



Bracing: A Short Course

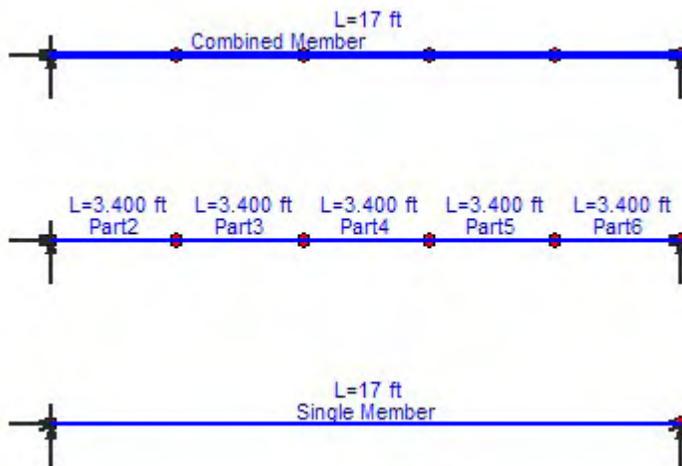
Members are unbraced by default in VisualAnalysis

Despite the amazing processing power available in modern computers they still cannot replace an engineer's ability to make sound engineering decisions. While [VisualAnalysis](#) is programmed to be as smart as possible, the software cannot always do the "right thing".

Conservative or Not?

Take lateral bracing in design checks as an example. VisualAnalysis tries to be conservative by using the member-element length as the unbraced length. This is a conservative approach as long as you model beams and columns with a single member element: a typical situation. It is also conservative if you split members only to provide connections for perpendicular or brace members that would provide lateral and/or torsional stability to the member: also a typical situation.

However, if you arbitrarily split a beam into five elements, so you can apply nodal loads to represent air conditioning units hanging from the beam, then the unbraced length assumption is no longer very good as the actual unbraced length may be five times greater than the length of each element.



With VisualAnalysis 5.x, the Combined Member feature, which allows you to view a chain of elements as a single entity for loading, reporting, and design, introduces additional complexity into the situation. The design software sees a Combined Member as a single member and it knows nothing about the intermediate connection points that may or may not exist or act as brace points. So in this case, the software may be overly conservative if you use the unbraced default.

Using the sketch above, we have three different ways to model a simply supported beam. The design software will assume unbraced lengths of 17 feet for the top and bottom models, and 3.4 feet for the middle model. However, you can easily change the bracing configuration in the design group.

Contents

- [Bracing: A Short Course](#)
- [New FAQ Answers](#)
- [QuickRFooting Update](#)
- [Printing to Adobe PDF?](#)
- [VA 6 Development](#)

New FAQ Answers

Customer Questions Answered

IES is updating the [FAQ Answers](#) system regularly. It is a great place to start if you have technical questions which are not addressed in a Help file.

Update a Non-Internet Machine

Most IES customers have Internet access on the machine where their software is installed and running and so the "Direct Update Links" method works. If you have a laptop or otherwise disconnected machine, you can still install updates from our web site: (Using VisualAnalysis as an example)

Combined Members: Advantageous or Not?

The advantages for using combined members are these:

- There are fewer members in your model, so reports are smaller, editing is easier.
- In design calculations the Cb factor is "smarter" because the complete moment diagram for a chain of member elements is seen rather than just a portion of the moment diagram when each element is checked separately.

The disadvantages of using the combined members:

- You could get away with default "unbraced" design groups if the only external bracing is at the end points of individual members.
- For continuous beams with interior supports, you may not be able to get deflection checks to work the way you would like. Use combined members on either side of the supports, but not crossing them.

Understanding Design Brace Options

Steel, Wood, and Cold-formed design all allow you to specify a variety of "brace points" along a member, in a number of different ways. This flexibility can be a bit overwhelming and so we are simplifying the options in the upcoming VisualAnalysis 6.0 release. But it is really not hard to understand the three ways to specify bracing:

- **Typical:** Regular or pattern bracing, say 1/4 points, or 1/3 points.
- **Fraction:** When a typical pattern does not work or you have irregularly spaced brace points. Here you just list the brace points as fractions between 0 and 1, such as "0.4, 0.8".
- **Specified:** You enter the absolute unbraced length that is used for all locations along the member.

In addition to the bracing type, you have to specify bracing for three different types of buckling: lateral buckling, column buckling, or torsional buckling (cold-form only). Though which you use depends on the type of member you are designing.

The Modify tab in Project Manager allows you to view design parameters as either "Beam Only", "Column Only", or "Beam-Column". These settings don't really change any of the parameters, but just make editing easier by renaming the bracing options and by hiding parameters that will not likely apply:

Bracing		Bracing	
Top Bracing	Pattern	Weak Bracing	Pattern
Pattern	Unbraced	Pattern	Unbraced
Bottom Bracing	Pattern	Strong Bracing	Pattern
Pattern	Unbraced	Pattern	Unbraced
Beam Deflectic		Column Deflect	
Strong Type	None	Column	
Weak Beam	None	Design As	
Design As		View As	Column Only
View As	Beam Only		

1. Visit the customer install or update page: www.iesweb.com/install.htm
2. Look for the second set of links on the page, under the heading "Alternate Update Files".
3. **Right-click** on the link for VisualAnalysis (currently it says VisualAnalysis 5.50.0019 Package)
4. Choose **Save Target As...** (or in Mozilla Firefox choose **Save Link As...**)
5. Save the file (currently install-va5-50-0019.exe) to a CD-R, flash disk, or whatever.
6. Move this file to a machine you need to install onto and then run it.

Offsets, End Releases, Supports

When using member centerline offsets along with member end releases, and simply supported nodes, you can easily run into an unstable structural configuration.

VisualAnalysis offers advanced modeling feature that when combined can leave you somewhat dumbfounded. Fortunately we can use the power of the software to help us explain the behavior so that we can understand how these features interact.

Beam Bracing

For beams, bent about the strong axis, you have to worry about lateral buckling. If the member is subjected to a positive bending moment, the top (+y) is in compression, so the "Top" brace condition will be used to determine the relevant unbraced length. For negative bending, the bottom (-y) is in compression so the "Bottom" brace condition will be used.

Column Bracing

Columns can have braces in either the weak direction or the strong direction. If you don't specify that your design group is a "column only" then you may not see the correct bracing options for a column. You may see three options: Top, Bottom, and Strong, in which case you can specify the top and bottom both the same as these are the "weak" braces.

How are Design Checks Made?

The design software will look at the unbraced lengths differently, depending on which check is being made. For strong bending, the unbraced length of the compression-side is used, unless the tension-side stresses control (unsymmetric shapes!). The checks can get fairly complicated and involved and you may not see all the complexity or nuances in the design check reports. For more details about what got checked, you can double-click on the design report to show "Complete Check Details".

Printing to Adobe PDF?

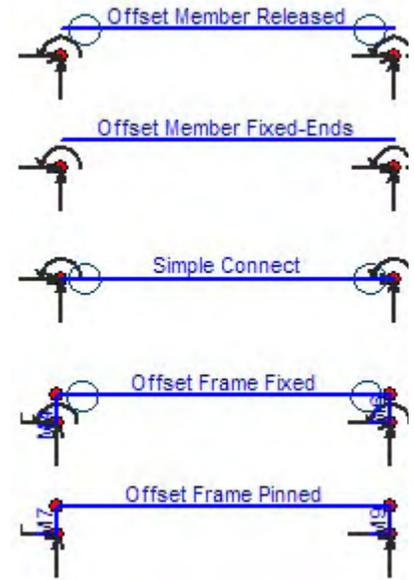
A customer had problems with Acrobat 7's Printer Driver...

A Customer Writes:

VisualAnalysis crashes when printing a Model View (picture) to Adobe Acrobat 7.0.8, what can I do?

IES Support Answers:

I too received a crash when I went to print the Model View from VisualAnalysis to my "Adobe PDF" printer driver. Then I ran Adobe Acrobat Reader, and checked for updates. Even though the "About Box" said I had 7.0.8 installed, this version still appeared in the update list, so I did the update, rebooted and went into my Start | Printers & Faxes menu item to look at the properties and settings for this printer driver.



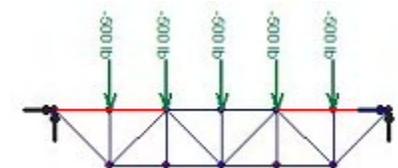
We have created a [sample project](#) that shows most of the possible configurations for using offsets and end releases on a single member. You can model a small frame (using 'rigid links') to simulate what the centerline offset feature does and it helps you to see the behavior more clearly.

Note that if you try to model the frame with pinned supports and end releases, it is unstable.

Understand 'Indeterminate'

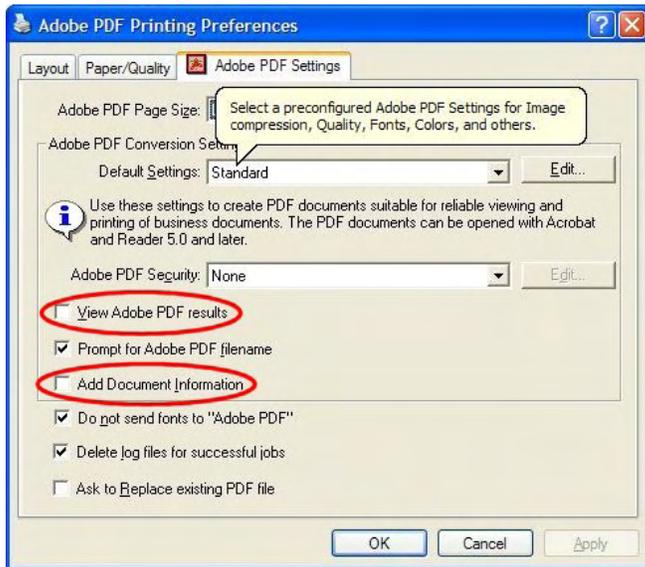
We all understand that if you have a simply supported beam in bending, the top is in compression. If you have a simply supported truss, the top chord is in compression too.

So why is it that some of the top chord members in this model are in tension?



Answer: The FRAME is indeterminate. There is a horizontal support at both ends AND the 'truss' members are not

Here I cleared the option to "View Adobe PDF Results" (Because I don't like it when Acrobat pops up every time I save a file!) and "Add Document Information" (I suspect this may be part of the crashing, because we don't write any document information from VisualAnalysis?)



Then I tried again from VisualAnalysis, and it seemed to work just fine.

When we print, whether to Adobe Acrobat or to ANY printer, we do exactly the same thing, and send the exact same commands, etc. So generally, if there is a crash it is the problem of the "print driver" software, not VisualAnalysis. Obviously we can and do make mistakes, but I think in this instance the problem lies somewhere in the printer driver software. You might try updating as I did and also playing around with different settings in the driver until you get it to work...

VA 6 Development

Progress on the Next Generation

Programmers have been continuing work on VisualAnalysis 6 implementing improvements in a number of areas as we get close to releasing a preview version for customer testing. Here are just some of the things we are creating to make your life easier and to improve the quality of your work:

Improved DXF Handling

Select which layers to import, more import types (e.g. polylines and arcs), template members (shape, material) on import, better reliability, etc.

Smart Sketching

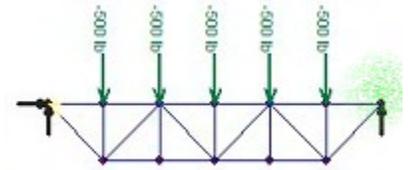
Model View sketching is getting smarter and easier with multiple grids, new layers (to replace cut-planes), and now we are adding smart member and plate splitting. If you drag a new member it will "snap" to an intersection with an existing member. If you drag a member or plate such that it intersects with another plate we will automatically split the plate element(s) to give you a connected mesh.

STAAD File Import

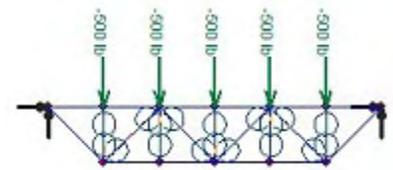
Big improvements have been made for reading industry standard .STD (STAAD) script files. This should help you migrate projects from other software packages or validate models in multiple tools.

released, making it a frame.

You get more expected behavior (top chord compression) if you make one end a 'roller' support:



You also get more expected behavior if you release the moments on the web members so that it is a TRUSS:



Of course, none of these models are 100% accurate, they all reflect assumptions that you make about the real structure, supports, and loads.

However, with all this sophistication and flexibility in the Finite Element Method, you must pay careful attention to both your boundary conditions (supports) and your connectivity in order to get the behavior you expect!

QuickRFooting Update

IES responds to customer needs

Build 1.00.0008 (October 9, 2006)

Check for updates of all products at:

www.iesweb.com/install.htm

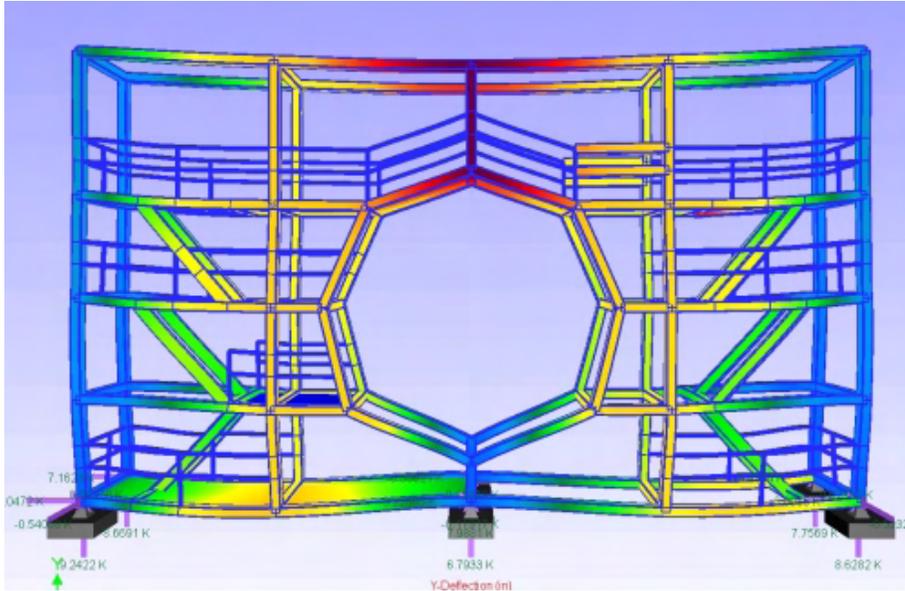
New Feature:

A menu command (Project | Copy Existing Load Set) was added to ease the tedium of creating multiple load sets for when you have loads of the same source that should be considered independently, for example wind from different directions. This prevents the

VA 6 Preview Image

A Screen Shot from the Lab

We test customer projects in VisualAnalysis 6 to make sure the tool stands-up to real-world projects. If you have a project you would like us to run through the new version to see how it will work, send it to IES Technical Support. We should have a test version available soon so that you can run your own tests as well...



need to enter those loads that are constant across load sets (e.g. dead load) every time.

Two Fixes:

1. If you chose the command, **Design | Size For Bearing Pressure** and the footing was already passing the bearing pressure check, it would always reset to a 3x3ft footing. The command worked correctly for the case where the bearing pressure check was failing.
2. The wrong value for punching shear capacity (V_c) was being used for cases where the aspect ratio of the column exceeded 1.5, and for cases where the punching shear perimeter was unusually large relative to the thickness of the footing.