A COMPARISON OF FINITE ELEMENT CODES
AND
RECOMMENDED INVESTIGATION METHODOLOGY

Michael A. Porter
Dynamic Analysis
Leawood, Kansas

Dennis H. Martens
Black & Veatch Pritchard, Inc.
Overland Park, Kansas

Charles S. Hsieh
Black & Veatch Pritchard, Inc.
Overland Park, Kansas

ABSTRACT
Significantly different results attained from the use of three Finite Element codes used in the analysis of a large complex model are discussed. Building on previous work by the authors regarding the comparison of stress results from several commercial FE codes used on a simple model, this paper recommends steps for an investigation methodology to aid in ascertaining results which are most representative, useful and correct.

DESCRIPTION OF PROBLEM
The current commercial finite element (FE) codes represent an effective analysis tool for the investigation of critical aspects of pressure containment equipment. The need to formulate an adequate model, apply the correct loadings and confirm the results represents a major portion of the engineer's analysis task. Once a valid model is constructed, the variability of results produced by the different FE codes remains a concern. To evaluate the magnitude of these variations, we will examine a relatively large and complex model. This model will be "solved" using three commercial FE codes. The results, in terms of the indicated Stress Intensity, will be compared to investigate the variability of the results.

For this comparison, the same model was used with each FE code, except for some minor variations required to adapt to the specific code. The results derived from the three codes were compared to closed-form solutions for pressure and thermal displacements. The variability of the indicated displacements and stresses were then compared to assess the ability of the commercial codes to give reasonably accurate and consistent results. This methodology of establishing a valid model of the problem, verifying the model by closed-form solutions, and comparing and interpreting the results forms a basic protocol that can be extended to other FE analysis work.

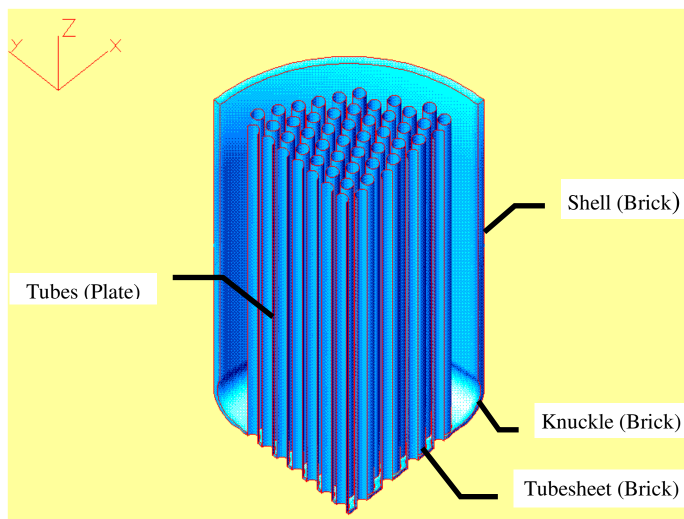

Figure 1 - Pictorial View of Heat Exchanger Model

FEA MODEL
The model used for this project was that of a shell and tube heat exchanger. Figure 1 illustrates the model pictorially. The various components of the exchanger model are indicated along with the type of element used in the model. Not indicated are the beam elements that were used to attach the tubes to the tubesheet. This form of connection resulted in some high indicated stresses at the attachment points. This modeling technique was employed as a means of limiting the model size. Since prior work had shown that the stress in the tubes was not a primary concern, the stress at these junctures was not evaluated.

Figure 2 illustrates a cross-section through the knuckle portion of the model. As may be seen, the material thickness in the knuckle portion of the vessel was less than the shell thickness and greater than the tubesheet thickness. Although a formal mesh convergence study

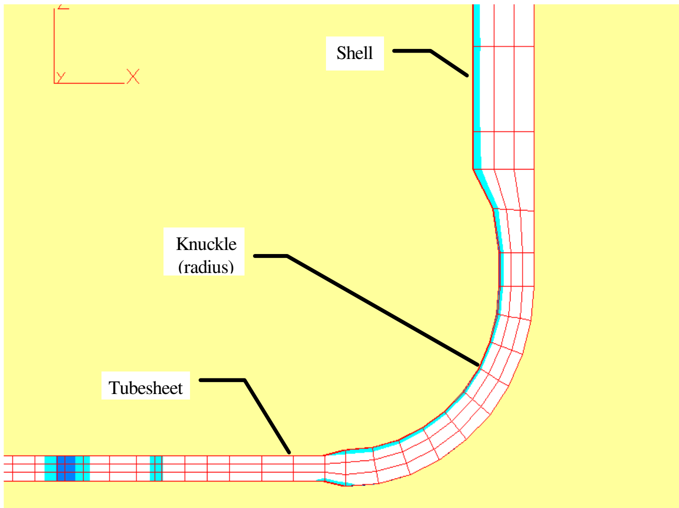

Figure 2 - Cross Section Through Knuckle Region of Model

was not conducted, several iterations of this geometry were employed to assure that the elements in this area were reasonably behaved.

The completed model has approximately 11,390 nodes that define some 8,500 elements. Of this total, approximately 4,500 elements were the brick elements used to model the shell, knuckle and tubesheet portions of the model. Symmetrical constraints were applied to the cut surfaces indicated in the model, so that only 1/8 of the exchanger (as illustrated) was required for the analysis. The assembled model had approximately 41,000 degrees of freedom. The solution times ranged from about 3 to 6 hours on Pentium 90-100 computers, depending on the code used.

COMPUTATION OF DISPLACEMENTS AND STRESSES

The original model was constructed using one of the popular PC based FE packages (code A). As a means of checking the results, a geometrically similar model (using most of the geometry of the original model) was constructed using another FE package (code B). The results (stress) obtained from the two models differed by approximately 30% in the critical knuckle region! In order to try to resolve the differences, several checks of the results were performed using closed-form "hand" approximations.

In the appendix, equations used to approximate the hoop stress, radial expansion and axial expansion are presented. All these equations may be found in Roarks's Formulas for Stress and Strain. Note that such a reference, in the authors' opinion, is a mandatory tool for anyone conducting FE analyses.

For this vessel, the computed hoop stress using the equations in the appendix is approximately 16,200 psi. The computed radial displacement is approximately 0.035" and the computed axial extension for the tubes and tubesheet is approximately 0.026". However, the above analysis ignores the restraint that the shell will contribute to the axial movement of the tubesheet. Thus, we would expect that the actual extension will be less than computed above.

When the FE results of the two models were compared with the values computed above, the displacements in the code B model were found to be almost an order of magnitude greater. Further examination of this model indicated that the Young's Modulus (E) value that had been entered in code B (by the author, unfortunately) was low by an order of magnitude. This value was corrected and the displacements and stresses were re-computed. The results from codes A and B using the correct E value were in close agreement, as will be discussed in the next section.

The use of the computed displacements facilitated identification of the incorrect exponent on the material E value. Reliance on a check of the hoop stress alone, as is often done when checking vessel analyses, would not have identified the problem. As may be seen in Appendix Equation 1, the hoop stress is independent of the material E value. The hoop stress in the model with the low E value was substantially the same as in the model with the correct E value! The maximum stress in the knuckle, however, was low in the low E model by nearly 30%. Thus, even though the hoop stress may "look" right in a model, this does not necessarily mean that the model is correct. More checks must be conducted. Checking the displacements is a key place to start.

In order to avoid further "operator entry error" problems between the models, the program FEMAP (1997) was used to translate one of the models into the native format for each of the other two codes. Following the solutions phase, the model results were examined using the FEMAP post-processing facilities.

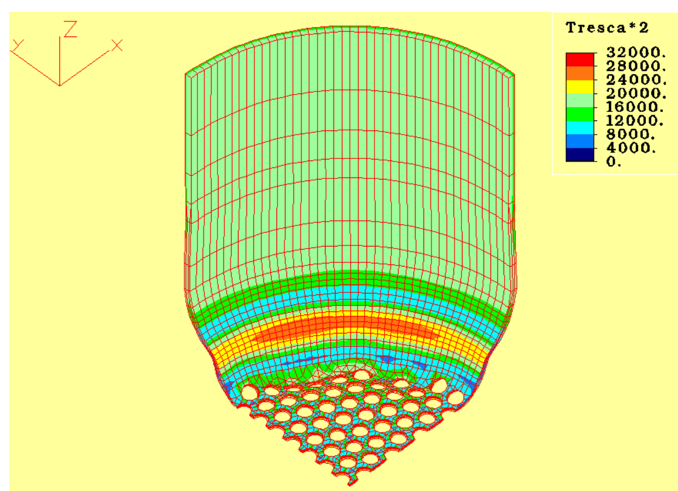

Figure 3 - Indicated Stress Intensity (Displaced Shape - Pressure Only)

INTERPRETATION OF RESULTS

Figure 3 illustrates the deflected shape of the shell portion of the vessel along with the indicated stress intensity as computed by one of the packages. As may be seen, there is a reverse bending that takes place in the knuckle region. The highest stresses are indicated in the region where this bending is at a maximum. The indicated hoop stress in the vessel away from the tubesheet was approximately 16,000 psi independent of the FE code used to compute the deflections and stresses. In addition, the region where the highest stress was indicated was approximately the same for all three FE codes examined. Figure 4 illustrates the deflected shape along with the magnitude of the displacements. All three FE codes indicated a radial displacement of approximately 0.036" and an axial displacement of approximately 0.024", indicating good agreement with the hand computations. Thus, on initial examination, the same model run on different processors seems to produce consistent displacement results.

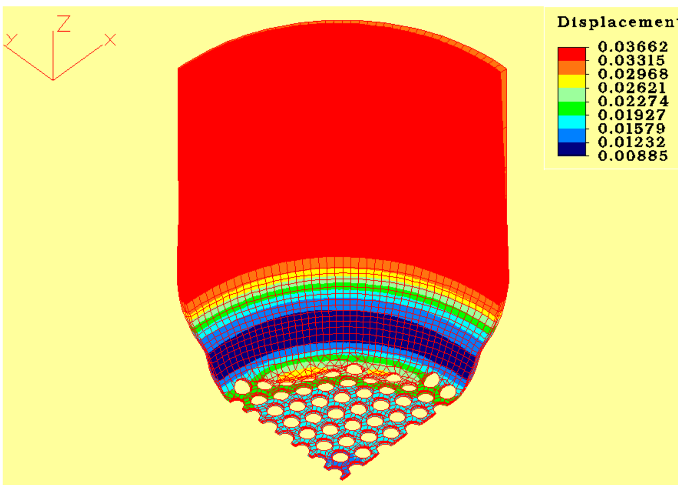

Figure 4 - Magnitude of Indicated Displacements (Displaced Shape - Pressure Only)

However, if we look at the magnitude of the maximum indicated Stress Intensity (in the knuckle region), we find that the agreement is less than satisfactory. As illustrated in Table 1, the results from one of the codes differs from the others by nearly 30%.

Table 1
Maximum Indicated Values

Code	Stress Intensity	Radial Displacement	Axial Displacement
A	25,963	0.036	0.026
B	25,760	0.036	0.026
C	19,137	0.035	0.024

In an earlier paper, Porter and Martens (1996) illustrated that, while the maximum indicated Stress Intensity on a model comprised of all plate elements might differ significantly depending on the FE code used, the agreement away from this point of maximum value was quite good. As a check to see if such agreement could be found for the brick elements being used in this case, the stress on the inside surface of the knuckle, as a function of the distance in inches from the point of highest indicated stress, was plotted.

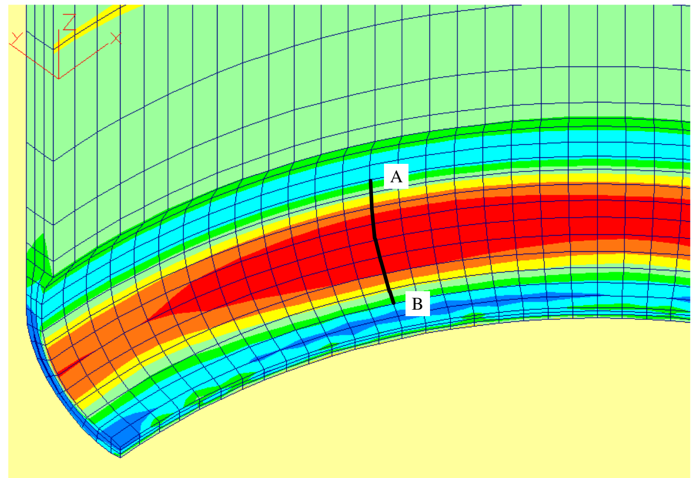

Figure 5 - Location of Stress Reporting Line

Figure 5 illustrates a section of the model through the knuckle near the region of highest indicated stress. The line A-B passes through the point of the highest indicated stress in the model. In Figure 6, we illustrated the stresses along this line (at nodal points) indicated by the three FE codes. The horizontal axis in Figure 6 is the distance away from the point of highest stress. As may be seen, the stress intensities indicated by two of the codes appear to be in close agreement, while those indicated by the third code (code C) are considerably lower.

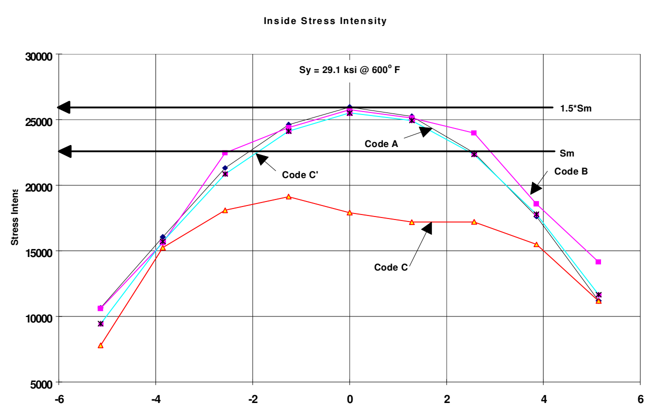

Figure 6 Stress Intensity Contours Through Point of Highest Stress

Code C, as indicated by the lower curve, uses an approximation recommended by Cook (1981) to avoid the stiffening effect on linear isoparametric elements caused by parasitic shear in bending. When this approximation in code C is replaced with an incompatible mode formulation (see curve Code C' on Figure 6), all three codes report nearly the same value. Note that testing the different element formulations in code C was facilitated by code C having the shortest run time. We would expect similar results with the other codes. In the following section, assuming that we had only the original (lower curve, code C) values for code C, we will discuss the results.

ATTEMPTED RESOLUTION OF DIFFERENCES

The nearly 30% difference between the highest stress intensity indicated by two of the codes and that indicated by the third code is, without question, too great a difference to ignore. The conservative, "safe" assumption would be that there is something wrong with code C or with the element formulation, and that the values reported by codes A and B are correct. The possibility exists, however, that code C is, in fact, reporting the correct value and that the use of codes A and B would result in an overly conservative design. In order to attempt to resolve this matter, an additional verification model was constructed.

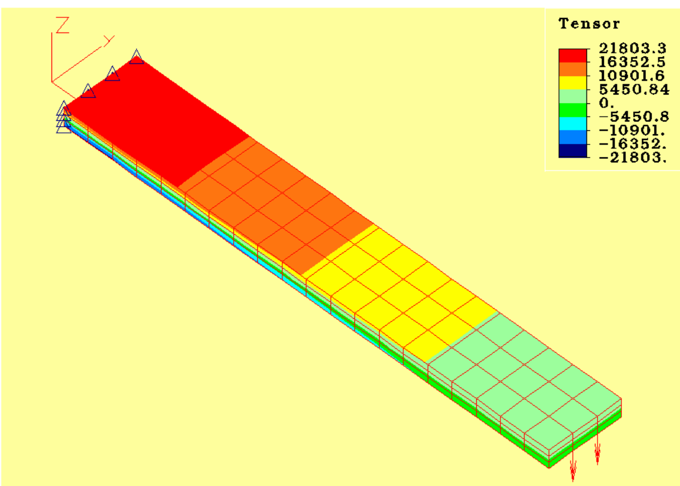

Figure 7 - Bar Model

Figure 7 illustrates the "Bar" model that was constructed to test the brick elements in the three codes. This model has dimensions of 40" long by 6" wide and 1.5" thick. The elements are meshed so that the aspect ratios of the elements are approximately the same as those used in the heat exchanger model. The bar is fixed at one end and loaded with 1,200 LB on the other end, as illustrated. Using the familiar Mc/I equation for the maximum stress at the point of attachment, we get a computed stress of 21,331 psi. The maximum deflection at the end, computed from $d = Fl^3/3EI$ is 0.505". Table 2 illustrates the stresses and deflections computed using the three FE codes.

Table 2
Results of "Bar" Model

Code	Maximum Indicated Stress	Deflection
A	21,855	0.496
B	21,803	0.496
C	21,704	0.496

As may be seen, the differences between the results indicated by the three codes are quite small and in good agreement with the closed form solution. Unlike the heat exchanger model, in the Bar model there is no large difference between the stress reported by code C and the other two codes. Thus, we have not identified a difference in the codes that would explain the differences reported in the heat exchanger models (except that the models only agree when the same element formulation is used). If an axial load is added to the Bar model to more closely approximate the loading of the elements in the knuckle region of the heat exchanger model, the indicated stresses get closer rather than farther apart, even though different element formulations are being used.

The indicated differences in the reported stresses based on element formulation is an unresolved issue. As users, we need better guidance on which element formulations to use and when to use them. It obviously makes a difference. But, as may be seen from the bar problem, the conditions under which various formulations may affect the answer is not always clear.

CONCLUSIONS

Modeling technique is very important in the development and use of FE models. Without a carefully considered, constructed and tested model, the results are questionable at best. As illustrated by the unintentional use of a low E value in the original code B heat exchanger model, the stresses can "seem" to be "OK," but in fact they can be low by a significant degree. Careful model construction *and* model verification are essential.

Even with carefully constructed and tested models, however, the answer is still dependent on the code and element formulation used for the solution. The vessel design using codes A and B would be more conservative than that using code C (and a different element formulation). At this point, there is no conclusive proof that the results from any of the codes (or formulation) are actually correct. All three codes seem to give the same answer when examining a simple problem (e.g. Bar) for which there is a handy closed-form solution. The loading and element shapes seem to be approximately the same in both the simple model and the more complex model, yet the stress intensity results differ dramatically for the more complex model.

At this point, the phrase "caveat emptor" (Let the Buyer Beware) would seem to apply to the use of commercial FE codes. Testing (using a simple model against known results) seems to indicate that all three of the codes examined give the same results. However, when used in an actual vessel analysis, the results differ dramatically. If code C (with the first element formulation) is reporting stresses that are 30% low, then a design at 1.5 Sm (ASME, 1996) would likely result in stresses that exceed yield.

How many of us really know the performance level of the codes that we and/or our vendors are using? And with the codes that we use, do most of us really know which element formulation to use for each particular application?

RECOMMENDATIONS

The authors recommend that the engineer/user carefully select an FE code that gives consistent and accurate results for the type of problem being investigated. Additionally, the engineer using the code needs to be aware of the element options available and the consequences of their use. The variability of FE code results can be significant and, in many cases very difficult to confirm. The engineer must be cautious, think through and verify the results of an FE analysis before using them as a basis for a design. Such checking of the FE solution is essential to the achievement of a practical and safe result.

The authors recommend that the following methodology be used for FE analysis:

1. Select the FE code that you will be using based on its proven applicability
2. Determine the design parameters that must be modeled to assure a valid analysis. Consider:
    - pressure loadings that must be applied to model elements to simulate actual conditions
    - thermal gradients and profiles that can be expected during various operating condition
3. Select the type of elements that will react correctly to the above design parameters and give reasonable results on the first attempt. Convergence of critical stressed areas is required.
4. Confirm the model displacements for pressure and thermal parameters.
    - use closed-form solutions for pressure induced strain
    - use closed-form solutions for thermal induced expansion
5. Question the results for all critical displacement and stress areas of the model. Confirm that the results appear to be intuitively correct.
    - movements are in the correct direction and magnitude
    - stresses on individual elements of the model are distributed adequately

Finally, as users of FE codes, we need to have better information about the validity of the codes we select. All three of the codes examined in this paper are advertised in the pages of Mechanical Engineering magazine as being suitable for the type of analysis conducted in this study. Although we have not tested other codes using this problem, we suspect that the scatter of answers might increase with the number of codes tested.

The authors recommend that the engineer using FE for design analysis develop a suite of real life application problems for verification of the FE codes being used. Ideally, these problems would have verifiable solutions and be made available to other engineers to test the codes that they are using. PVP should consider taking the lead in the development of such a suite.

REFERENCES

ASME Boiler and Pressure Vessel Code Section VIII Divisions 1 and 2, 1996, The American Association of Mechanical Engineers, New York, NY

Cook, Robert D., 1981, Concepts and Applications of Finite Element Analysis, John Whiley & Sons, Inc., New York, NY, pp 190-195

FEMAP, 1997, Enterprise Software Products, Inc., 415 Eagleview Boulevard, Exton, PA. 19341

Porter, M. A. and Martens, D. H., 1996, "A Comparison of the Stress Results from Several Commercial Finite Codes with ASME Section VIII, Division 2 Requirements," PVP Vol. 336, ASME, pp 361-371

"Roark's Formulas for Stress and Strain," Sixth Edition, 1989, McGraw-Hill Book Company, N.Y. pp 518-519

APPENDIX - Check Calculations

Hoop Stress
The hoop stress in the shell, away from the ends, should be approximately equal to:

$Sh = Pr/t$ Equation 1

where: P = Pressure (790 psi)
r = radius of vessel (59")
t = vessel wall thickness (2.875")

Radial Expansion
The expansion of the vessel in the radial direction may be related to the hoop stress by:

$dr = Sh * r/E$ Equation 2

where: E = Young's Modulus for the vessel material (29×10^6 psi)

Axial Extension
If we assume that the axial extension of the vessel proportional to the pressure load acting on the tubesheet and the axial stiffness of the tubes, we may compute this extension using:

$F = P*(Ats - Atb)$ Equation 3

$Sa = F/Atb$ Equation 4

$e = Sa/E$ Equation 5

$dl = e * l$ Equation 6

where: F = Pressure force on tubesheet
Ats = Area of tubesheet (10,936 sqin)
Atb = Area of tube bundle (1,268 sqin)
e = Strain in tubes (in/in)
l = Length of tunes (126")
dl = Axial extension of tubes and tubesheet