

MSC SOFTWARE REVIEW: MULTIFRAME & VISUAL ANALYSIS

By Jacques Cattan

In the January issue of Modern Steel Construction, I reviewed six structural analysis software packages; this month I'm reviewing two more and in the future will expand the reviews to other types of engineering software. If there are programs you're interested in having reviewed, fax the information to Scott Melnick at 312/670-5403 or email melnick@aiscmail.com.

LET ME START WITH THE SAME DISCLAIMERS I DID LAST TIME: the opinions I express in this article are only mine and in no way do they represent AISC or Modern Steel Construction. Also, to put things into perspective, I am a software user, not a programmer. Finally, I did not evaluate the software for its accuracy (a full-time job in itself), but rather its ease of use.

Why is ease of use so important? Because computer and software technology is changing so rapidly that structural engineers frequently have the opportunity to upgrade or even switch to a better product. Software vendors should take advantage of today's technology to make a structural engineers work faster and easier (and of course more accurate), and as such it should be easier for structural engineers to evaluate and learn a new software package.

DOS vs. Windows. After the first set of reviews appeared in January, MSC received a large number of mostly complemen-

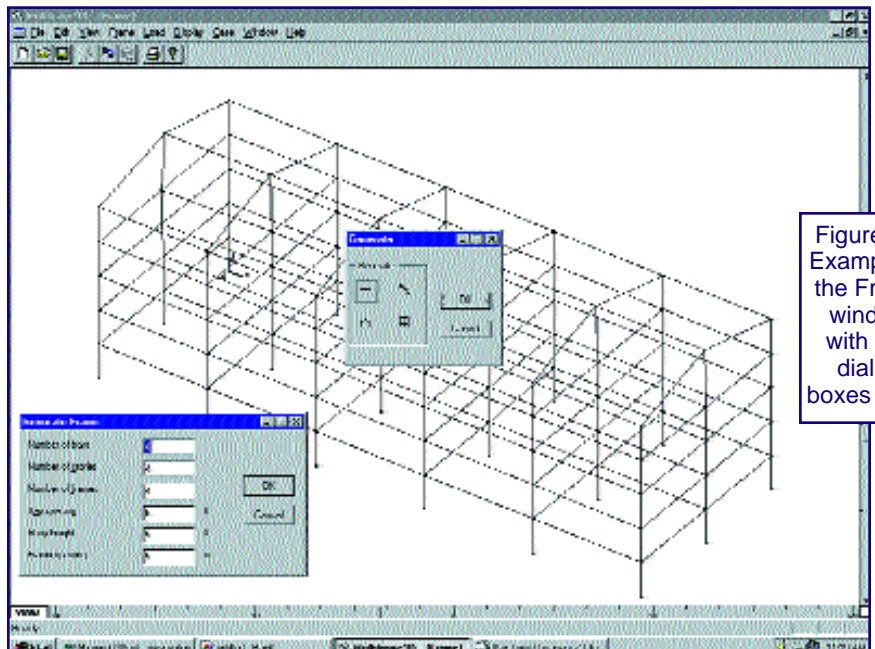


Figure 1.1
Example of
the Frame
window
with two
dialog
boxes open

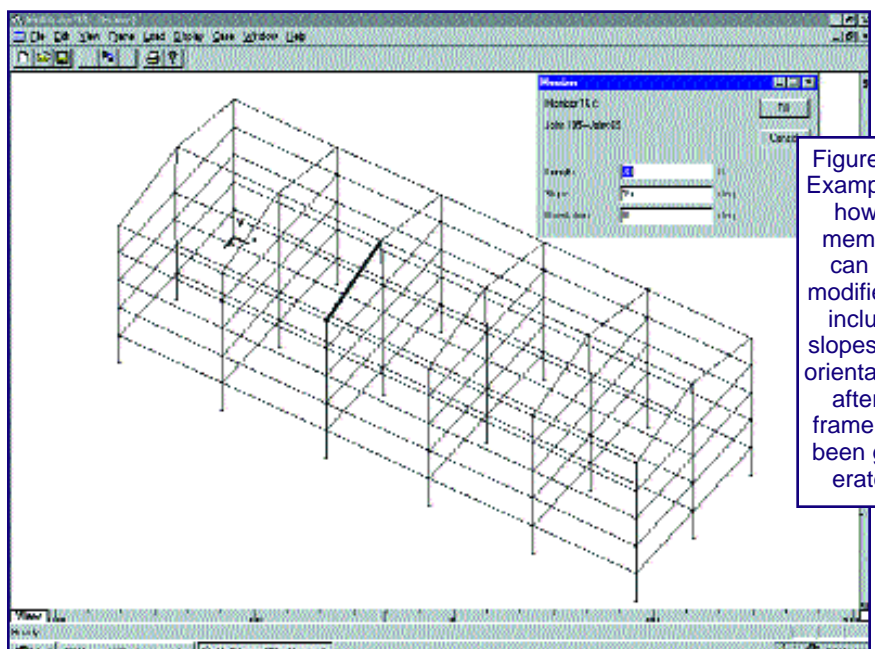


Figure 1.2
Example of
how a
member
can be
modified to
include
slopes and
orientations
after a
frame has
been gener-
ated

tary-comments. Most of the critical responses, however, focused on the amount of attention I gave to the look and “feel” of a program; in essence I strongly espoused the benefits of a good GUI (Graphical User Interface) and others claimed that text-based systems are more than adequate. Why am I so pro GUI? Because when I design I look at a series of plans and sketches that are trying to portray a three-dimensional structure. I examine drawings, not spreadsheets, nodes or frames. I layout floors, capturing the main architectural and mechanical features, stack them up and try to find an optimum structural system. It is a visual concept, not a series of tables with calculations. Calculation sheets are needed tools, but should come after this visualization process. Computers are ideal tools for number crunching, but let’s not forget that structural engineering is also about design.

Garbage in, garbage out.

For most structural applications, the software packages on the market today are all about equally accurate since they typically use similar mathematical models. In common structural engineering applications we use linear elastic analysis and most programs include a P-Delta analysis, so we are not talking about heavy-duty finite element analysis. My point is if a readily available structural engineering program is inaccurate it is easier to spot, as opposed to a general finite element analysis package (such as ABAQUS, ALGOR, ANSYS, etc.) that we sometimes need to use in order to push the envelop. Garbage in garbage out applies equally to computers and to a room full of engineers needed to check each others work. Simplifications, assumptions and judgments about a structure in order to model it leads to errors and over design. No amount of analytical or computational tools can make up for engineering experience. Finally, I live in

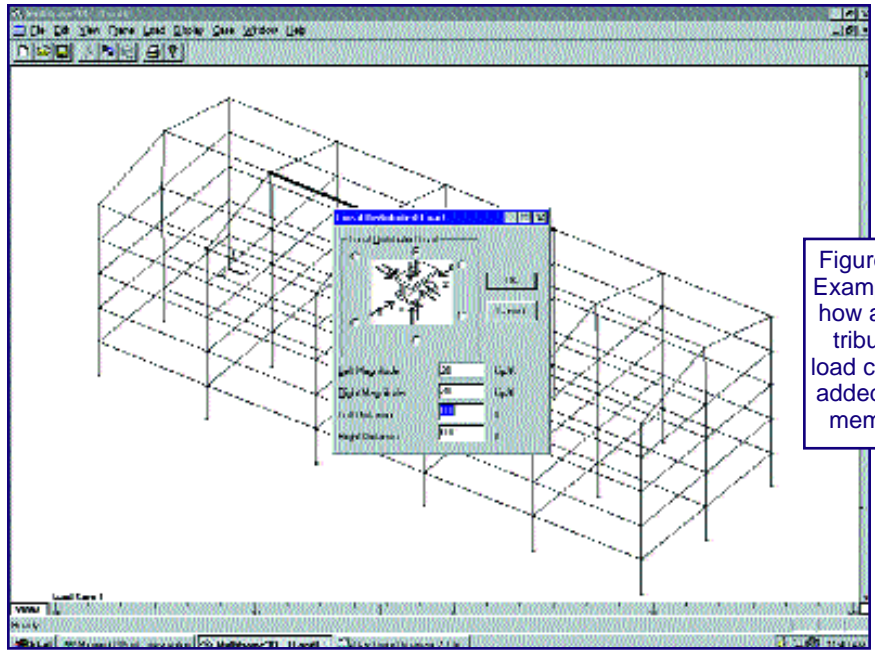


Figure 1.3
Example of
how a distributed
load can be
added to a
member

1997: the time of the Pentium II and MMX technology, the decade when every PC is shipped with 16 MB of RAM, a GUI environment and plenty of hard disk space for even a non linear plastic analysis. As Dilbert is fond of saying, technology is not for wimps.

Every effort is made to review the latest versions of all available structural analysis programs. Inadvertently, in the January issue an older version of ETABS was used and we apologize.

MULTIFRAME

(Version 5.0b2)

Formation Design Systems
5550 Scotts Valley Dr., Suite 200
Scotts Valley, CA 95066
ph: 408/440-0702
fax: 408/440-0705
<http://www.formsys.com>
For more information, please circle number 92.

(This program was previously only available as a Macintosh program. The reviewed program is the beta release of the new Windows version.)

Organization: This graphical program is organized around six windows.

- * **Frame** is where the geometry is defined, member sizes and restraints are assigned.
- * **Data** is a spreadsheet view of the model, including geometry and loads.
- * **Load** is the window in which loads are assigned, load cases are formulated and analysis is performed.
- * **Result** is a spreadsheet form of resulting moments, forces and displacements.
- * **Plot** is a graphical representation of the results.
- * **CalcSheet** is a built-in spreadsheet.

Frame window: This was a very easy and intuitive program to use. When the program is launched, the user is in the frame window to start a model. Creating frames is simple with the generation dialog windows: they look like any other Windows application. Figure 1.1 shows a frame I used and a couple of dialog boxes. Figure 1.2 shows how a member can be modified to include slopes and orientations after a frame has been generated. Steel shapes can be assigned to members by

Figure 1.4 Inside the Frame window, you can display load magnitudes and other features

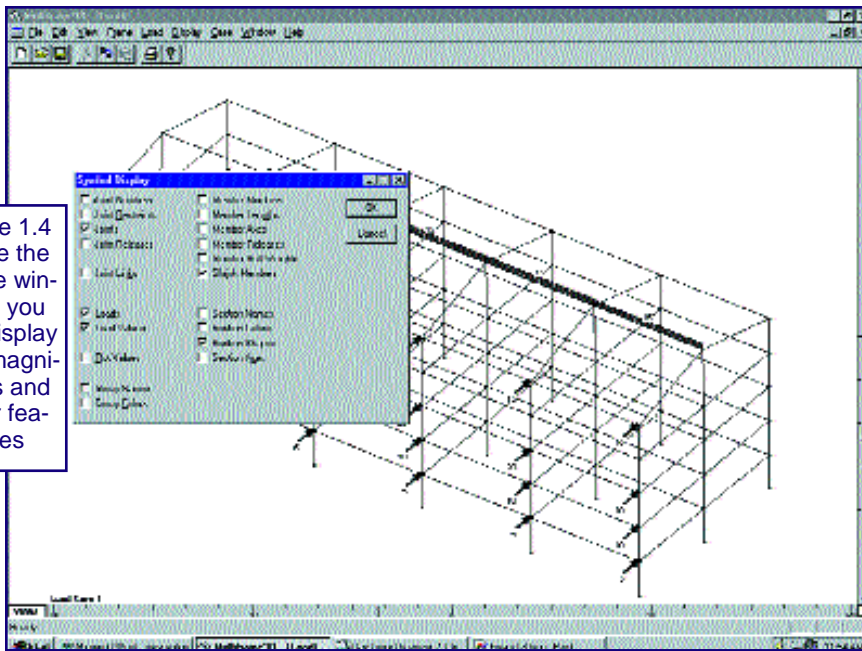


Figure 1.5 The Results window is made up of tabs, just like in Excel

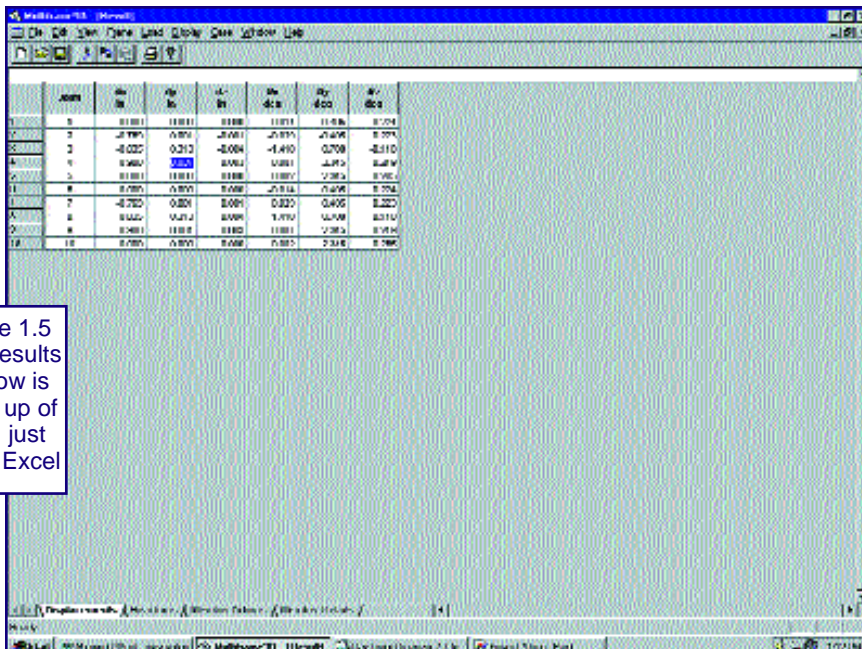
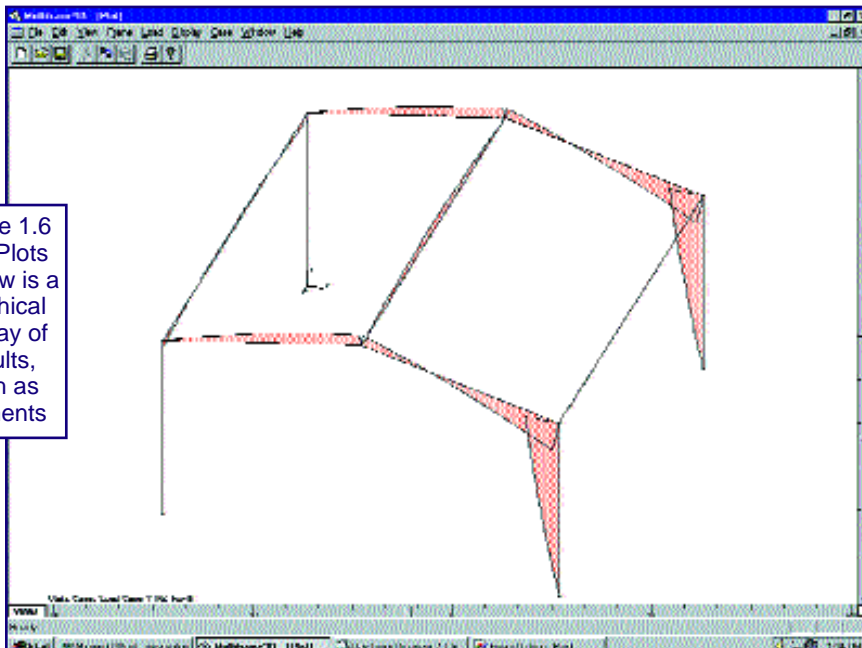


Figure 1.6 The Plots window is a graphical display of results, such as moments



selecting from a database; the user can do it on a member per member basis or by selecting all sloped or all horizontal or vertical through the edit menu much like a word-processor program. Selecting a view can be done through a default menu or by dragging the angle at the bottom or the right of the screen. Subdividing members is simple again through a drop down menu. It is very easy to display and group elements for viewing and analysis purposes, it is also simple to link joints to simulate rigid floors. Adding a new member is a simple click and drag process on a grid.

Load window: Once the frame is defined, adding loads and load combinations is just as simple. Again the window looks just like the frame window except now different features are activated. Figure 1.3 shows how a distributed load can be added to a member. Simply drop the load menu down, select if you want it with respect to the global or local axis assign a value and accept. Of course like any windows application, by pressing the shift key you can select several members and through the display menu you can display load magnitudes and many other features just like in the frame window as shown in Figure 1.4. Once the loads and load cases are defined the user can analyze the structure again through a drop down menu command.

Results window is straight forward (Figure 1.5). It is made of tabs just like an Excel spreadsheet. One can look at a variety of data, copy, cut or print any of it.

Plot window is a graphical display of results such as moments (Figure 1.6), shears, stresses and displacements. By selecting a member a designer can see the details; if you are looking for the values at a particular location simply move the vertical dotted line and read the results (Figure 1.7).

CalcSheet window is a spreadsheet for calculations. Of

course it is possible to have multiple views of just about everything (Figure 1.8).

This is a very easy program to use. Unfortunately, at this time the steel design menu has not been released so code checking must be done outside Multiframe. A couple of weaknesses are plate stresses and code loads generation. I did read in the Manual that Version 4D had seismic spectra for dynamic analysis but the version I had did not have this feature. Finally, I could not find any P-Delta analysis capabilities in this version.

VISUAL ANALYSIS

Integrated Engineering Software
Visual Analysis 3.0

Operating System: Windows

For more information, please circle number 87.

Organization: This program is organized around one window and a series of wizards (such as View Manager, Analysis, New Elements, and Load).

Model Edit window. The Model Edit window is the default window at start up. Files are organized by project when you start a new project you get the new project wizard where you can specify if the model is 2-D or 3-D, setup headers and job information as well as the type of structure, frames or trusses. Setting up a standard frame or structure is very easy through—you guessed it—a wizard. As shown in Figure 2.1, many pre defined structures are available. And because plate elements exist in Visual Analysis, a user can model floors, tanks and other plate structures. Creating members is as easy as dragging your mouse—and by setting an adequate size grid, nodes snap to it easily. Once the members are created you can assign properties to a members or a series of members by double clicking or selecting member or node numbers from the wizard, as shown in Figure 2.2. A standard AISC

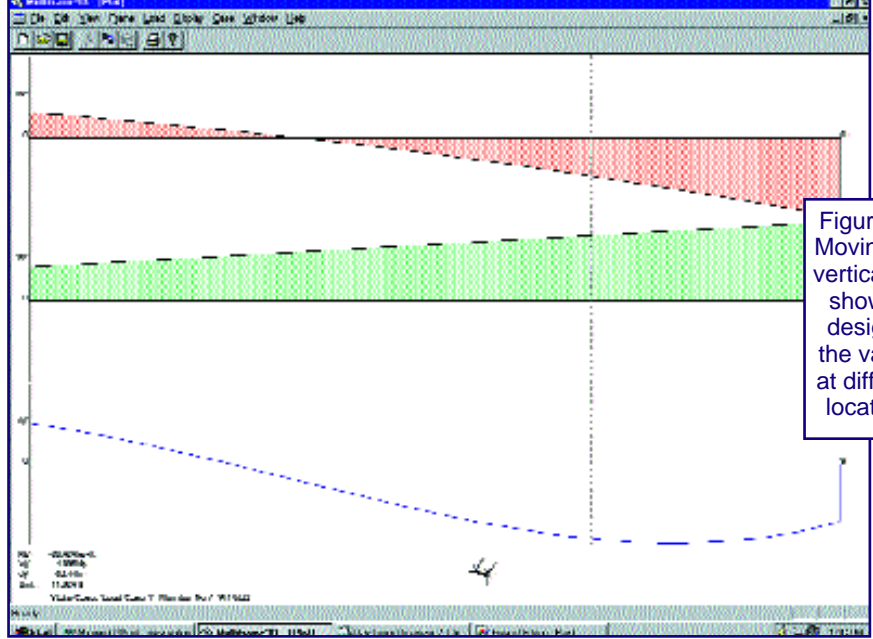


Figure 1.7
Moving the vertical line shows a designer the values at different locations

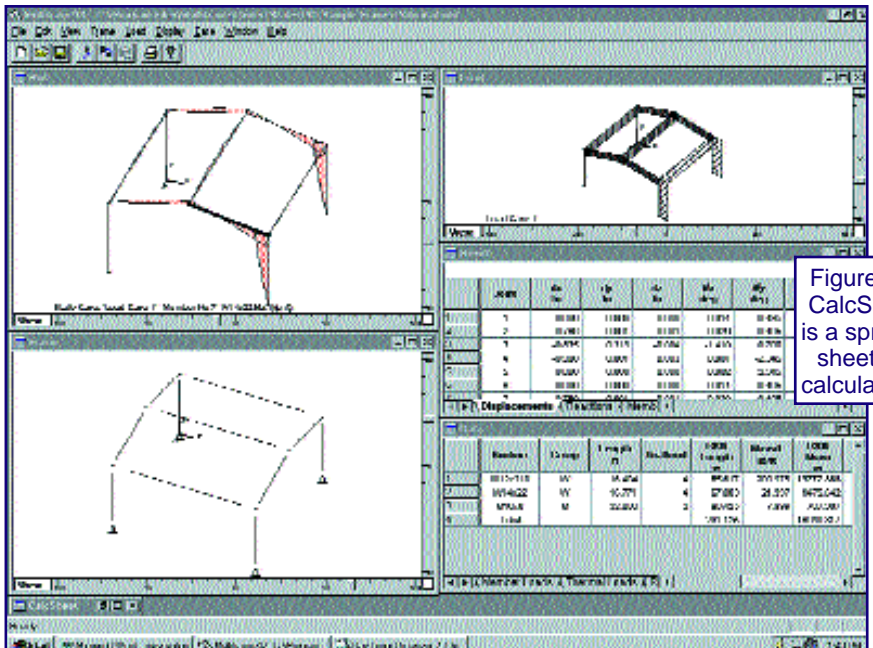


Figure 1.8
CalcSheet is a spreadsheet for calculations

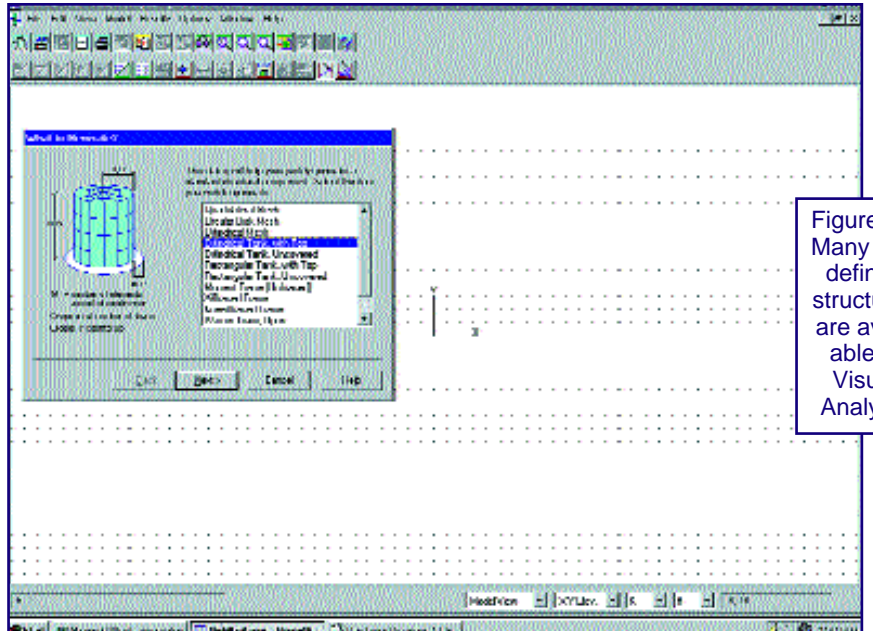


Figure 2.1
Many pre-defined structures are available in Visual Analysis

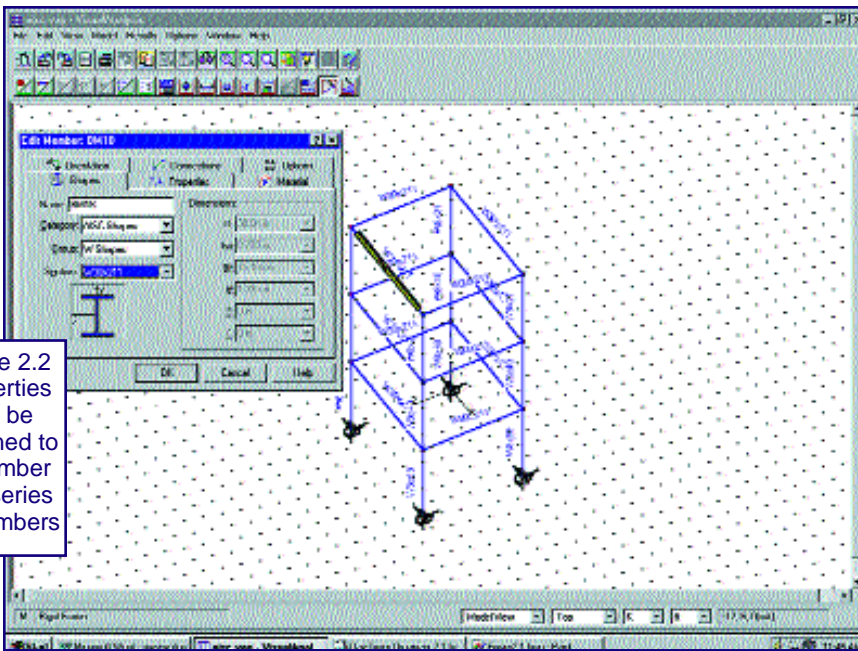


Figure 2.2 Properties can be assigned to a member or a series of members

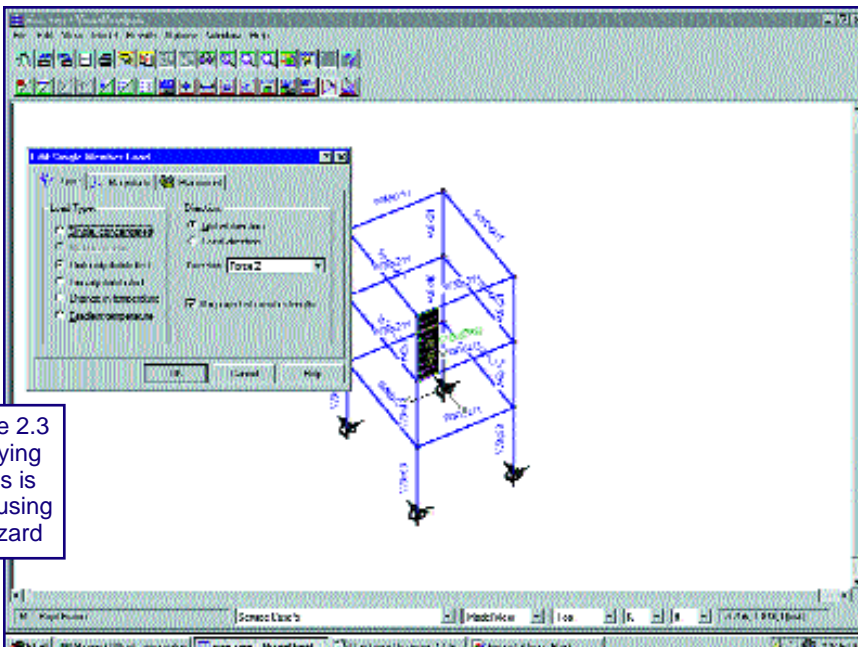


Figure 2.3 Applying loads is easy using a Wizard

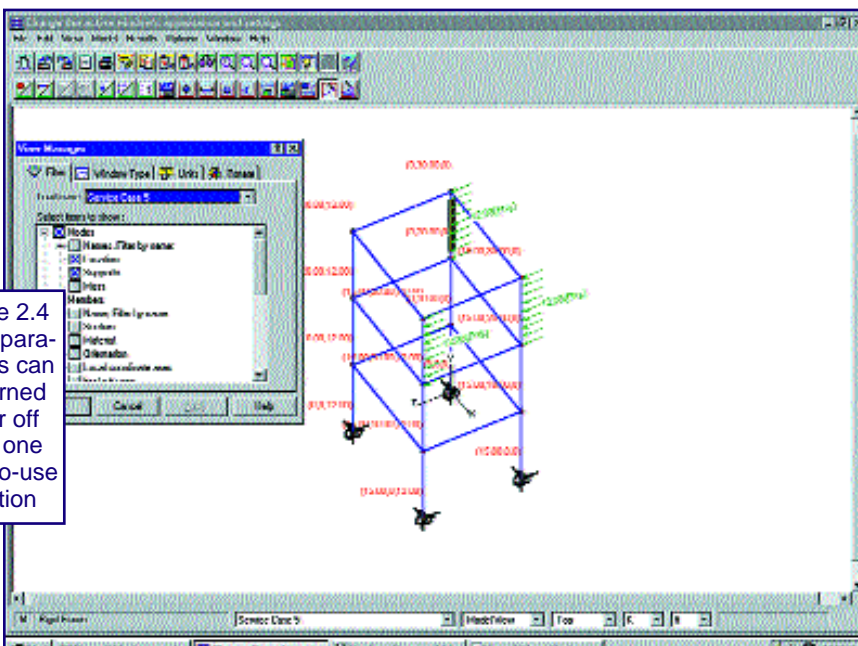


Figure 2.4 Many parameters can be turned on or off from one easy-to-use location

database of shapes is available including W-shapes, channels, tees and angles. Besides drawing members and plates in the model edit window one defines loads and load cases. The loads are termed as service loads and the load cases, which are consequences of service loads, are termed response cases. Again, they are very easy to setup using a wizard.

The only built in load generation capabilities are seismic loads; any other loads need to be computed, reduced and applied to the frame. Applying loads is easy, again through the wizard. Figure 2.3 shows the wizard and a load I applied on one column in the z direction. A couple of nice things about the loads wizard: modal results of seismic loads can be combined in several ways (CQC, SRCC, etc.) and LRFD combinations can be generated automatically, which makes selecting members from the AISC Manual direct. Unfortunately, the version I had did not have steel design capabilities. Before I leave the model window, I think the View Manager is worth mentioning. It is one of the easiest ways to manage views I have run across. The user can turn on and off many parameters from one easy-to-use location, as shown in Figure 2.4.

Analysis. On to analysis. Analyzing the structure is simplified by a wizard that walks the user through the steps. Once the analysis is complete, the user can view the results of every service load and every combination of service loads one has asked for. I asked for LRFD combinations and a first order static analysis, Figure 2.5 shows the results for every equation created by Visual Analysis. You can focus on a combination and using the handy view manager display a displaced shape, moments, axial forces and a variety of stresses resulting from the load condition as shown in Figure 2.6. Last but not least, a series of reports can be generated and

customized similar to the one shown in Figure 2.7.

In short these two programs are very easy to use and are impressive. With today's computational power, I still wish we could have analysis packages that include complete code load generations, complete steel design and non linear analysis for seismic loads. I know most of us are still using ASD but for the others maybe the day will come. I hope you've enjoyed this first year of software reviews and I look forward to revisiting some programs and to expanding my horizon to other type of software packages relevant to steel design.

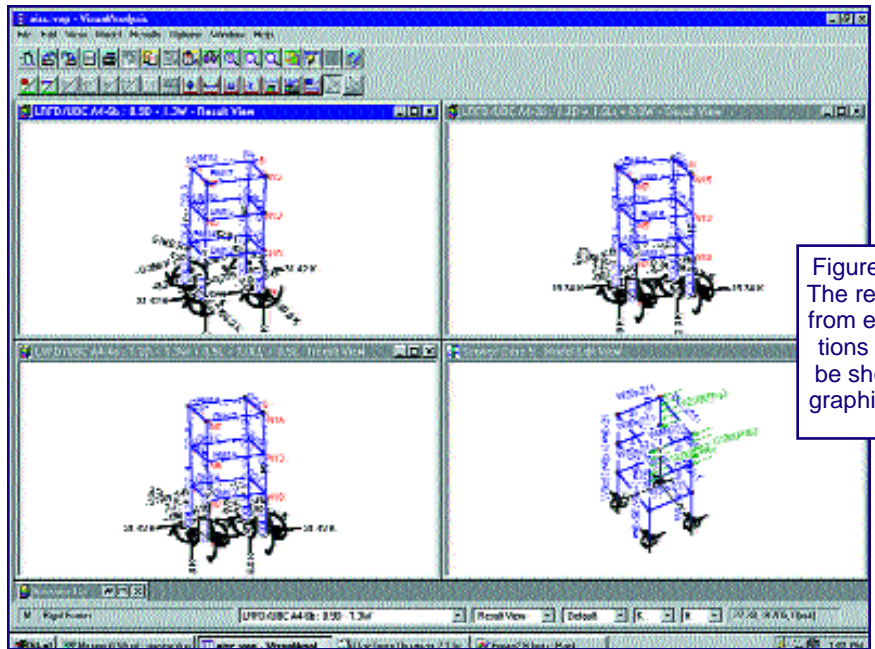


Figure 2.5
The results from equations can be shown graphically

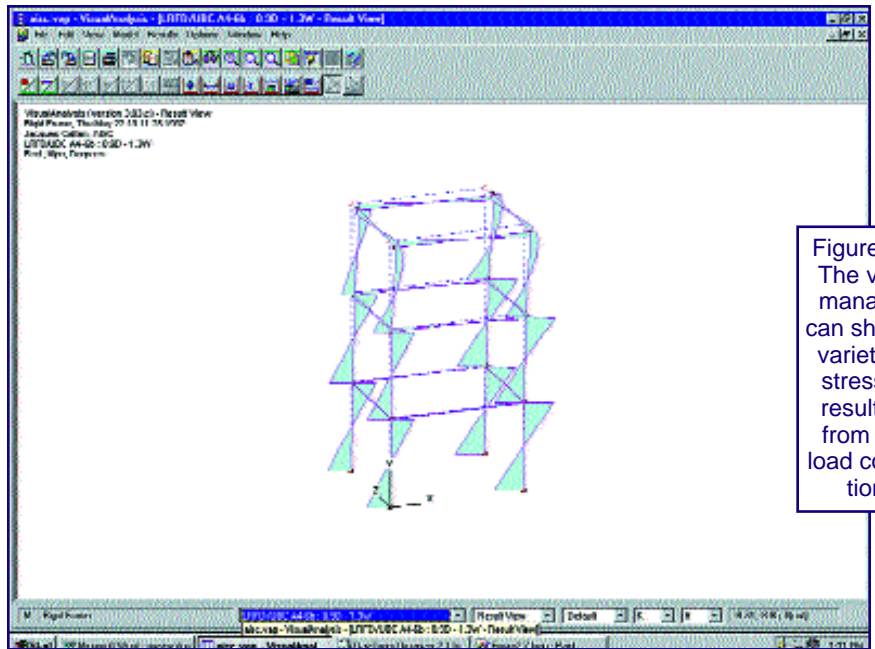


Figure 2.6
The view manager can show a variety of stresses resulting from the load condition

**PROGRAMS REVIEWED IN
JANUARY 1997**

ETABS
(Computers & Structures, Inc.)
Please circle #31

RISA 3D
(RISA Technologies)
Please circle #40

ROBOT V6
(Metrosoft Structures)
Please circle #51

STAAD-III
(Research Engineers, Inc.)
Please circle #34

STRAP
(ATIR Engineering Software)
Please circle #28

WinSTRUDL
(C.A.S.T.)
Please circle #27

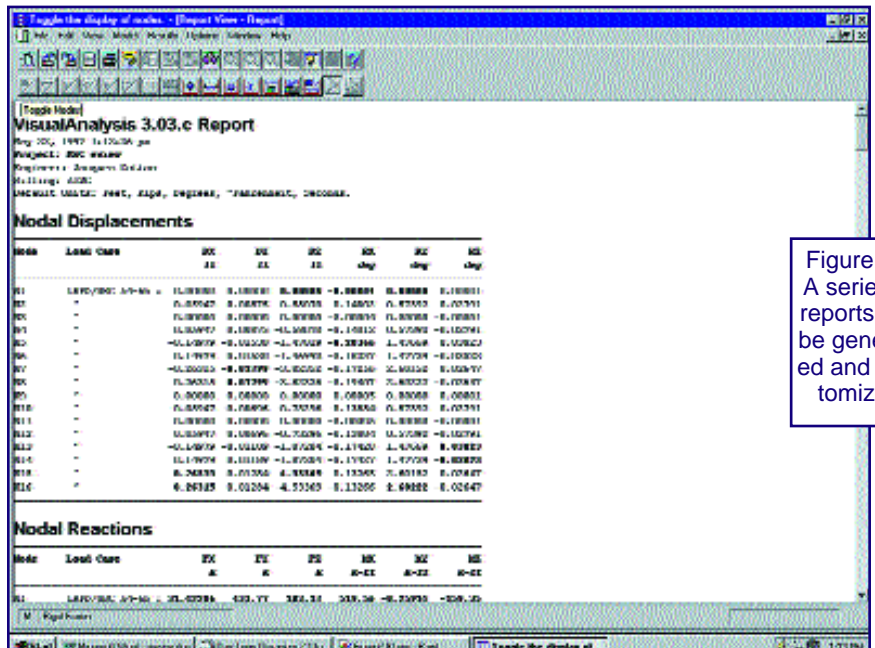


Figure 2.7
A series of reports can be generated and customized