FLAT PLATE PUNCTURE TEST CONVERGENCE STUDY

Douglas J. Ammerman
Sandia National Laboratories

Michael Yaksh
NAC International

Chi-Fung Tso
ARUP

David P. Molitoris
Westinghouse Electric Company

Spencer D. Snow
Idaho National Laboratory

Jeffrey Jordan
Savanna River National Laboratory

ABSTRACT

The ASME Task Group on Computational Mechanics for Explicit Dynamics is investigating the types of finite element models needed to accurately solve various problems that occur frequently in cask design. One type of problem is the 1-meter impact onto a puncture spike. The work described in this paper considers this impact for a relatively thin-walled shell, represented as a flat plate. The effects of mesh refinement, friction coefficient, material models, and finite element code will be discussed. The actual punch, as defined in the transport regulations, is 15 cm in diameter with a corner radius of no more than 6 mm. The punch used in the initial part of this study has the same diameter, but has a corner radius of 25 mm. This more rounded punch was used to allow convergence of the solution with a coarser mesh. A future task will be to investigate the effect of having a punch with a smaller corner radius. The 25-cm thick type 304 stainless steel plate that represents the cask wall is 1 meter in diameter and has added mass on the edge to represent the remainder of the cask. The amount of added mass to use was calculated using Nelms' equation, an empirically derived relationship between weight, wall thickness, and ultimate strength that prevents punch through. The outer edge of the plate is restrained so that it can only move in the direction parallel to the axis of the punch. Results that are compared include the deflection at the edge of the plate, the deflection at the center of the plate, the plastic strains at radius r=50 cm and r=100 cm, and qualitatively, the distribution of plastic strains. Because cask designers are using analyses of this type to determine if the shell will puncture, a failure theory, including the effect of the tri-axial nature of the stress state, is also discussed. The results of this study will help to determine what constitutes an adequate finite element model for analyzing the puncture hypothetical accident.

INTRODUCTION

The Task Group on Computational Modelling for Explicit Analyses in the ASME Boiler and Pressure Vessel Code committee was set up in August 2008 to develop a quantitative finite element modeling guidance document for the explicit dynamic analysis of energy-limited events. This guidance document will be referenced in the ASME Boiler and Pressure Vessel Code Section III Division 3 and NRC Regulatory Guide 7.6 as a means by which the quality of a finite element model may be judged.

In energy limited events, which the guidance document will address, ductile metallic materials will suffer significant plastic strains to take full advantage of their energy absorption capacity. Accuracy of the analyses in predicting large strains is therefore essential.

One of the issues that this guidance document will address is the issue of the quality of a finite element mesh, and in particular, mesh refinement to obtain a convergent solution. That is, for a given structure under a given loading using a given type of element, what is the required mesh density to achieve sufficiently accurate results.

One portion of the guidance document will be devoted to a series of element convergence studies that can aid designers in establishing the mesh refinement requirements necessary to achieve accurate results for a variety of different elements types in regions of high plastic strain. These convergence studies will also aid reviewers in evaluating the quality of a finite element model and the apparent accuracy of its results.

IAEA TS-R-1[1] requires radioactive material transportation packages to withstand a free drop from 1-meter onto a puncture probe. This is one type of problem that traditionally results in large strains within the package. The ability of the package to withstand this event can be determined via finite element analyses. To assure a "quality" analysis, the finite element mesh of both the package and the punch need to be sufficiently refined. The goal of this paper is to demonstrate the degree of mesh refinement that is required to obtain an accurate solution. The puncture problem discussed in this paper is slightly less severe than the regulatory puncture problem in that it has a punch corner radius of 25 mm instead of the regulatory minimum radius of 6 mm. This was done so the mesh refinement study would focus on bending of the plate representing the cask instead of localized shear at the corner of the punch. Future work in this area will investigate the increased level of mesh refinement necessary when the smaller punch radius is considered.

PUNCH CONVERGENCE STUDY PROBLEM STATEMENT

The convergence study was performed using a circular flat plate impacting the punch at its center. The remainder of the cask was represented by added mass at the outer circumference of the plate. Figure 1 shows the geometry of the problem. The problem is analyzed using quarter symmetry. The plate being impacted is 1.016 m in diameter and 25.4 mm thick moving downward with an initial velocity of 4.47 m/sec.

The initial value of the edge mass was calculated based on the traditional Nelms' equation [2] that has been used to size cask wall thickness to prevent puncture. This equation: $WH=2.14S_ud^{1.6}t^{1.4}$, in English units, where H=40 inches, S_u=70,000 psi, d=6 inches, and t=1 inch, gives W=65,840 lbs. Taking into account the weight of the plate and a slight reduction to limit the initial strains, gave a total edge mass of 27,700 kg (in the quarter symmetry model, only ¼ of this mass was applied). A second set of analyses was performed with the edge mass equal to twice this value.

The material for the punch was assumed to be mild steel with an elastic modulus of 208 GPa, Poisson's ratio of 0.3, density of 8000 kg/m^3, a yield stress of 290 MPa, and a hardening modulus of 2.08 GPa. The material for the plate was a 304 stainless steel with a power-law formulation for the true stress-strain curve represented by $\sigma = \sigma_y + A\varepsilon^n$, with σ_y = 208 MPa, A = 1323 MPa, and n =

0.74819. This material has elastic parameters and density similar to the punch. A coefficient of friction of 0.3 is assumed between the plate and the punch.

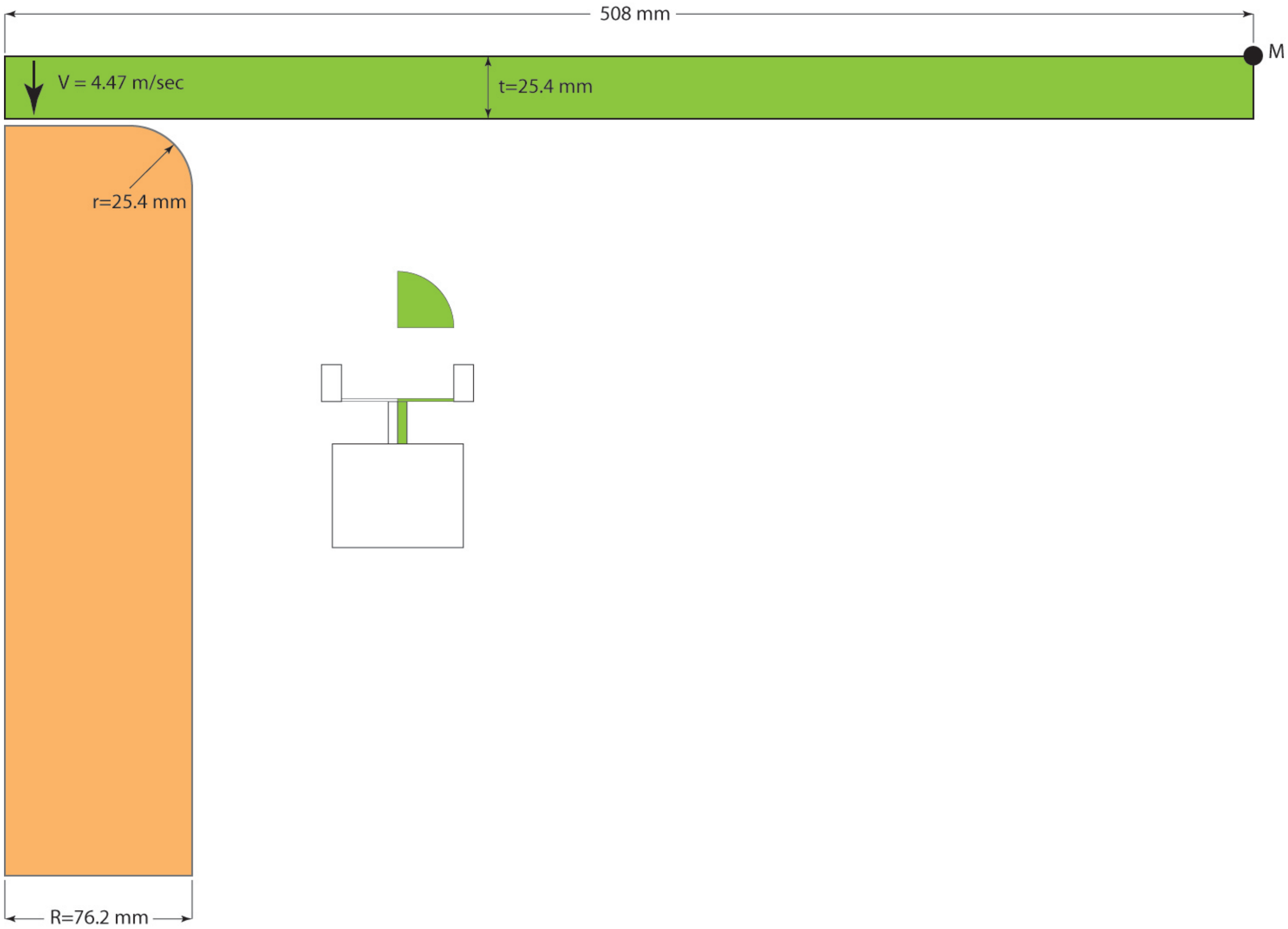


Figure 1. Quarter symmetry punch problem geometry

FINITE ELEMENT MESHES AND BOUNDARY CONDITIONS

The puncture problem was run with finite element meshes 2, 3, 5, 7, and 9 elements through the thickness of the plate. Element aspect ratios of 0.5, 1, and 2 were used in the region of interest, where the plate bends over the punch. For all of the quarter symmetry meshes a symmetry boundary condition (no displacements into the plane of symmetry) was applied. In addition, at the outer edge of the plate the displacements in the radial and tangential directions were constrained (the outer edge could only move in the direction of the initial velocity). At the base of the punch only the vertical displacements were fixed, the punch was free to expand in the radial direction. For all plate meshes, a compatible punch mesh was developed. Early attempts to use a single punch mesh for all of the analyses resulted in undesirable behavior due to the large mismatch in element stiffness at the contact. Figure 2 shows one example of the complete finite element mesh. Figure 3 shows details in the region of the punch/plate interface for all the meshes.

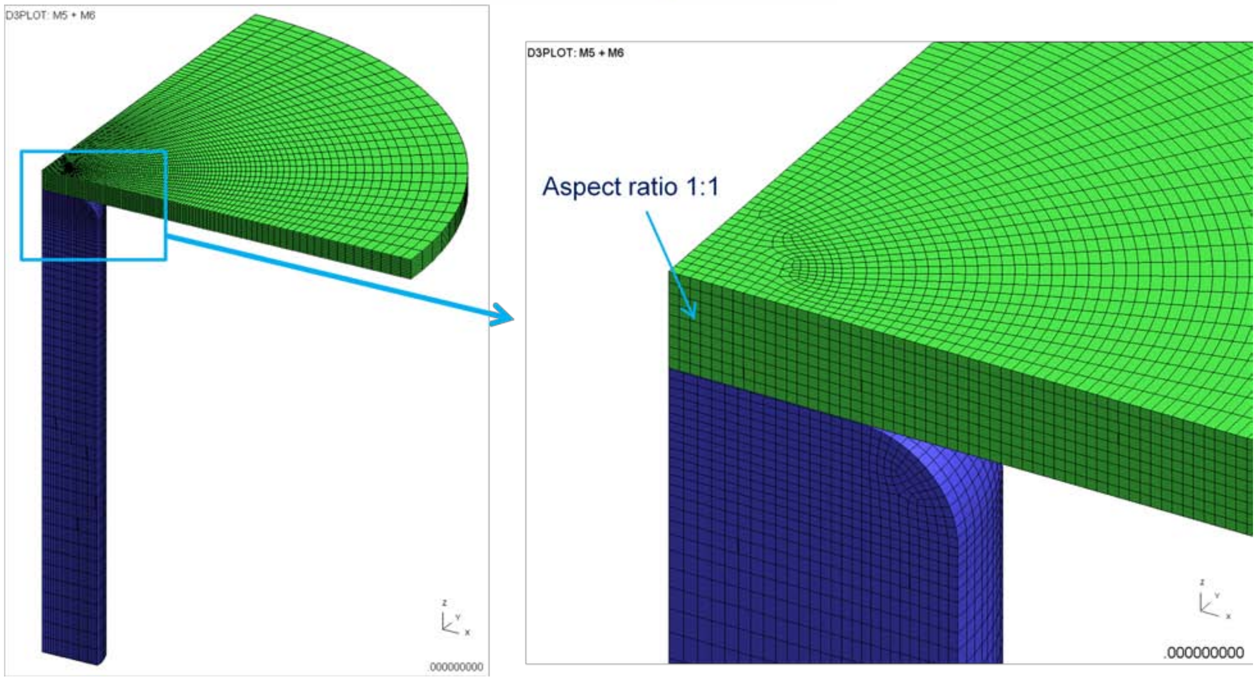


Figure 2. Mesh for the case with 9 elements through the plate thickness and an element aspect ratio of 1.

FINITE ELEMENT ANALYSES PERFORMED

All of the analyses were performed using both single point integration hexahedral elements and fully integrated selectively reduced elements. The analyses were done with the finite element codes LS-DYNA (by NAC International, ARUP, and Westinghouse Electric Company) and ABAQUS-Explicit (by Idaho National Laboratory and Savannah River National Laboratory).

The output parameters that were compared were:

- the vertical deflections at the top center and top outer diameter of the plate
- the equivalent plastic strain at the top center element and at the top and bottom elements located just outside r=5 cm and r=10 cm (integration point values)
- qualitatively, the contour plots of plastic strain.

Meshes for Aspect Ratio of 2 | Meshes for Aspect Ratio of 1*

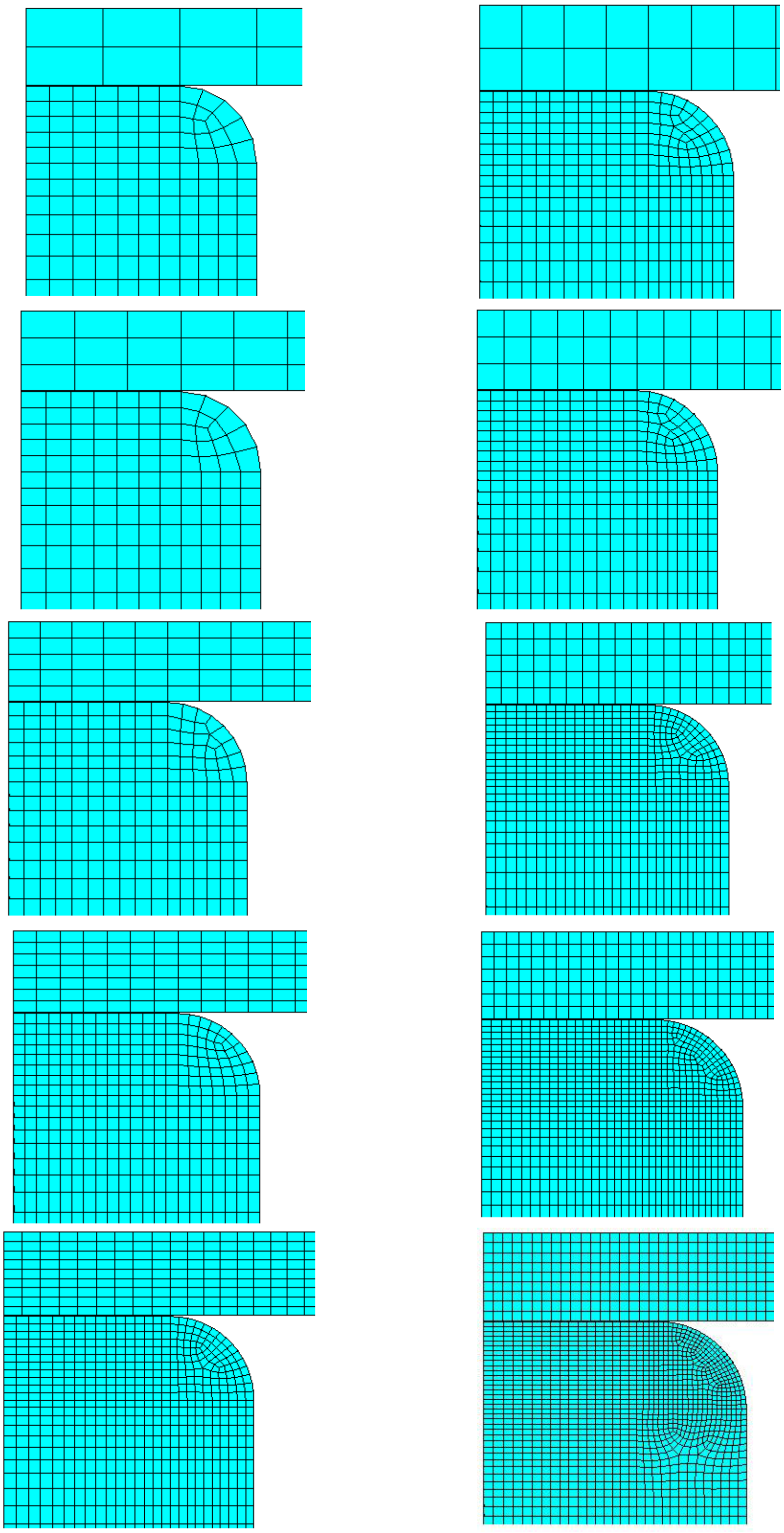
2 elements through the thickness

3 elements through the thickness

5 elements through the thickness

7 elements through the thickness

9 elements through the thickness



*For an aspect ratio of 0.5 the same punch mesh was used but there were twice as many elements along the length of the plate

Figure 3. Detailed meshes in the region of plate/punch contact

RESULTS

The results of the analyses performed by the different users and different codes were very similar. Table 1compares the different analytical results for the case with 7 elements through the plate thickness and an aspect ratio of 1. The edge mass used in this case was twice the calculated Nelms' value to exaggerate the deformation and strains. The plastic strain reported for the fully integrated selectively reduced element is the average of the element integration points.

Table 1. Code-to-code comparison of results for the 7-elements through the thickness case with a 55,340 kg edge mass

		Equivalent Plastic Strain					Deflection (cm)	
Analysis	**Integration Scheme**	**Top Center**	**Top @ r=5 cm**	**Bot @ r=5 cm**	**Top @ r=10 cm**	**Bot @ r=10 cm**	**Top Center**	**Top @ Edge**
SRNL - Abaqus	reduced	0.279	0.648	0.418	0.215	0.205	1.10	20.28
SRNL - Abaqus	full	0.276	0.639	0.407	0.214	0.206	1.09	20.29
Westinghouse - DYNA	reduced	0.280	0.618	0.399	0.212	0.202	1.38	20.34
Westinghouse - DYNA	full	0.277	0.620	0.398	0.211	0.202	1.36	20.37
INL - Abaqus	reduced	0.278	0.620	0.397	0.212	0.202	1.38	20.36
INL - Abaqus	full	0.276	0.617	0.394	0.211	0.202	1.37	20.36
Arup - DYNA	full	0.280	0.626	0.412	0.213	0.204	1.36	20.37

Figure 4 shows the results of the convergence study for the case where the element aspect ratio is 1. Note that the seeming lack of convergence for the plastic strains at the top of the plate are due to the values being taken at integration points, rather than at the surface of the elements. As the mesh gets finer, the location of the strain measurement gets closer to the top of the plate. This also affects the radial location of the integration point. With respect to edge deflection, it is seen that a fairly converged solution is obtained even with the relatively coarse mesh of three elements through the thickness. Results for all of the output variables from the Westinghouse Electric Company analysis using reduced integration are given in Table 2.

Figure 5 shows selected deformed shapes of the results. The two upper figures are for the coarsest mesh used in this study and the two lower ones are for the finest mesh used.

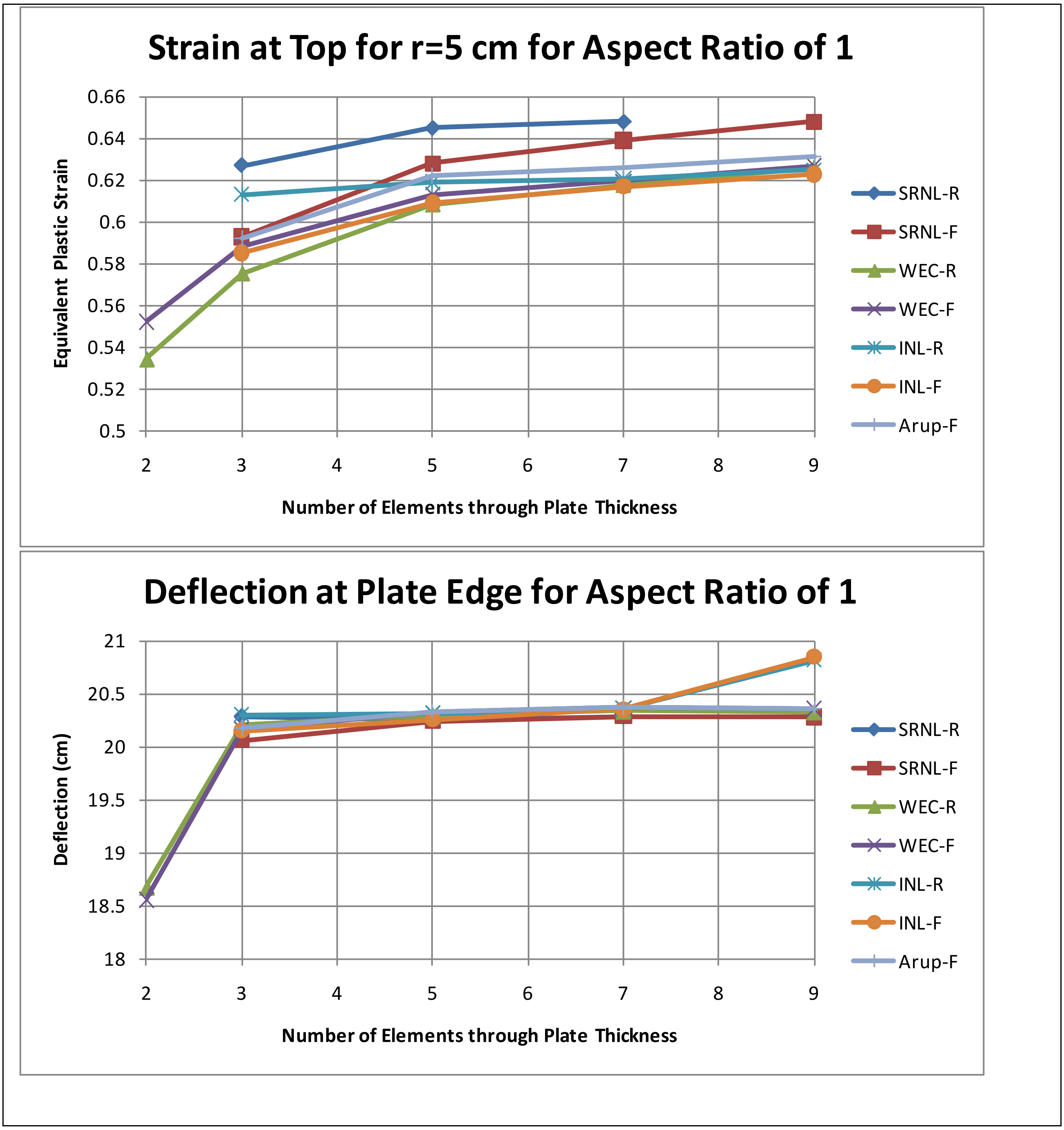


Figure 4. Results of convergence study for element aspect ratio of 1

Table 2. Convergence results for Westinghouse Electric Company analysis with reduced integration and 55,340 edge mass

Elements thru Thickness	Aspect Ratio	EPS	EPS just outside r=5 cm		EPS just outside r=10 cm		Max Δy (cm)	
		Top Center	Top	Bottom	Top	Bottom	Top Center	Top OD
2	2	0.3063	0.4358	0.3138	0.1876	0.1730	0.477	18.478
2	1	0.2718	0.5344	0.3966	0.2008	0.1901	0.447	18.674
2	0.5	0.2823	0.5658	0.4231	0.2056	0.1946	0.550	18.806
3	2	0.2725	0.5180	0.3500	0.1996	0.1908	1.383	20.072
3	1	0.2757	0.5754	0.3987	0.2057	0.1936	1.422	20.204
3	0.5	0.2939	0.6154	0.4502	0.2067	0.1981	1.423	20.232
5	2	0.2824	0.5941	0.4250	0.2035	0.1939	1.386	20.284
5	1	0.2769	0.6083	0.4106	0.2094	0.1985	1.389	20.300
5	0.5	0.2848	0.6240	0.4199	0.2113	0.2012	1.381	20.285
7	2	0.2799	0.6178	0.4100	0.2089	0.1993	1.368	20.356
7	1	0.2800	0.6175	0.3991	0.2123	0.2017	1.382	20.337
7	0.5	0.2853	0.6310	0.4094	0.2133	0.2034	1.365	20.312
9	2	0.2783	0.6224	0.4058	0.2108	0.2006	1.371	20.360
9	1	0.2805	0.6259	0.3868	0.2135	0.2035	1.375	20.330
9	0.5	0.2817	0.6262	0.3771	0.2145	0.2047	1.374	20.311

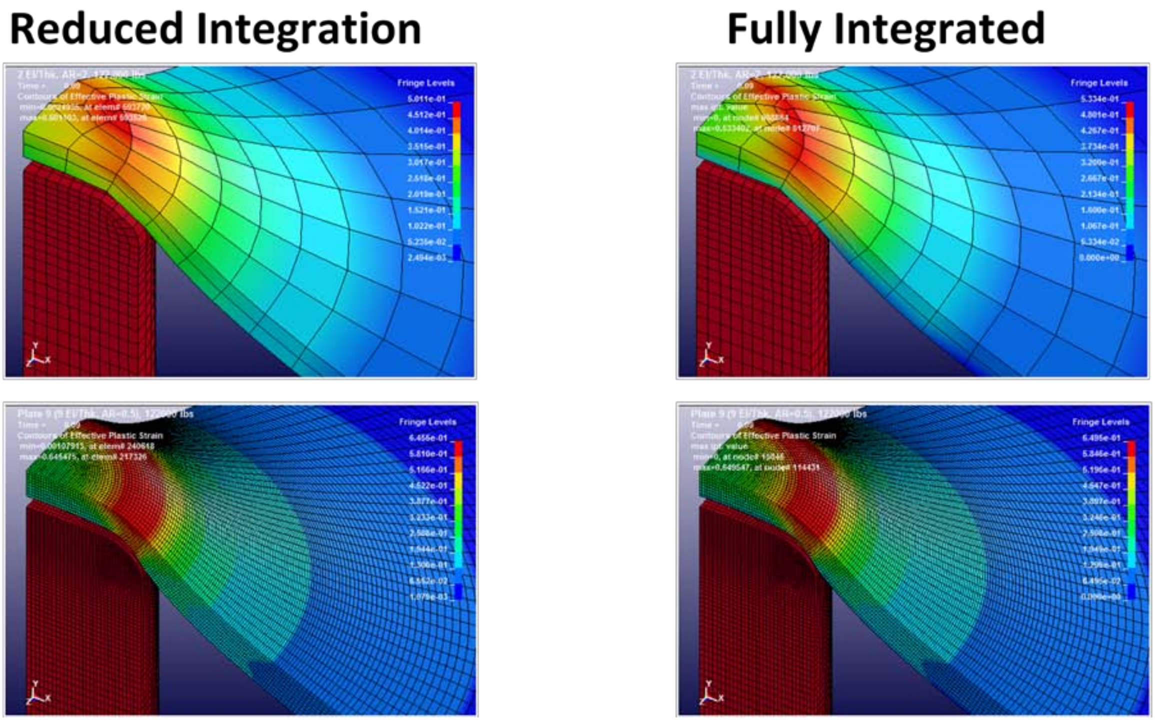


Figure 5. Selected deformed shapes for the 55,340 kg edge mass analyses

CONCLUSIONS

During the conduct of this convergence study problem several key findings were made.

- It is important to have similar meshes in curved surfaces that are contacting each other to avoid mismatch in curvature,
- A very careful characterization of the problem is needed because default parameters in the codes (or the implementation typically used by a user) may result in differences in the results and the implementation of an option within a code may be different, even if the option has the same identification,
- Even experienced analysts may not choose the same parameters for a loosely defined problem, and
- For a well defined problem, different codes and different analysts obtain the same results.

For this problem, where there is a fairly gentle bend at the pin corner, the relatively coarse meshes provided a converged solution. This was largely due to the fact that the plate deformation was being primarily driven by a large membrane strain that develops in the plate as it stretches over the punch. Under a membrane dominated loading condition, mesh refinement becomes less important. This problem does not have a singularity, such as a sharp corner, that could influence results and would require a higher degree of mesh refinement to capture the true behavior.

REFERENCES

[1] IAEA, "Regulations for the Safe Transport of Radioactive Material", TS-R-1, Vienna, Austria, 2005.

[2] Nelms, H. A., "Structural Analysis of Shipping Casks, Vol. 3, Effects of Jacket Physical Properties and Curvature on Punching Resistance", ORNL TM-1312, Oak Ridge National Laboratory, Oak Ridge, TN, 1968.