On Using Finite Element Analysis for Pressure Vessel Design

Michael A. Porter
Dynamic Analysis
Lenexa, Kansas

Dennis H. Martens
Black and Veatch Pritchard
Overland Park, Kansas

Pedro Marcal
PVM Corporation
Cardiff, California

ABSTRACT

This paper presents a practical review of the use of PC-based Finite Element software in the analysis of typical pressure vessel components. The authors discuss element type selection criteria and features. Some of the different element formulations are discussed. Modeling parameters and convergence procedures are examined. Practical evaluation tolerances are discussed.

INTRODUCTION

The ASME Section VIII vessels that are in general use throughout the refining and chemical industries have demonstrated an excellent safety record. However, as these industries mature, there is an ongoing need to reduce the capital and facility maintenance costs, including the cost of pressure vessels, related piping and infrastructure. This cost control requirement is manifested in the increase in the allowable design stress utilized in ASME Section VIII Division 1. The resulting reduced design safety factor for pressure vessels combined with the need to reduce piping costs presents increased challenges to the design engineers.

To this end, design engineers must use their experience and the latest design tools to maintain reasonable safety levels while providing the most cost effect design. One tool being used on an ever increasing basis is Finite Element (FE) analysis software. The current capabilities of FE software on desktop computers provide pressure vessel design engineers with the ability to employ FE analysis on a nearly routine basis. For example, FE investigations are often used to augment tools such as the Welding Research Bulletin WRC-107 for assessing the effect of piping-imposed loading on vessel nozzles.

Pressure vessel design engineers must have a reasonable understanding of FE fundamentals to adequately use this design tool. The engineer must determine the most appropriate modeling approach, select the proper elements and solution technique to assure a reasonable analysis. The engineer must also determine if the model is reacting correctly and presenting reasonable results. This paper presents some of the basic information that is needed to accomplish these tasks.

REVIEW OF PREVIOUS WORK

In a previous paper (Porter and Martens, 1996), the authors compared the results of an analysis of a typical thin wall nozzle using five different commercial FE codes. The linear elastic analysis used thin shell/plate elements to model the nozzle and shell. In a subsequent paper (Porter and Martens, 1998), the same model was analyzed using 3-D Solid (brick) elements. As with the first paper, the analysis was conducted using linear elastic assumptions. In another paper (Porter, Martens and Hsieh, 1997), a comparison of the results obtained using brick elements with different commercial FE codes on a heat exchanger model was presented.

The expressed purpose of these papers was to evaluate the differences that could be expected from solutions using different FE codes to evaluate real life problems. In addition, the differences that could be expected between the use of shell and brick elements in the analysis of thin wall vessels were investigated.

In general, the variation in the reported stress values between the different codes was found to be relatively insignificant, assuming that model geometry was constant and that a similar element formulation was employed. For the thin wall nozzle model, no significant differences were reported between the shell and brick model solutions. However, for both the shell/plate element model and the brick element model, the effect of different element formulations proved to be very significant.

ANALYSIS GUIDELINES

Based on these previously published papers and a significant amount of practical experience with both pressure vessel and FE analysis in general, the authors present the following guidelines for FE analysis of pressure vessels.

Element Selection

Once the geometry of the object to be analyzed is defined, the first task is to select the type of element that is to be employed. For most pressure vessel analyses, the element selection is made from three categories of elements: axisymmetric solid elements, shell/plate elements and 3-D brick elements. Although nearly all problems can be solved using 3-D brick elements, the other two types offer significant reductions in the solution time and effort where they are applicable. Often, this reduction in solution effort is significant enough to make the use of FE analysis feasible where it might not be with 3-D bricks.

Axisymmetric Elements

The axisymmetric element represents a significant reduction in both model creation and solution effort when its selection is appropriate. The primary consideration is that both the geometry and the loading must be symmetrical about the axis of revolution for this element to be validly employed. The head of a vessel under pressure loading is an example of a good candidate for the use of axisymmetric elements. Often, however, vessels have non-axisymmetric mechanical loading and/or thermal profiles. One side of the vessel is often at a significantly different temperature than the other side of the vessel. This is especially true for horizontal vessels. For these cases, the use of axisymmetric elements will introduce inaccuracies into the solution; another type of element will likely produce better results.

Axisymmetric elements have often been employed in the analysis of bolted flange connections. Although they offer a very significant reduction in analysis effort, they are not generally appropriate for that application. The load from the bolts is periodic rather than axisymmetric. Although some of the more costly codes have a means of overcoming this limitation, periodic loading cannot be properly modeled with the axisymmetric elements in most PC-based FE codes. As more and more detail of the flange/gasket interaction is desired (e.g. Porter and Martens, 1994), the validity of the axisymmetric element flange model becomes more and more questionable.

Shell/Plate Elements

Many commercial FE codes blur the distinction between shell and plate elements. Technically, shell elements can represent a curved surface, while plate elements are constrained to be flat. For practical purposes, so long as the ratio of the thickness (t) of the vessel wall to the radius of the vessel (r) is less than 0.1 ($t/r < 0.1$), both shell and plate elements will produce acceptable results. This assumes that a significant number of elements around the circumference are employed to adequately represent the curvature of the vessel. For example, Ha (1995) has indicated that with a minimum of 96 elements around the periphery of a nozzle, convergence is assured.

In addition to the ratio of the vessel thickness to the radius, there is a general requirement that thin plate elements used in many commercial FE codes be used to model only structures that have a thickness (t) that is small in comparison to their planar dimensions (L). This is usually expressed as $t/L \leq 0.1$. It is important to note that this requirement is directed at the object being modeled and not on the individual element. A 12-in thick slab could still be modeled using plate elements providing that the plan dimensions of the slab were greater than 10-ft.

Two of the potential problem areas in the use of thin plate elements in the analysis of pressure vessels are the assumption of a linear stress distribution and the lack of shear deformation. Thick plate and shell elements are formulated to overcome most of these difficulties. The use of higher order (8-noded quadratic, 12-noded cubic etc.) elements can result in the reduction of the number of elements required to achieve the desired degree of accuracy.

3-D Solid Brick Elements

The brick element is probably the most general of all the elements. It can be used in almost all situations. However, the effort involved to use this element, both in model creation and solution, is often not justifiable for the analysis of thin wall vessels. This is especially true when the common 8-noded linear brick element is used. With 8-noded linear brick elements it is essential that a minimum of 3-5 elements be used through the thickness of the vessel. This restraint combined with the necessity to maintain an aspect ratio of less than 5:1 causes the number of elements required to increase rather quickly. The large number of elements adversely affects both the modeling effort and solution times. The use of higher order brick elements, if available, can substantially reduce number of elements required. However, the reduction in the number of required elements may not be enough to offset the higher solution effort required per element.

Element Formulation

Almost as important as the type of element selected is the formulation of the element. Historically, analysts with a relatively strong background in the underlying theory of finite element formulation did most FE analysis work. The introduction of easier to use FE codes with automatic meshing capabilities and, to some extent, competitive pressure, has led to much analysis work being done by engineers with little or no background in the theory. This makes the evaluation of element formulation difficult for many users.

It is unlikely that a comprehensive list detailing which element to use for every possible analysis situation could be developed. There are, however, some methods that the average engineer can use to evaluate element formulations. The classic test used to evaluate elements is called the “patch test” (see Zienkiewicz and Taylor, 1989, Chapter 11). In the patch test, a model consisting of a few elements, is tested by applying displacement boundary conditions. This model is expected to produce a uniform stress result. Satisfaction of the uniform conditions, even for distorted elements, implies a balance of external work with internal work, i.e. conservation of energy. A variant of this test can be used to evaluate the suitability of various elements for a particular application.

As an example, Porter et al. (1997) reported that a difference of approximately 30% was noted in the same model depending on the elements that were used. The only difference between the elements seemed to be the formulation. However, when the elements were tested using a simple cantilever beam model, no difference was noted. Thus, the difference seemed to be unexplained. In this example, an application of the patch test would have pinpointed the difference.

In essence, the patch test compares the stresses (and displacements) reported by two groups of elements (regular shaped and distorted elements). The two results should be relatively close. In the above referenced paper, only regular shaped elements were compared using the simple cantilever beam model. In the actual model being discussed, the elements at the point of question were somewhat distorted due to the curvature of the vessel. Thus, the simple cantilever beam model was not a very good way to test for differences in the element formulations.

Figure 1 illustrates the displacement in a curved bar similar to the geometry in the referenced paper. The bar is approximately 40” long, 6” wide and 1.5” thick. The mesh is 20x3x3 in the respective directions. The bar forms a 90-degree section of a 25” cylinder. It is fixed at one end and loaded on the other end with a 1000-LB load as shown. The distortion of the elements, in this case, is due to the curvature of the bar.

The brick elements used in the model in Figure 1 are linear, 8-noded elements that allow incompatible modes. The indicated displacement is within about 2% of the displacement computed using the Roark/Young (1989) closed form solution. Figure 2 illustrates the same geometry using linear, 8-noded brick elements that do not allow incompatible modes. This indicated displacement is only about 58% of the closed form solution value. The corresponding stress difference is on the order of 30%, about the same as noted on the model used in the earlier paper. Thus, the difference noted between the values reported by the various programs was due to a difference in element formulation.

In comparing the results of this rather simple model with the closed form solution, it is clear that the elements not allowing incompatible modes do not accurately report the displacements and stresses for this geometry. Conversely, the elements that allow incompatible modes do seem to accurately report the displacements and stresses. Thus, for this geometry, the elements with incompatible modes should be used.

The testing of elements to verify their applicability to a specific problem is an essential part of the analysis task. The development of such a test need not be particularly time consuming. The models shown in Figures 1 and 2 require only a few minutes to construct and run using current FE codes. This is a small price to pay to avoid using the wrong element formulation for an application.

Model Geometry

After the type of element has been selected, the model geometry must be considered. In evaluating the geometry, there are several prime considerations. In addition to the necessity to accurately represent the actual geometry of the vessel or component of the vessel, one must consider the loading and support (boundary) conditions and the mesh to be employed. The extent of the vessel or component modeled is also of prime concern when the decision (usually for purposes of economy) is made to model only part of an overall system.

Model (Attenuation) Length

When modeling nozzles on vessels, it is normal practice to model only that portion of the vessel and nozzle affected by the stresses in the intersection region. The attenuation length ($2.5 * \sqrt{rt}$) provides a good estimate of the length over which the localized stress discontinuities may be expected. This distance away from the nozzle will not, however, be sufficient for the construction of an accurate nozzle model. One must also consider the effect of the stresses produced by the boundary conditions on the model. To ensure that the (perhaps artificial) stresses produced by the boundary conditions do not affect the stresses at the nozzle intersection, we recommend that the model be constructed with a minimum length of $7.5 * \sqrt{rt}$ away from the nozzle-to-shell junction in each direction.

Meshing

The accuracy of the FE model is highly dependent on the mesh employed, especially if higher order (cubic, quadratic etc.) elements are not used. In general, a finer mesh will produce more accurate results than a coarser mesh. At some point, one reaches a point of diminishing returns, where the increased mesh density fails to produce a significant change in the results. At this point the mesh is said to be "converged." This process of refining the mesh and evaluating the results is normally referred to as a "mesh convergence" study or analysis. Although many FE codes contain "error estimates" of one sort or another, mesh convergence remains the most reliable means of judging model accuracy.

Coarse meshes almost always under-report the stresses in a model. It is not uncommon to have maximum reported stresses on the order of less than 50% of the converged stresses on a coarsely meshed model. Thus, without consideration of mesh convergence, gross errors in stress estimates are quite possible. As mentioned earlier, Ha (1995) concluded that 96 elements around the periphery of a nozzle are required for convergence. This number is considerably higher than the number typically used by many analysts. If higher order elements are used, good results can be obtained with fewer elements. Either mesh convergence analysis or a reliable error estimate is absolutely necessary to quantify the analysis results. Typically, an increase of less than 5% in the stress levels after a doubling of mesh density or an "error estimate" of less than 0.05 will ensure that the indicated stresses are within 5-10% of the "converged" values.

Some FE codes employ an adaptive process (Zienkiewicz and Taylor, 1989) to automatically refine the mesh and/or increase the order of the elements to reach the desired degree of accuracy. When available, these processes can save a lot of manual effort. They do not, however, completely relieve the engineer of the need to check the results.

Another important consideration in meshing is the aspect ratio of the elements. In general, a 1:1 (1:1:1 for brick elements) is considered ideal, which means that square (or cubic) elements are best. The farther one strays from this ideal, the less accurate the results become. Up to a ratio of approximately 3:1 the results are usually within ~5-10% of what would be expected from the ideal elements. Elements with an aspect ratio greater than 5:1 should be used only to transmit load from one section of the model to another. The stress results from such high aspect ratio elements cannot be trusted, thus they should not be used in areas where the stress values are of concern. The angle where the sides of the elements meet is also important. For four-sided elements, the ideal is 90 degrees. Angles of less than 60 degrees (or greater than 120 degrees) for four-sided elements should be avoided. For three-sided elements, the ideal is 60 degrees.

With 3-D brick elements, the number of bricks that must be used through the thickness of the vessel must be considered. For the commonly used 8-node, linear brick element, an absolute minimum of three elements though the thickness is necessary. Significantly better results may be obtained using five elements though the thickness. Alternately, higher order 20+ noded brick elements may be used with a significant reduction in the number of elements required. Use of these higher order elements may result in an overall reduction of required computational effort.

Boundary Conditions

One of the most significant sources of errors in FE modeling is the inaccurate (or inappropriate) modeling of the loads and restraints on a model. For example, fully fixing (restraining) all of the nodes on the end of a pressure vessel does not represent the same condition as does a normal head. With any normal head, the radial stiffness is not infinite, thus radial expansion under pressure loading will occur. This cannot be the case when the nodes are fixed. As long as the vessel is modeled with a length of at least $7.5 * \sqrt{rt}$ away from the nozzle, however, this restraint approximation should have little effect in the nozzle region.

Figure 3 illustrates the boundary conditions and loads on a shell-nozzle intersection model. In this case, the nodes on end A of the shell are fully restrained (all translations and rotations of the nodes are restricted). This (rather unrealistic) restraint simulates the shell being attached to some unmovable object. Since end A is more than $7.5\sqrt{rt}$ from the nozzle, however, this restraint will have little or no effect on the stresses at the nozzle-shell intersection. If we had wanted to assume that the vessel continued on past end A, we could have used a symmetrical boundary condition. The symmetrical boundary condition always restrains translations normal to the plane of symmetry and the rotations in the plane of symmetry. In this case, we would have restrained translation in the X direction and the rotations about the Y and Z axes.

End B has the rotations about the Y and Z axes restrained. Nodal loads are applied to simulate to the end load developed by the internal pressure. The end load due to a closed end must be included in the analysis of vessels under pressure loading. This requirement includes the nozzle in a nozzle/shell junction. Since many piping programs (which are often used to generate the loads on nozzles) do not typically report this load, it is easy to overlook. It is not, however, typically a trivial load. For example, the pressure-generated end load on a 24" diameter nozzle at 165 psi is nearly 75,000 LB!

Forces and moments may be applied to a model in a number of different ways. The particular method chosen is often a function of the pre-processor used to generate the model. Some pre-processors will automatically distribute a force or moment along a line or surface, such as the end of the nozzle. The forces on the end of the shell at end B in Figure 3 are applied in this manner. Other programs do not make this calculation. With such programs, especially when the node spacing is non-uniform, it is often easier to use a "spider" or "wagon wheel" as described by Porter and Martens (1996) to distribute the load. Such a "spider" has been used on the nozzle model as illustrated in Figure 3. When such a technique is used, it is important that the connection between the "spider" and the vessel be configured in such a way that artificial rigidity is not introduced into the nozzle

Checking the magnitude of the boundary loads after a run can identify possible problems in a model. Even though we are using a modern computer-based tool, the sum of the forces and moments about each axis must remain equal to zero when the body is at rest.

CLOSING

The guidelines presented are intended as a starting point for the engineer tasked with conducting an FE analysis of a pressure vessel component. It is hoped that they will prove helpful. In the end, however, no set protocol of canned "we solve everything automatically" can guarantee an accurate analysis for every project. Good engineering judgment must be the guide. Four additional "rules" may help:

1. Develop and test simple models (for which you have a closed form solution) that are similar to at least parts of the more complicated model. For example:

The hoop stress in the vessel away from discontinuities should be:

$\sigma_h = \mathrm{Pr}/t$ where: P = internal pressure
r = radius of vessel
t = wall thickness of vessel

The radial expansion of a vessel due to pressure should be approximately:

$\delta_r = \sigma_r * r / E$ where: E = Youngs modulus

The thermal expansion of a component should follow:

$\Delta L = \alpha L \Delta T$ where: L = length of component
α = Coefficient of thermal expansion
ΔT= Temperature change

These and other examples can be found in Porter et al (1997).

2. Apply these closed form solutions to the actual model to check that the model is behaving properly.

3. If the result does not "look" right, it probably is not.

4. If you're not sure, get help.

REFERENCES

Ha, J.L, Sun, B.C., and Koplik, B., 1995, "Local Stress Factors of a Pipe-Nozzle Under Internal Pressure," *Nuclear Engineering and Design*, 157, Elsevier Science S.A., Lausanne, Switzerland, pp. 81-91.

Martens, D. H., and Hsieh, C. S., 1998, "Finite Element Investigation of a CBA Reactor for the Effects of Thermal Loading," *PVP Vol. 368*, ASME, New York, NY, pp. 139-146.

Porter, M. A. and Martens, D. H., 1994, "Investigation and Repair of Heat Exchanger Flange Leak," *PVP Vol. 278*, ASME, New York, NY., pp. 133-143.

Porter, M. A. and Martens, D. H., 1996, "A Comparison of the Stress Results from Several Commercial Finite Element Codes with ASME Section VIII, Division 2 Requirements," *PVP Vol. 336*, ASME, New York, NY., pp. 341-346.

Porter, M. A., Martens, D. H., and Hsieh, C. S., 1997, "A Comparison of Finite Element Codes and Recommended Investigation Methodology," *PVP Vol. 359*, ASME, New York, NY., pp. 241-246.

Porter, M. A. and Martens, D. H., 1998, "Stress Evaluation of a Typical Vessel Nozzle Using PVRC 3D Stress Criteria: Guidelines for Application," *PVP Vol. 368*, ASME, New York, NY., pp. 297-301.

Young, W. C., 1989, *Roark's Formulas for Stress and Strain*, McGraw-Hill Book Co., New York, NY, p. 251 and Table 16.S.

Zienkiewicz, O. C., and Taylor, R. L., 1989, *The Finite Element Method,* Vol. 1, McGraw-Hill Book Co., New York, NY, Ch. 14.