Mechanical Engineering News

COADE, Inc.

For the Power, Petrochemical and Related Industries

August, 1992

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 will serve as the official notification vehicle for software errors discovered in those Mechanical Engineering programs offered by COADE.

TABLE OF CONTENTS

PC Hardware for the Engineering User (Part 14) 1
CAESAR II Development 3
proVESSEL AND CodeCalc Developmen 3
COADE Seminars 3
Maximum Shear Stress Intensity in CAESAR II 4
Finite Element Piping and Vessel Stresses 7
AISC Unity Checks on Pressure Vessel Legs 8
Piping Failure Caused by Elastic Follow-Up 10
CAESAR II Specifications 12

PC Hardware For The Engineering User (Part 14)

In recent weeks, COADE has received a number of telephone calls from users reporting that moving up from 386 machines to 486 machines, caused the software to behaving differently. In general, 486 machines run COADE software in an identical manner as 386 machines. Any differences can be attributed to:

- Different configurations: All COADE products are configured for the hardware via a configuration program which is invoked automatically by the installation program. Additional computation configuration parameters can be set and controlled by the user. Depending on how these configuration parameters are set, the same program can behave in a completely different manner on two different computers. An example would be the case of two **CAESAR II** configurations where one has ALL_STRESS_CASES_CORRODED turned on, and the other does not. Completely different stress values will result from these two computers. Another example is the case of two **proVESSEL** configurations, one of which set the help text background and foreground to the same color. This resulted in *invisible* help text on the one computer.

- Different DOS versions: As discussed in the October 1991 issue of *Mechanical Engineering News*, the older keydisk versions of COADE products are not compatible with DOS 5.0 if it is loaded in high RAM. The problems reported here are:

  - The computer locks up when the keydisk is accessed. This typically requires the system to be powered off to restart.

  - A “stack over flow” or “stack under flow” is reported, the program aborts, and control returns to the main menu.

  - An “unexpected heap error” is reported, the program aborts, and control returns to the main menu.

  The solution here is to upgrade to the current version of the software, or remove the statement DOS=HIGH from your CONFIG.SYS file.

- Different Hardware: Even if two identical (looking) computers are purchased from the same vendor on the same day, there is a good chance that something is different between them. This could range from different hard drive controllers to different parallel ports. If a hardware problem is suspected, the system diagnostics should be run. In the end, it may be necessary to take the cover off and visually determine the difference between the machines.

ESL Incompatibility

Recently a problem with Notebook computers connected to a network has surfaced. Since the typical Notebook does not have the internal room for a network card, network usage currently requires a Xircom Network Adaptor. This device is connected to the parallel port of the Notebook, the same port the ESL uses.

Unfortunately, the Xircom Network Adaptor is an active device, i.e. they take over control of the parallel port. ***The result of this action is that all parallel type hardware locks are incompatible with this device.***

The vendor of the ESLs has been in contact with Xircom over the last eight months in an attempt to work out a solution to this problem. After multiple attempts, Xircom will no longer return telephone calls to the ESL vendor. ***Until this problem is resolved, COADE users should avoid the Xircom Network Adaptor.***

ESLs and Multiple Computers

As most users know, the External Software Lock (ESL) is the method employed to protect the COADE software products, **CAESAR II**, **PROVESSEL**, and **CodeCalc**. Most users are also aware that each copy of the software purchased is a license to use the software on a single machine.

COADE has never objected to users installing the software on multiple machines, as long as only a single machine is in use at any given time. The intent here is to acknowledge that users may want to work at home or take a portable computer into the field. As long as the ESL is attached, the terms of the license agreement are met.

Additionally, the COADE ESL also provides the facility whereby the software can be date or run limited. This capability is realized by storing the CPU time and date on the ESL each time an analysis is performed.

The COADE ESL also provides the ability for COADE to modify the data stored on the ESL interactively via a telephone call. This procedure requires that the CPU time/date stamp on the users computer and the COADE computer be identical.

For these reasons, checks have to be made to insure that the time and date values stored on the ESL always move forward. The storage of the CPU time and date as well as the associated checks on these values is performed on all ESLs, even if they are unlimited.

The time/date checks performed on the ESL can deny access to the software when the ESL is moved between computers, ***if the machines do not have the correct time/date settings***. For example, assume computer A is set to January 1, at 10 am, and computer B is set to January 1, at 2 pm. An ESL could be moved from computer A to computer B without any problem. However, attempting to move the ESL back to computer A would result in the software being inaccessible for 4 hours. This mishap is magnified if several machines are located in close proximity and the ESL is moved about in an attempt to run more than a single copy of the software, ***which is in violation of the license agreement***.

For those users who need to legitimately move an ESL between an office desk top machine and a machine at home or a portable in the field, ***insure that the time/date settings on all of the computers are set correctly.*** The "last ESL access date" can be viewed by striking the "F2" key from the main menu of any of the COADE software programs. The "last ESL access time" can not be viewed.

When moving an ESL between computers the user may encounter another problem, hardware incompatibility. Depending on the brand of computer and its components, users may find that the software does not find the ESL. This problem is usually attributed to one of the following items:

- The parallel port of the computer is not generating enough power to access the memory of the ESL. This is usually the result of an inferior or bad board. The two solutions are to replace the port, or attach a printer and turn it on. (A "write only" parallel port will also cause this problem.)

- An extension cable or switcher box is in use. Extension cables should be limited to 2 feet and must be ribbon cables. Coaxial cables usually do not work. Switcher boxes may disable the ESL, similar to the effects of a bad printer cable.

- Certain printers have been known to cause ESL access problems. The October 1991 issue of Mechanical Engineering News discussed this problem with regard to the Hewlett Packard Laser Jet IIIP. Since that time the ESL manufacturer has tested these printers (as have additional COADE users) without any problems.

  This problem can still occur, and can be either the printer, the parallel port, or the ESL. If the ESL allows access to the software on a different computer, it is a safe bet that the ESL is not the problem.

Users who encounter problems with their ESL should contact COADE. We can replace the ESL upon receipt of your original, although if the problem is hardware, this will not solve anything. As of **CAESAR II** Version 3.18, **PROVESSEL** Version 2.6, and **CodeCalc** Version 5.1 the ESL "last access date" value can be modified over the telephone (for those users who have been careless with their date setting). Any additional problems not addressed here should be brought to COADE's attention.

CAESAR II DEVELOPMENT

The next version of **CAESAR II** (3.18) is well underway. This release of the software is tentatively scheduled for August or September of 1992. This version of the software will include:

- A new generic neutral file interface which provides two way transfer capabilities between the **CAESAR II** input file and third party software. All data contained in the input file can be written to or read from this neutral file.
- An interface to **LIQT**: This interface will read an ASCII output file of dynamic pipe forces generated by Stoner's **LIQT** (liquid transient) program. Once this data is available, the interface can generate a **CAESAR II** DLF curve for use in force spectrum analysis problems.
- The Canadian piping codes Z183 and Z184 will be updated to the 1990/1992 edition, of these codes.
- Additional valve and flange data bases will be available.
- The file manager will be improved to allow users to log different disk drives or directories without exiting from **CAESAR II**.
- A library of pipe size specifications will be incorporated that will enable the user to specify pipe diameters and wall thicknesses according to either ANSI, JIS BIN, or BS specifications.

COADE has begun work on the 386 version of **CAESAR II**, which will enable access to the extended memory available on most present day computers. The 386 version is being developed using the WATCOM FORTRAN and C compilers. The first module to be completed using these 32 bit compilers is BIGPRT and will be distributed with the 3.18 release.

PROVESSEL AND CodeCalc Development

The latest version of **PROVESSEL**, version 2.6, shipped to all users current on their updates in July 1992. This version of **PROVESSEL** included:

- Saddle Wear Plate Size Estimation
- Expansion of existing Material Data Base
- New Base/Skirt Design/Analsis Module
- ASME A91 Code Updates
- Updated File Handler
- On-Line error processing

The latest version of **CodeCalc**, version 5.1, also shipped in July to all users purchasing this update. This version of **CodeCalc** included:

- ASME A-91 Code Update.
- Base Ring and Skirt Analysis/Design program added.
- Thin-Walled Expansion Joint program added, App-BB.
- Flanged and Flued Expansion Joint program added, App-CC.
- New file manager: enables copy/delete, change drive etc.
- New material database: 1051 materials, in Code order.
- Tube Sheet program updated to TEMA A-91 1991 Errata
- AISC unity checks on angle sections.

COADE Seminars

All Houston seminars are now held at COADE headquarters, at 12777 Jones Road, in Houston, Texas. Due to space limitations, these seminars are limited to fifteen attendees, therefore early registration for a particular seminar is desirable. ***An important point to note for seminar attendees flying in from out of town, Houston has two airports, Intercontinental and Hobby. Intercontinental is approximately twenty five minutes from our office, Hobby is about an hour away - use Intercontinental.***

The **CAESAR II** seminar in May included a presentation on expansion joint manufacturing and usage by Flexonics, including slides and expansion joint models. COADE will continue to provide time for manufacturers to make non-sales related presentations for the benefit of the attendees. The following table lists the seminar schedule for the remainder of 1992 and 1993. The pipe stress introduction seminars are targeted towards those engineers who: are unfamiliar with PC operations, unfamiliar with **CAESAR II**, or are new to pipe stress analysis.

Pipe Stress Seminars

October 14-16, 1992	Introduction
October 19-23, 1992	Statics & Dynamics
January 20-22, 1993	Introduction
January 25-29, 1993	Statics & Dynamics
March 1-5, 1993	Statics & Dynamics
May 5-7, 1993	Introduction
May 10-14, 1993	Statics & Dynamics
September 20-24, 1993	Statics & Dynamics
November 3-4, 1993	Introduction
November 8-12, 1993	Statics & Dynamics

Pressure Vessel Seminars

October 7-9, 1992	ASME Code, **CodeCalc** and **PROVESSEL**
February 3-5, 1993	ASME Code, **CodeCalc** and **PROVESSEL**
May 19-21, 1993	ASME Code, **CodeCalc** and **PROVESSEL**
October 20-22, 1993	ASME Code, **CodeCalc** and **PROVESSEL**

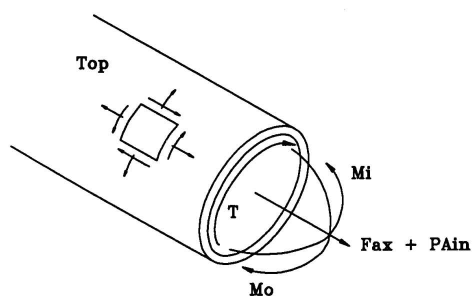
Maximum Shear Stress Intensity in CAESAR II

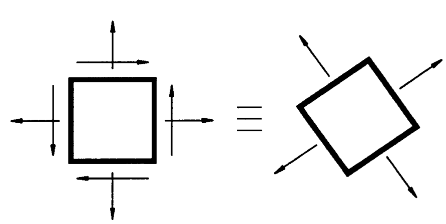
Listed in the load case stress summary and in the expanded stress report is a calculated item labeled 3D maximum shear stress. **CAESAR II** Version 3.18 will properly label this column maximum 3D stress intensity. (Stress intensity is twice maximum shear). This is one value calculated by **CAESAR II** which is independent of piping code definition. In fact, the maximum 3D stress intensity calculation is the closest **CAESAR II** comes to replicating "text book" stress analysis - three dimensional stress on thick wall cylinders. This article will develop the formulae used by **CAESAR II** and then suggest possible uses.

One method of correlating failure of a material test specimen with the failure of the same material under any variety of loads is through the measure of maximum shear stress. Here, the yield of the test specimen in simple axial load is transformed into an equivalent maximum shear stress at yield. Any combination of loads in any orientation which transform into the same or greater maximum shear stress will have met this failure criteria. Using maximum shear stress as a mode of failure is quite common in engineering and its use is evident (in two dimensions) in the piping codes.

In their formulation of calculated stress, the piping codes do not consider all of the piping loads acting simultaneously. Instead, circumferential (hoop) stress calculations are used to set a minimum wall thickness for pressure containment. Then, in several independent calculations, the other mechanical loads on the pipe (deadweight and thermal strain, for example) are evaluated. In most cases, these loads are longitudinal in orientation, only shear due to torque is considered, and no radial component is included. (Of course, the maximum bending stress is on the outside fiber of the pipe where the radial pressure stress is zero.) Code-defined stress intensification factors (SIFs) accommodate reduced strength of piping components such as tees and elbows. Rather than reducing allowable stresses for these components, the SIFs increase the calculated moments (and therefore stresses) so that a common allowable can be maintained.

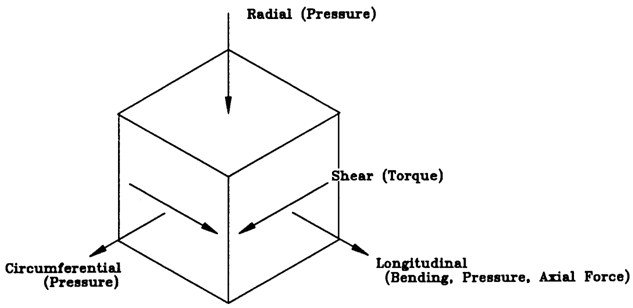
For those situations where "code-defined" stress calculations may not provide all the stress information an engineer wants, this stress intensity calculation may prove helpful. Here, all of the major components in the load case which cause stress are included: pressure, axial force and bending moment in the longitudinal direction, pressure (hoop) in the circumferential direction, torque shearing these two directions, and pressure, again, but in the radial direction. This total view of the fully-loaded stress element can be used to evaluate stress levels independent of the piping codes. The **CAESAR II** calculations make no distinction between stress (or failure) category such as expansion or sustained.





Mamimum 3D stress intensity is calculated using all the load components in the load case.

CAESAR II calculates stresses at four points across the pipe cross section - top outside, top inside, bottom inside and bottom outside. The top / bottom orientation is determined by the resultant bending moment acting on the pipe such that the maximum bending moment is at the top fiber and the minimum bending moment is at the bottom fiber. The longitudinal, circumferential and shear stresses (2D) are then rotated in space so that the principal stresses are found (no shear) at these four points. The last principal stress is simply the radial term which is zero outside the pipe and equal to the pressure inside the pipe.



The local loads which make up these principal stresses are similar, if not identical, to those common in the piping codes. Mechanical loads in the axial, bending, and torsional directions and the pressure in the line are used with the corroded section modulus of the pipe to calculate local stresses at these four points through the pipe. The stresses calculated are as follows:

Radial Stress:

$$\sigma_{ri} := P \cdot \left(\frac{r_i^2}{r_o^2 - r_i^2}\right) \cdot \left(1 - \frac{r_o^2}{r_i^2}\right)$$

$$\sigma_{ro} := 0$$

Circumferential Stress:

$$\sigma_{hi} := P \cdot \left(\frac{r_i^2}{r_o^2 - r_i^2}\right) \cdot \left(1 + \frac{r_o^2}{r_i^2}\right)$$

$$\sigma_{ho} := 2 \cdot P \cdot \left(\frac{r_i^2}{r_o^2 - r_i^2}\right)$$

Longitudinal Stress:

Components:

$$\sigma_{ax} := \frac{F_{ax} + \pi \cdot r_i^2 \cdot P}{A_{xs}} \qquad \sigma_b := \frac{\sqrt{(M_i \cdot i_i)^2 + (M_o \cdot i_o)^2}}{Z}$$

Stresses:

$$\sigma_{lto} := \sigma_{ax} + \sigma_b \qquad \sigma_{lbo} := \sigma_{ax} - \sigma_b$$

$$\sigma_{lti} := \sigma_{ax} + \left(\frac{r_i}{r_o}\right) \cdot \sigma_b \qquad \sigma_{lbi} := \sigma_{ax} - \left(\frac{r_i}{r_o}\right) \cdot \sigma_b$$

Shear Stresses due to Torsion:

$$\tau_o := \frac{T}{2 \cdot Z}$$

$$\tau_i := \frac{T}{2 \cdot Z} \cdot \left(\frac{r_i}{r_o}\right)$$

Terms:

σ_{ri} = radial pressure stress (inside)

σ_{ro} = radial pressure stress (outside)

σ_{hi} = hoop pressure stress (inside)

σ_{ho} = hoop pressure stress (outside)

σ_{ax} = axial stress

σ_b = bending stress

Longitudinal Stresses:

σ_{lto} = outside top

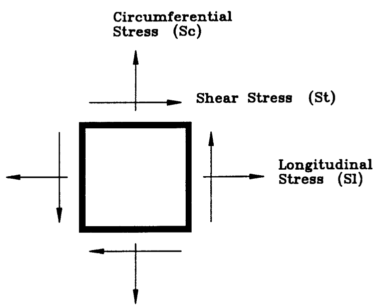
σ_{lbo} = outside bottom

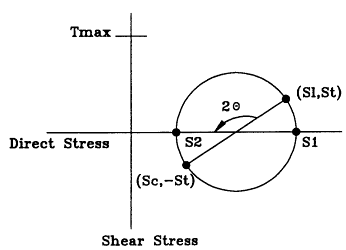
σ_{lti} = inside top

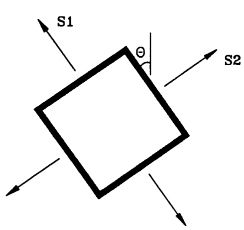
σ_{lbi} = inside bottom

P = pressure
r_i = inside radius of pipe
r_o = outside radius of pipe
F_{ax} = axial force due to mechanical loads
A_{xs} = area of pipe cross section
M_i = in-plane bending moment
i_i = in-plane stress intensification factor
M_o = out-plane bending moment
i_o = out-plane stress intensification factor
Z = section modulus of pipe
T = torque

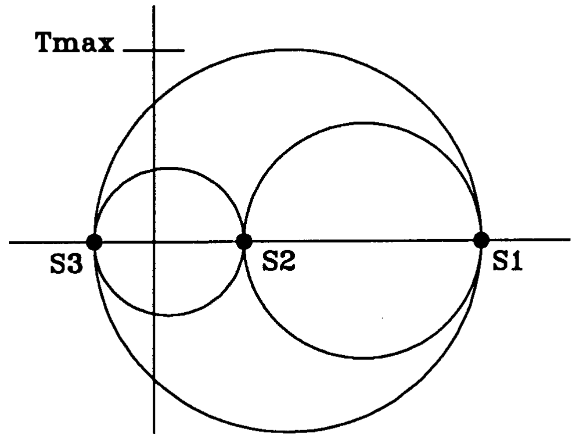
The two dimensional stress elements laying on the pipe surfaces are rotated to find the principal stresses in these planes. Mohr's Circle is a common method of conceptualizing this rotation of the circumferential, longitudinal, and shear stresses. To do this, the direct stress and shear stress on two sides of the element are plotted on a direct stress / shear stress plot. These two points lay on the circumference of a Mohr's Circle. Where this circle intersects the zero shear line is the magnitude of the principal stresses on the rotated element.







Since the radial stress is perpendicular to the rotated element and no shear term exists on the inside or outside surfaces of the pipe, the radial stress can be dropped on the zero-shear line of Mohr's Circle and serve as the principal stress in the third direction. ***The maximum stress intensity is simply the greatest difference between two of these three principal stresses. This difference is also (by definition) twice the maximum shear stress on the element.***



Again, this calculation is performed at four points through the pipe. The final step in producing the 3D maximum stress intensity listed in the **CAESAR II** report is to select the maximum of these four values at each node in the piping system.

What are some uses for this three dimensional maximum stress intensity? Recalling that this stress reflects basic yield analysis more so than the "code-defined" calculations of stress, maximum stress intensity naturally fits as a measure of stress in those situations not addressed by the code. And what would be considered an allowable limit on this stress? Of course the material yield would compare directly to the stress intensity at failure of the test specimen. It would be up to the user to compare the stress intensity to the specified minimum yield stress or the more conservative hot allowable stress.

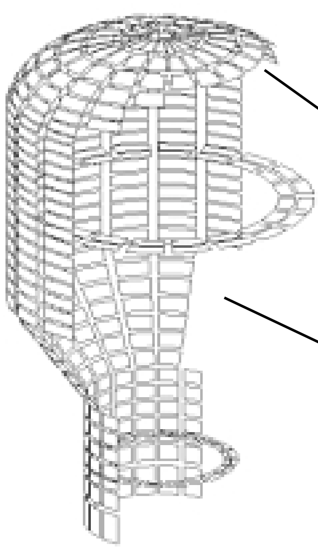
One obvious use of the maximum stress intensity is in the evaluation of loads on non-pipe elements. Another use is an additional check where the code-defined stresses on an element are close to the code allowable. If the stress intensity is also close to the allowable, then there may be reason to further check the model for accuracy. An unconservative or inaccurate model should be modified to reflect the true, as-built, conditions so that this region is modelled as accurately as possible. (This sort of sensitivity check should be made even without use of the stress intensity.)

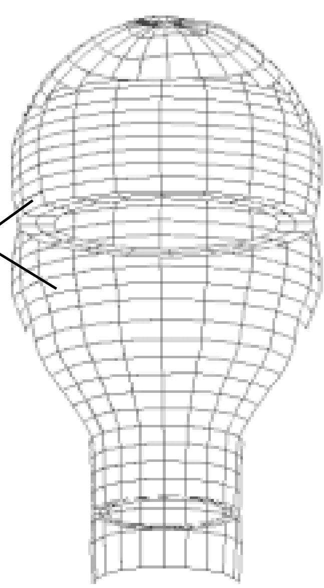
Although many of the piping codes do not establish an allowable stress for the "mixed-mode" loads of the operating case (sustained & cyclic or primary & secondary), there are situations where the stresses in the operating condition may be significant. Again, exceeding an allowable stress indicates yield (of at least the highest stressed fiber) but not necessarily "failure" of the element. It may be useful to examine the maximum stress intensity in elevated temperature systems (systems operating in the creep range). Here, thermal or secondary stresses may contribute to creep damage which is normally caused by primary stresses. Even if these primary stresses are below their allowable, creep damage is still possible if the operating case maximum stress intensity exceeds the allowable.

Maximum 3D stress intensity, as developed by **CAESAR II**, follows the basic definition of three dimensional stress on an element rather than the piping code's stress equations. This value is one more piece of information for the user's evaluation of the accuracy or acceptability of the piping system model.

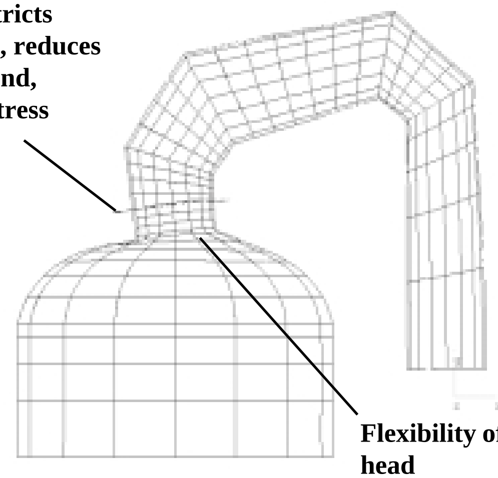
Finite Element Piping and Vessel Stresses

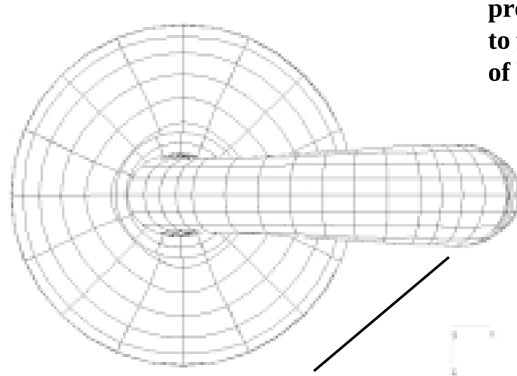
With the release of Version 2.5 of **Fe/Pipe** a detailed stress analysis of vessel heads, miters, "large diameter-thin walled components", cones, annular plates and bends is possible. The models shown below were generated using Version 2.5 of **Fe/Pipe**. Each input took less than five minutes to enter.

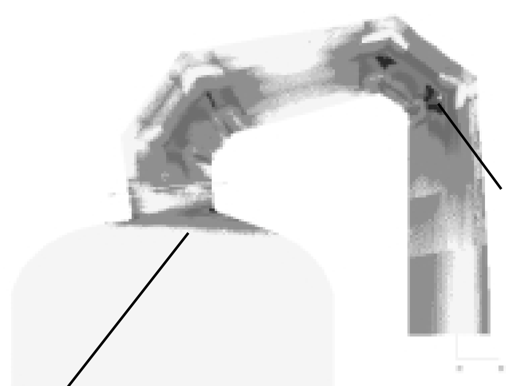




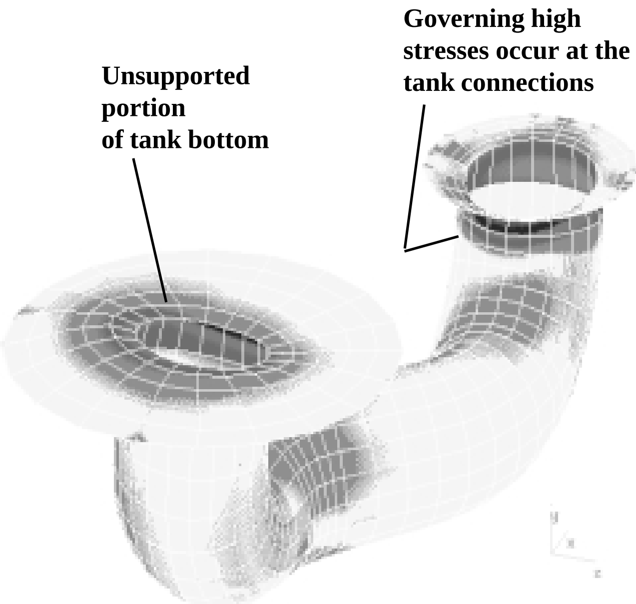
Cutaway view of vessel transition with internal stiffening rings

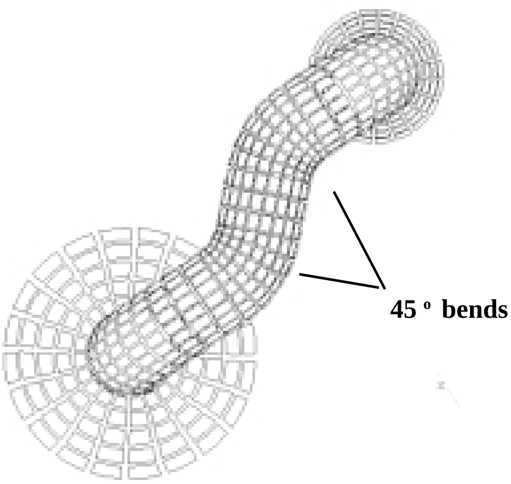






Vessel Head with Mitered Overhead Line 24" diam.





42" Tank By-Pass Piping

Stress intensification factors and flexibilities can be generated for heads, miters and other components that have not been put through the standard Markl type fatigue tests. (SIF's from **Fe/Pipe** have been compared to SIF's from Markl type tests for those components where tests exist.) These SIF's and flexibilities can be used back in **CAESAR II** to generate better load and stress calculations. THIS IS PROBABLY THE MOST PRACTICAL, COMMON USE FOR FINITE ELEMENT CALCULATIONS. FE FLEXIBILITIES OFTEN TIMES SIGNIFICANTLY AFFECT BEAM-TYPE PIPING RESULTS. LOADS ARE REDISTRIBUTED IN THE PIPING SYSTEM SO THAT A COMPLETELY DIFFERENT PICTURE OF THE STRESS AND LOAD STATE IS REFLECTED.

AISC Unity Checks on Pressure Vessel Legs

A common support system for vertical pressure vessels is the structural steel leg. Typical support systems use three to eight legs, of either wide flange or angle shapes. This article will focus on the procedures used to size theses legs and the software COADE provides to assist in these procedures.

A very important point to note is that a wide flange shape is symmetric about its two cross sectional axes, an angle shape is not. It is only with the 1989 version of the AISC code that angle shapes could be adequately checked against combined axial and bending overloads. (This statement is true of the AISC code only. Several non U.S. structural codes have included angle checks for several years.)

The loads on the leg support system are: weight, wind, earthquake, and/or piping induced. These loads produce axial compression and bending. Typical design practice (as discussed in numerous vessel texts) assumes the compressive load is W/N, where W is the total weight load and N is the number of legs. Similarly, the bending moment is assumed to be 4M/ND, where M is the overturning moment, N is the number of legs, and D is the outer diameter of the vessel. The resulting axial load and bending moment are then used in the first two terms of AISC interaction equation H1-1. If the simplification of this equation yields a value less than 1.0, the structural member is adequately sized.

In the above scenario two assumptions were made. First, the loads on the legs were obtained from W/N and 4M/ND. These equations assume the load on all of the legs is the same. Second, the use of only two terms of the AISC interaction equation assumes a planar problem. To explore the effects of these assumptions the analyst must begin with a model of the system which will yield the true loads on the legs.

Consider a vertical vessel supported by four legs. The vessel sits on wide flange columns, four feet high, W8X15. (The column tops are connected to the center of the vessel with beams of the same cross section.) The vessel has a diameter of five feet, a wall thickness of a quarter inch, and a height of fifteen feet. The vessel is assumed to be filled with water and experiences a wind pressure of twenty pounds per square foot, with a drag coefficient of 0.6. These parameters result in the following approximate global loads:

Vessel steel weight (Fy) = 2449 lbs
Vessel water weight (Fy) = 18073 lbs
Structural steel weight (Fy) = 390 lbs
Wind load (Fx) = 900 lbs

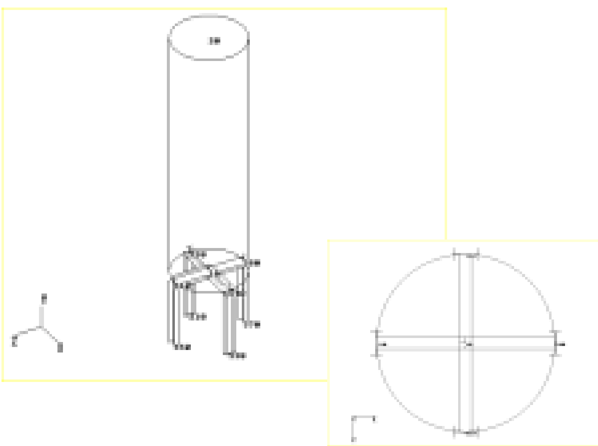
Typical practice is to take these loads and determine the loads acting on individual legs. This would yield:

$$Fy = (2449 + 18073 + 390) / 4 = 5228 \text{ lbs}$$

$$Mz = 4 * (900*7.5) / (4*5) = 1350 \text{ ft-lbs}$$

As expected, we obtain only a single bending moment. Should this moment be applied as strong axis bending or as weak axis bending? What happens to these loads if the wind direction is not aligned with one of the global axis?

To answer these questions and to test the validity of the above assumptions, a **CAESAR II** model of this vessel was constructed. Typical **CAESAR II** plots of the combined model are shown below. Note the plan view and the orientation of the legs.



The initial review of the **CAESAR II** run verified that both the "wind only" case and the "weight only" case produced the correct (anticipated) loads. Now the combined load case results can be used with confidence.

The relevant results from the combined load case (weight + wind) are summarized for three different wind directions in the table below.

CASE 3 (SUS) W+WIN

	WIND at 0		WIND AT 30			WIND AT 45		
POINT	FY	MZ	FY	MX	MZ	FY	MX	MZ
510	3628	198w	3842	-637s	106w	4096	-901s	-1w
520	-3567	-343	-3782	0	-427	-4036	0	-527
530	6829	1166w	6614	-637s	1075w	6360	-901s	966w
540	-6768	1597	-6554	0	1513	-6300	0	1413
550	5228	1275s	6028	-825w	1104s	6360	-966w	901s
560	-5168	0	-5968	-1283	0	-6300	-1413	0
570	5228	1275s	4428	143w	1104s	4096	1w	901s
580	-5168	0	-4367	657	0	-4036	527	0

The "s" and "w" indicate strong and weak axis bending.

A quick scan of these results shows that for all three wind directions, there is always a leg that exceeds the anticipated axial load and bending moment. (For equilibrium there must also be a leg that experiences loads and moments below the anticipated values.)

Once the loads and moments on the legs are known, the AISC interaction equations can be applied. Which legs need to be checked? If the approximate method is used, i.e. M' = 4M/ND, then the load and moment combination needs to be checked twice, once for strong axis bending and once for weak axis bending. The results from **CodeCalc** for the approximate method are shown in the table below.

Strong Axis Bending Unity Check = 0.1326

Weak Axis Bending Unity Check = 0.5479

If the more exact load distribution is used, the following members should be checked:

<u>Wind at 0</u>

530-540, F = 6768, M = 1596; note this is weak axis bending
550-560, F = 5228, M = 1275; note this is strong axis bending

<u>Wind at 30</u>

550-560, F = 6028, M_{strong} = 1104, M_{weak} = 825

<u>Wind at 45</u>

530-540, F = 6360, M_{strong} = 901, M_{weak} = 966

These loads represent the worst loading combinations on the legs. Additional legs and loads can be checked as needed in other conditions. (Since these loads include bending about both axis, the **CAESAR II** AISC module must be used, since **CodeCalc** addresses bending about one axis only. This is consistent with the current procedures found in Pressure Vessel texts.)

In generating the input for the AISC module, an effective length factor (k) of 1.5 was used. Non-braced legs can be approximated by a column with end conditions between guided-fixed (1.0) and guided-hinged (2.0), thus an effective length factor (k) of 1.5 can be used. Diagonal bracing which prevents translation at the top of the legs results in a smaller effective length factor. The output from the **CAESAR II** AISC module is shown in the table below.

```
  Member      Axial                  Fy     Lngth X    UC 1   Unity Chk
  i Node      Bend Y                 Ky     Lngth Y    UC 2   Equation
  j Node      Bend Z                 Kz     Lngth Z    UC 3   Compact
------------------------------------------------------------------------

 {Wind @ 0}
 W8X15          6768.          36000.00       48.      .000     .518
     530.         0.               1.50       48.      .000  H1-3
     540.     19152.               1.50       48.      .518  Yes

 W8X15          5228.          36000.00       48.      .000     .133
     550.     15300.               1.50       48.      .000  H1-3
     560.         0.               1.50       48.      .133  Yes

 {Wind @ 30}
 W8X15          6028.          36000.00       48.      .000     .353
     550.     13248.               1.50       48.      .000  H1-3
     560.      9900.               1.50       48.      .353  Yes

 {Wind @ 45}
 W8X15          6360.          36000.00       48.      .000     .386
     530.     10812.               1.50       48.      .000  H1-3
     540.     11592.               1.50       48.      .386  Yes
```

From these results we can see that the axial stress in the legs is less than 0.15 times the allowable axial stress. (We know this because AISC equation H1-3 was used and equations H1-1 and H1-2 yield 0.0.) Comparison of the first two Unity Check (interaction ratios) values show good agreement between the two methods. However, the skewed loading directions produce Unity Check values which lie between 100% strong axis bending and 100% weak axis bending. This is as expected.

Could one then use the approximate method, and evaluate the Unity Check for "weak axis" bending only? No, this will produce overly conservative results, and in the case where the Unity Check is greater than 1.0 no guidance is offered toward resizing the member. Similarly, using the approximate method and evaluating the Unity Check for "strong axis" bending produces nonconservative results.

The previous discussion is based on using wide flange (symmetric) sections for the support legs. The 1989 revision to the AISC code included a procedure for utilizing the interaction equations for angle sections. This new procedure (new for AISC) involves a rather complex method to obtain the axial and bending allowable stresses. The necessary steps are outlined below (referenced equations are from AISC pages 5-309 through 5-323):

1) Compute the stress reduction factor (Q) using equations 4-3a, 4-3b, or 4-3c.

2) Compute the allowable bending stress for minor axis bending using equations 5-1a, 5-1b, or 5-1c.

3) Determine the cross sectional moments of inertia about the principal axis (of the angle) and the corresponding section modulii.

4) Compute the value of Fob using either equation 5-5 or 5-6.

5) Compute the allowable bending stress for major axis bending using equation 5-3a or 5-3b.

6) Compute the coordinates of the shear center of the angle.

7) Compute the elastic flexural-torsional buckling stress, Fe. This requires solving equation C4-1 or C4-2.

8) Compute the equivalent kl/r value based on Fe.

9) Compute Cc and finally the axial allowable Fa.

10) Apply the Unity Check equations as usual.

The following conclusions can be drawn after a review of the preceding paragraphs.

1) A more exact determination of the loads on the legs is required instead of the approximations W/N and 4M/ND.

2) The complete AISC interaction equations should be employed. This will include the axial as well as both bending terms.

3) Legs constructed of angle sections must be evaluated using the 1989 AISC procedure for angles. It is incorrect to employ the procedures for symmetric sections to angles.

Piping Failure Caused by Elastic Follow-Up

This article has been submitted for publication by Mr. J. Czerwinski, Mr. J. Gephart, and Mr. L. Kimball of Brown & Root Power, Houston, Texas. It is a condensed version of their paper "Case Study: Hot Reheat Piping System Failure Caused By Unforeseen Elastic Follow-Up Condition", presented at the Power-Gen Conference in December of 1991. The unit involved was base loaded and had operated successfully for 105,000 hours.

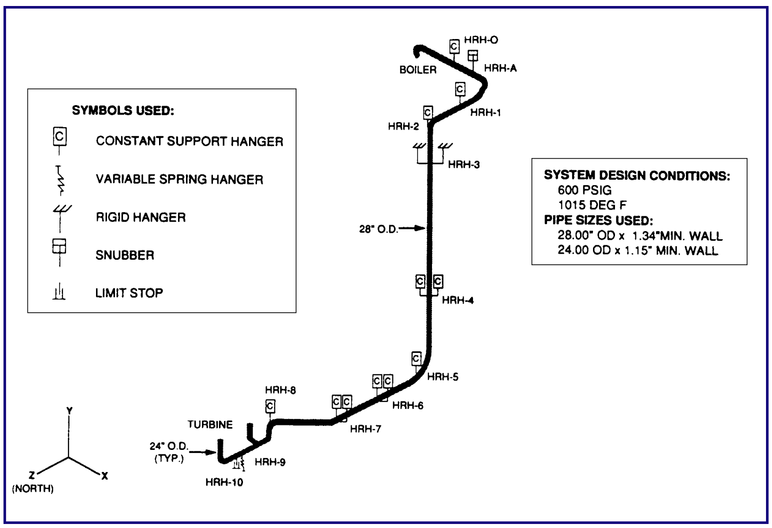
It has been proven many times that high energy piping systems are not maintenance free and have a finite service life. A variety of conditions can cause elevated temperature, high pressure steam piping to deteriorate. This article focuses on one particular failure mode, elastic follow-up, which has been observed in the field.

Piping systems are typically analyzed assuming only elastic behavior. The calculated elastic stresses and stress ranges are then compared to the ANSI/ASME B31.1 Power Piping Code allowable stress and stress ranges to confirm compliance.

However, the Code also recognizes that consideration must be given to piping that is "subjected to strain concentrations due to elastic follow-up of the stiffer or lower stressed portions." Because it is difficult to predict, this is a concern that is mostly overlooked in final design, fabrication and installation of high temperature piping that operates in the creep range of its material.

The Code describes several piping system geometries that can be the catalyst for elastic follow-up. They include:

1) Smaller pipe that operates in a higher stress range than the larger or stiffer pipe to which it is connected.



2) The introduction of reducers or other configurations in which the pipe section modulus becomes smaller.

3) Pipe material that is, or becomes, locally weaker.

4) The use of insufficient offset to absorb the expansion strain of the major portion of the piping system.

An important assumption of elastic analysis that is acknowledged by the code is that piping systems self-stress relieve. This phenomenon occurs when the thermal expansion stresses in a piping system tend to "relax in the hot condition." The expansion stresses then appear in the cold condition. Because they are not applied continuously, these stresses are then treated as secondary stresses. Furthermore, expansion stresses are compared to an allowable stress range which is equivalent to the weighted sum of the Code's cold (Sc) and hot (Sh) stress allowables.

However, the pipe routing and restraint configuration may not allow high temperature piping to self-stress relieve. This is a form of elastic follow-up.

If this is the case, the unrelieved expansion stresses become analogous to primary or sustained stresses, such as those due to deadweight and pressure. Under these conditions, while not stipulated by the Code, it is believed appropriate to add these primary-like expansion stresses to the sustained stresses and compare the sum against only the material's hot sustained allowable (Sh).

The subject unit of this article was placed into operation in 1970. The unit had undergone nearly 300 full thermal cycle start-ups and approximately nine full load turbine trips. In Late June, 1988, while under evaluation, a leak occurred in the Hot Reheat Piping. The circumferential weld connecting the north turbine lead to the reducer of the fabricated lateral at the header termination failed, forcing the unit off line. The failure occurred in the hot Reheat Piping System which had design conditions of 600 psig at 1015 degrees F.

Prior to this failure, there had been no other know failures or repair welding performed on the system. Subsequent non-destructive examination of other welds revealed major cracks in two other nearby welds besides the one that failed.

The conventional elastic stress analysis of the "as exits" Hot Reheat piping indicated that the calculated code stresses did not exceed present day Code allowables during normal operations. The maximum calculated sustained Code stress at the failure location was only 2,114 psi. The expansion stress range at the failure location was calculated to be 8,292 psi. However, the highest thermal stress within the system was calculated to occur at a weld adjacent to the failure. At this weld, considerable creep damage and severe cracking was found. It is now widely known that piping that operates at around 1000 degrees F is prone to creep damage. The degree of creep damage is directly affected by the magnitude of sustained (i.e., primary) stress applied.

The stresses calculated from the original elastic analysis were not high enough to cause the pipe to fail in only 105,000 hours of operation. This would initially lead to evaluator to blame inadequate welds, fabrication error, installation oversights, or improper operation for the failure. ***Still, the amount of visual and creep damage found, points to the existence of primary stresses above and beyond those***

calculated. In this case there is a strong possibility that the elastic analysis did not fully reflect the actual conditions contributing to the stresses in the pipe.

The modelling of the fabricated lateral which had a two inch thick saddle posed a special problem. The actual stress intensification factors were difficult to estimate. In addition, because of the saddle's thickness and size, there probably existed a thermal gradient across its depth. This, in turn, created a local thermal expansion differential. This differential probably caused the saddle to act like a restraint along the top of the lateral, much like an oversized welded hanger lug. ***Unfortunately, conventional elastic stress analysis does not allow such differentials to be considered.***

While an elastic-plastic analysis, using finite element modelling techniques, was considered to further study the stresses at the lateral, the lack of definable local boundary conditions made this technique infeasible from a practical standpoint. Regardless, one of the problems with the subject piping system that caused the elastic follow-up condition is that the system was close coupled. Close coupling occurs when a system is inflexible by virtue of its routing and restraint scheme.

One method of verifying if the piping was close coupled is to test its sensitivity to small changes in the boundary conditions. Close coupling occurs if the calculated restraint loads and piping stresses vary greatly with only minor boundary changes that are well within installation and operating tolerances.

The combination of the inflexible close coupling of the turbine piping risers, the influence of the axial restraint and the lateral's geometry prevented the system from self-stress relieving locally. The piping became highly sensitive to small changes in displacements. This favors the potential for accelerated creep damage. The more highly stressed piping near the turbine is then subject to excessive plastic deformation and strain concentrations due to elastic follow-up from the lower stressed stiffer portions of the piping (i.e., the header).

To recognize the possibility of elastic follow-up conditions, requires an understanding of the phenomenon and experience in analysis, design, and evaluation. The standard elastic stress analysis did not indicate a problem. However, an experienced piping engineer should recognize that the piping is closed coupled around the turbine. Sensitivity analyses involving variations of the turbine movements, cold spring and limit stop gaps would provide sufficient data to show that the pipe could be susceptible to elastic follow-up. Therefore the secondary stresses convert to the primary level. Hence, it is probable that the actual primary stresses, especially the bending stresses, at the least stiff regions of the subject area, far exceed those calculated by the elastic analysis.

In conclusion, creep damage results from the application of inordinately high sustained (or primary) stresses. These stresses can be due to unanticipated piping movements and rotations, unrelieved expansion stresses becoming primary in nature and improper application of pipe restraints. ***This can happen even though a conventional elastic stress analysis may predict that the system satisfies the B31.1 Code.*** Unanticipated piping displacements can be the result of numerous situations including the improper application of rigid elements, such as welded attachments, poorly designed restraints, installation errors, incorrect cold spring and thermal gradients within a pipe element or attached equipment.

The pipe routing and hanger/restraint configuration must prevent stress concentrations resulting from elastic follow-up. That is, the piping must be supported so that self stress relieving of the thermal expansion stresses cannot occur. If self stress relieving does not occur, then the expansion stresses become primary in nature and must be included with the piping sustained stressed for Code compliance evaluation.

CAESAR II Specifications

Listed below are those bugs/errors/omissions to the **CAESAR II** program that have been identified since the last newsletter. These items are listed in two classes. Class 1 errors are problems or anomalies that might lead to the generation of erroneous results. Class 2 errors are general problems that may result in confusion or an abort condition, but do not cause erroneous results.

Class 1

1) <u>Dynamic Restraint Load Module</u> - For independent support motion spectrum jobs in which the pseudostatic component is included, the restraint loads on rotational restraints are computed incorrectly. This error exists only if the pseudostatic component is included and the restraints are of type RX, RY, or RZ. This error exists in all 3.x versions, and will be corrected in version 3.18.

2) <u>Piping Input Module</u> - In Versions 3.0 through 3.17, the "Block Rotate" module of the LIST processor could erroneously free a specified zero displacement. This error exists only for values specified as 0.0, when the rotation angle specified is 90 degrees.

   For example, assume displacements of: DX=0.0, DY=3.0, and DZ=2.0, with a requested rotation of 90 degrees about the "Y" axis. The resulting "rotated" displacements would be: DX=2.0, DY=3.0, and DZ=FREE. This is incorrect, DZ should be 0.0.

This error is corrected in Version 3.18.

3) Piping Error Check Module - An error has been found in the Piping Error Check module which affects partially modified uniform load specifications.

For example, assume on element 40 to 50 the uniform load specification is: UX=10, UY=15, and UZ=20. Assume several elements later on 100 to 110, it is necessary to change the UY value to 30. The user would see the following auxiliary screen:

Original	Modified	Comment	Actual
UX=10	UX=10	duplicated forward value	UX=0
UY=15	UY=30	user modified value	UY=30
UZ=20	UZ=20	duplicated forward value	UZ=0

Since the only value specified on 100 to 110 element was UY, the Error Checker assumed incorrectly that the UX and UZ values should be zero. This caused the duplication of the UX and UZ loads to stop at element 100 to 110.

The work around for this error (until Version 3.18 is released in August or September) is to completely specify all three uniform load vectors any time a new uniform load auxiliary screen is pulled up. In the above example, if the UX and UZ values had been respecified (as 10 and 20 respectively) along with the UY value, the uniform loads would have been duplicated forward as anticipated.

This error exists in all 3.x versions, and will be corrected in Version 3.18.

4) Dynamic Results Module - An error has been found in the dynamic results summation module when the ASME NC or ND codes are used. The error involved the use of the peak pressure variation in the modal summation routine, causing a pressure stress to be computed and summed for each mode. This produced very large values of the "code" stress. This error exists in Versions 3.0, 3.1, 3.15, 3.16, and 3.17. It will be corrected in Version 3.18.

5) Mechanical Stress Computations - An error has been found in the mechanical stress computation routine. The value of the otrsional stress for the NC and ND codes was inadvertently multiplied by the B1 index. This error exists in Versions 3.0, 3.1, 3.15, 3.16 and 3.17. It will be corrected in Version 3.18.

Class 2

1) Piping Input Module - When requesting "help text" for the insulation cell, the display routines do not properly convert the default density for calcium silicate into the current active units system in the explanatory text. However, the proper density value is converted and presented in the table below the text. This error exists in all 3.x versions, and will be corrected in version 3.18.

2) Piping Input Module - A memory allocation problem prevents access to the "help facility" when on the main "LIST" display screens. This error exists only in Version 3.17.

3) Piping Input Module - The screen attributes (color) for the material id cell and the Y/N cells are still displayed in the black on white combination. This error exists only in Version 3.17.

4) Piping Input Module - The screen attributes (color) for the valve & flange selection screen default to combinations which are indistinguishable on lap tops and other single color screens. This can be corrected by resetting color values 5 and 8 via the C2SETUP program (option 9 from the main menu). This error exists only in Version 3.17.

5) WRC297, SIF, & FLANGE Module - The cursor pad can not be used to select option 5 from this menu. You must strike the "5" key to activate this option. This error exists in Version 3.16 and 3.17.

6) WRC297, SIFs & Flange Module - An error exists in the error checking routine for Bonney Forge Sweepolets which requires the weld type to be 0, i.e. as welded. Any attempt to change the weld type to 1, ground flush, results in an error preventing the computation. This error exists in Version 3.15 through 3.17.

7) External Interface Menu Module - The "hot" keys can not be used to select options from the second column of the menu. You must use the cursor pad to select these options. This error exists only in Version 3.17.

8) Output Processor - Version 3.17 incorporates a routine to verify the current input file produced the current output file. This routine relies on the OTL file generated by the piping error checker.

Users moving up to Version 3.17 from older versions will not have OTL files unless they error check their input again. This does not require another analysis. Simply enter the input, quit and select the option “Start the Run”. This will invoke the error checker and build the necessary OTL file, allowing output processing. (A warning will be generated that the input has an earlier time/date stamp than the output file and that input echo from the output is disabled. This is expected since starting the run rewrites the input file. If you need to produce an input listing from the output module you will have to rerun the analysis.)

Users analyzing structural only jobs will find they can not enter the output processor. This is due to the fact that the piping error checker was skipped for the current job. To remedy this problem, create a piping model with a single dummy pipe (1" long, 1" diameter) and attach it to the structure using the “INCLUDE” option. Starting the run from the piping input module will invoke the error checker and generate the necessary OTL file

.

9) Piping Error Check Module - Version 3.17 allows material id 21 to be used as a “user defined” label. A limit in the error checker limits material ids to 20 *for all but the first element in the job*.

10) Dynamic Output Module - An error exists in a file open statement which prevents the output from being sent to a disk file if the disk file already exists. This error exists in all 3.x versions, and will be corrected in version 3.18.

11) Buried Pipe Input Module - An error exists in Version 3.17 which prevents the graphics display (plot) of the piping system within this module.

12) Buried Pipe Input Module - A conversion error caused the display of the default pipe expansion coefficient to be displayed incorrectly when unit systems other than the default English system are used. This error is a display error only, in Versions 3.1 through 3.17.

COADE Engineering Software
12777 Jones Rd. Suite 480, Houston, Texas 77070 713-890-4566 (Tel) 713-890-3301 (Fax)