Mechanical Engineering News

FOR THE POWER, PETROCHEMICAL AND RELATED INDUSTRIES

COADE ENGINEERING SOFTWARE

VOLUME 26 DECEMBER 1998

COADE, Inc. and Research Engineers Plan Merger

By Richard Ay and Tom Van Laan

In early August this year Research Engineers, Inc. (REI) approached COADE, Inc. with a proposal to merge the two companies. REI, a public company, is a world leader in PC-based structural steel design and analysis with programs such as STAAD/Pro. REI intends to build the world's source for plant engineering software and a union with COADE is a major step in that direction. Initial negotiations resulted in a Letter of Intent (LOI) being signed on 17 August. The LOI is posted on our web site in the Press Release section.

REI recognizes COADE's success and position in the plant engineering industry and wishes to maintain that culture in the new company. COADE, too, wants to maintain its position both philosophically and geographically. This situation prompted COADE management to reflect on what works for COADE and why it works. To that end, the resulting COADE shareholder's agreement to sell to REI includes several paragraphs on how COADE business will continue in the new organization. **> continued on p.2**

IN THIS ISSUE:

TANK Version 2.00 includes updated codes and Windows.

> see story page 2

New releases of PVElite and CodeCalc scheduled for January 1999.

All COADE programs are now available as native 32 bit Windows applications.

CAESAR II Version 4.10 introduces fatigue analysis.

> see story page 18

The COADE Mechanical Engineering News Bulletin is published periodically from the COADE offices in Houston, Texas. The Bulletin is intended to provide information about software applications and development for Mechanical Engineers serving the power, petrochemical and related industries. Additionally, the Bulletin serves as the official notification vehicle for software errors discovered in those Mechanical Engineering programs offered by COADE. (Please note, this bulletin is published only two to three times per year.)

COADE, Inc. and Research Engineers Plan Merger

> continued from p 1

Says COADE president, Tom Van Laan, “Whether customer, dealer or employee, I know what you want to hear - ‘Don't rock the boat’ and now we have that in writing. But at the same time, the true benefit to all from this merger derives from the integration of our operations, retaining our common strengths, and eliminating our individual weaknesses.”

The resulting agreement to sell (also on our web site) was signed by both parties on October 15. The deal should be complete by the end of 1998. The management team at COADE believes the acquisition by REI will result in an overall-stronger company with greater development, service and growth opportunities.

CAESAR II Version 4.10

By Richard Ay and Tom Van Laan

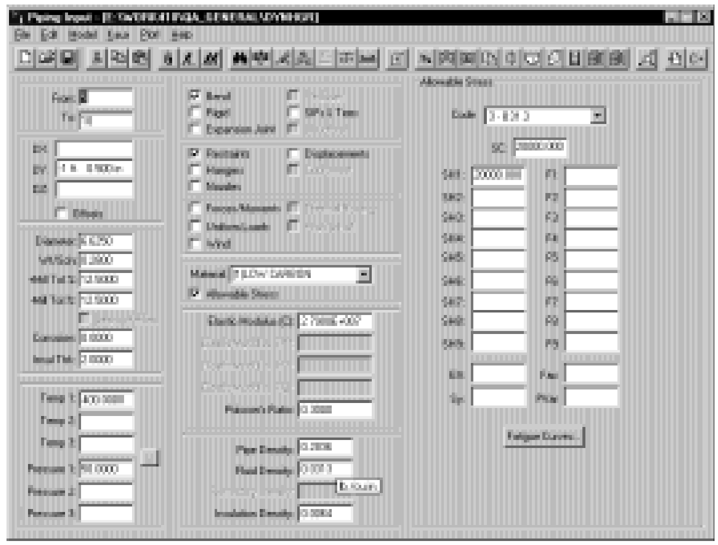
Work on **CAESAR II** Version 4.10 began as soon as Version 4.00 shipped. Version 4.10 is primarily a technical update, dealing with the updates to several codes, and the finalization of the TD/12 (British Gas Transmission) code. A by product of these enhancements is the ability to perform fatigue analysis, and handle up to nine different temperatures and pressures.

The major enhancements to the software include:

- 9 temperatures, 9 pressures, 9 displacement sets, and 9 force/moment sets
- Update of piping codes (CODETI, NC, ND, B31.1, B31.3)
- Finalization of TD/12 piping code
- Fatigue capabilities including cumulative damage
- Increase in number of load cases to 99
- Reactivation of the input LIST facilities
- Printing capabilities for graphical renderings
- Saving graphics images to BMP files
- On-Line User’s Guide and Quick Reference Guide in PDF format
- Plotting of structural restraints

> As a result of the recent republication of ASME Section VIII Division 1, and the changes incurred, both **PVElite** and **CODECALC** will be updated for a January 1999 release.

Additional enhancements have been made throughout the program. These improvements include: tool tips showing the expected units for each field in the piping input processor, a 30 minute input save reminder has been added, and optional on-line registration has been added. The figure below shows the revised input spreadsheet with the units tool tip for fluid density activated.



These tool tips are activated by positioning the mouse cursor over the input cell for a few seconds. The figure above also shows the expanded input fields for the allowable stress and the cyclic reduction factor, which correspond to the nine possible temperature cases. The “…” button to the right of the “Temp 3” field is used to bring up a dialog box where the nine temperatures and pressures can be specified.

TANK Versions 1.60/2.00 (Windows)

By Richard Ay

The **TANK** program has been revised to include the latest Addenda for API-650 and API-653. For API-650, Addendum 4 was published in December 1997, while for API-653, Addendum 2 was published in December 1997. These addenda necessitated a number of changes to the software, as listed in the tables below. Additionally, a number of modifications have been made as a result of user requests and code interpretations. These changes are also noted in the tables below.

API-650 Changes:

- The material database has been updated to reflect the changes to Table 3-2. This involved removing both A442 materials.

- Appendix F no longer forces a redesign utilizing Appendix A.
- Appendix I changed the equation for the maximum deflection by raising a term in the denominator to the 3rd power.
- Section 3.4.2 has been incorporated, which insures that the bottom plate diameter is at least D + 2 inches.
- A modification has been made to the implementation of Section 3.5.2, to include the bottom shell course thickness.
- Allowances have been made to enable metric jobs to utilize 6 mm plate as the minimum thickness instead of 0.25 inch plate.

API-653 Changes:

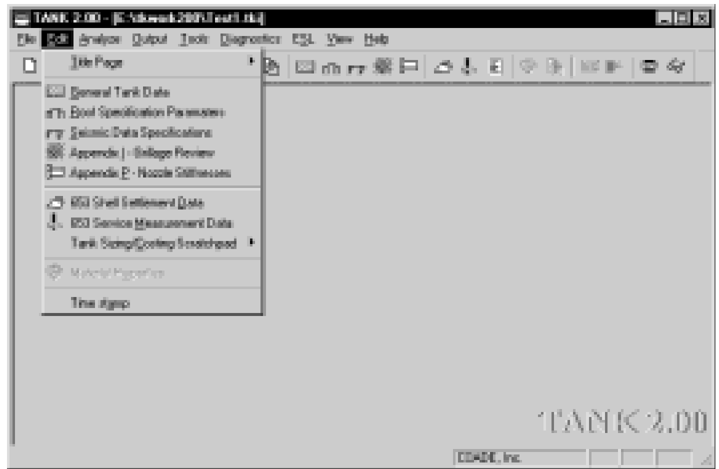
- The equations for determining the allowable stress have been modified as per the recent addendum.
- Incorporated a recent Code Interpretation stating that the Appendix M reduction factor should be applied to both terms in the allowable stress determination.
- Modified the basic thickness equation in accordance with the recent addendum (it no longer subtracts 1 foot from the fluid height).
- Incorporated the new computations for the allowed hydrotest height.
- Modified the allowed settlement measurement points from 30 ft to 32 ft around the circumference, in accordance with the recent addendum.

The next release of **TANK**, Version 1.60, is scheduled for December 1998. Version 1.60 will include all of the changes listed in the tables above. ***Users should note that this will be the last DOS version of TANK***. All subsequent versions will be native Windows programs.

Version 2.00 of **TANK** is the first Windows version, and is being released with Version 1.60. Both versions will reside on the same CD-ROM. Version 1.60 and Version 2.00 are technically equivalent. Both versions share the same features and capabilities, and both yield the same results. Input files can be passed between these two versions.

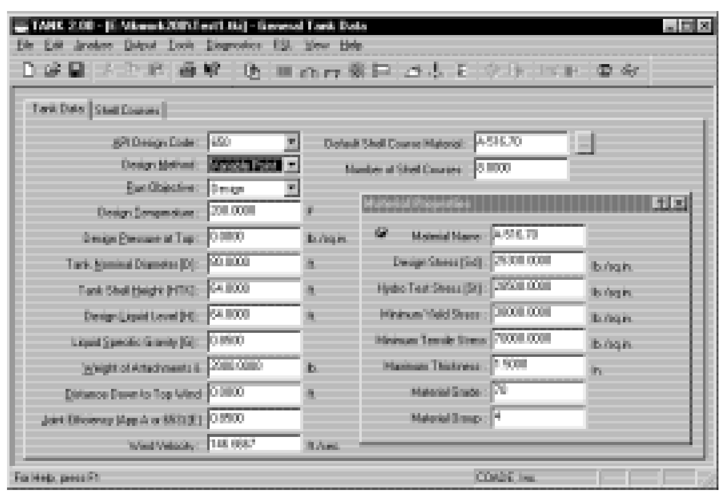
Version 2.00 is targeted toward Windows 95/98/NT 4.0 systems. Version 2.00 will not run under Windows 3.1x or Windows NT 3.51. Version 2.00 relies on standard 32 bit Windows components, which are part of the current Windows (95/98/NT 4.0) operating systems. These standard components allow improved functionality, in the areas of file management and data presentation. Version 2.00 also provides HTML help, which requires the presence of Internet Explorer to function properly.

The primary emphasis in the conversion of **TANK** to Windows was to keep the interface layout as close as possible to the previous DOS versions, while taking advantage of standard Windows components. Users familiar with the layout of the DOS version should feel comfortable with Version 2.00 immediately. The screen below shows the **TANK** Main Menu, with the input menu expanded.

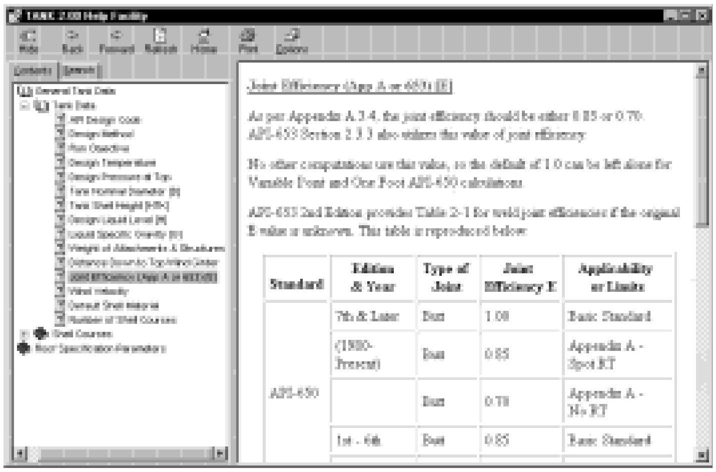


Where appropriate, tool bar buttons have been provided to allow quick selection of frequently used options. Each tool bar button corresponds to a text based menu item, as shown in the figure above. Selections can be made by pressing the tool bar button, or by picking the desired option from a menu.

When an input menu or tool bar item is selected, a "tabbed" dialog box is presented to allow user input. A typical input dialog box is shown in the following figure.

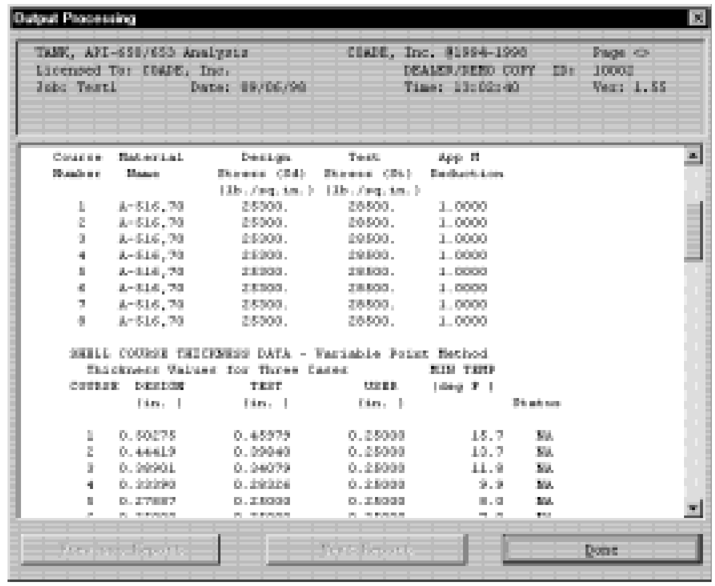


Each input item (edit box) is shown with descriptive text and the expected units. When this information is not sufficient, the on-line help can be activated. **TANK** Version 2.00 implements the HTML help system suggested for current software. (This help system requires the presence of Internet Explorer somewhere on the system.) A typical help window is shown in the figure below.

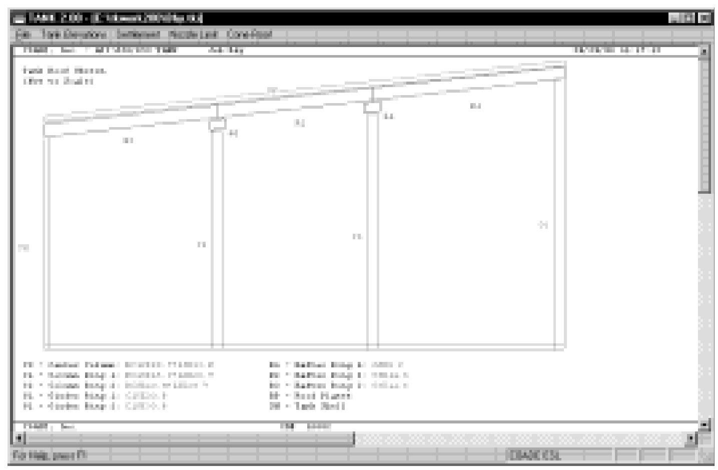


When appropriate, the help system refers to related sections of the API codes. This can be seen in the figure above. Note that the help system is organized in a hierarchical fashion, similar to a book. Each book (shown in the left pane above) represents a dialog box. Chapters represent tabs on the dialog, with pages representing each item. A built in search feature enables rapid navigation in the help system.

Once the input has been specified, and successfully passed error checking, the analysis can commence. Following the analysis, the results can be viewed in tabular report form, or graphically. The reports present the results segregated by topics, which correspond to the main input categories. As in the help system, the output reports reference code sections when appropriate. A typical output report is shown in the following figure.



As an alternative to text based report review, results can be viewed graphically for shell settlement, nozzle interaction diagrams, and supported cone roof design. A sample output graphic from a supported cone roof design is shown in the figure below.



COADE Hosts its First Dealer Conference

By Richard Ay

During the week of June 17th, COADE hosted its first dealer conference. Dealers from thirteen countries (and six continents) attended the three day conference, in COADE's Houston offices. Countries represented at the conference were: Australia, Brazil, Canada, China, England, France, Germany, Singapore, Italy, Romania, South Africa, The Netherlands, and the United States.

The conference began with a presentation on the strength and stability of COADE, the tremendous growth over the last four years, and future plans. The conference also detailed the activities necessary for COADE's day to day operations. This included the technical activities related to software development (development, QA, Beta testing, and technical support), sales policies and procedures, and marketing activities, including what COADE can do for its dealers.

The second and third days of the conference centered on COADE software products, their strengths, the current competition, and suggested sales methods. The conference wrapped up with a "round table" discussion forum to address dealer selected issues.

All attendees received a conference binder containing all of the presentations, marketing materials, forms, demonstration scripts, the demonstration CD, a conference CD, and a conference poster. The conference CD contains all of the presentations, example files, forms, documents, and the latest versions of all COADE products. (An engraved momento was also given as a token of our appreciation for attending the conference.)

After hours activities included a golf game, a cocktail party, a suite at the Astrodome to watch a baseball game, and several smaller gatherings. (The most exciting event of the conference occurred when many attendees found themselves trapped in one of the elevators in the COADE building. One hates to be late for a party.) These events and the day to day interaction proved to be valuable in building individual relationships between the dealers and the COADE staff members.

In order to evaluate the conference, attendees were asked to fill out an evaluation form. The results of this evaluation by the dealers reveals that:

- In general the conference exceeded expectations, everyone left satisfied and looking forward to the next dealer conference.
- Suggestions for the frequency of the conference ranged from every year to every four years, with most suggesting every other year. Several dealers requested an advanced agenda so that more people from different departments (sales, technical) could attend.
- Most dealers felt that COADE's office in Houston is the best place to hold this conference. "All facilities and resources essential to the conference are at COADE."
- Most dealers felt that four days would have been more suitable given the amount of information presented.
- In summarizing the usefulness of the presentations, "I found all presentations well presented and thoroughly relevant. The first day was most useful in terms of defining COADE philosophies, future visions, product emphasis, etc. This gives the dealers a framework and gives confidence when dealing with end users. The 'how to sell' sessions were equally relevant, but suffered due to lack of time."

- Dealers also commented on the benefits of discussing ideas and problems with other COADE representatives.

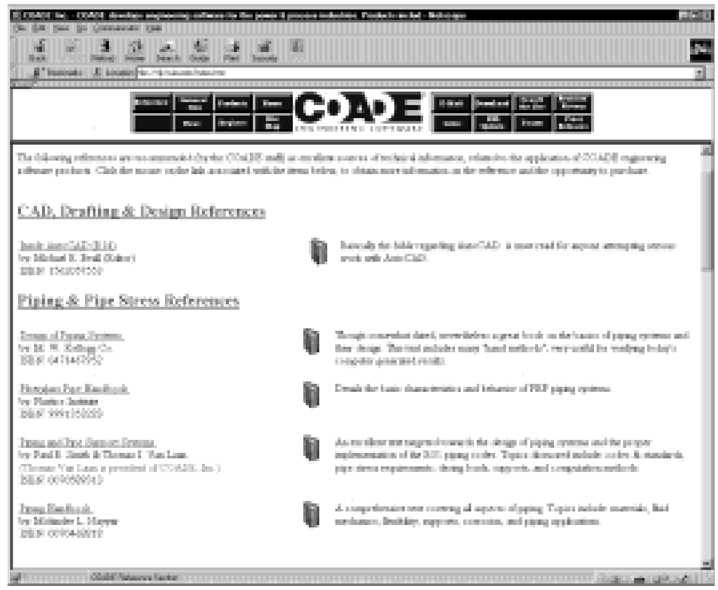
The entire staff at COADE would like to extend our thanks and appreciation to all those who attended this conference. The comments and suggestions made will be used in preparing the next conference.

COADE WEB Site Update

By Richard Ay

Usage of COADE's WEB site continues to increase, with the average number of monthly visitors approaching 3000. Many users have discovered this site to be an excellent source of news, information on software usage, and software updates. Usage of the discussion forums has also increased.

Recently, a new ***Reference Section*** has been added to the site. The creation of this section is a result of many requests for reference materials on the applications addressed by COADE software. This reference section lists those publications recommended by the COADE staff. Links are provided from each reference to Amazon's (the noted internet book seller) site for those interested in additional details or purchase information. This reference section is shown in the following figure.



The COADE WEB site also offers discussion forums. These forums are intended to allow users to offer their opinions about software usage and applications. These forums are a means to distribute information to our user base. Everyone is urged to contribute, when you feel you have something to offer on a particular topic.

Modeling Large D/d Tees

By Richard Ay

When building piping models, the modeling technique for tees is an important detail for the correct application of SIFs (stress intensification factors). The piping codes define the equations to be used in the determination of the tee SIF, based on the geometry of the fitting. Piping programs construct models using infinitely thin 3D beam elements to represent the pipes. Particulars such as diameter, thickness, elastic modulus, density, expansion coefficient, and length are just properties of these infinitely thin elements. ***Piping software does not have the concept of large or small diameter***. This is an important point, which if missed by the analyst can lead to erroneous stress solutions. (Note, according to B31.1 Section 104.8.4, moments are to be taken at the junction point of the legs, unless the designer can demonstrate the validity of a less conservative method.)

When modeling a piping system with branches, the analyst would typically continue down the header pipe, just flagging a certain node as a tee. Later in the model, the branch coding would begin at this tee node and continue on. Such typical coding is show in Figure 1 below.

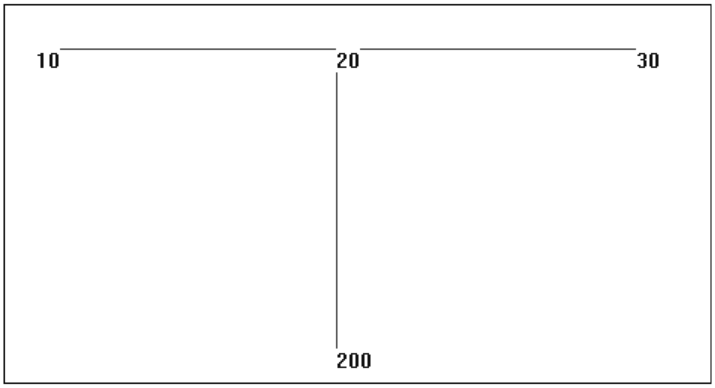


Figure 1 – Typical Tee Coding

From a “stress” point of view, the fact that node 20 is a fitting (a certain type of tee), is accounted for in the computation of the SIFs applied to the three elements 10-20, 20-30, and 20-200. The bending stress at node 20 is computed (for each of the three elements) and then multiplied by the appropriate SIF value. Note that the software sees the elements as depicted in Figure 1, i.e. infinitely thin sticks. In reality, the actual model from the analyst’s point of view is as shown in Figure 2.

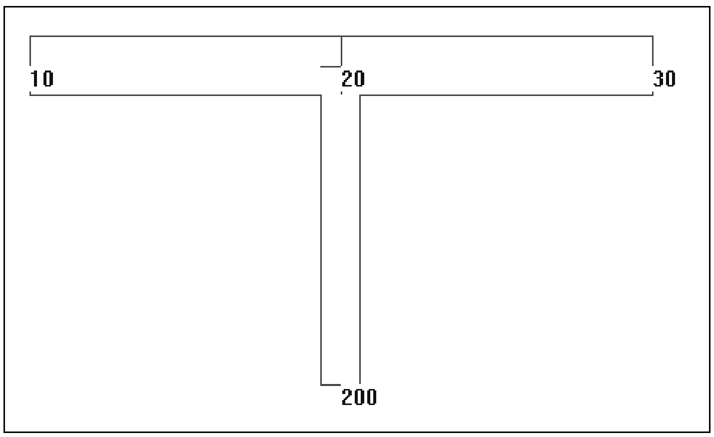


Figure 2 – Analyst’s View of a Typical Tee

Notice that in Figure 2, the three pipes frame into node 20. This is because the fitting is not modeled (typically) as a physical entity as it would be in a CAD system. For SIF calculations, all the stress program needs to know is what type of tee exists, and where it exists. Notice in Figure 2 the difference in diameters between the header and the branch pipes. The header is a 6” diameter, standard schedule pipe, while the branch is a 4” diameter, standard schedule pipe.

In Figure 2, the run pipe elements 10-20 and 20-30 both meet at node 20, as they should. However, the branch pipe, 20-200, also extends from node 20. In reality, the branch element starts at the surface of the header pipe (on offset of 6.625/2 and runs to node 200. So modeling to the tee center point introduces some *minor* error in the elemental stiffness matrix for the branch element. How much is *minor*, when does this become a problem that can not be ignored, and how can this problem be avoided?

As a general rule, as long as the distance from the tee node to the surface of the run pipe does not affect the overall stiffness of the model, the extra length of the branch can be ignored. Once the distance between the tee node and the surface of the run pipe is of sufficient length to affect the stiffness of the system, a more accurate model is called for. Items to consider here are: the length of the branch to a support, the distance down the branch to a point of high stiffness (valve or equipment), and the extra length of the branch when modeled to the center of the header.

For example, consider the configuration in Figure 3. Here we have a 24” diameter, standard schedule header with a 4” diameter, standard schedule branch. The offset distance from the tee node (1020) to the surface of the header is 12”, which is almost three times the branch diameter. If something else is modeled at node 1200, such as a guide, the moments on the branch may be inaccurate. Further complicating matters, the branch SIF will not be applied at the surface of the header, where it intersects the branch, but at the tee node 1020 (in accordance with the code recommendations).

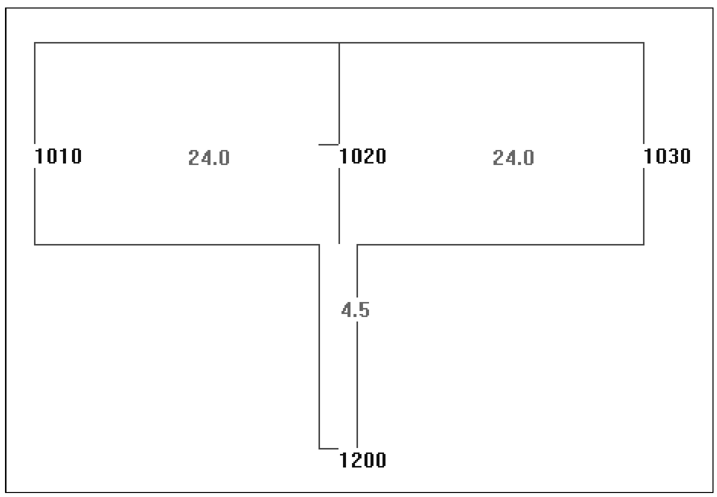


Figure 3 – A Large D/d Tee

One way to properly consider such a fitting is to break the branch element (1020-1200) into two elements, as shown in Figure 4. The first element, 2020-2025, is a dummy element – it does not really exist in the real world. However, 2020-2025 provides a connection point for the branch element, 2025-2200. There is also a location (2025) at which the SIF can be applied. (Note that dummy elements should be modeled as zero weight rigid elements, therefore only transferring forces and moments between their nodes.)

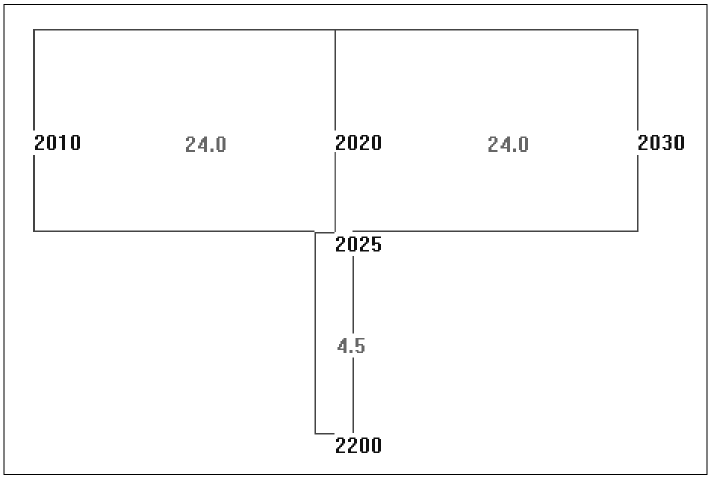


Figure 4 – A Large D/d Tee with Dummy Element

The steps to constructing the model shown in Figure 4 are as follows. First, code the model as if the tee at node 2020 could be modeled as normal (as in Figure 3). Once the proper tee type has been specified for node 2020, use the **CAESAR II** SIF Scratchpad to obtain the SIFs as per the current piping code. Make special note of the branch SIF values, they will be entered manually for the branch element.

Once the proper SIFs are known, break the branch element into its two components. The element 2020-2025 should be designated as a rigid with zero weight. On the element 2025-2200, indicate SIFs will be specified (i.e. check the check box). Specify node 2025 as the tee node, ***but leave the tee type blank***. Below the field for the tee type, specify the previously acquired values for the SIFs.

The model now contains the proper length for the branch element, and the correct SIF can be applied at the tee end of the branch.

How Code Case 2290 Impacts Vessel Software

By Scott Mayeux

The year 1998 marked several changes that are being made to the ASME Code that will effect the thickness requirement for vessels constructed according to the rules of the Boiler and Pressure Vessel Code. The major change is reflected in Code Case 2290. This Code Case was approved on June 17, 1998 and effectively increases the design allowable stress value S for many materials. In previous years, the design factor used on allowable tensile stresses was based on a factor of 4.0. For example, a material whose ultimate tensile strength was 70000 psi would have an allowable stress of (70000/4) or 17500. Of course the allowable stress for a material decreases as the temperature increases. Additionally, there are several other considerations such as yield, creep and fatigue that set the allowable stresses in certain temperature regimes. Code Case 2290 reduces the factor from 4.0 to 3.5. Now the same material will have an allowable of (70000/3.5) or 20000 psi. To see the difference this makes let's review the basic ASME formula for determining the required thickness of a cylindrical shell under internal pressure. It is as follows:

$T_r = (P * R) / (S * E – 0.6 * P)$ per UG-27

Where:

T_r = required thickness (in)

P = total pressure (psi)

R = corroded inside radius (in)

S = allowable tensile stress from Section II Part D at design temperature (psi)

E = applicable joint efficiency

In our case T_r = (1000.0 * 96.0 / (17500.0 * 1.0 – 0.6 * 1000.0) = **5.6805** inches

Using Code Case 2290 and an allowable tensile stress of 20000 yields the following:

Thickness Due to Internal Pressure (TR):
= (P*R)/(S*E-0.6*P) per UG-27 (c)(1)
= (1000.00 * 96.0)/(20000.0 * 1.0 - 0.6 * 1000.0)
= **4.9485** in.

The difference in thickness is (**5.6805 - 4.9485**) = 0.7320 inches. This decrease in thickness represents a substantial savings in weight and cost of the vessel. It should also be noted that this change to the allowable stress tables now makes Section VIII Division 1 much more competitive with foreign pressure vessel codes that have similar design requirements.

This Code Case also comes with some other changes. One change is to the hydrostatic test requirement. The previous test factor was 1.5 and has now been reduced to 1.3. If this change had been neglected, the vessel designed using Code Case 2290 may have been overstressed during hydrotest. Also, the Code Case suggests using the lower allowable stresses for items (such as flanges) where a slight distortion could cause a failure or leakage. Doing so will design a thicker, stiffer flange that would be less likely to leak.

Of course existing vessels may be re-rated using the new allowables. However, those doing so should be very careful. Critical aspects such as metallurgy and minimum design metal temperature requirements should be checked to insure compliance with the current Code requirements.

The Code Case can be obtained directly from ASME. Information on acquiring this case can be found at ASME's web site (www.ASME.org). This Code Case includes revisions to about 750 materials. If you are a current user of PV Elite 3.3 or CodeCalc 6.0, you can download the latest build from the COADE's web site. This build incorporates these revisions in the form of a new database and program execution files.

Conversion of Legacy P&ID Drawings Using CADWorx/P&ID

By Robert Wheat

Why are P&ID drawings so important?

For many decades, companies have been creating Process and Instrumentation Diagrams (P&ID) for the design and management of their chemical or process facilities. Standards have been created by groups such as the Instrument Society of America (ISA), which dictate the way we develop these drawings. P&IDs contain important diagrammatic information, the first source to which engineers and operations personnel turn for solving problems or increasing capacities. Engineering design cannot even begin without accurate P&IDs.

Prior to the mid-eighties, drawings were created with paper and pencil which provided nothing more than a visual aid. After that period, AutoCAD and the computer provided the world with the capability of creating these drawings electronically. From that period on, companies longed for the ability to store (and retrieve) more information within these drawings. In the beginning, blocks with attributes were used to store information that could be extracted in a rudimentary manner. This system could provide information from the drawing such as exact quantities of valves, instruments, line numbers and different types of process equipment. This provided engineers with an early advantage in the organizational aspect of a project.

Today, P&ID programs that work in the AutoCAD environment are expected to, and can, achieve much more. COADE's **CADWorx/P&ID** can actually store information in an external database, and the drawings can be modified from many different paths. Volumes of information can be stored, retrieved, created, viewed, and processed all from a central relational database and drawing system.

What should be done with old P&ID drawings?

Are all those old P&ID drawings that contain no more than blocks and attributes worth saving? Should they be redrawn with a new system such as **CADWorx/P&ID**? To redraw them would mean hours of drafting time that could lead to errors in an already perfect system. What about symbols that are represented in the old system that might not be in the **CADWorx/P&ID** program? To forget about the old drawing and start a new one seems like an awful waste. What are the options? With hundreds of attributes present in the drawing, is there is a better way to handle this? Yes, **CADWorx/P&ID**.

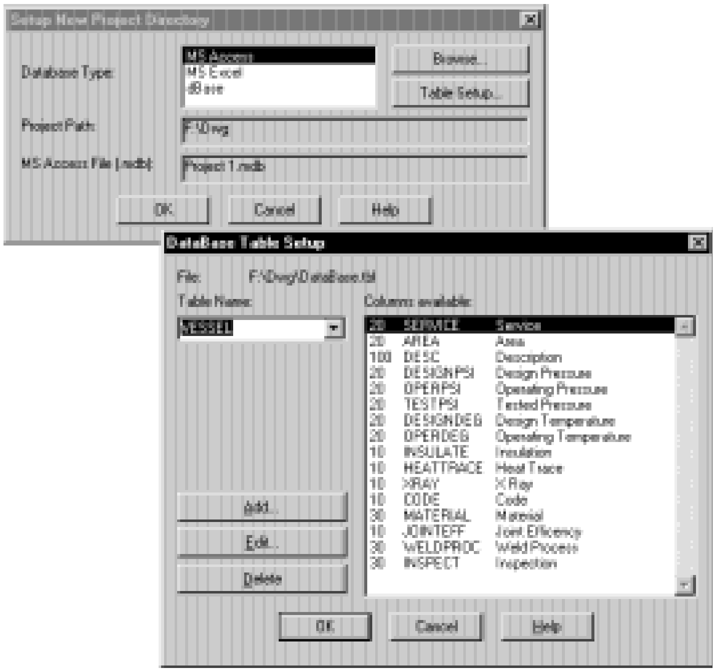
CADWorx/P&ID provides a very simple method of converting any present P&ID drawing into a system that works within its own environment. There are two issues that must be addressed. The first matter concerns the existing drawing and how to place information within it that allows conformity to the new **CADWorx/P&ID** system. The second is how to reuse existing blocks without causing problems for the new system. The process- experienced developers at COADE decided that the new system within **CADWorx/P&ID** would need to be dynamic for just those reasons.

Dynamic database structure…

The first problem is to deal with all the blocks present in the existing drawing. These blocks could have many attributes with labeled or assigned values. These drawings could be complete or may need modification. As is, they will not work with within **CADWorx/P&ID**.

The dynamic part of **CADWorx/P&ID** has to do with the database used. The first part of the database, or the tables to be more accurate, is fixed. But the second part of each table is dynamic in nature. Being dynamic means that the user can configure that portion of the tables in any way desired. Each column can have any name the user wishes within the table.

Now, this was the simple part. The developers at COADE decided to create a program that would associate the name of the column to the name of any existing attribute within any block. That means that when attributes are changed, the database changes. Change the database, and the attributes change. There nothing magical – just a simple association. The figure below shows the database table setup facility. This is used when the project is initially setup.



There are 8 different tables within the **CADWorx/P&ID** that the user has control over. These tables represent the major categories in any P&ID system. They are DOCUMENTATION, INSTRUMENT, LINE, MECHANICAL, MISC, NOZZLE, REDUCER VALVE and VESSEL. As shown above in the table setup, the vessel table can have any number of columns that can be associated with the attributes in the drawing.

Adding Extended Entity Data (xdata) and a row in the database…

Bringing in an existing drawing does not automatically associate the attributes with the columns in the tables. This requires running an additional command named *XDATAADD*. This command allows the user to choose which table will be used with the block and associated attributes. This command has 8 different options. The options are the same as the tables within the **CADWorx/P&ID** database.

> Command: **_XDATAADD**
> Line/Instrument/VAlve/MEchanical/Nozzle/Reducer/MIsc/<VEssel category>:
> Select a polyline, line, block or group: select a block or process line(s)
> Select objects: return to finish the selection

This function adds identification to the object selected and makes it recognizable to the program by placing an entry (or row) within the appropriate table. From then on, the user can modify the database or the drawing. If an object selected has labeled attributes, these values are placed into the database table.

What about the line or polyline that represents a process line within the existing drawing? These are not blocks. They do not have any attributes associated with them. The one piece of information that may be associated with these lines is the tag number.

Once again, the *XDATAADD* function can be used to attach identification and provide a row in the table. Utilizing the editing facility within **CADWorx/P&ID** allows the tag to be placed within the line. This part is not totally automatic, but provides an easy way of re-tagging the line so that it to will act as the blocks and attributes do. By utilizing the *TAGNUMBER* function, the line can have a tag number that reacts with the database table. In fact, the tag on the drawing reacts with the line as if it were the line. The edit facility within **CADWorx/P&ID** allows operation on these tags or the process lines. If anyone of the three is updated — tag, line, or row in table — the other two automatically update as well.

Combining process line segments into one…

Process lines can be in multiple segments. Does this capability add a row for each segment of an apparent single process line? Yes, it adds rows for each segment selected. The developers at COADE knew this would not be the desired representation. An additional command would be needed that allows the user to combine the line segments into a single process line. The *COMBINELINE* function allows individual process line segments to be combined into a single identification and table entry.

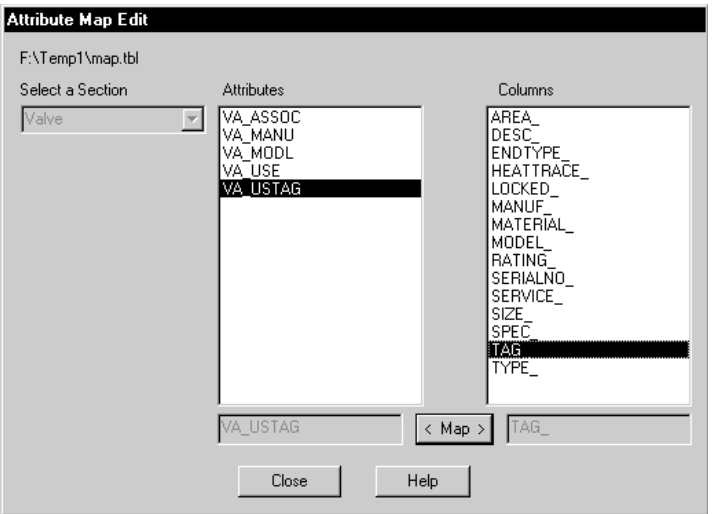
> Command: **_COMBINELINE**
>
> Pick lines in process direction...
> Select objects: *pick the starting end of the process line*
> Select objects: *pick the remainder with any selection method*

This combines all process lines segments selected into one. The user needs to go through each drawing and manually pick the process line segments that need to be represented as one.

Additional mapping facility…

Mentioned earlier was reference to the fixed portion of the database table which contains some of the most important associations desired (SIZE, TAG, SPEC, etc.). Unlike the dynamic portion of the database mentioned above, this table is fixed and cannot be modified. What if the user wants to associate an attribute in the existing P&ID drawing with one of these column names? What if the user does not want to use the same name for the column and the attribute? **CADWorx/P&ID** addresses this concept also.

When the *XDATAADD* function is used, it looks at the block and checks to make sure that all attributes have been associated with table column names. If an attribute is found where a match does not exist, a dialog is used to warn the user. This dialog allows the attribute to be mapped with any column name available within the table chosen. This dialog is shown below.



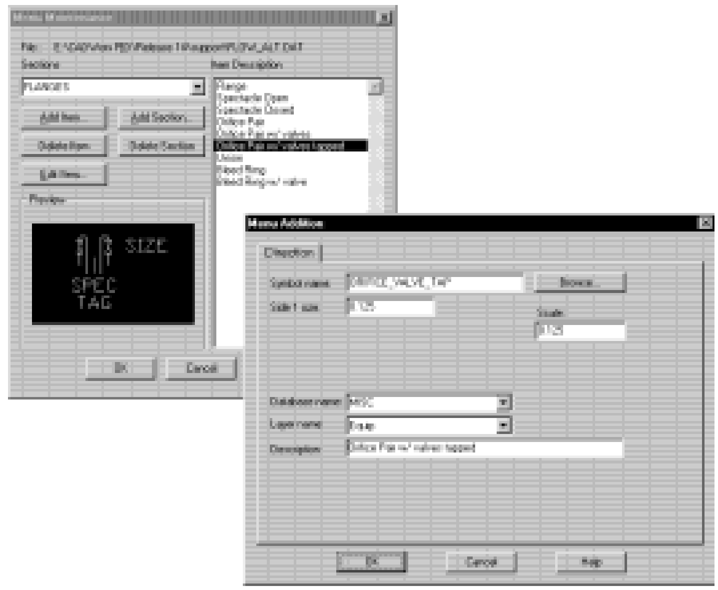
In the upper left part of the dialog, the *Table* combo box is labeled the same as the option selected on the command line of *XDATAADD*. The *Attributes* list box provides the attributes which need mapping. The program automatically finds the attributes that are not associated with any column name. The *Columns* list box provides all the column names available within the selected table. All the user is required to do is pick the attribute in the left list box and pick a corresponding column name in the right list box. When this selection has been made, the *Map* button at the bottom makes the association. All these mappings are stored in the project directory in a file called MAP.TBL. This file can also be edited by using the MAPEDIT function.

Mappings are permanently stored as mentioned above. Therefore, the mapping dialog does not appear again as long as the attribute has either the same name or is mapped to an appropriate table name. This allows the fixed and dynamic portion of the database tables to be mapped to any other P&ID AutoCAD drafting system used today or yesterday. Match the dynamic portion of the database table names to the attribute names or make sure that the attribute names are mapped to either the fixed or dynamic portion of the database tables names. Could it be any easier?

What about all the existing blocks…

Now, icing for the cake. A mapping facility is automatically invoked when xdata is added to the block as mentioned above. A mapping edit function is also available which allows attributes to be associated with database table column names as the user wishes. But what about using the blocks and associated attributes from another P&ID drafting system while inserting them into the drawing with the **CADWorx/P&ID** system? This, too, is possible.

CADWorx/P&ID allows the user to easily add blocks to the system with the *MENUSYMBOL* function. The facility provides dialogs that allow the user to place any block for use within the **CADWorx/P&ID** program. There are 9 different types of insertion routines that are used by the program, which are all available for use with the other program's blocks. The dialog below shows an example of this facility.



If the program tries to insert a block that has attributes that do not have the same names as the database column names, the program searches the MAP.TBL to find proper associations. When found, it associates this foreign attribute into the **CADWorx/P&ID** system as if it were its own. And as mentioned above, it will update in the drawing if the database is changed or vice-versa. No other program can achieve such simplicity for the user.

Summary…

Even without database use, the program operates similarly to the process outlined above. The only difference is that there are a limited number of fixed fields available. Regardless of whether databases are used, **CADWorx/P&ID** can read and write the existing P&ID drawings much the same way.

To the developers of **CADWorx/P&ID**, dynamic structures were the only solution available that would make this product competitive. Simplicity, as with all COADE products, is the key to making this easy to operate. Software products which are easier, and more flexible to use will eventually replace those without such simplicity and configurable characteristics.

A Survey of Impact Load Analysis in CAESAR II

By David Diehl

Introduction

Impact loads on piping systems can be evaluated by a number of methods in CAESAR II. Usually a preferred method is chosen and the analysis is run. Few opportunities exist to take the time to run through all the methods to evaluate the benefits and shortcomings of each. This article will review one common impact load – a relief valve discharge – to introduce such a study.

Impact loads on or in piping systems are not uncommon. Relief valve discharge, water- or steam hammer, and slug flow are typical examples. In each case a force is applied at a point for a period of time and then removed. The force in relief valve discharge is the jet force on the relief piping or vent stack. The force in hammer loads is the temporary pressure imbalance on upstream and downstream elbows as the pressure wave flies through the run connecting the elbows. The force in the slug flow problem is the force required to change the momentum of the much more massive liquid in a vapor line at each change in direction.

The system response to this dynamic load can be less than, greater than, or equal to the static forces mentioned above. The maximum dynamic response at any time to an impact load is twice the response to the static load. This ratio of dynamic response to static response is termed the dynamic load factor or DLF. The DLF reflects the relationship between the timing of the event – its attack time or ramp up and duration – and the dynamic characteristics of the piping system – its natural frequencies. A single event, then, will have a varying effect on different systems and these DLF's can be plotted for a range of frequencies. This is what CAESAR II produces in its response spectrum generator. If you know the static load and the "significant" frequency of the piping system, you are well on you way to evaluating the system response to an impact load.

I chose a relief valve discharge for this analysis survey. Relief valve manufacturers produce values for the steady state thrust load on their valves. They also define the opening times for their valves. Such clean information is harder to come by when defining rapid valve closure for hammer loads or slug size for change in momentum calculations. Additionally, the Power Piping Code, ASME B31.1, provides guidelines for evaluating relief valve loads in Appendix II – Nonmandatory Rules for the Design of Safety Valve Installations.

No matter what approach is used to evaluate this impact load, we will need to define the magnitude of the static load exerted on the relief piping. CAESAR II has a thrust load generator to do this. The calculations are based on the thermodynamics of the escaping gas in an open discharge system. It was expanded from a steam application into a general liquid and gas application. It's useful when you do not have better data available but it requires you to enter several valve and fluid properties that may also be unavailable. One nice feature, though, is its ability to size the diameter of the vent stack to avoid blowback at the bottom of the stack. We will take the easy way out here and use the "total outlet reaction force" listed in the manufacturer's catalog.

After a review of the types of impact analysis available in CAESAR II, I will build up an example, compare the results, and draw some conclusions.

What methods of analysis can we use?

Static equivalent method

Simply take the static thrust load and multiply it by the dynamic load factor (DLF) for the frequency of interest and apply that load in a static analysis. Its that simple. Well, part of it is simple. We're getting our static thrust load directly from the manufacturer. But how do we get that DLF? In the worst case the DLF would be 2 so two times the static load would be a simple, conservative approach. Appendix II provides some assistance here. Figure 3-2, "Dynamic Load Factors for Open Discharge System", (see Figure 1) charts a dynamic load factor against the term t_o/T, where t_o is the opening time of the relief valve and T is the period of vibration of the piping system. Now 1/T is frequency, so what we have here is system response for a range of frequencies as a function of the valve opening time. The manufacturer gives us the opening time so we can almost get a DLF directly from this chart. What we're missing is the frequency or period of the significant mode of vibration – the mode that will be excited by this event. Until we get a better number we will stick with the conservative DLF of 2.

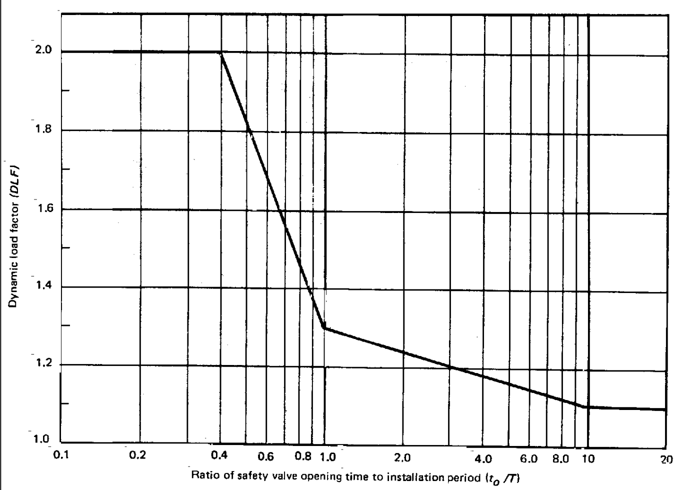


FIG. 3-2 DYNAMIC LOAD FACTORS FOR OPEN DISCHARGE SYSTEM
This Figure Is Based on Curves From *Introduction to Structural Dynamics,* J. M. Biggs, McGraw-Hill Book Co., 1964

Figure 1

There are some conditions on using this approach. B31.1 says it well in Appendix II Paragraph 3.5.1.3: "For structures having essentially one degree-of-freedom and a single load application, the DLF value will range between one and two depending on the time-history of the applied load and the natural frequency of the structure." The code goes on to work an example where the valve sits on a rigidly supported line. This eliminates the line's participation in the event and it is only necessary to determine the characteristic frequency of the valve assembly alone, independent of the connecting pipe. That's not what we have here. Our line is not rigidly supported and it will participate in the response to the event. The next question, then, does this system respond as would a single degree-of-freedom (DOF) system? Certainly this system is a collection of mass points each with their own DOF's so you can't say that this system has only one DOF. But, as we will see, this approach holds true if we replace the phrase "one DOF" with "one mode of vibration".

Response spectrum method

If more than one mode of vibration is excited by the event then the simple static equivalent approach may not be conservative. The response to each mode of vibration must be evaluated and then combined to approximate the total system response. The response spectrum method is suited for this task. The response spectrum method is commonly used for seismic analysis and we adjusted it in CAESAR II to apply to impact loads as well. Here, each mode's response is calculated based on its own DLF. These modal responses are combined by the absolute (ABS) or the square root sum of the squares (SRSS) methods to determine the total system response to the impact. This combination method washes away any semblance of structural reality but it does provide a good "statistical" view of the system response.

With a more sophisticated analysis comes more data requirements. For a response spectrum analysis we need a response spectrum. For seismic analysis the response spectrum takes the form of ground response (e.g. ground acceleration) as a function of frequency. For our impact load we will need a DLF as a function of frequency. How detailed will this data be? We can dumb it down to a constant value of 2 for every frequency of vibration. That may be better than the static equivalent load since the modal analysis also considers the location of the load in determining each mode's response to the event[1]. A flat DLF of 2 across the entire frequency range will allow each mode to respond at its own level. But a flat DLF of 2 is not sensible. Lower modes of vibration may be too flexible to reach this maximum of 2 and higher modes may be too rigid to respond this way. A typical force response spectrum starts at a DLF of zero (at an extremely low frequency), builds up to a possible maximum of 2 and then settles down to a DLF of 1 at high frequencies. The rise time and overall duration of the transient event set these trends in the response spectrum. Figure 3-2 in B31.1 Appendix II follows these trends. The low frequency, flexible response is ignored[2] but the drop from a DLF of 2 to a conservative 1.1 is set by the opening time of the valve. If you know the opening time of the valve, Figure 3-2 can be used to develop a more realistic force response spectrum for the event. The valve manufacturer gives us the valve opening time.

You can also build your own response spectrum. CAESAR II makes this easy by providing a processor that converts an event time-history into a response spectrum. To complete a time-history of the event, we will have to know not only the opening time of the valve but also the overall duration of the event. How long will the safety relief valve stay open? A conservative approach here would be to run the event long enough so that even the lowest expected mode of vibration reaches a DLF of 2.

[1] I compare this to striking a glass with a spoon. The same event (hitting the glass) produces a different response (the sound) depending on where you strike the glass. The modal response and therefor system response is sensitive to the point of application.

[2] The drop in response at low frequencies is a function of the overall duration of the event. Figure 3-2 assumes the worst case where the event doesn't end. This approximation has no impact (excuse the pun) except on extremely low frequencies – frequencies not found in piping systems.

Time-history method

Modal time-history is the most complete dynamic analysis in CAESAR II. Several impact loads can be scheduled to occur at different times – something that cannot be done in the response spectrum method. This is what you need to model a hammer load or slug load traveling through a piping system. System damping is also considered in time-history analysis. Damping will reduce the response to the applied load. Time-history analysis maintains the shape of each mode's response to the event – you get pluses and minuses in the output. Time-history's true structural response is a great improvement over the statistical nature of the response spectrum results. The results are referenced in time. The results are constructed like a movie where each analysis time step represents another frame in the movie. Program output is created at set times through the analysis. These reports serve as snapshots of the system loads, displacements and stresses at these intervals throughout the movie. CAESAR II output also includes a valuable report holding the maximum response for each output value along with the time at which it occurs. We even show the "movie" through an animation of the system displacement through time.

But once again, this improvement comes at a price. The results are very event specific and sensitive to changes to the piping model and the applied transient loads. Finer results require a finer model definition. There are also choices to control the calculations. What is the analysis time step? How many modes should be included in the analysis? How much time should be included in the analysis? What is the damping? What are the report times? These questions must be answered before a good run can be made. Many of these input questions are addressed in a previous newsletter article – see the June 1994 issue.

What's the best approach?

A static equivalent has its place. In many cases a static equivalent works because only one mode of vibration is excited and that mode takes on the shape of simple cantilever bending. A static equivalent load that creates a similar cantilever-bending shape will closely approximate the modal response found in a dynamic analysis. The big problem with static approaches to dynamic simulation is that they focus on the applied load rather than the equally important dynamic characteristics of the system. This mistake leads to the obvious and expensive addition of supports to directly carry these loads rather than simpler modifications to the system's modal characteristics. Impact load evaluation using a static equivalent load is quick and easy but it does not highlight the dynamic relationship between the applied load and system response.

The response spectrum is easy and provides more clues for problem diagnosis. More input is required but the added results are worth the effort. A response spectrum is required to define the event. You can use Figure 3-2 from Appendix II to build your own response spectrum for relief valve analysis and you can convert any other time-history into a response spectrum using the generator found in CAESAR II. The impact or shock results will show which mode of vibration is the largest contributor to the overall response. If the stress is excessive, you can change the response by (usually) increasing the frequency of the offending mode of vibration. Adding a restraint at a point of large modal deflection will do the trick. CAESAR II helps by displaying this mode shape in animation to show where the movement occurs.

Time-history analysis is exact but exacting in its requirements. Time-history is best saved for well-defined events that have several loads timed throughout the system. Only time-history analysis can maintain the timing relationship between loads in the system. Slug flow and water hammer analyses are good candidates for this type of analysis. Once again, refer to the water hammer sample in the June 1994 issue of Mechanical Engineering News to see what's required.

An example

Relief valve subsystem

The piping system used in this analysis is rather simple. It consists of a 12 meter riser (yes, it's metric) with an 11 meter horizontal run and a relief valve near its midpoint. The relief valve piping is unattached to the vent stack. This has its good and bad points. Unattached, the piping is not restricted in its thermal growth. But lacking the restraint of a hard-piped vent stack, the valve piping can also move about when the valve opens. This freedom increases the system's response to the impact load perhaps to the point where the discharge piping no longer remains in the vent stack. The system is a 12-inch, extra-strong (XS), A106B pipe supported at the bottom by an anchor and by two spring hangers at the elbow and terminus. The system operates at 280C (535F) and 60 bar (855 psi) and has 100 mm of calcium silicate insulation. Although B31.1 provides the assistance, I'll still use B31.3 for the stress analysis. My limit for sustained plus occasional stresses is 1.33 times the hot allowable stress. See Figure 2.

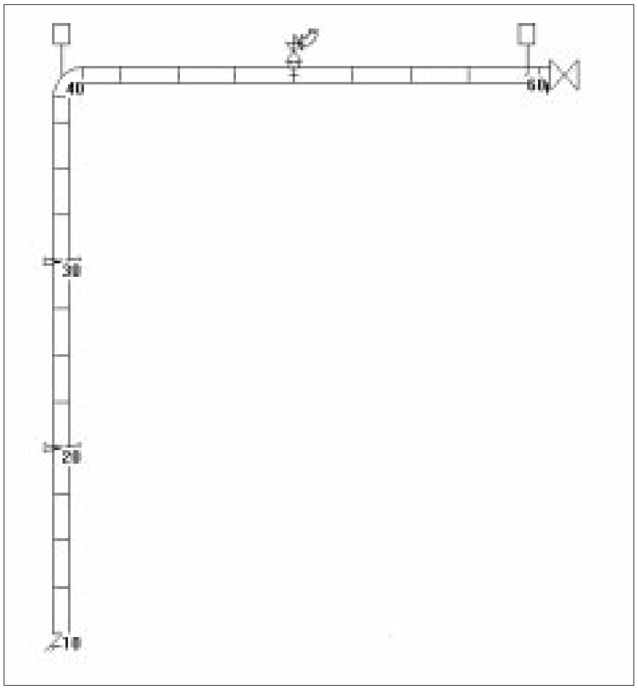


Figure 2

I made several adjustments to the model in an attempt to create more accurate results. I was concerned about the six-inch weld-o-let coding for the relief valve connection to the header. I put in a short piece off the 12-inch line to satisfy the tee specification in CAESAR II and ran a rigid, weightless element to the header wall. I also added more modes simply by using the Break command. This smoothed out the mass distribution for the dynamic analysis. I ran all dynamic analyses to 100 Hz. instead of using the default cutoff of 33 Hz. Extracting more modes will not take too much time and I may end up with lower, more accurate results because of the dropping DLF. I will leave it as a future exercise to evaluate the significance of these modeling adjustments. It would be good to go back and check to see if any of these changes were necessary so I'll know better next time.

The Crosby Safety Valve

The relief valve thrust load could be estimated using the thrust load generator in CAESAR II but I decided to pull that number directly from a manufacturer's catalog. I selected Crosby's Class 900 3 x L2 x 6 HCI ISOFLEX relief valve shown in Catalog No. 420. The total outlet reaction force at 60 bar is tabulated as 14,826 N. (3333 lbf.) The catalog also specifies the opening and closing time for these valves as 80 ms and 250 ms, respectively. The valve opening time sets the high frequency response and that's what's used in the Figure 3-2 chart to set the modal DLF. (See Figure 1.) The one term that's uncertain is the overall duration of the event.

The static equivalent analysis

I propose two static equivalent cases – one with the "dumb" 2 and another with a DLF calculated from Figure 3-2 in Appendix II. To get the Figure 3-2 value you need to know the opening time of the valve and the frequency of the excited mode of vibration. (If you need to run modal analysis to find the frequency you might as well finish the task with the response spectrum method.) The term t_o is the opening time of the valve and that is 80 ms. T is the period of the excited DOF or mode of vibration and 1/T is the frequency of that mode. Peeking ahead to the response spectrum analyses, the first mode of vibration is the major contributor to the system response and that frequency is 1.3 Hz. Enter Figure 3-2 with a to/T of (0.080 * 1.3) or 0.1 to see that even the "smart" DLF is 2. Only one static equivalent will be necessary.

The response spectrum analyses

We will look at three response spectrum loads – one with a flat DLF of 2, one with the Figure 3-2 spectrum, and one with a spectrum generated directly from a time-history. The first spectrum assumes that the impact load will affect all modes of vibration at the maximum value of 2 or twice the static response. This response spectrum is unnecessarily conservative but it is a starting point. A more sensible and probably the preferred approach is the ASME B31.1 Appendix II response spectrum. Figure 3-2 can be converted to a response spectrum by dividing the horizontal axis by the valve opening time - t_o. In our case t_o is given in the Crosby catalog as 80 ms. The third response spectrum, based on a specific time-history requires more input – what is the total duration of this transient event? Does the valve stay open for a quarter second, one second, ten seconds or more than one minute? Not knowing that answer, I tested several of these events feeding the time-histories into CAESAR II's response spectrum generator to find the shortest duration that would still affect low frequencies of vibration. (See Figure 3.) Clearly, low frequency response is governed by the overall duration of the event. A one-second event reaches the maximum DLF of 2 at 0.5 Hz. Since no modes are expected below 0.5 Hz., a one-second duration will be conservative. The assumed time-history of the event, then, is 80 ms to open, 1000 ms at full open and 250 ms to close. The closing time has no influence on the frequency content of the impact load. This data is fed through the response spectrum generator to build the third shock analysis. These three response spectra are shown in Figure 4.

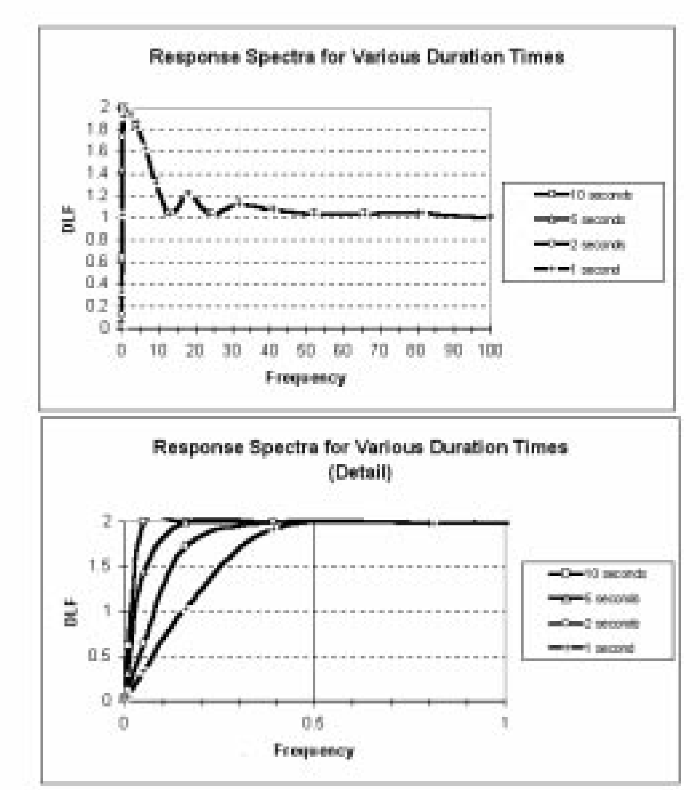


Figure 3

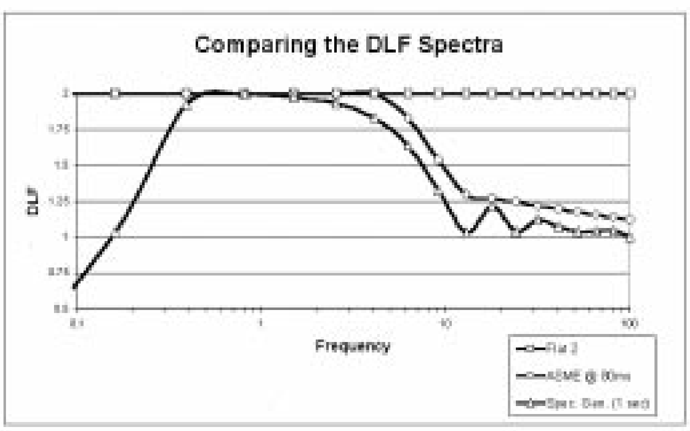


Figure 4

The time-history analysis

As illustrated above, a one-second duration is all that will be needed to fully excite a mode of vibration as low as 0.5 Hz. We will use the same event timing for the complete, modal time-history analysis. Longer events will not produce greater results while shorter events may have a reduced response. Time-history analysis also includes the effect of system damping. Using the suggested USNRC Regulatory Guide 1.61 will be helpful here since that guide defines damping terms for various structural systems in power plants. The guide lists 2% (0.02) critical damping for 12-inch or greater pipe and that's what we'll use in the analysis. Damping will reduce the overall response to the impact load. Any yielding in the system during the event may increase the damping to 3% critical.

System Evaluation

I have limited the results published in this article to what I believe is significant to the subject – a survey of impact analysis. Anyone running dynamic analysis knows that there's a lot of data that can blur the picture. I chose to examine the high stress points – the 12-inch elbow (nodes 38, 39 & 40) and the weld-o-let connecting the relief valve to the 12-inch line (node 50). In structural terms, I am concerned about the hanger displacement near the end of the run (node 60) and the alignment of the valve piping with the vent stack (node 550).

The stresses for each method of analysis are summarized in Table 1. These are the sustained stresses plus the occasional stresses from the impact load. With a maximum allowed stress of 168 MPa[3], the system is clearly overstressed. There are some trends that are interesting. First the elbow. The static equivalent stresses are highest and the response spectrum stresses that drop off as the DLF spectrum is refined. This is expected since, as the spectrum is refined, the higher modes' impact is reduced due to the falling DLF. What, perhaps, is not expected is the higher stresses in the time history analysis. I believe these stresses are higher because of the mode summation method. Time-history maintains the true modal displacements and it just so happens that at 437 ms, the major modes in this response are all squeezing the elbow together. The SRSS approach in the response spectrum method reduces this effect. Why did the static equivalent method work so well here? The results from the response spectrum method (not listed here) indicate that the response from the first mode of vibration is the major contributor to the elbow stress. The plot of the mode shape (see Figure 5) confirms that this shape is similar to simple cantilever bending, a shape that's matched by the static equivalent load. The weld-o-let is a bit different. The static equivalent load is exceeded by the flat DLF of 2. At the weld-o-let, the fourth mode is the major contributor and the shape taken by the fourth mode cannot be matched by a simple static equivalent load – **the static equivalent method may not be conservative for these "more complex" shapes**.

[3] The limit is k*Sh; here, k=1.33 and Sh= 126 MPa.

Location	2*static	Resp. Spec. - Flat DLF 2	Resp. Spec. - ASME	Resp. Spec. - from Time-History	Time History	Time to max. (ms)
Bend:						
38	219	192	189	187	203	437
39	204	179	176	174	187	437
40	188	169	167	165	174	437
Weld-o-let:						
50	165	171	151	142	149	100

Table 1 – Stresses (MPa)

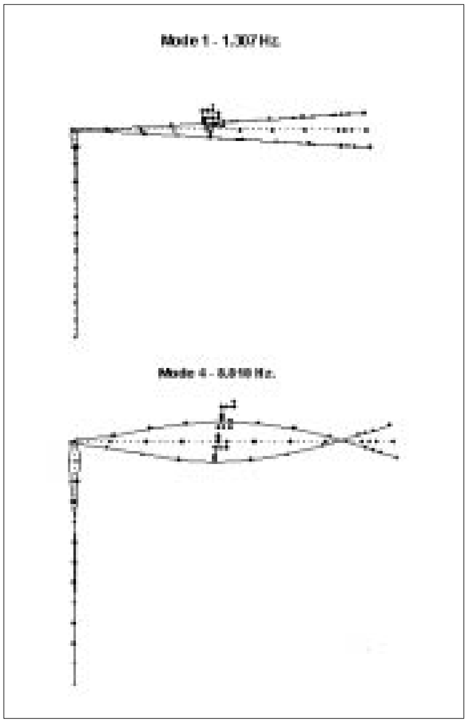


Figure 5

It's easy to get too focused on the system stresses – the reports list and compare the actual and allowable values for you. Other, more subtle, checks must be made as well. Two are discussed below but other considerations must be made. For example, this line starts at an anchor. What is the nature of this anchor? Is it a tee connection where stresses should be properly evaluated or is in a nozzle connection where the impact loads are checked. Table 2 summarizes the results for the position of the relief piping (Node 550) under the vent stack and the spring load (Node 60) at the end of the run. I've summed the shock results and operating results to build the combined effects and listed these numbers in the column "Operating + …". The same trend in the output magnitude is recognized at the relief piping as with the elbow stresses – static equivalent is the highest, followed by time-history and then the response spectra. Again, the relief piping is not "hard piped" to the vent stack. The valve piping moves over 70 mm both horizontally and down. Depending on the size of the vent stack, the valve piping may clash with or exit the stack. This contact is not addressed in the analysis but should be a concern to the engineer in the design and installation of the system. For the hanger we see a Grinnell Figure 98 Size 12 spring was selected for installation at 60. This spring has a recommended maximum load of 12,020 N. and an absolute maximum – "bottom-out" – load of 13,000 N. Clearly, this spring will bottom-out during the relief valve firing. This observation certainly invalidates this analysis. Currently, CAESAR II does not consider nonlinear effects in dynamic analysis. Note, too, that while the algebraic summation used in the modal time-history analysis aggravated the elbow stresses and valve deflections it actually helped to reduce the load (and deflection) at the hanger. The hanger is located beyond the "nodal" point of the fourth mode of vibration (the point that looks like a support point in Figure 5). This means that, instead of adding to the first mode displacement, it subtracts from it to produce the smallest load of the bunch.

Type of Analysis	Operating + …	Deflection of 550 in X (mm)	Deflection of 550 in Y (mm)	Load on hanger at 60 (newton)
Static:	F	73.5	-86.4	-18896
Response Spectrum:	DLF2	71.0	-75.1	-19085
	ASME080	70.9	-74.9	-19084
	CROSBY	70.4	-73.8	-18999
Time History:	CROSBY	71.9	-79.2	-18743

Table 2 – Structural Results

Conclusions

The advantages and shortcomings of the various ways of evaluating an impact load were illustrated through a relief valve discharge analysis. The results were quite similar for whichever approach was taken! Generally, the additional analysis "horsepower" thrown at the task produced a lower response and that's encouraging since you get a return on the time invested in collecting the required data. The exception to this trend is the time-history analysis – the most accurate dynamic analysis in CAESAR II. Here, the maximum stresses are greater than the more course response spectrum method. I suspect the reason lies in the modal summation techniques used in these two methods. The response spectrum method takes the square root sum of the modal squares while time-history sums the modes directly. The SRSS method reflects the magnitude of the modal components but it is insensitive to their signs. Time-history, on the other hand, keeps the signs on the modal magnitudes and there may be times when all the major modes have the same sign and produce a sum greater than the SRSS method. I evaluated sin(ωt) at the times (t) when the system responses were highest for each of the modes of vibration (ω) and there is a conjunction between the modes at these times. These modal maxima must also be correlated with their modal displacements and, at least for modes 1 and 4, the mode shapes show the elbow closing for first half cycle indicating that they are truly additive at these times. I reran the spectrum analyses with absolute modal summation (rather than SRSS) and confirmed that the stresses were indeed much higher – about 25% higher.

If a few more accurate numbers are all you get from a dynamic analysis it would be hard to categorically recommend such an approach. There are other benefits. The modal results indicate that the first mode of vibration provides the greatest contribution to the overstress at the elbow. You can take this knowledge to the animated mode shapes and see that the first mode of vibration is the horizontal run bouncing up and down, opening and closing the elbow – similar to the simple cantilever bending. The best approach to reducing the overall stress is to reduce this first mode response. You do this by altering the first mode, either increasing or decreasing its frequency. Recalling that the dynamic load factor drops with increasing frequency (see Figure 4), it would be prudent to increase the frequency of the first mode. This is done by increasing the stiffness of the system or reducing its mass [sqrt(k/m)]. Changing the stiffness of the support at the end of the horizontal run would be an obvious choice. Many systems do not have such obvious choices but looking at the mode shape of the troublesome mode will provide clues to where a support will be most effective. These clues are not available in a static equivalent analysis.

One final observation goes back to the static equivalent method. It's a better understanding of the term "structures having essentially one degree-of-freedom" as it's used in several texts including Appendix II of B31.1. The static equivalent to a dynamic load is only valid for single degree-of-freedom (DOF) systems. This piping system is not a single DOF system but it acts like one. The modal analysis methods in CAESAR II allow us to think of each mode of vibration as an independent, single DOF system. As long as only one mode is activated, and if the static load can replicate this mode shape, the response to the static equivalent load is similar to the full dynamic analysis. In many systems a single mode predominates. Just review the shock output from CAESAR II to confirm that the major modal component accounts for the lion's share of the total response. So these single DOF approximations work quite well – in most cases.

What about the relief line? I'm late as it is with this article so a write-up on the redesign must wait for the next newsletter. That might give you time to send in your comments and questions. In the meantime I'll post the data files and charts on our web site.

Fatigue Analysis Using CAESAR II Version 4.10

By Tom Van Laan

One of the major new features in CAESAR II Version 4.10 is the addition of the ability to perform fatigue analyses. For most piping codes supported by **CAESAR II**, this is an extension to, rather than an explicit part of, the code requirements (however it is an explicit part of the IGE/TD/12 Pipework Stress Analysis for Gas Industry Plant code).

Fatigue Basics:

Piping and vessels have been known to suffer from sudden failure following years of successful service. Research done during the 1940s and 1950s (primarily advanced by A. R. C. Markl's "Piping Flexibility Analysis", published in 1955) provided an explanation for this phenomenon, as well as design criteria aimed at avoiding failures of this type. The explanation was that materials were failing due to fatigue, a process leading to the propagation of cracks, and subsequent fracture, following repeated cyclic loading.

Steels and other metals are made up of organized patterns of molecules, known as crystal structures. However, these patterns are not maintained throughout the steel producing an ideal homogenous material, but are found in microscopic isolated island-like areas called grains. Inside each grain the pattern of molecules is preserved. From one grain boundary to the next the molecular pattern is the same, but the orientation differs. As a result, grain boundaries are high energy borders. Plastic deformation begins within a grain that is both subject to a high stress and oriented such that the stress causes a slippage between adjacent layers in the same pattern. The incremental slippages (called dislocations) cause local cold-working. On the first application of the stress, dislocations will move through many of the grains that are in the local area of high stress. As the stress is repeated, more dislocations will move through their respective grains. Dislocation movement is impeded by the grain boundaries, so after multiple stress applications, the dislocations tend to accumulate at grain boundaries, eventually becoming so dense that the grains "lock up", causing a loss of ductility and thus preventing further dislocation movement. Subsequent applications of the stress cause the grain to tear, forming cracks. Repeated stress applications cause the cracks to grow. Unless abated, the cracks propagate with additional stress applications until sufficient cross sectional strength is lost to cause catastrophic failure of the material.

The fatigue capacity of a material can be estimated through the application of cyclic tensile/compressive displacement loads with a uniaxial test machine. A plot of the cyclic stress capacity of a material is called a fatigue (or endurance) curve. These curves are generated through multiple cyclic tests at different stress levels.

The number of cycles to failure usually increases as the applied cyclic stress decreases, often until a threshold stress (known as the endurance limit) is reached below which no fatigue failure occurs, regardless of the number of applied cycles. An endurance curve for carbon and low alloy steels, taken from the ASME Section VIII Division 2 Pressure Vessel Code is shown in the following figure.

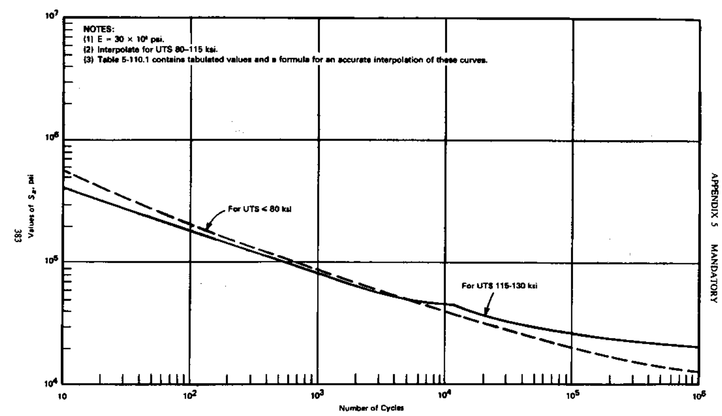


FIG. 5-110.1 DESIGN FATIGUE CURVES FOR CARBON, LOW ALLOY, SERIES 4XX, HIGH ALLOY STEELS AND HIGH TENSILE STEELS FOR TEMPERATURES NOT EXCEEDING 700°F

Fig. 5-110.1

Fatigue Analysis of Piping Systems:

Cyclic loads on piping (primarily thermal expansion or vibration loadings) are found to cause fatigue failure in piping systems. The fatigue design criteria required by the piping codes today are basically identical to those proposed by Markl in the 1950s. The codes typically limit the expansion stress range in piping to a formula which generally fits the fatigue curve of the material.

The IGE/TD/12 code does, on the other hand, present specific requirements for true fatigue evaluation of systems subject to a cyclic loading threshold. Furthermore, ASME Section III, Subsection NB and ASME Section VIII Division 2 provide guidelines by which fatigue evaluation rules may be applied to piping (and other pressure retaining equipment). These procedures have been adapted, where possible, to **CAESAR II**'s methodology.

Fatigue analyses can be done through the following steps:

1) **Assigning fatigue curve data to the piping material:** This is done on the Allowable auxiliary screen. Fatigue data may be entered directly, or read in from a text file (a number of commonly used curves have been provided). Users may define their own fatigue curves as defined in Appendix A below.

2) **Defining the fatigue load cases:** This may be done in either the static or dynamic load case builders. For this purpose, a new stress type, *FAT*, has been defined. For every fatigue case, the number of anticipated cycles must also be defined.

3) **Calculation of the fatigue stresses:** This is done automatically by **CAESAR II** – the fatigue stresses, unless explicitly defined by the applicable code are calculated the same as **CAESAR II** calculates stress intensity, in order to conform to the requirements of ASME Section VIII, Division 2 Appendix 5. (The IGE/TD/12 is currently the only piping code supported by **CAESAR II** which does have explicit instructions for calculating fatigue stresses.) The equations used in the calculation of fatigue stresses are documented in Appendix B below.

4) **Determination of the allowable fatigue stresses:** Allowables are interpolated logarithmically from the fatigue curve based upon the number of cycles designated for the load case. For static load cases, the calculated stress is assumed to be a peak-to-peak cyclic value (i.e., thermal expansion, settlement, pressure, etc.), so the allowable stress is extracted directly from the fatigue curve. For harmonic and dynamic load cases, the calculated stress is assumed to be a zero-to-peak cyclic value (i.e., vibration, earthquake, etc.), so the extracted allowable is divided by 2 prior to use in the comparison.

5) **Determination of the allowable number of cycles:** The flip side of calculating the allowable fatigue stress for the designated number of cycles is the calculation of the allowable number of cycles for the calculated stress level. This is done by logarithmically interpolating the "Cycles" axis of the fatigue curve based upon the calculated stress value. Since static stresses are assumed to be peak-to-peak cyclic values, the allowable number of cycles is interpolated directly from the fatigue curve. Since harmonic and dynamic stresses are assumed to be zero-to-peak cyclic values, the allowable number of cycles is interpolated using twice the calculated stress value.

6) **Reporting the results: CAESAR II** provides two reports for viewing the results of load cases of stress type *FAT*. The first of these is the standard stress report, which displays the calculated fatigue stress and fatigue allowable at each node. Stress reports may be generated individually for each load case, and show whether any of the individual load cases in isolation would fail the system.

   However, in those circumstances where there is more than one cyclic load case potentially contributing to fatigue failure, the Cumulative Usage report is appropriate. In order to generate this report, the user selects all of the FAT load cases which contribute to the overall system degradation. The Cumulative Usage report lists for each node point the usage ratio (actual cycles divided by allowable cycles), and then sums these up for total Cumulative Usage. A total greater than 1.0 indicates a potential fatigue failure.

An example follows:

Static Analysis Fatigue Example:

Consider a sample job that potentially has several different cyclic load variations:

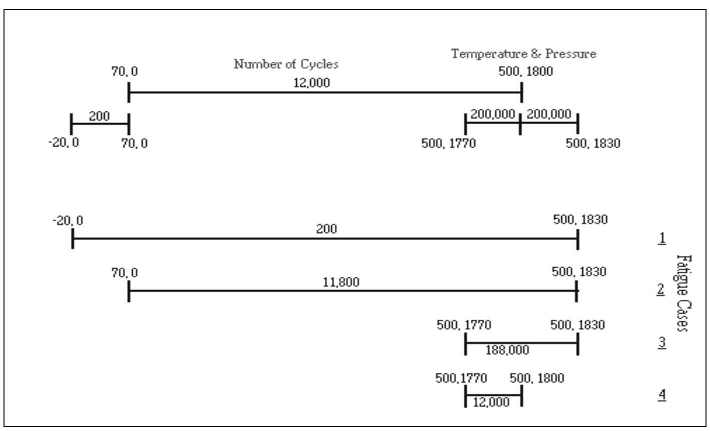
1) Operating cycle from ambient (70º F) to 500º F (12,000 cycles anticipated)

2) Shut down external temperature variation from ambient (70º F) to -20º F (200 cycles anticipated)

3) Pressurization to 1800 psig (12,000 cycles anticipated)

4) Pressure fluctuations of plus/minus 30 psi from the 1800 psig (200,000 cycles anticipated)

In order to do a proper fatigue analysis, these should be grouped in sets of load pairs which represent the worst-case combination of stress ranges between extreme states. These load variations can be laid out in graphical form. The figure below shows a sketch of the various operating ranges this system experiences. Each horizontal line represents an operating range. At the each end of each horizontal line, the temperatures and pressures defining the range are noted. At the center of each horizontal line, the number of cycles for each range is defined.



Using this sketch of the operating ranges, the four fatigue load cases can be determined. The procedure is as follows.

Case 1: Cover the absolute extreme, from –20 F and 0 psi to 500 F and 1830 psi. This occurs 200 times. As a result of this case, the cycles for the ranges defined must be reduced by 200. The first range (-20,0 to 70,0) is reduced to zero, and has no contribution to additional load cases. The second range (70,0 to 500,1800) is reduced to 11,800 cycles. The third and fourth ranges are similarly reduced to 199,800 cycles.

These same steps can be used to arrive at cases 2 through 4, reducing the number of "considered" cycles at each step. This procedure is summarized in the table below.

Segment	-20, 0 to 70, 0	70, 0 to 500, 1800	500, 1770 to 500, 1800	500, 1800 to 500, 1830
CASE				
Initial	200	12,000	200,000	200,000
After 1	0	11,800	200,000	199,800
After 2	0	0	200,000	188,000
After 3	0	0	12,000	0
After 4	0	0	0	0

This table is then used to set the load cases as cycles between the following load values:

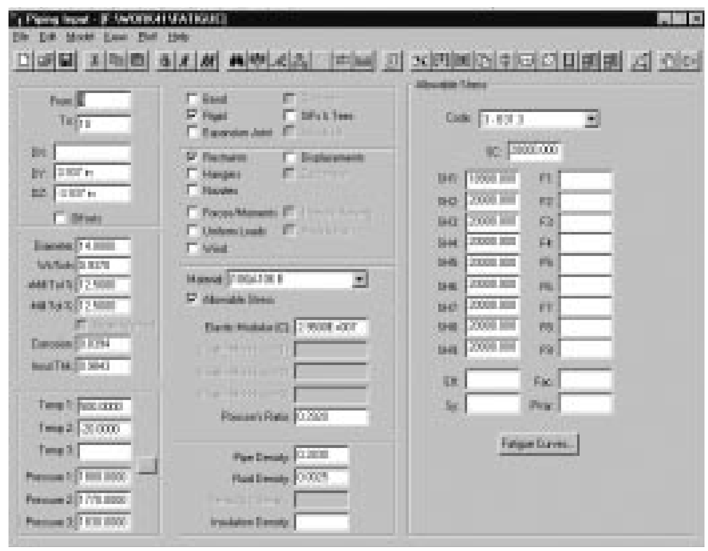
1) Between -20º F, 0 psig and 500º F, 1830 psig (200 cycles)

2) Between 70º F, 0 psig and 500º F, 1830 psig (11,800 cycles)

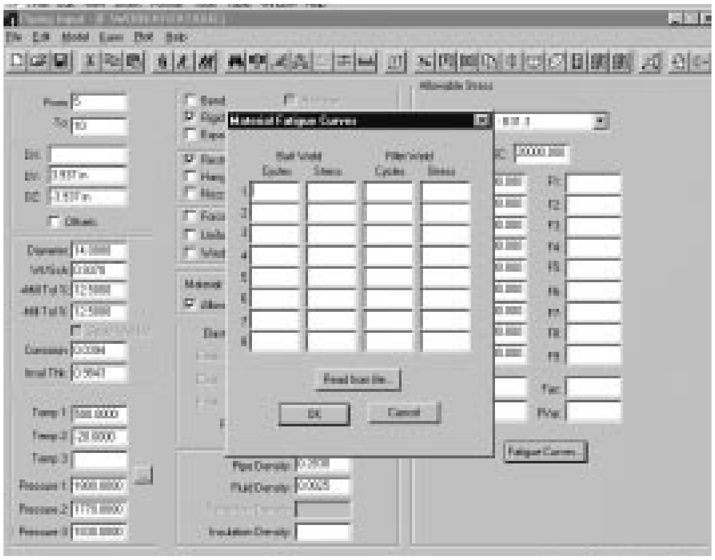
3) Between 500º F, 1770 psig and 500º F, 1830 psig (188,000 cycles)

4) Between 500º F, 1770 psig and 500º F, 1800 psig (12,000 cycles)

These temperatures and pressures are entered as operating conditions accordingly:



It is next necessary to enter the fatigue curve data for the material. This is done by clicking the **Fatigue Curves…** button, revealing the Material Fatigue Curve dialog box. This can be used to enter two sets of fatigue curves for the material – one for butt weld fittings and one for fillet weld fittings (note: this distinction is currently implemented only for the IGE/TD/12 code –fatigue analyses under all other codes are evaluated only against the butt weld curve). Up to eight Cycle vs. Stress data points may be entered to define the curve; interpolations are made logarithmically. Data points should be entered top down, from fewest number of cycles to greatest number of cycles.

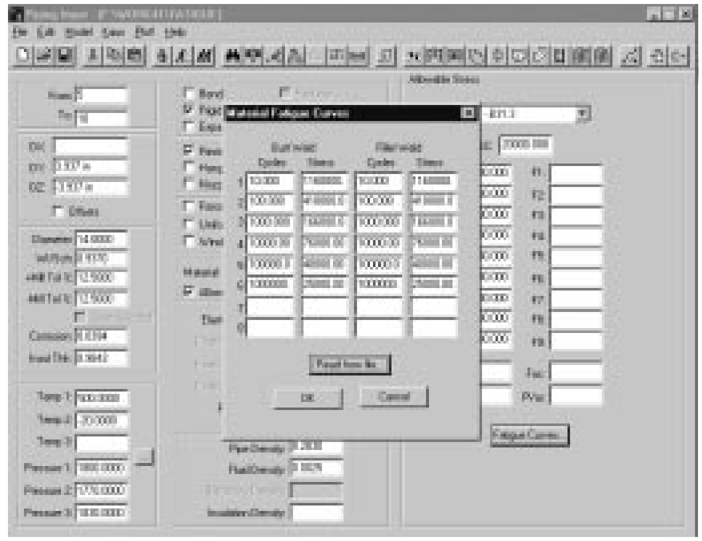


Fatigue curves may be alternatively acquired from a text file, by clicking on the **Read from file…** Button. This displays a list of all \CAESAR\SYSTEM*.FAT files.

Shipped with the program are the following fatigue curve files (the user may easily construct additional fatigue curve files, as described in Appendix A below):

5-110-1A.FAT	ASME Section VIII Division 2 Figure 5-110.1, UTS < 80 ksi
5-110-1B.FAT	ASME Section VIII Division 2 Figure 5-110.1, UTS = 115-130 ksi
5-110-2A.FAT	ASME Section VIII Division 2 Figure 5-110.2, Curve A
5-110-2B.FAT	ASME Section VIII Division 2 Figure 5-110.2, Curve B
5-110-2C.FAT	ASME Section VIII Division 2 Figure 5-110.2, Curve C
TD12AL.FAT	IGE/TD/12 Figure 1 S_R-N Curve (Aluminum)
TD12ST.FAT	IGE/TD/12 Figure 1 S_R-N Curve (Carbon/ Austenitic Steel)

In this case, for A 106 B low carbon steel, operating at 500° F, 5-110-1A.FAT is the appropriate selection. This fills in the fatigue curve data:



At this point, the job can be error checked, and the load cases can be set up.

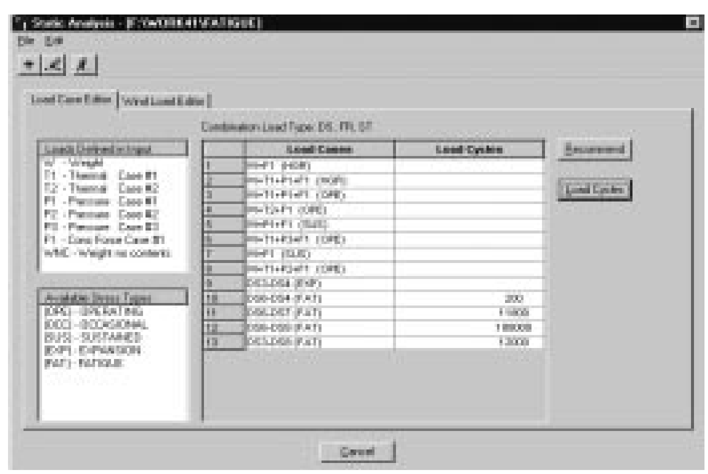
The static load case builder offers a new stress type, *FAT* (fatigue). Selecting this stress type:

1) invites the user to define the number of cycles for the load case (dragging the *FAT* stress type into the load case or pressing the **Load Cycles** button opens the **Load Cycles** field),

2) causes the stress range to be calculated as per the fatigue stress method of the governing code (currently this is stress intensity for all codes except IGE/TD/12),

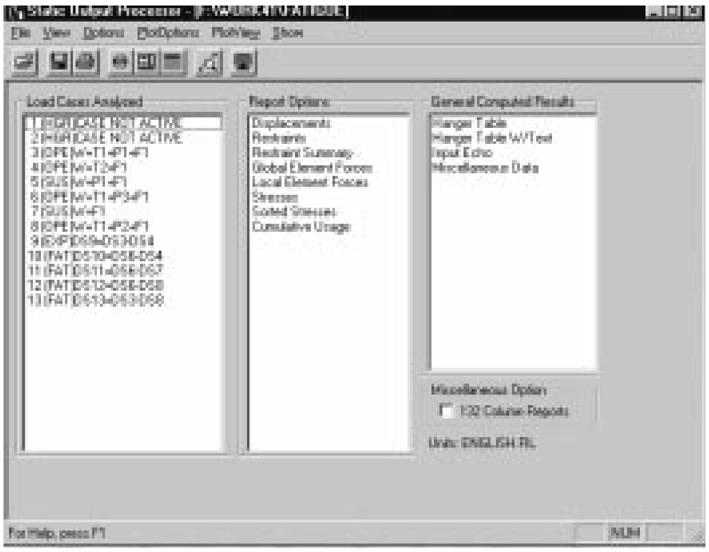
3) causes the calculated stress range to be compared to the full value extracted from the fatigue curve, and

4) indicates that the load case may be included in the Cumulative Usage report.

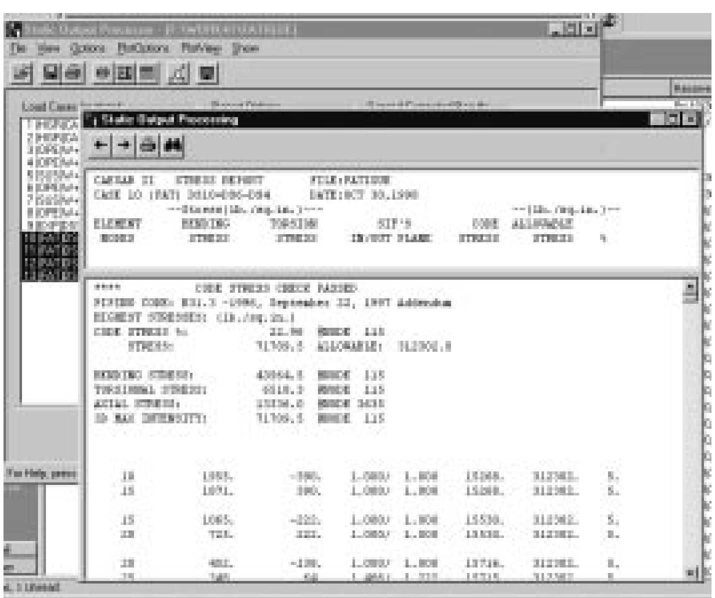


The last four load cases represent the load set pairs defined earlier.

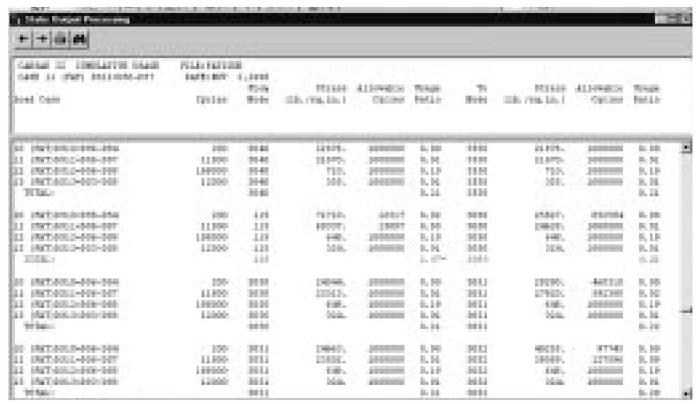
Once the job has been run, note that the presence of a *FAT* stress type adds the **Cumulative Usage** report to the list of available reports.



The fatigue stress range may be checked against the fatigue curve allowable for each load case by simply selecting it along with the **Stresses** report. Review of each load case shows that all stress levels pass.

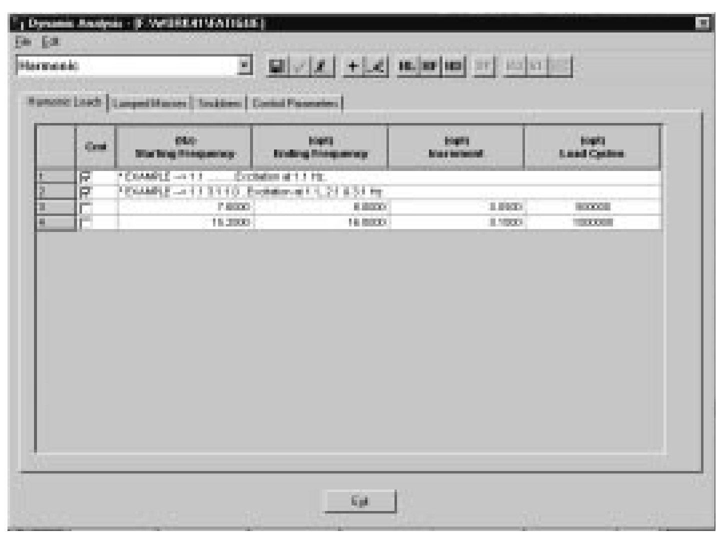


However, this is not a true evaluation of the situation, because it is not a case of "either-or". The piping system is subjected to all of these load cases throughout its expected design life, not just one of them. Therefore, we must review the **Cumulative Usage** report, which shows the total effect of all fatigue load cases (or any combination selected by the user) on the design life of the system. This report lists for each load case the expected number of cycles, the allowable number of cycles (based upon the calculated stress), and the Usage Ratio (actual cycles divided by allowable cycles). The Usage Ratios are then summed for all selected load cases; if this sum exceeds 1.0, the system has exceeded its fatigue capabilities. In this case, it is apparent that the sum of all of the cyclic loadings at node 115 can be expected to fail this system:

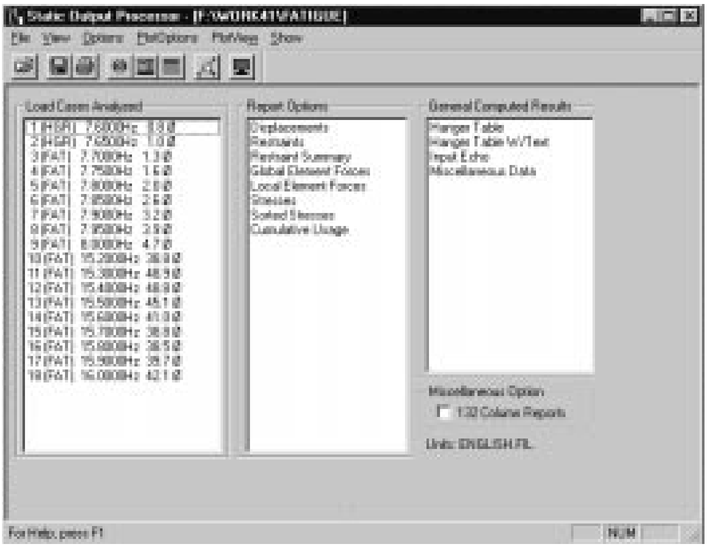


Fatigue Capabilities in Dynamic Analysis:

Fatigue analysis capability is also available for harmonic and dynamic analyses as well. Harmonic load cases are entered as they always have been; they may be designated as being stress type *FAT* simply by entering the number of expected load cycles on the harmonic input screen:

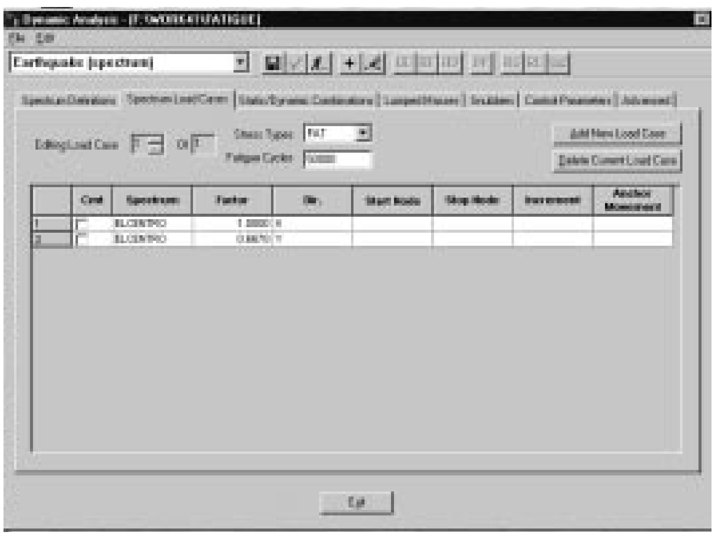


This produces the same types of reports as are available for the static analysis; they can be processed as discussed earlier.



The only difference between the harmonic and static fatigue analyses is that for harmonic jobs, the calculated stresses are assumed to be zero-to-peak calculations, so they are compared to only *half* of the stress value extracted from the fatigue curve. Likewise, when creating the **Cumulative Usage** report, the number of allowable cycles is based upon *twice* the calculated stress.

For other dynamic applications (response spectrum and time history), the stress type may be identified as fatigue by selecting the stress type from the drop list for the **Load Case** or **Static/Dynamic Combination**, and by entering the number of expected cycles in the provided field.



Note that as with the harmonic analyses, the calculated stresses are assumed to be zero-to-peak calculations, so they are compared to only *half* of the stress value extracted from the fatigue curve. Likewise, when creating the **Cumulative Usage** report, the number of allowable cycles is based upon *twice* the calculated stress.

Appendix A – Creating the .FAT Files

The **.FAT** file is a simple text file, containing the data points necessary to describe the fatigue curve for the material, for both butt welded and fillet welded fittings. A sample FAT file is shown below.

```
* ASME SECTION VIII DIVISION 2 FATIGUE CURVE
* FIGURE 5-110.1
* DESIGN FATIGUE CURVES FOR CARBON, LOW ALLOY,
* SERIES 4XX,
* HIGH ALLOY AND HIGH TENSILE STEELS FOR
* TEMPERATURES NOT
* EXCEEDING 700 F
* FOR UTS <= 80 KSI
*
0.5000000 - STRESS MULTIPLIER (PSI); ALSO
* CONVERTS AMPLITUDE TO FULL RANGE
* BUTT WELD
       10    580000.0
      100    205000.0
     1000     83000.0
    10000     38000.0
   100000     20000.0
  1000000     12500.0
        0         0.0
        0         0.0
*
* FILLET WELD (NONE SPECIFIED, USE SAME AS BUTT
* WELD)
*
       10    580000.0
      100    205000.0
     1000     83000.0
    10000     38000.0
   100000     20000.0
  1000000     12500.0
        0         0.0
        0         0.0
*
```

This text file can be created using any available text editor. Note that any line beginning with an asterisk is treated as a comment line. It is highly recommended that comment lines be used so that the data can be related back to a specific material curve.

The first actual data line in the file is a stress multiplier. This value is used to adjust the data values from “zero to peak” to “peak to peak” and to convert the stress levels to psi (the entered values will be divided by this number). Following this line is the data table for the “butt weld” fittings. This table consists of eight lines, of two columns. The first column is the “cycle” column, the second column is the “stress” column. For each value in the “cycle” column, the corresponding stress value from the material fatigue curve should be listed in the “stress” column.

Following the “butt weld” table is the “fillet weld” table. (Note that optional comment lines are used to separate the two tables – these comments aid in the readability of the data file. This will help when creating and verifying your own tables, use comments liberally.) The “fillet weld” table also contains eight lines of two columns.

In both tables, the number of cycles increases as you work down the table. If there is not enough data to utilize all eight lines, unused lines should be populated with zeroes.

Appendix B – Calculation of Fatigue Stresses

For the IGE/TD/12 piping code, the computation of fatigue stresses are detailed in Section 5.4.4 of that code. This section of the code states: “The principal stress in any plane can be calculated for any set of conditions from the following formula:”

$$\frac{1}{2}\left[[S_h + S_a] \pm \sqrt{[S_h - S_a]^2 + 4 S_q{}^2} \right]$$

“This should be used for establishing the range of stress, due regard being paid to the direction and sign.”

For all other piping codes in **CAESAR II**, the fatigue stress is computed as the stress intensity, as follows:

```
3D MAXIMUM SHEAR STRESS INTENSITY

  SI = MAX OF:    S1OT - S3OT
                  S1OB - S3OB
                  MAX(S1IT,RPS) - MIN(S3IT,RPS)
                  MAX(S1IB,RPS) - MIN(S3IB,RPS)

Where:
  S1OT  = MAXIMUM PRINCIPAL STRESS, OUTSIDE TOP
        = (SLOT+HPSO)/2.0+SQRT(((SLOT-HPSO)
          2.0)^2+TSO^2)
  S3OT  = MINIMUM PRINCIPAL STRESS, OUTSIDE TOP
        = (SLOT+HPSO)/2.0-SQRT(((SLOT-HPSO)
          2.0)^2+TSO^2)
  S1IT  = MAXIMUM PRINCIPAL STRESS, INSIDE TOP
        = (SLIT+HPSI)/2.0+SQRT(((SLIT-HPSI)
          2.0)^2+TSI^2)
  S3IT  = MINIMUM PRINCIPAL STRESS, INSIDE TOP
        = (SLIT+HPSI)/2.0-SQRT(((SLIT-HPSI)
          2.0)^2+TSI^2)
  S1OB  = MAXIMUM PRINCIPAL STRESS, OUTSIDE BOTTOM
        = (SLOB+HPSO)/2.0+SQRT(((SLOB-HPSO)
          2.0)^2+TSO^2)
  S3OB  = MINIMUM PRINCIPAL STRESS, OUTSIDE BOTTOM
        = (SLOB+HPSO)/2.0-SQRT(((SLOB-HPSO)
          2.0)^2+TSO^2)
  S1IB  = MAXIMUM PRINCIPAL STRESS, INSIDE BOTTOM
        = (SLIB+HPSI)/2.0+SQRT(((SLIB-HPSI)
          2.0)^2+TSI^2)
  S3IB  = MINIMUM PRINCIPAL STRESS, INSIDE BOTTOM
        = (SLIB+HPSI)/2.0-SQRT(((SLIB-HPSI)
          2.0)^2+TSI^2)
RPS   = RADIAL PRESSURE STRESS (INSIDE)
HPSI  = HOOP PRESSURE STRESS (INSIDE, FROM LAME'S
        EQN)
HPSO  = HOOP PRESSURE STRESS (OUTSIDE, FROM LAME'S
        EQN)
 SLOT = LONGITUDINAL STRESS OUTSIDE (TOP)
 SLIT = LONGITUDINAL STRESS INSIDE  (TOP)
 SLOB = LONGITUDINAL STRESS OUTSIDE (BOT)
 SLIB = LONGITUDINAL STRESS INSIDE  (BOT)
 TSI  = TORSIONAL STRESS INSIDE
 TSO  = TORSIONAL STRESS OUTSIDE
```

PC Hardware/Software for the Engineering User (Part 26)

By Richard Ay

Many users are still surprised when they find that a certain software program has an update available. These updates are typically necessitated by *bug fixes*, and occasionally include minor enhancements. All modern, well supported software programs experience these updates, sometimes called builds. Recently, a number of users have expressed concern over whether or not the second update for Windows 95 is required. This article discusses the state of Windows 95 and its revision history. *In addition, please note that Microsoft released Service Pack 4 for Windows NT 4.00 during the later part of October*. Items in this Service Pack will also be briefly discussed.

Windows 95 has seen two major updates from Microsoft. The first update, known as “Service Pack 1” has been readily available from a number of web sites since its release by Microsoft. This service pack is a necessity if you are running Windows 95, it corrects a number of problems with essential components of the operating system. The service level of the operating system can be checked by:

- From the “start” button, click on “Settings\Control Panel”.
- From “Control Panel”, click on the “System” icon.
- On the “General” tab, the operating system version (including Service Pack level) is listed at the top right.

Windows Service Pack 1 is usually reported as Version 4.00.950a.

Windows 95 Service Pack 2

The second update to Windows 95, known as “Service Pack 2” is not available as a complete unit. Service Pack 2 was distributed to OEMs (Original Equipment Manufacturers) only. Existing Windows 95 installations were left on their own, and have had to download the individual components which make up Service Pack 2, if available.

How do you know if you should attempt to update your Windows 95 system to Service Pack 2? That depends on individual usage patterns, and what Service Pack 2 updates. The contents of Service Pack 2 can be divided into four categories, as defined by Microsoft. These categories are: Hardware Support, Internet and Multimedia, Networking and Communication, and Additional Features. The following paragraphs summarize the contents of Service Pack 2. *(The following information has been condensed from a document on Microsoft's web site.)*

Hardware Support:

FAT32	This enhancement improves Windows 95 handling of large hard disks, up to 2 terabytes in size. This fix is included on new PCs, ***but is not*** available for download from Microsoft's web site.
DriveSpace	The disk compression utility has been improved to handle drives up to 2 gigabytes in size. . This fix is included on new PCs, ***but is not*** available for download from Microsoft's web site.
Power Management	Support has been added for advanced power management, multi-battery PCs, disk and modem power down. This fix is included on new PCs, ***but is not*** available for download from Microsoft's web site.
Storage Enhancements	Support has been added for IDE bus mastering and other removable media. This fix is included on new PCs, ***but is not*** available for download from Microsoft's web site.
PCMCIA Enhancements	Added support for low voltage PCMCIA cards. This fix is included on new PCs, ***but is not*** available for download from Microsoft's web site.
CDFS Enhancements	Support added for ISO 9660 disks up to 4 gigabytes and CD-I format CD-ROMs. This fix is included on new PCs, ***but is not*** available for download from Microsoft's web site.
PCI Binding & Docking	Support for PCI docking stations improved. This fix is included on new PCs, ***but is not*** available for download from Microsoft's web site.
IRQ Routing	Added support for new PCI interrupt routers. This fix is included on new PCs, ***but is not*** available for download from Microsoft's web site.

Internet and Multimedia:

Internet Explorer 4.0	Microsoft's internet browser. This fix is included on new PCs, and is available for download from Microsoft's web site.
Internet Connection Wizard	Adds a simple configuration procedure for connecting to the internet and the sign-up process. This fix is included on new PCs, and is available for download from Microsoft's web site.
Internet Mail & News	The SMTP and POP3 clients are improved. This fix is included on new PCs, and is available for download from Microsoft's web site.
NetMeeting	Allows teleconferencing over the internet. This fix is included on new PCs, and is available for download from Microsoft's web site.
Personal Web Server	Allows publishing and hosting HTML pages. This fix is included on new PCs, and is available for download from Microsoft's web site.
DirectX 2.0	Includes a new high performance graphics and sound for games. This fix is included on new PCs, and is available for download from Microsoft's web site.
Active Movie	Includes the next generation video format. This fix is included on new PCs, and is available for download from Microsoft's web site.
OpenGL	Added support for OpenGL (graphics standard) libraries. This fix is included on new PCs, and should be available for download from Microsoft's web site soon.
Intel MMX Support	Added support for software development targeting the Pentium MMX chip. This fix is included on new PCs, ***but is not*** available for download from Microsoft's web site.

Networking and Communications:

Dial-Up Networking	User interface for dial-up connections improved. This fix is included on new PCs, and is available for download from Microsoft's web site.
Voice Modem Support	Support for voice modems added. This fix is included on new PCs, and is available for download from Microsoft's web site.
Service for Netware Directory Services	Full client support for Novell NetWare 4.x added. This fix is included on new PCs, and is available for download from Microsoft's web site.
32-bit DLC	Added 32 bit support for SNA host connectivity. This fix is included on new PCs, and is available for download from Microsoft's web site.
Infrared Support	Support added for infrared devices. This fix is included on new PCs, and is available for download from Microsoft's web site.
Desktop Management	Added support for desktop management interface 1.1. This fix is included on new PCs, and should be available for download from Microsoft's web site soon.
NDIS 4.0	Added support for new NDIS 4.0 network drivers. This fix is included on new PCs, ***but is not*** available for download from Microsoft's web site.

Additional Features:

Display Enhancements	Adds support for dynamic changes to screen resolution and color depth. This fix is included on new PCs, and should be available for download from Microsoft's web site soon.
Imaging	Allows viewing of various graphics file formats. This fix is included on new PCs, and is available for download from Microsoft's web site.
Fonts	Adds support for HP Laserjet 4 grayscale fonts. This fix is included on new PCs, and should be available for download from Microsoft's web site soon.
MSN 1.3	the latest version of the MSN client. This fix is included on new PCs, and is available for download from Microsoft's web site.
Miscellaneous Fixes	Updates to OLE components, Windows messaging client, and Microsoft Fax. This fix is included on new PCs, and is available for download from Microsoft's web site.
Auto Scandisk	Allows Scandisk to run on bootup following an abnormal shutdown. This fix is included on new PCs, ***but is not*** available for download from Microsoft's web site.
Online Services Folder	Added client software for AOL 3.0, CompuServe 3.0, CompuServe WOW, and AT&T Worldnet. This fix is included on new PCs, ***but is not*** available for download from Microsoft's web site.

As stated above, the necessity of these Service Pack 2 items depends on the usage of a particular PC. The average engineering, CAD, or word processing workstation doesn't need any of these updates. However, workstations with large hard disks, or workstations performing communications tasks should consider the updates available.

Windows NT 4.00 Service Pack 4

Service Pack 4 for Windows NT 4.00 was released during the later part of October. This service pack includes all updates and enhancements that have been issued by Microsoft since NT 4.00 was first released. Service Pack 4 includes:

- Management enhancements, including fixes for the "year 2000" problem.
- Security enhancements
- Support tools
- Networking enhancements
- Other enhancements

This service pack is available for download from Microsoft's web site.

Note, COADE Recommends that you do not upgrade to Service Pack 4 at this time, due to potential problems with Aladdin's ESL Drivers.

CAESAR II Notices

Listed below are those errors & omissions in the **CAESAR II** program that have been identified since the last newsletter.

1) **Piping Error Checker:** An error was discovered in the setup of uniform load vectors #2 and #3. The loads entered were only applied to the element where the specification was made, instead of being duplicated forward. This error affects all builds of Version 4.00 up to 980917.

2) **Element Generator:** An error was discovered in the generation of fixed end forces and moments when the "straight pipe pressure stiffening" option was activated. This error was corrected in the build of 981012.

3) **Miscellaneous Processor:** A units conversion error was discovered in the manipulation of the flange allowable stresses. This error was corrected in the build of 981012.

4) **Structural Input Processor:** Several "memory allocation" problems were found and corrected. These problems prevented some users from creating the model successfully. These problems are corrected in the build of 981012.

5) **Input Echo from Output:** A problem was discovered with the input echo module, when activated from static output – the configuration parameters could not be reported. This problem was corrected in the build of 981012.

TANK Notices

Listed below are those errors & omissions in the **TANK** program that have been identified since the last newsletter.

1) **Computation Module:** The computation of the bottom plate diameter did not include the 1 inch protrusion required by API-650 Section 3.4.2. The correction for this has been included in Versions 1.60 and 2.00.

2) **Computation Module:** The computation for the annular base plate width did not include the thickness of the bottom shell course. The correction for this has been included in Versions 1.60 and 2.00.

CODECALC Notices

Listed below are those errors & omissions in the **CODECALC** program that have been identified since the last newsletter. These corrections are available for download from our BBS and WEB sites.

1) **TEMA Tubesheet Input:** TEMA tubesheet, the tube yield stress value was not being passed into the analysis program causing an error to be generated when the analysis was attempted.

2) **Nozzle Input:** The cylinder external pressure length dialog did not appear when the design external pressure was entered. This caused an error to be generated when the analysis was attempted.

3) **Leg & Lug Input:** The program did not allow for the entry of pipe leg information.

4) **General Input:** Various problems were noted with respect to metric units. Some input values, such as allowable stresses were not converted properly into user units. In some cases the temperature used to access the material database was not converted to Fahrenheit causing the incorrect stress value to be returned from the material database.

5) **External Pressure Processing:** On some length/diameter ratios the program would hang the computer.

6) **Titanium Stress Values:** Stress values at intermediate temperatures (150 degrees etc.) for some titanium materials did not agree with the code. Those values were interpolated and usually resulted in an allowable lower than the Code value.

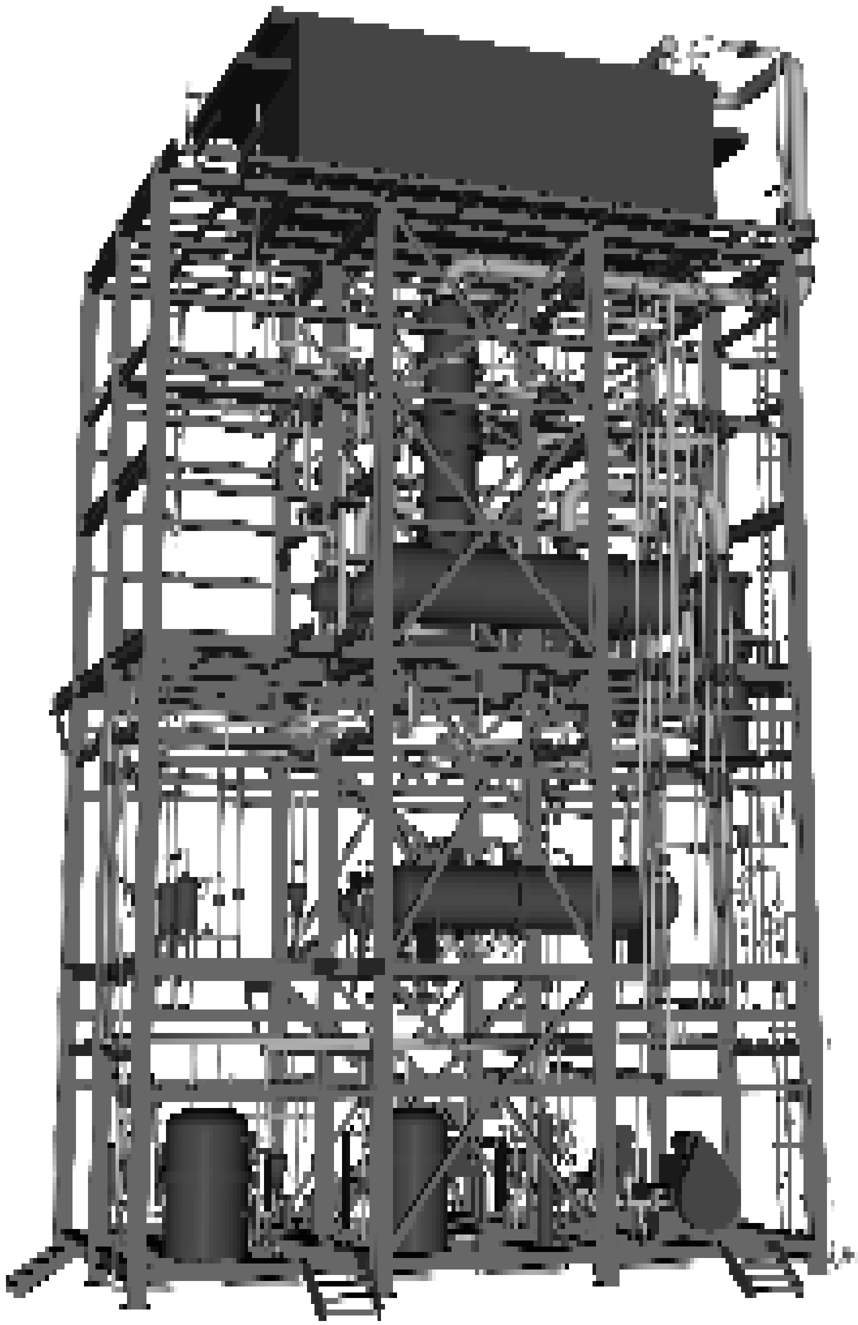
The above problems with **CODECALC** 6.00 have been fixed in an update (build), downloadable from the COADE website.

PV Elite Notices

Listed below are those errors & omissions in the **PV Elite** program that have been identified since the last newsletter. These patches are available for download from our BBS and WEB sites.

1) The problems mentioned above in **CODECALC** 6.00 also existed in the PV Elite component analysis module.

2) **AISC Unity Check:** The leg centerline diameter for legs on cylinders was written improperly to the input file causing the analysis module to use a smaller diameter in the AISC unity check.



CADWorx/Pipe Model

COADE Engineering Software

12777 Jones Rd. Suite 480
Houston, Texas 77070

Tel: 281-890-4566
WEB: www.coade.com

Fax: 281-890-3301
E-Mail: query@coade.com

BBS: 281-890-7286
CompuServe: 73073,362