Proceedings of the 15th International Symposium on the Packaging and Transportation of Radioactive Materials
PATRAM 2007
October 21-26, 2007, Miami, Florida, USA

Finite Element Mesh Considerations for Reduced Integration Elements

Gordon S. Bjorkman, Jr.
U.S. Nuclear Regulatory Commission

Jason M. Piotter
U.S. Nuclear Regulatory Commission

ABSTRACT

Finite element models of spent fuel casks and canisters that are typically used in impact and impulse analyses may contain tens of thousands of nonlinear elements. These models use explicit time integration methods with small time steps. To achieve reasonable run times, fully integrated elements are replaced with under-integrated elements that use reduced integration procedures. When fully integrated these elements produce a linear strain distribution. Reduced integration, however, results in a constant strain distribution, which requires more elements through the thickness of the canister shell to achieve the same accuracy as fully integrated elements. This paper studies the effect of the number of reduced integration elements through the thickness of the canister shell and the ratio of element height to shell thickness on the accuracy of the strains in regions of high through-thickness bending, such as the junction between the shell and base plate. It is concluded that mesh refinement has a significant effect on the maximum plastic strain response in such regions and that a converged solution may not be attainable within practical limits of mesh refinement, if the results are based solely on the maximum plastic strain on a cross section at a structural discontinuity. However, a converged solution is obtained by investigating the response of other elements on the same cross section that are not located on the surface of the stress concentration at the structural discontinuity. Based on the results, a “rule of thumb” is proposed for mesh refinement in a region of severe structural discontinuity wherein reasonably proportioned reduced integration solid elements are used and plastic strains are evaluated

INTRODUCTION

Impact and impulse problems can be solved using either implicit or explicit methods. Since explicit methods require inverting a diagonal mass matrix to solve for accelerations, as opposed to inverting a stiffness matrix to solve for displacements as required by implicit methods, explicit methods are much faster. Thus, impact and impulse analyses often lead to the use of explicit time integration methods because of their computational efficiency especially when dealing with the small time steps needed to resolve high frequency shock waves. For large models, typically containing tens of thousands of nonlinear elements, computer run times may be excessive when using fully integrated elements. Therefore, to achieve reasonable run times fully integrated elements are replaced with under-integrated elements that use reduced integration procedures. For three dimensional models, run-time reductions of an order of magnitude are often achieved using reduced integration instead of full integration.

Drop impact analyses of spent fuel transportation casks that are conducted to satisfy the requirements of 10 CFR 71.73 typically have shell and lid components constructed of 8-node brick elements. When fully integrated, i.e., 2x2x2 Gauss quadrature, these elements produce a linear strain distribution. Reduced integration, however, results in a constant strain distribution, which requires more elements through the thickness of the canister shell to achieve the same accuracy as fully integrated elements. This paper studies the effect of the number of reduced integration elements through the thickness of the cylindrical shell and the ratio of element height to shell thickness on the accuracy of the strains in regions of high through thickness bending, such as the junction between the shell and lid or shell and base plate. The loading case considered here is an internal pressure pulse. A follow-up paper will address drop impact.

METHODOLOGY

To evaluate the performance of reduced integration constant stress 8 node solid elements, a finite element cylindrical shell wedge model was constructed. The 1.27 cm (0.5 inch) cylindrical shell is 305 cm (120 inches) long and 173 cm (68 inches) in diameter and is welded to a 15 cm (6.0 inch) thick base plate. The number of solid elements considered through the shell thickness is 2, 3 and 5. To determine the effect of the longitudinal height of the elements on response, the longitudinal height is varied from a ratio of height to thickness of 0.156 to 5.0. Figure 1 is a sketch of the finite element model at the intersection of the shell and base plate that shows the element height (h), shell radius (R) and shell thickness (t) for the case of 5 elements through the thickness. It is this first layer of shell elements adjacent to the base plate where response is evaluated, since this is the location of the highest plastic strains under drop impact or internal pressure pulse, and is also the location of the weld between the canister shell and base plate.

The internal pressure loading in the analysis is applied to the shell only and consists of a pressure pulse with a linear rise time of 5.0 msec to a constant pressure of 5.52 MPa (800 psi). The shell and base plate materials are stainless steel with a stress strain curve of the form, $\sigma = K\varepsilon^n$, where, K, is 1104 MPa (160,000 psi) and, n, is 0.279. The LS-DYNA computer code [1] was used for this study.

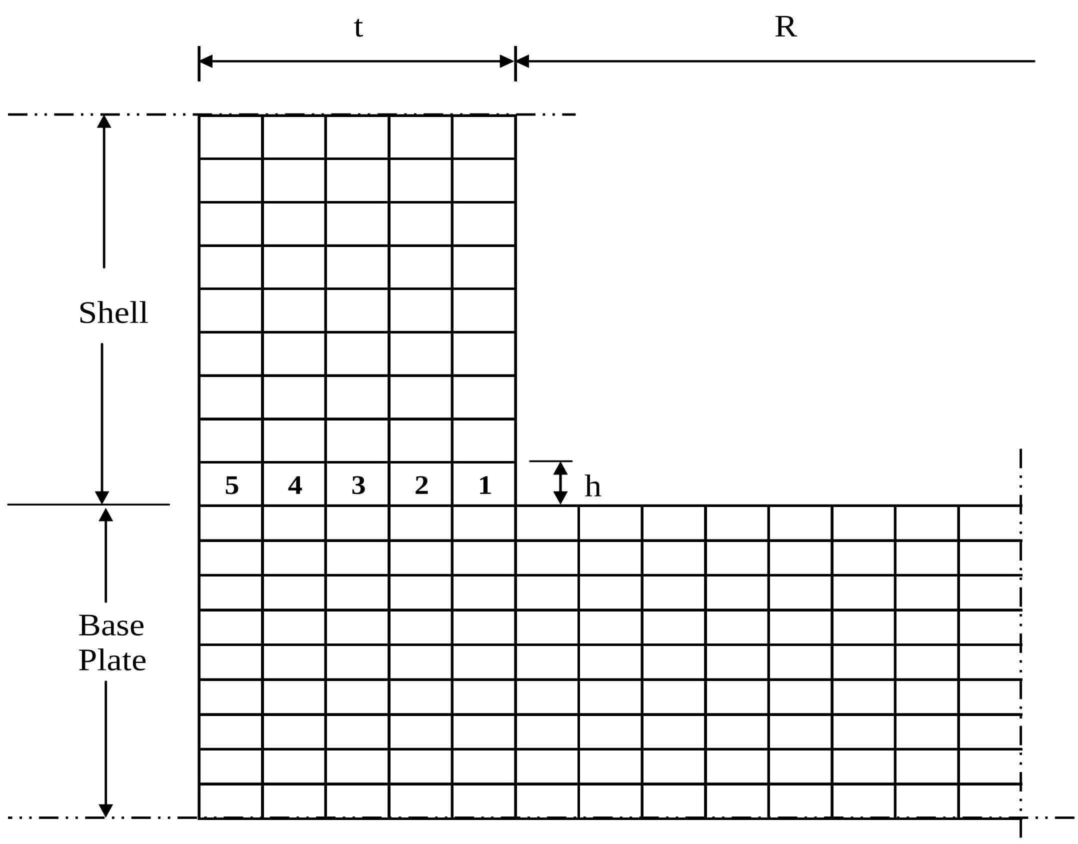

Figure 1: Sketch of the finite element model at the intersection of the shell and base plate showing the element height (h), shell radius (R) and shell thickness (t) for the case of 5 elements through the thickness.

RESULTS

For energy limited events, acceptance criteria based on strain is the best measure of containment boundary integrity and, therefore, it is the effective plastic strain in each element through the thickness that is tabulated at the time when maximum response is achieved. It is important to note that maximum response always occurs in Element 1 (i.e., the element on the inside radius at the bottom of the shell) regardless of the number of elements through the thickness. Figure 2 plots the maximum effective plastic strain in Element 1 for the case of 5 elements through the thickness against the ratio of element height to shell thickness. It is clear from this Figure that the ratio of element height to thickness at the shell to base plate junction has a dramatic effect on the accuracy of the maximum strain response. Figure 3 is the same as Figure 2, but also includes the results for 2 and 3 elements through the thickness. These results show that mesh refinement, both in the number of elements through the thickness and the relative height of the elements, has a significant influence on maximum strain response.

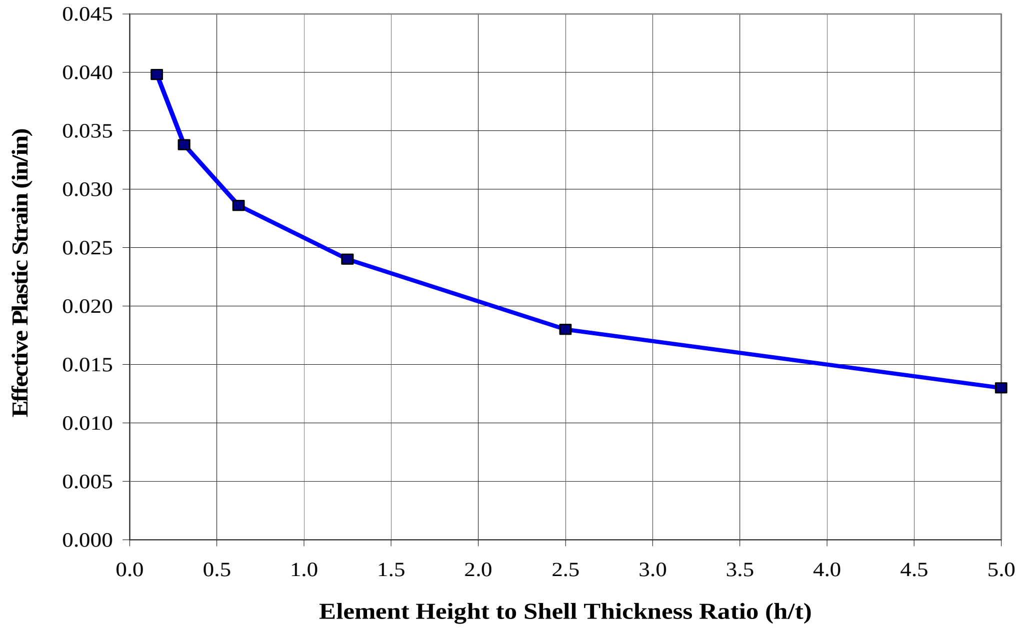

Figure 2: Maximum effective plastic strain for 5 elements through the thickness plotted against the ratio of element height (h) to shell thickness (t).

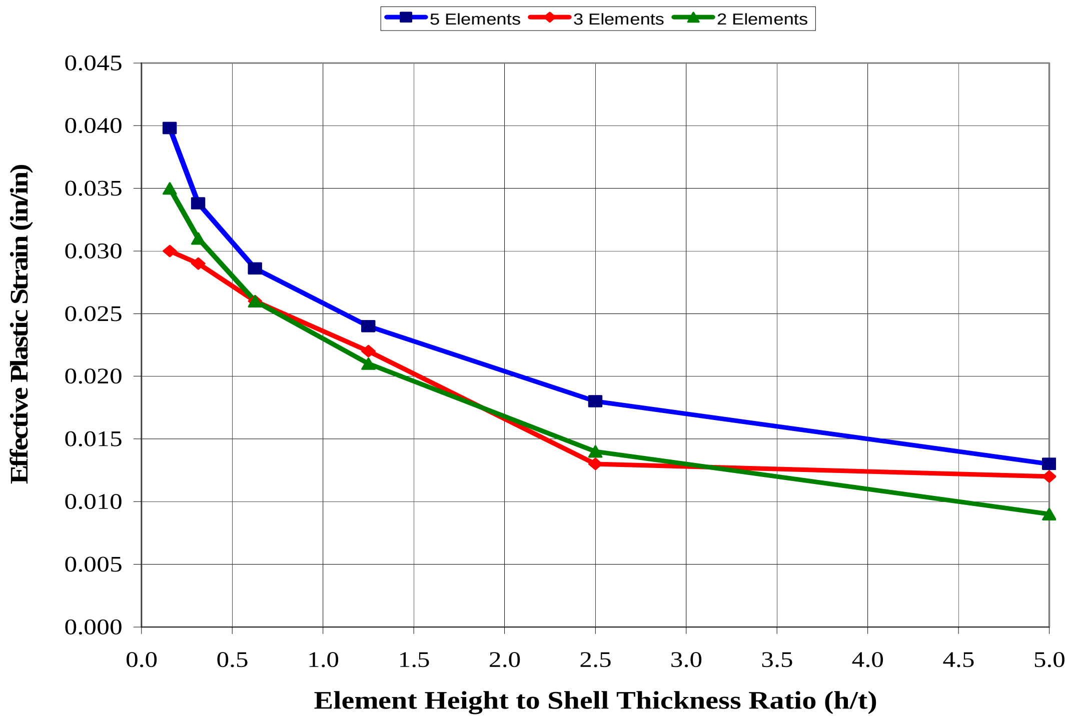

Figure 3: Maximum effective plastic strain for 2, 3 and 5 elements through the thickness plotted against the ratio of element height to shell thickness.

The results also show (see for example Figure 2) that a converged solution for maximum effective plastic strain was not obtained even for the finest mesh used in this study. This is because Element 1 on the inside of the cross section is located directly on the surface of the stress concentration at the structural discontinuity. However, Element 5 on the outside of the cross section is not located on the surface of a stress concentration, and the plot of maximum plastic strain verses the ratio of element height to thickness given in Figure 4 shows that convergence is achieved for a ratio of element height to thickness of 0.31. Thus the maximum effective plastic strain that should be used in a structural evaluation is 0.034 in/in in Element 1. This is the strain that would be associated with the bending moment at the structural discontinuity. The effect of stress concentration can be incorporated separately by using a stress (strain) concentration factor.

Therefore, an acceptable finite element mesh using reduced integration solid elements in a region of gross structural discontinuity would consist of 5 elements through the thickness with an element height to shell thickness ratio (h/t) of no greater than approximately 0.3. (This translates into an element aspect ratio of (0.3)(t)/(t/5) = (0.3)(5) = 1.5, where the aspect ratio is here defined as the ratio of element longitudinal height to the element radial length.)

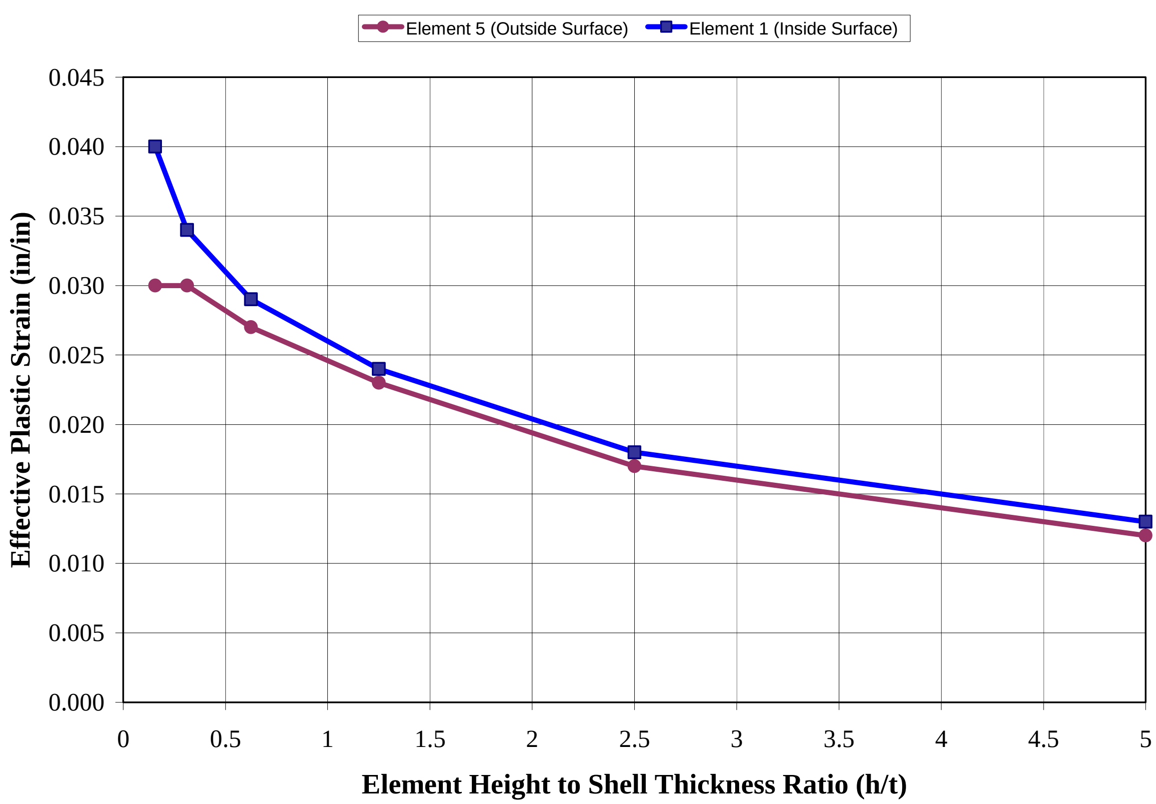

Figure 4: Maximum effective plastic strain verses the ratio of element height to thickness for Element 1 (inside surface) and Element 5 (outside surface)

Proposed "Rule of Thumb"

For the purpose of calculating the maximum effective plastic strain at the location of a hard structural discontinuity, it is proposed that approximately 15 uniformly shaped elements occupy an area equal to the shell thickness squared, t^2, at the location of the discontinuity. This proposed *"Rule of Thumb"* is shown in Figure 5. Thus, if there were 4 elements through the thickness, 16 elements (4 across and 4 high) would occupy the area t^2 .

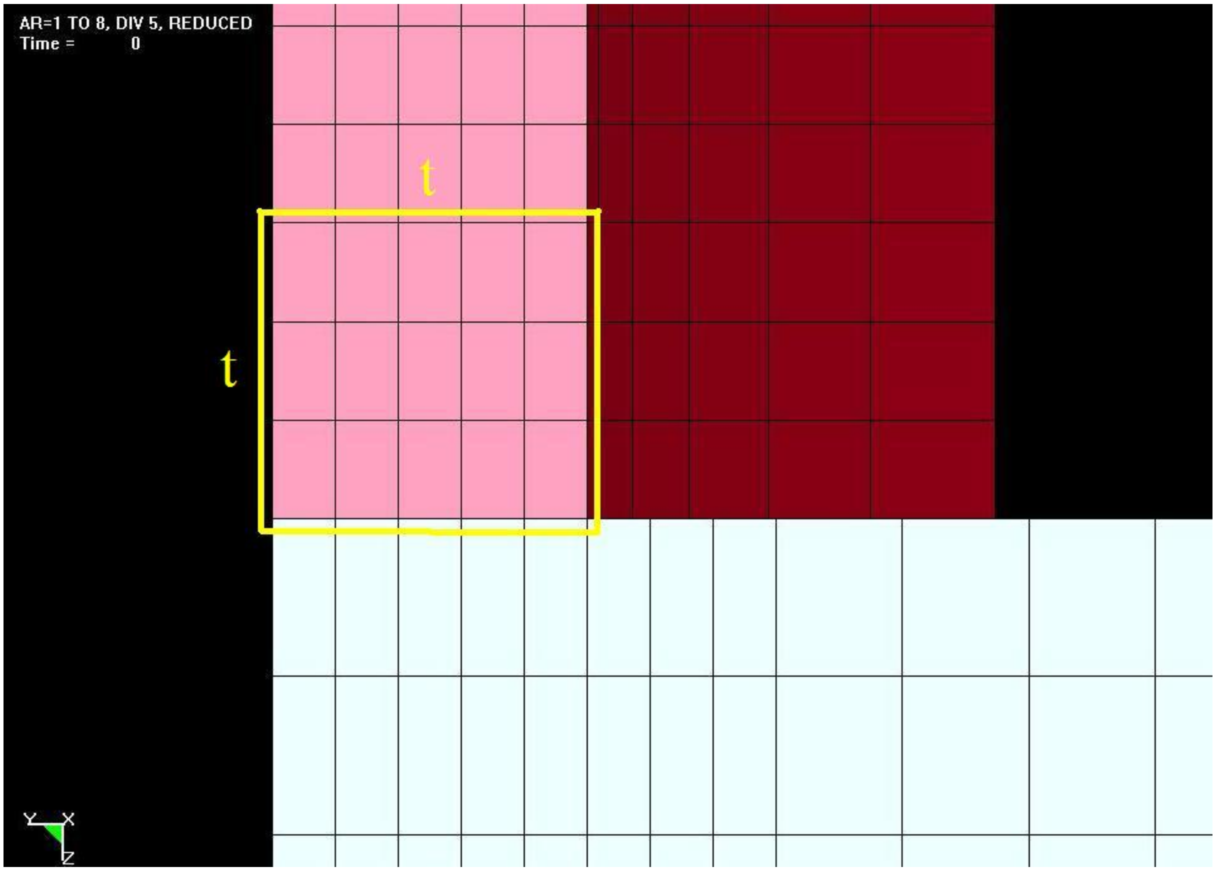

Figure 5: It is proposed that approximately 15 uniformly shaped elements occupy an area equal to the shell thickness squared, t^2, at the location of a hard structural discontinuity.

CONCLUSION

The use of reduced integration elements requires special attention to mesh refinement in regions of high strain gradient, such as at structural discontinuities. It has been shown that mesh refinement has a significant effect on the maximum plastic strain response in such regions, and that a converged solution may not be attainable within practical limits of mesh refinement, if the results are based solely on the maximum plastic strain on a cross section at a structural discontinuity. However, a converged solution is obtained by investigating the response of other elements on the same cross section not located on the surface of the stress concentration at the structural discontinuity,. Based on these results an acceptable finite element mesh using reduced integration solid elements in a region of gross structural discontinuity would consist of 5 elements through the thickness with an element height to shell thickness ratio (h/t) of no greater than approximately 0.3. (This translates into an element aspect ratio of (0.3)(t)/(t/5) = (0.3)(5) = 1.5, where the aspect ratio is here defined as the ratio of element longitudinal height to the element radial length.)

For the purpose of calculating the maximum effective plastic strain at the location of a hard structural discontinuity, it is proposed that approximately 15 uniformly shaped elements occupy an area equal to the shell thickness squared, t^2, at the location of the discontinuity. This proposed *"Rule of Thumb"* is shown in Figure 5. Thus, if there were 4 elements through the thickness, 16 elements (4 across and 4 high) would occupy the area t^2 .

REFERENCES

1. Livermore Software Technology Corporation, LS-DYNA Computer Code, Version 970.