Analysing FCC hot spots

Finite element analysis can be employed to improve the safety and quality of old or new designs and troubleshoot problems in the field

KENNETH FEWEL
Technip Stone & Webster Process Technology

Hot spots on FCC units are a common occurrence but relatively little is known about the implications for long term operation. Standard procedures in a FCC plant are to monitor on a scheduled basis the thermal scans of their equipment and use steam to cool if hot spots exceed a certain temperature threshold. Although this has served operators in a practical manner for years, there are larger safety issues that should not be ignored. ASME has developed standards for analysing thermal stresses that are suitable for design, maintenance and operation. Operators should be aware of these methods and take advantage of them for their hot spot maintenance and procedures.

Finite element analysis (FEA) is a method that enables accurate state-of-the-art analysis of stress and strain in all types of solid materials for all types of loadings, including thermal. Thermal stresses are usually calculated for a design that is working perfectly to insulate the steel shell from high temperatures inside the FCC vessel. FEA has been used to validate traditional calculation and more accurately define the state of temperature, stress and strain that will occur under normal operation in FCC units made from composite, refractory-lined steel, materials.

Recently, as a result of hot spots appearing on FCC equipment, there was a need to analyse and determine the severity of stresses and strains under hot spot operation. This has provided an excellent opportunity to use FEA to explore these situations using quantitative analysis and make recommendations to operators about the safety of their particular situation.

Before further explaining FEA, it is important to understand why hot spots occur and how stress and strain occurs.

Hot spot causes

Hot spots are caused by several mechanisms of operation that come together like a 'perfect storm'. They do not appear randomly as there are reasons why hot spots form in FCC equipment.

1. Erosion by catalyst

Catalyst is, by nature, irregular in shape and very hard compared to carbon or stainless steels. The refractory used to line steel vessels or piping is designed to provide insulation and strength for stresses and thermal strains; it is not designed purely for erosion resistance. Therefore, the material is susceptible to erosion by catalyst. Fortunately, this insulating refractory is thick, usually 4-5in in piping and vessels. However, catalyst can quickly erode through this thickness with the help of the next mechanism.

2. Thermal stress and strain

a. Wall stress: thermal wall stress can easily reach levels that will crack the refractory. In many designs some cracking cannot be avoided. Longitudinal cracks in the refractory outer diameter of lining will form as soon as temperatures reach operational levels. This is due to thermal wall stresses in the refractory. These stresses have been determined from research experiments by Wygant and Crowley[1] among others.[2] The inner diameter of the lining is very hot, particularly in a regenerator, typically 1300°F (700°C). The inner refractory will go into compression when internal temperatures start to rise. The outer surface of the lining may not expand as much as the steel. Thus the lining will go into tension stress and the refractory may not bear upon the inner diameter of the steel piping. If so, thermal wall stress can cause the lining in the outer edges to crack.

b. Thermal bending stress: thermal expansion forces in the piping and vessel can crack the refractory in bending, often transverse to the longitudinal direction. In addition, compression bending stress can open gaps between the steel and lining, leading to large gaps through which vapour and catalyst can easily circulate. These are the most damaging stresses to the refractory that can occur. The resulting hot spots will exacerbate the problem, making the thermal stress and gap worse.

3. Worm holes

As a direct result of wall stress cracks in the refractory lining and gaps opened by thermal bending of the refractory on the outside of the lining, there is a strong possibility that the hot vapours and catalyst will find alternate paths through the piping. The wall stress cracks are probably not large enough for catalyst to circulate through, and they may fill with catalyst instead. A larger problem is the thermal bending strain in the refractory. Thermal bending stress and strain will open existing cracks or form new cracks large enough for catalyst to circulate through. Worm holes can then form, leading to catalyst circulating past the steel, overheating, and rapid erosion of the steel skin from the inside out.

The most apparent result of a combination of these mechanisms is a hot spot. The steel will often reach temperatures in excess of piping or vessel design values, typically around 650°F (340°C). When the skin reaches temperatures higher than design, the situation requires cooling steam to keep the temperatures under control. This is standard maintenance procedure, but what are the resulting safety and long term implications? When is a shutdown necessary? If a shutdown is planned, what needs to be done? Is the steel and/or lining material ruined or can it be reused after shutdown?

Usually, the answers are not so clear to the operator as a potentially dangerous situation arises. What if the steam pressure is lost temporarily? What if the temperature rises? Continuous monitoring of the steam cooling system is rarely done. The thermal scans are not continuously reviewed either, but done only periodically. How can an operator know that his plant is safe for the next day or week, or until the next scheduled shutdown?

These questions should not be answered by rough estimates or guesses. Hot spots, once formed, cause additional thermal stress. When a hot spot appears, the situation will worsen and thermal stress and strain can quickly become unstable.

How hot spots cause thermal stress

Take, for example, a thin plate held rigidly at the edges and heated with a torch at the centre of the plate. The material in the centre will begin to expand and thermal stress will occur. The edges, if truly rigid, will prevent expansion from occurring in the plane of the plate. Once the stresses exceed the proportional limit, the plate will begin to warp out of plane. Imagine that this happens on the surface of a large catalyst transfer pipe. The steel must bulge outwardly but will be resisted by the geometrical constraints of the adjacent cooler pipe and, to a lesser extent, refractory anchors. But the gap between the steel and refractory is larger now and the circulation of catalyst is naturally increased. The hot spot temperature is increased and the thermal stress becomes higher. This instability will not stop until one of the following mechanisms occurs:

1. The catalyst streaming in builds up to fill the void left by the steel strain and the gas bypass flow is reduced or eliminated.
2. The strain hardening in the steel, already in the plastic region, reaches a point of equilibrium before the rupture point is reached. This is usually assisted by a geometric constraint and/or steaming.
3. The steel reaches rupture point. Catalyst and vapours begin to spew from the opening.

Obviously every operator hopes for number 1 or 2 to occur. Despite denials, number 3 has been known to happen.

Based on the Goodier[3] equation for thin circular plates unconstrained at the edges, the following relationship can roughly estimate the thermal stress rise due to additional temperature rise in the elastic region:

$$d\sigma_r = d\sigma_t = d\tau_{rt} = 0.5 \cdot E \cdot \alpha \cdot dT$$

where:
σ_r = radial stress in the hot spot
σ_t = tangential stress in the hot spot
τ_{rt} = shear at the edge of the hot spot
dT = temperature change
E = elastic modulus of the material
α = coefficient of thermal expansion

Of course this must be added to the normal operating weight, pressure and thermal stress state which existed before the hot spot formed. The sample field data employed in **Figure 1** show that geometric configuration and pre-stress matter greatly because there is much scatter in the equivalent stress data. Where pre-stress is largely compressive, hot spot stress will simply add more compressive stress to the negative components of pre-stress. Where pre-stress is mostly tensile, the hot spot will lower or reverse the tensile components and raise the compressive components. In addition, geometric conditions such as crevices, bending stress and other stress risers will play an important part. Finally, vibration will also increase the general state of stress with reversing components that will complicate the situation. Vibration stress is almost always present in an FCC and is largely disregarded, despite the fact that it can be significant.

Combining all of these factors makes a traditional analysis very challenging. It can be done, but the results will be highly uncertain, could be inaccurate, and likely misleading.

Methods are available to determine the complete state of stress and to mitigate the causes. This is where the application of FEA modelling can help.

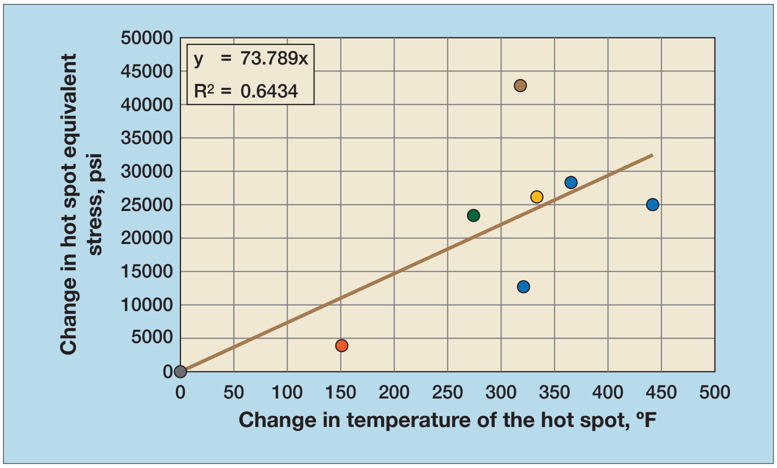

Figure 1 Rise in equivalent stress as a function of rise in hot spot temperature

The FEA modelling approach to lined pipe hot spot analysis

An FEA model can analyse hot spots in detail. With the assistance of thermal scans, the hot spot can be replicated in a model. By doing so, the state of stress caused by the hot spot can be determined with a high degree of accuracy. These thermal stresses can be compared with ASME allowable to determine if continued operation is safe. **Figure 2** illustrates a FCC lined pipe hot spot model and the associated thermal state.

ASME Section VIII, Division 2, has developed an analysis method for determining the safety of steel thermal stress.[4] The ASME piping code B31.3[5] can be used also, but this will result in more conservatism as the piping code is based on traditional calculations. Both have been used for most studies, as a check against one another. The

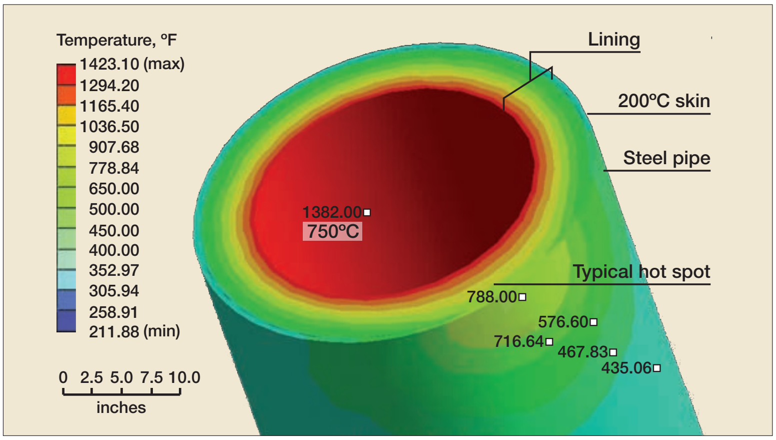

Figure 2 Thermal gradients in typical lined FCC pipe modelled with heat transfer through both materials using FEA

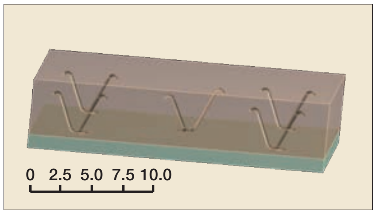

Figure 3 FEA sub-model of steel plate, refractory lining and stainless anchors

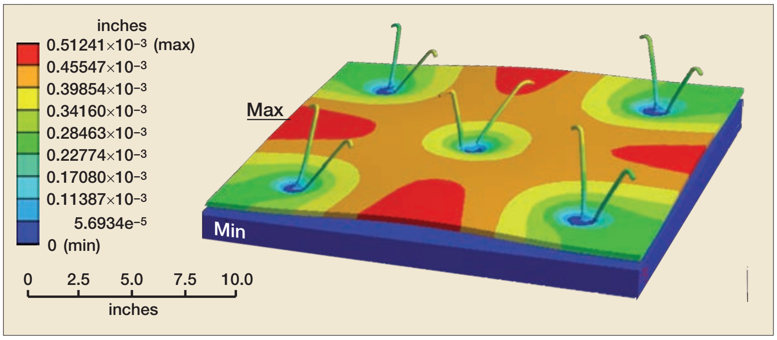

Figure 4 ANSYS FEA sub-model of steel and lining pulling apart. Colours show deformation with highest shown in red and lowest in blue. Refractory has been cut away to show the anchors clearly

results were similar although stresses checked against B31.3 had a slightly lower margin of safety.

Refractory stress can be checked also. Although allowable material limits are not available in ASME, the manufacturer's strength properties can be used with a safety factor applied. Compared to steel, refractory strength is very low. Although compressive strength has been stated as high as 5000 psi, tensile or rupture stress strength, determined from bending specimens, is generally less than 1500 psi, even though stainless needles are used in many formulations to improve tensile strength. Fortunately, the elastic modulus is generally less than steel and therefore flexibility is greater. In addition, the flexibility of the anchors is fairly high and this isolates some of the steel strain from the refractory strain and vice versa (see **Figures 3** and **4**).

Despite the use of anchors and needles, refractory is very susceptible to cracking under operating conditions due to thermal stress. If the design is not analysed carefully with the use of FEA, areas of critical bending stress in the refractory are usually missed. This is because most piping is analysed with a one-dimensional piping stress program. These programs do not provide for placing discrete refractory material in the piping model. Although the added strength of the refractory can be factored in, stress in the refractory material often cannot be determined. If the refractory reaches the bending rupture point, the contribution to overall strength is lost. Yet the one-dimensional approach would not take this into account. Therefore, this approach could be non-conservative if the refractory strength is factored in and the refractory is actually breaking.

FEA is the most advanced means to analyse the refractory-lined pipe in FCC systems. No other method can supply the detailed stress state or the design quality for both steel and refractory lining in one model.

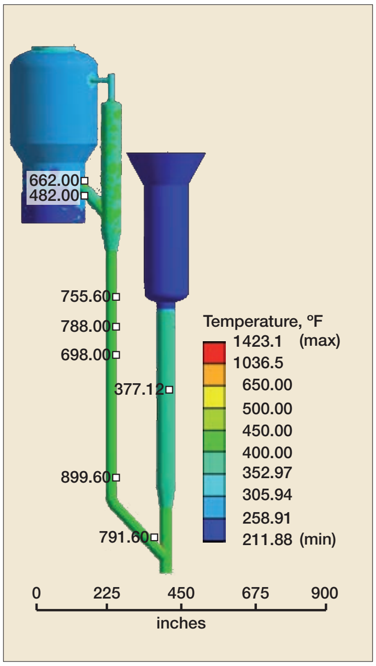

Figure 5 Field case situation: hot spot temperatures found from thermal scans, steam on

Case history: FCC hot spot FEA

A major European operator was experiencing an increasing number of hot spots on transfer lines and nozzle connections. The plant had vessels bulging, pipes rupturing, and hot spots occurring across the piping system's slide valve, cyclone dip-leg nozzle and other transfer lines. The long standpipe and riser transfer piping had no expansion joint so thermal stress and strain were suspected. A transfer line model was set up to determine the complete state of stress in the affected piping. **Figure 5** shows hot spot tagged by temperature with existing steam cooling.

In addition to a model of normal operation under weight, pressure and thermal loads, a second model was set up and the hot spots were added.

Figures 6 to **8** show the temperature contours of model setup and resulting stresses and deformations for typical parts of a catalyst transfer system.

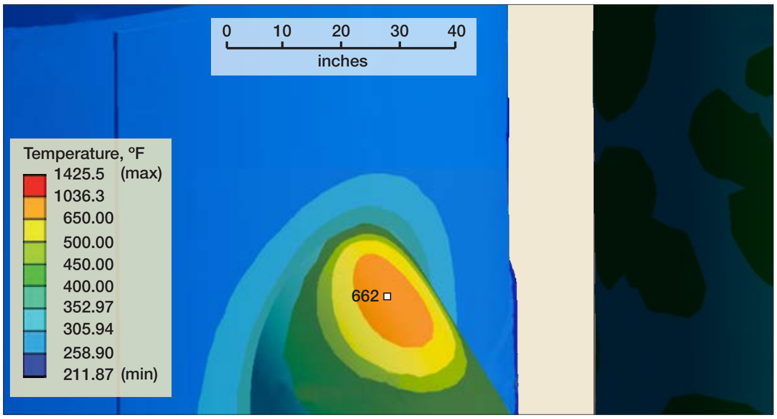

Figure 6 FEA replicated hot spot on regenerated catalyst withdrawal well nozzle, top side. High stress existed here because of normal operating thermal tension load in the regenerated catalyst standpipe

The results of the study were very revealing. **Figure 9** shows computed displacements later verified by the operator. **Figures 10** to **11** illustrate the stresses in the reactor cyclone dipleg skin and lining. **Figure 12** summarises safety factors. All of the thermal hot spot stresses could pass ASME Section VIII, Div 2, part 5 rules. The operator decided to continue running the plant to the next scheduled shutdown with confidence.

Pre-stress and hot spots: causal relationships

In almost every case, high levels of operating stress and strain coincide with hot spot locations. The hot spot temperature can be correlated by the operating stress and strain at the hot spot. **Figure 13** illustrates this with a graph of hot spot temperature vs steel operating stress.

Although the data are scattered, the relationship between pre-stress and hot spots is clear. Faulty thermal design causes high operating stress and this leads to cracking of the refractory, perhaps during the first start-up. From there, it may be only a matter of time before the hot spots appear.

Use of FEA to analyse FCC designs

The detailed design process for FCC equipment should

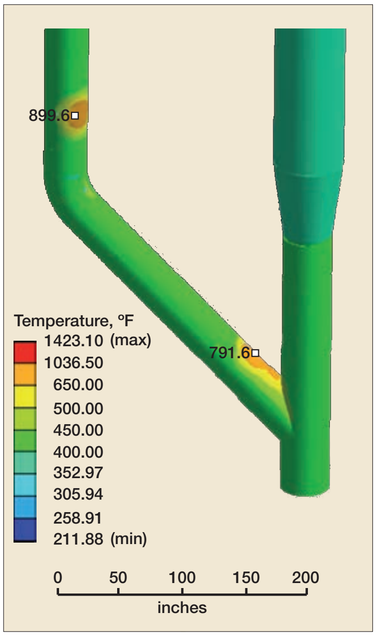

Figure 7 Standpipe to riser wye piece; hot spots appear in high bending stress areas

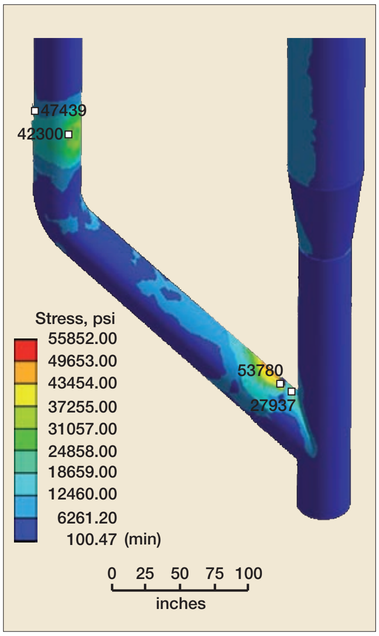

Figure 8 Stresses found with FEA in the wye, probably due to bending failure in the refractory

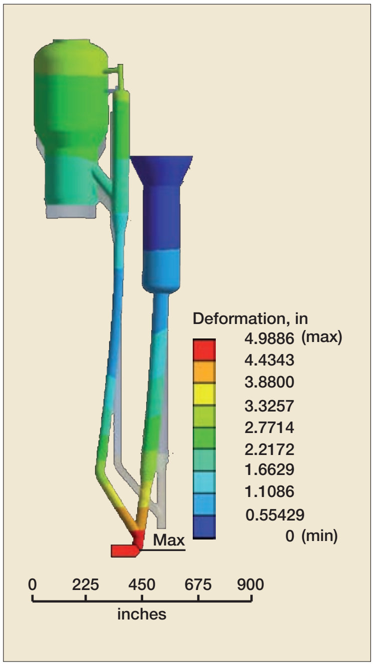

Figure 9 As a benefit of an accurate thermal strain state, FEA will yield very accurate thermal displacements. The operator measured a 4in lateral movement with hot spots, matching the FEA closely

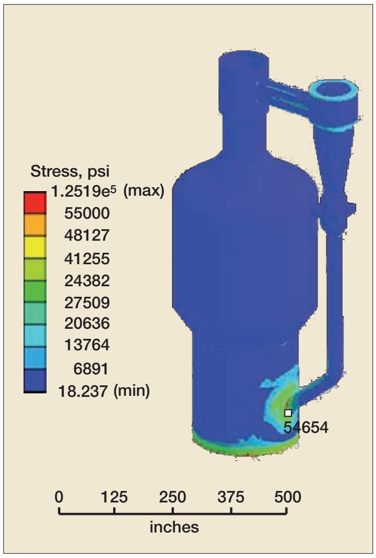

Figure 10 Skin stress on an external FCC reactor with hot spot around the nozzle connection

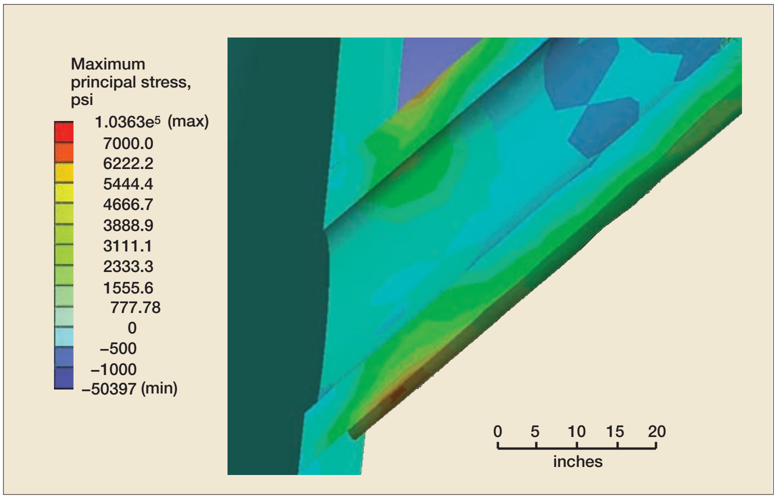

Figure 11 Dipleg refractory lining stress contours at the hot spot indicating ultimate failure of refractory throughout thickness

rely on an FEA stress model to determine the complete thermal stress state before an FCC is built. FEA will reveal where thermal stress and strain is high, allowing custom changes to the design and possibly preventing hot spots. This premise is based on experience with hot spots in the field. Most, if not all, cases are associated with thermal stress and strain, the key causal elements in hot spot formation. If these are eliminated, erosion is the likely reason for a hot spot to form and that could take many years to happen.

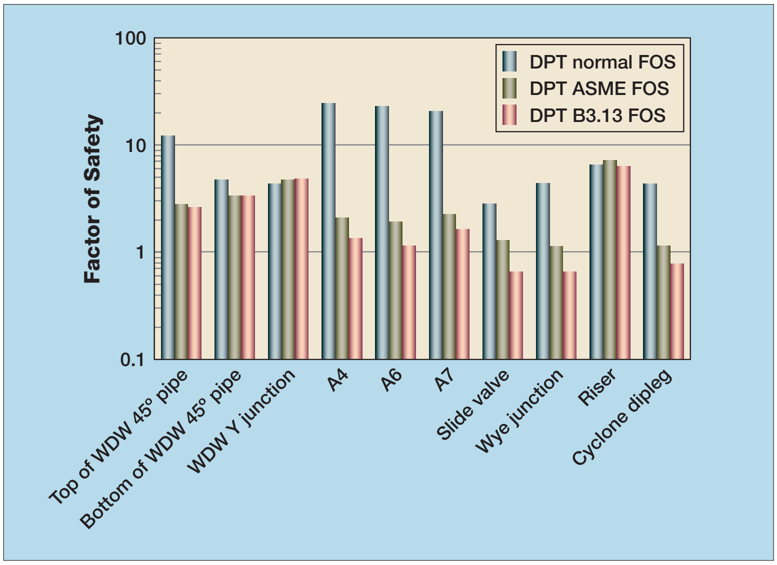

Figure 12 Factor of safety (FOS) comparison for dead weight, pressure and thermal (DPT) case at new condition vs hot spot operation. A factor of safety of 1 or greater is a passing grade, indicating that computed stress is lower than allowable stress. Normal FOS indicates as designed. For hot spot cases, ASME indicates Sec 8, Div.2 evaluation and B31.3 is a piping evaluation

Once a hot spot has appeared, an FEA analysis can determine if the stresses are high enough to cause alarm. Variables include temperature, size, prestress, position, geometric factors and material properties. It is difficult to know with certainty if a hot spot is a safety issue without a detailed FEA analysis.

The analysis can take all of these factors into consideration: anchors, refractory strengths, refractory expansion coefficients, piping material, weight, pressure and thermal stress. Even vibration and fatigue can create damaging refractory stress and strain to occur, leading to hot spots and the possibility of rupture. This can be analysed with a dynamic stress FEA model of the piping or vessel.

Conclusion

FEA is a maturing technology used more frequently as the tools become more widespread, user friendly and real world effective. With proper setup, verification of computations and experience to evaluate the results, this engineering tool can be a means to improve the quality of old or new designs and troubleshoot problems in the field with the vision like no other method. Do not discount FEA as unnecessary, too complicated or expensive as the investment returns can greatly advance engineering and product quality.

References

1 Wygant J F, Crowley, *American Ceramic Society Bulletin*, Vol. 43, No. 3, 1964.

2 Buyukozturk, Oral, Tseng, *Journal of the American Ceramic Society,* Vol. 65, No. 6, Jun 1982.

3 Goodier J N, Thermal Stress, *ASME J of Applied Mechanics,* Vol 4, No 1, 1937.

4 ASME Section VIII, Division 2, Part 5: Design by Analysis, 2007.

5 ASME Process Piping, B31.3, 2006.

Kenneth Fewel is a Senior Supervising Engineer with Technip Stone & Webster Process Technology. With 40 years of industry engineering experience, he holds a bachelor's degree in mechanical engineering from Southern Methodist University and a master's from the University of Texas. PE licensed, he holds seven US patents, received numerous professional awards, and is a Fellow of the American Society of Mechanical Engineers. *Email: kfewel@technip.com*