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

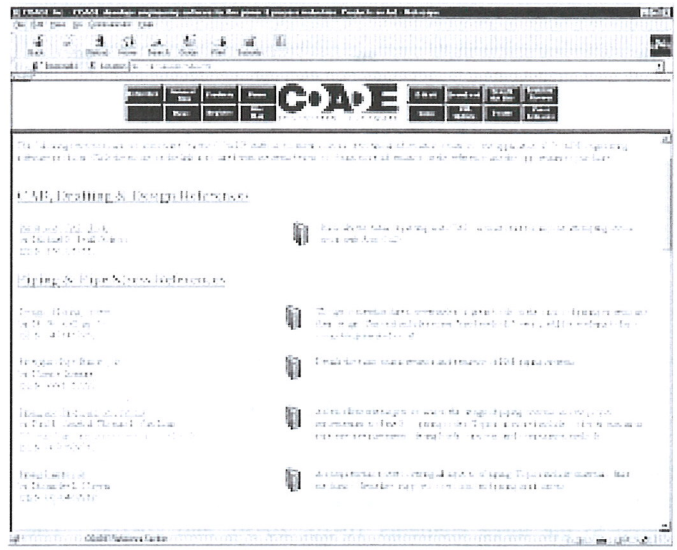
The entire staff at COADE would like to extend our thanks and appreciation to all those who attended this conference. The comments and suggestions made will be used in preparing the next conference.

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.

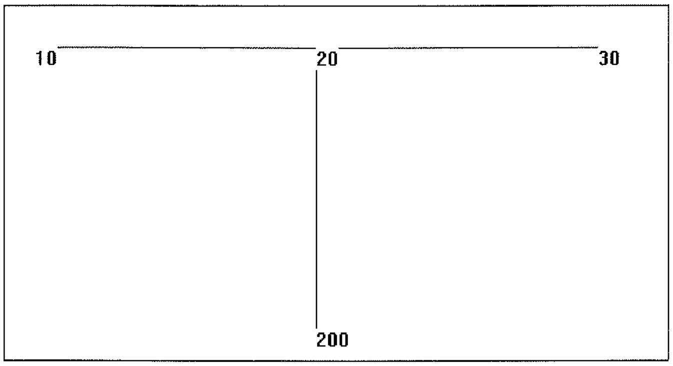


Figure 1 – Typical Tee Coding

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.

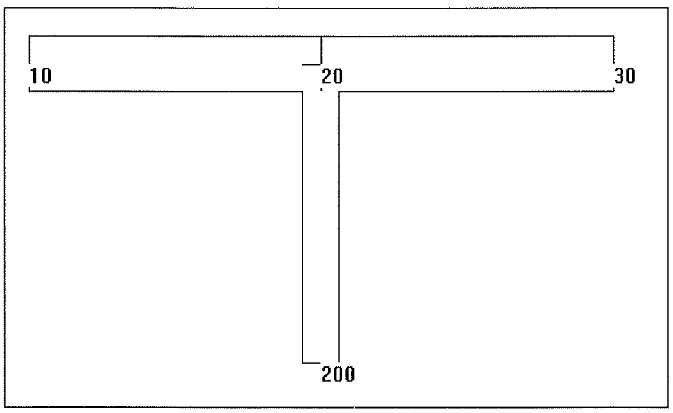


Figure 2 – Analyst's View of a Typical Tee

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).

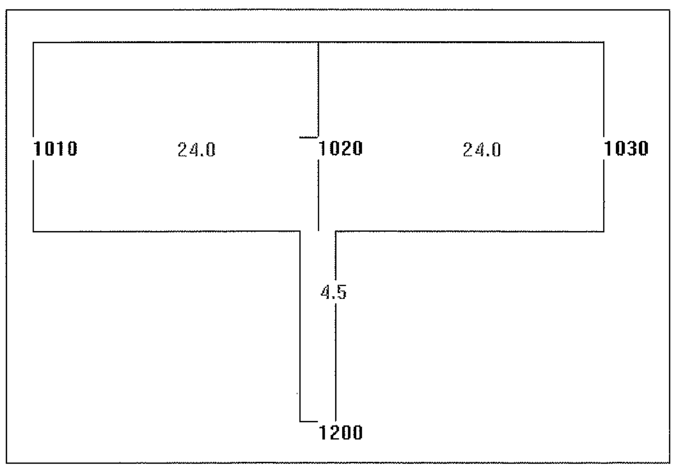


Figure 3 – A Large D/d Tee

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.)

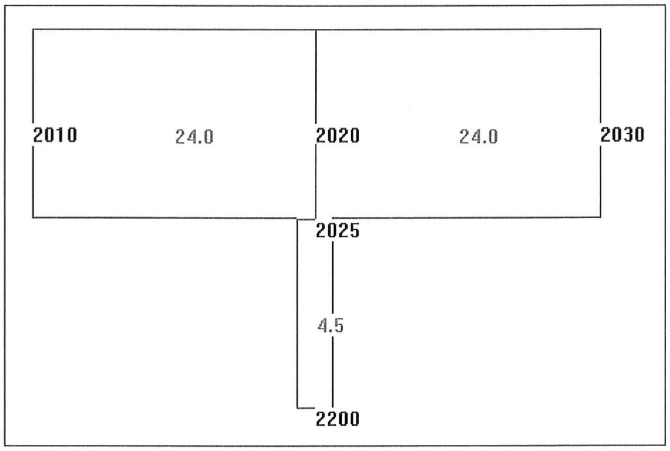


Figure 4 – A Large D/d Tee with Dummy Element

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:

$$T_r = (P * R) / (S * E - 0.6 * P) \text{ per UG-27}$$

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:

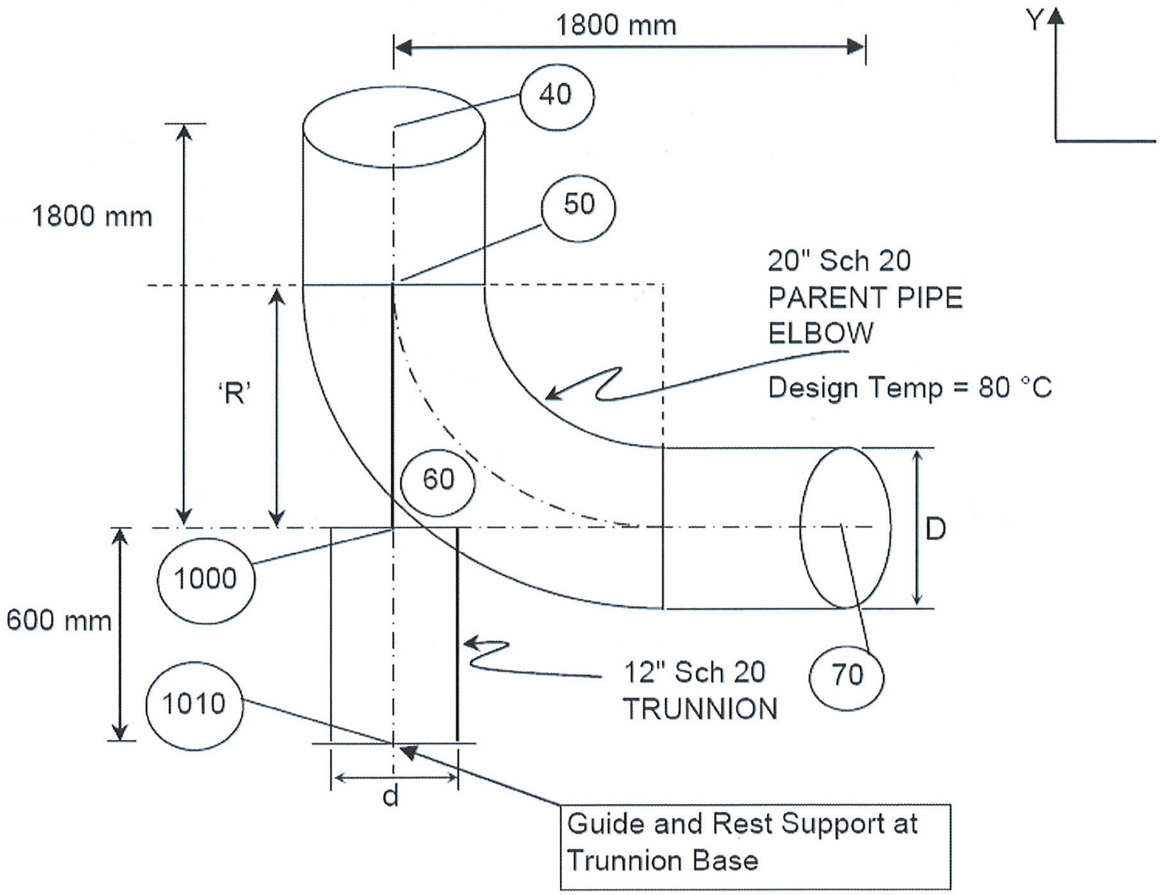
Thickness Due to Internal Pressure (TR):
= (P*R)/(S*E-0.6*P) per UG-27 (c)(1)
= (1000.00 * 96.0)/(20000.0 * 1.0 - 0.6 * 1000.0)
= **4.9485** in.

Trunnion at Vertical Elbow

Trunnion at vertical elbow shall be modelled using two elements:

a. one rigid element from the elbow end point to the elbow corner along tł of the pipe (parent pipe design temperature)

b. pipe element with trunnion size and length (average of pipe design and ambient temperature)



iv. Trunnion at Pipe Elbow (Continued)

Trunnion at Horizontal Elbow

Trunnion at horizontal elbow shall be modelled using one rigid and two pipe elements:

a. one rigid element from the elbow end point to the elbow corner along the centreline of the pipe (parent pipe design temperature)

b. pipe elements with trunnion size and length (average of pipe design and ambient temperature)
   1. from end of the rigid element up to support point
   2. from support point to edge of trunnion