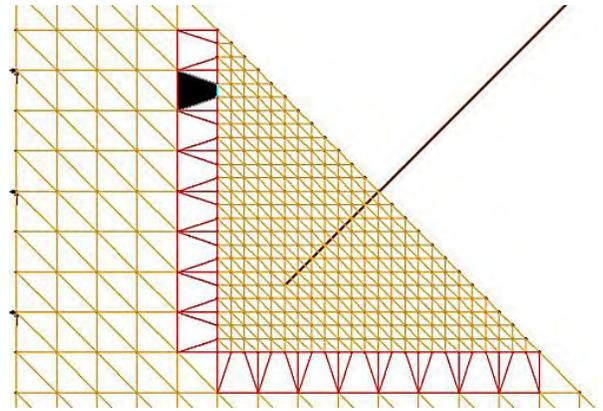


This model is certainly easy to construct, but may miss the mark completely if your goal is to understand the plate behavior and stress under an axial load in the brace member. If you look at the plate results you can see highly concentrated stresses at the load point, where only half a dozen plate elements are very highly stressed. But is this accurate? If you go back to the photo, you can see that the HSS member extends into the plate some distance, in fact, the highlighted plate elements (green/yellow) in the result view above would all be "covered" by the member, so we probably cannot trust these stress results at all.

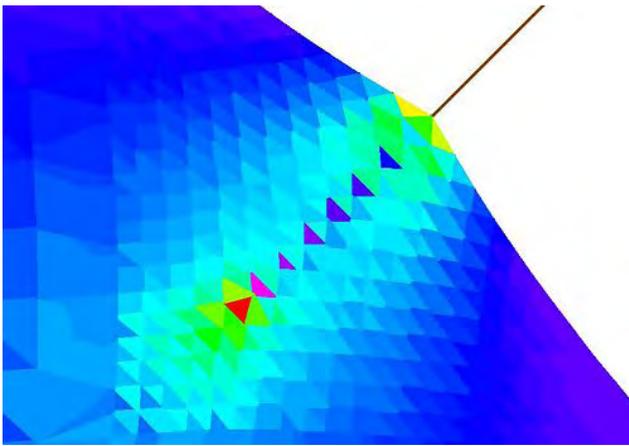
Mesh Refinement and Stiffness

We could make a more refined model that might better capture the plate stresses and behavior by making two changes. First we will refine the mesh in the area where we have high stress variations. This is a little tricky.

It is very easy in VisualAnalysis to select a group of plate elements and split them to refine the mesh. The difficulty comes when you try to split some, but not all of the plates (important for keeping solution times reasonable, reducing report size, etc.). In the above refinement we have created a transition-zone to ensure plates are connected properly. To start we refined the plates in a triangular region of the model. Next we deleted plates between the refined area and the unrefined area. Finally we drew in some triangle and quadrilateral (one is colored black above) elements to connect all the nodes between the two areas. It was easy to draw in one "square" of 3 elements and then generate copies of them to fill in the row or column.



Next we will "extend" the member into the plate mesh, creating seven short members between plate nodes to help model the stiffness of the member as it overlaps the plate.



You can see how this helps distribute the load of the member across the plate more uniformly, our stress concentrations are also reduced in magnitude. This model may be accurate enough, but again it depend on what you want to learn. If you are interested in the stresses under the welds between the brace and the plate then this model is probably insufficient even if we ignore issues of convergence, which have been covered in the *VisualAnalysis User's Guide* and elsewhere.

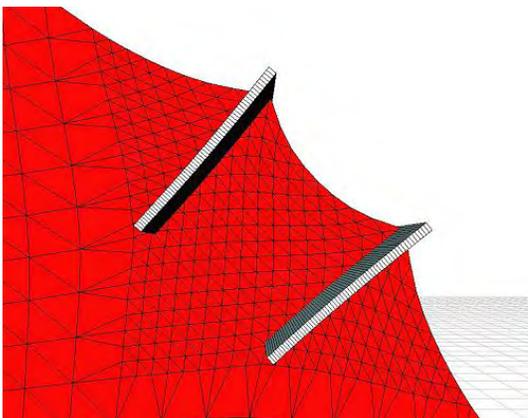
Going One Step Further

We have still not accurately modeled the force transfer between the HSS member, which is connected at two flanges (separated by perhaps 6 inches). We could modify our model to account for the fact that the brace force is divided between

as follows:

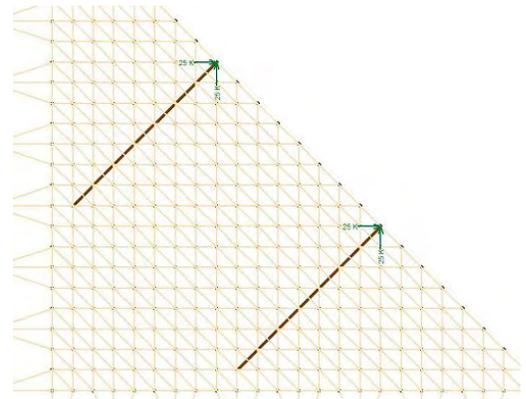
This model now provides a substantially better representation of the connection geometry than our original version, and will likely yield more accurate results. In this model we have sized the member elements to

match the flange dimensions of the HSS member.



Conclusions

This short investigation illustrates some "art of modeling" techniques you might use as you construct models for your projects. There is much more to finite element modeling to discover and understand, that cannot be covered here, so we refer you to a textbook like "*Finite Element Modeling for Stress Analysis*" by Robert D. Cook (ISBN 0471107743) for a more thorough and technical discussion of these concepts.

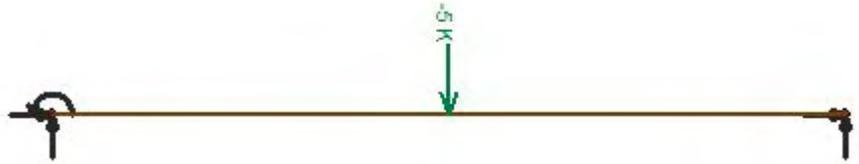


Shear Diagrams and Numerical Analysis

Secret Mysteries of Finite Element Analysis Revealed!

Despite popular opinion, [VisualAnalysis 5.5](#) is an approximate numerical program rather than a substitute for idealized hand calculations. VisualAnalysis uses discrete mathematics to calculate results that are quite accurate but not what you might expect. Designed for very small academic projects and also for projects with thousands of members, there are some necessary trade-offs to be had. One place where this can show up dramatically is when looking at shear results for a beam with concentrated loads.

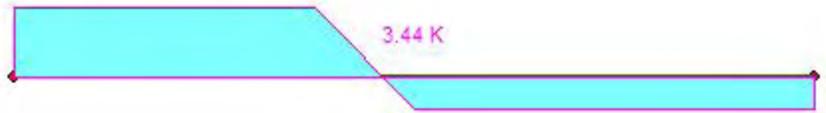
Here for example is a Result View showing the default shear diagram for a beam with a point load in the middle. Notice the **shallow slope** where we would expect a vertical line!



Explaining the On-Frame Diagrams

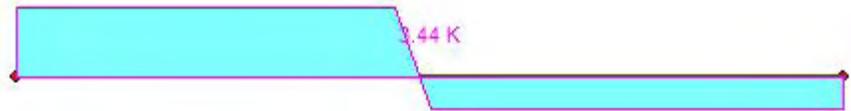
The "on-frame" moment and shear diagrams are rather crude by default, interpolating results at just **nine** locations along the length of a member, and not even intelligently chosen locations, but evenly distributed points. The reason for this is simply performance. If you have many members shown, then it still works!

Many IES customers are analyzing over 40 load cases and combinations, so the variations in load-points can be significant and we have not tried to optimize our result reporting as we would if we were creating a diagram manually. For moment diagrams this is more than sufficient for 95% of all typical loading patterns. For shear diagrams, however, this approach can be misleading and leave you thinking there is an error in the software.



Tweaking the View

You can improve this **Result View** default slightly, using the Project Manager's Filter tab. Under the Members section you may increase the "Display Sections" value to **twenty-five** for a sharper shear diagram. If you regularly work with smaller-sized projects, you may wish to permanently set this default for future new windows, using **Tools | Customize Appearances**, on the **Filters** tab.) Here is the same **Result View** with the improved "resolution" setting, the line is closer to vertical, but still not very good:



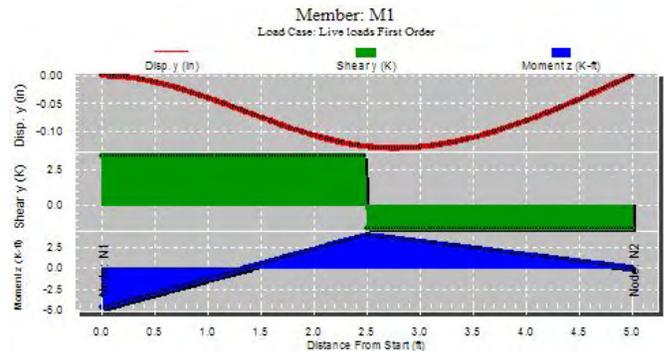
Member Graphs to the Rescue!

If you need really accurate graphs, you can select a member (or a chain of members) in the **Result View** and **Right-Click** the

mouse, choose **Member Graph** from the Context Menu and view a very nice looking graph that uses more like 200 points along each member element. The shear diagrams are significantly sharper. The **Member Graph** associated with the above problem is shown here, with shear in green and a "near vertical" discontinuity in the plot.

What About Text Reports?

Tabled numerical results (including shear values) provide a similar balance between performance and precision. The *Member Internal Forces* table will report eleven result points by default, but you can set this value in the **Table Options** dialog (**double-click** on a table in a report). There are other report items that are designed to report extreme values and these automatically scan the member results and display various extremes.



QuickMasonry 2.0 Beta

QuickMasonry Testing

Be the first to try out [QuickMasonry 2.0](#) and help IES track down last-minute defects before we release this fantastic upgrade! We are offering a very short beta test period on this new tool for masonry design. If you use [QuickMasonry 1.0](#) or if you plan to purchase QuickMasonry, now is your chance to make sure we have it working at 100%.

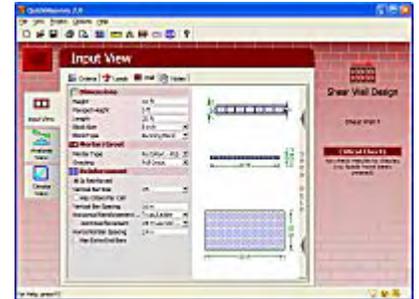
Automatic Free Upgrades

QuickMasonry 2.0 will be an automatic, **free upgrade** to QuickMasonry 1.0 customers. Customers have provided excellent feedback over the last year on what was missing in version 1.0. As soon as we have the final release available, customers will be able to update to the new version.

New Features

We have also expanded the already comprehensive capabilities in version 2.0 to support the latest MSJC code, added the strength provisions and more:

- Clay Masonry Design
- More Calculations Exposed
- Load Combinations Report
- Improved Interaction Diagrams
- [More Details Here...](#)



ASCE 7-05 Seismic

Dude! Are you shaking?

In a previous [VirtualWork](#) we discussed the insanity of ASCE 7 wind load combinations and how VisualAnalysis 6 would be helping you implement the provisions without forcing you to be as ridiculously meticulous as the code writers, cackling in their evil committee meetings, had plotted.

Today they have us shaking in our boots again as we reconsider the seismic load combinations introduced in [November's VirtualWork](#). This is almost like going from undergraduate to graduate work, where you learn that all the assumptions in mechanics of materials were never really true: the world is more complicated than you had originally believed.

We forgot to include the requirement of combining the orthogonal horizontal forces in 100% + 30% combinations, which doubles the number of cases to consider.

Combinations for Strength

- $1.2D + p_x Q_{ex} + 0.3p_z Q_{ez} - 0.2S_{ds}D + L + 0.2S$
- $1.2D + p_x Q_{ex} - 0.3p_z Q_{ez} - 0.2S_{ds}D + L + 0.2S$
- $1.2D + p_x Q_{ex} + 0.3p_z Q_{ez} + 0.2S_{ds}D + L + 0.2S$
- $1.2D + p_x Q_{ex} - 0.3p_z Q_{ez} + 0.2S_{ds}D + L + 0.2S$
- $1.2D + p_z Q_{ez} + 0.3p_x Q_{ex} - 0.2S_{ds}D + L + 0.2S$
- $1.2D + p_z Q_{ez} - 0.3p_x Q_{ex} - 0.2S_{ds}D + L + 0.2S$
- $1.2D + p_z Q_{ez} + 0.3p_x Q_{ex} + 0.2S_{ds}D + L + 0.2S$
- $1.2D + p_z Q_{ez} - 0.3p_x Q_{ex} + 0.2S_{ds}D + L + 0.2S$

Combinations with Over Strength

- $1.2D + \Omega_x Q_{ex} + 0.3\Omega_z Q_{ez} - 0.2S_{ds}D + L + 0.2S$
- $1.2D + \Omega_x Q_{ex} - 0.3\Omega_z Q_{ez} - 0.2S_{ds}D + L + 0.2S$
- $1.2D + \Omega_x Q_{ex} + 0.3\Omega_z Q_{ez} + 0.2S_{ds}D + L + 0.2S$
- $1.2D + \Omega_x Q_{ex} - 0.3\Omega_z Q_{ez} + 0.2S_{ds}D + L + 0.2S$
- $1.2D + \Omega_z Q_{ez} + 0.3\Omega_x Q_{ex} - 0.2S_{ds}D + L + 0.2S$
- $1.2D + \Omega_z Q_{ez} - 0.3\Omega_x Q_{ex} - 0.2S_{ds}D + L + 0.2S$
- $1.2D + \Omega_z Q_{ez} + 0.3\Omega_x Q_{ex} + 0.2S_{ds}D + L + 0.2S$
- $1.2D + \Omega_z Q_{ez} - 0.3\Omega_x Q_{ex} + 0.2S_{ds}D + L + 0.2S$

Fortunately, VisualAnalysis 6 will be coming to the rescue early in 2007 to generate all these combinations, if they are required. What remains is to figure out how to use the over strength combinations in design work.

Needle in a Haystack?

Finding the Source of Crashes

FAQ: Why Does It Crash?

We regularly get "crash reports" from customers with a message like this.

"The software has crashed, do you know why? Can you fix it NOW?!"

While we would love to be able to push the "Easy Button" and fix everything, the situation is a little more uncertain.

Living with Windows

Application crashes in Windows are all too frequent in any application that we run. We are regularly getting crashes in just about every program including MS Word, Excel, Outlook, VisualStudio (the MS C++ compiler), QuickBooks, Act! and even Windows itself. The major cause of the problem is that the systems are simply too complex: layers upon layers of code from a variety of manufacturers built on layers of hardware components from hundreds of vendors.

Hardware Abstraction

Your video card driver could very well be the source of crashes, especially if you make frequent use of the **Picture View** in VisualAnalysis. Your printer card driver could be causing problems as well, and so we recommend that you try updating these device drivers to see if that helps with your system's stability.

Background Processes

On top of the system complexity you have the added complications of interacting "services" such as Norton Antivirus, and the two dozen other sub-systems that may be running in the background and "hooked" into various Windows operations each interpreting, blocking, changing, re-ordering messages, etc. In other words, drastically altering the behavior of the basic Windows system.

Isolating The Problem

The bottom line is that locating the source of the crashing is like finding the "needle in the haystack", unless we have some additional clues or debugging aides.

For example, we have been collecting crash reports in [VisualAnalysis 5.5](#) for 16 months and this has enabled us to locate and fix dozens of crashing problems and potential crashing problems, but in the last six to nine months, we have learned very little from these reports, as the remaining crashes are from various low-level systems that we are not able to trace with the current details in our reports. We hope to improve the diagnostic capabilities in **VisualAnalysis 6.0**, where we will be eliminating support for Windows 98 and Windows NT systems, which hamper our abilities.

Explaining Your Situation

If you experience crashes or system "hangs", then the most important information **IES Support** can get from you is any recognition of a pattern in the crashing (i.e., "It always happens after I do X", or "It only crashes if I run the software all day").

Any clues from you that will help us narrow down the search for offending code from 200,000 lines of code to 100 lines or even narrowing our search down to a large module like "LRFD steel design", can help.

The holy-grail for **IES Support** is a small example project file, along with step-by-step instructions for reproducing a problem. We can run this through our debugging tools and locate the offending line of code.

Protection & Backing Up

Your first defense is to use our latest products and the latest updates available. We still regularly get crash reports from customers using a release that is over 12 months old. (Updates are free and usually take less than 5 or 10 minutes to download.)

The best approach for you to take is to make use of and understand our crash-protection system and "history file" mechanisms in VisualAnalysis and to make sure they are turned on. Make regular "checkpoint" saves of your work to avoid loss. You can pay attention to changes made in other systems (hardware or software) on your machine so that you can recognize when some change has negative consequences on other software packages. You can try disabling certain "add-ins" to Windows in a trial-and-error fashion to see if they are contributing to the instability problems, etc. You can try running VisualAnalysis on a different machine to see if you experience the same behavior in a different environment.

IES Customer Partnership

IES engineers and customers have the same goals in mind: **zero crashes, 100% stability**, an engineer's dream system! Every single failure in VisualAnalysis (or any IES product) puts IES at risk and we do not treat that lightly. Together we can create systems that offer excellent feature sets, incredible performance and a stable and robust platform for structural engineering.