Proceedings of the 17th International Symposium on the Packaging and Transportation of Radioactive Materials
PATRAM 2013
August 18-23, 2013, San Francisco, CA, USA

Mesh Convergence Studies for Thick Shell Elements Developed by the ASME Special Working Group On Computational Modeling

David P. Molitoris
Westinghouse Electric Company LLC

Gordon S. Bjorkman
U.S. Nuclear Regulatory Commission

Chi-Fung Tso
Arup, London, UK

Michael Yaksh
NAC International

ABSTRACT

The ASME Special Working Group on Computational Modeling for Explicit Dynamics was founded in August 2008 for the purpose of creating a quantitative guidance document for the development of finite element models used to analyze energy-limited events using explicit dynamics software. This document will be referenced in the ASME Code Section III, Division 3 and the next revision of NRC Regulatory Guide 7.6 as a means by which the quality of a finite element model may be judged. One portion of the 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 element 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.

In this paper, the authors present the results of a convergence study for an impulsively loaded propped cantilever beam constructed of LS-DYNA thick shell elements using both reduced and selectively reduced integration. A large load is applied to produce large deformations and large plastic strains in the beam. The deformation and plastic strain results are then compared to similar results obtained using thin shell elements and hexahedral elements for the beam mesh.

INTRODUCTION

Explicit finite element codes, such as LS-DYNA [1] and ABAQUS [2], are used to analyze storage casks and transportation packages for energy-limited events, such as drop impact, puncture and aircraft crash. These codes have evolved to become sufficiently sophisticated and robust as to be able to predict the response to such events with reasonable accuracy. Such results are only achievable, however, by analysts who possess intimate knowledge of structural behavior and an understanding of how to properly construct a finite element model to produce accurate results using these codes.

In the hierarchy of complexity for finite element structural analysis is explicit dynamic analysis, which is typically used to solve crash and impact problems, followed by implicit dynamic analysis, which is typically used to solve problems in forced vibrations and ground motion, and finally static analysis. Engineers who first encounter explicit dynamics codes, and who may be well versed in static analysis and implicit dynamics, soon become aware of the new challenges presented by crash and impact problems using explicit dynamics codes. To help address these new challenges, the ASME Special Working Group

(SWG) on Computational Modeling for Explicit Dynamics was formed in August 2008. The purpose of the SWG is to create a quantitative guidance document for the development of finite element models used to analyze energy-limited events using explicit dynamics codes. This guidance document will support the use of strain-based acceptance criteria being developed for Section III, Division 3, WB/WC 3700 [3]. This document will become a Non-Mandatory Appendix in the ASME Code Section III, Division 3 [4] and be referenced in the next revision of NRC Regulatory Guide 7.6 [5] as a means by which the quality of a finite element model may be judged.

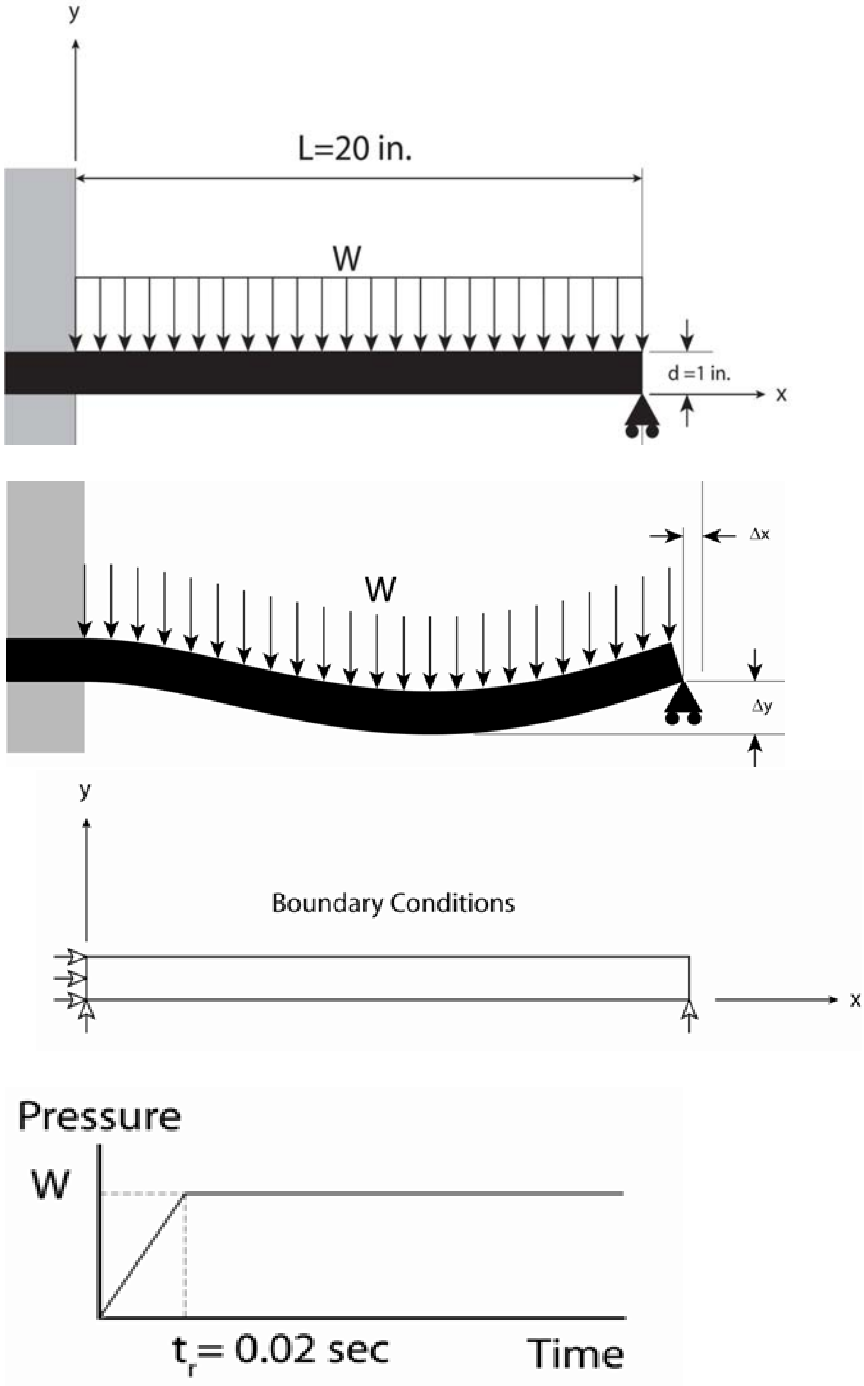

Figure 1: Propped cantilever convergence problem

Of all the considerations that go into constructing an accurate finite element model, the choice of element type and level of refinement of the element mesh are the most fundamental. Therefore, the SWG's first undertaking was to develop 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 accuracy of results. These convergence studies will be incorporated into the guidance document for use. One such study has been completed for thin shells in [6].

In this paper, the authors present the convergence study results for a propped cantilever beam constructed of thick shell elements using LS-DYNA. The problem is investigated using a range of mesh densities and a multiple number of in-plane integration points through the thick shell. The beam is loaded by a uniformly distributed load that is ramped up over a finite time to a constant value. The load is set large enough to induce large plastic deformation. The beam convergence problem is illustrated in Figure 1.

CONVERGENCE PROBLEM DEFINITION

The propped cantilever beam is 20 inches long with a 1-inch square cross section. The beam is built-in on the left end (ux=rz=0) and simply supported at the two bottom corners (uy=0) as shown in Figure 1. The beam is analysed as a plane-strain problem with only one element through the width (z-directon). Therefore, all nodes in the model are restrained from displacement in the z-direction (uz=0).

The beam is assumed to be a stainless steel material with a yield strength of 30 ksi. The material model is a power-law hardening model: $\sigma = \sigma_y + A \cdot \varepsilon_p^n$, with σ_y = 30 ksi, strength coefficient A = 192 ksi, and hardening exponent n = 0.74819. The material has an elastic modulus E = 28e6 psi, a poisson's ratio $\nu = 0.3$, and a mass density of $\rho = 7.39e{-4}$ lbm/in^3.

The loading is a downward uniformly distributed load of magnitude W applied on the top surface of the beam as shown in Figure 1. The pressure load ramps from zero to value "W" over a time interval of t_r = 0.02 seconds, and then remains constant. The pressure loading is defined to remain vertically oriented throughout the beam deflection. For this problem, a pressure load of W=500 psi is used to induce large plastic deformation.

The problem is evaluated with the range of mesh densities shown in Table 1. The beam is meshed with 1, 2, and 3 elements through the beam thickness using element aspect ratios of 5, 2, 1, 0.5, and 0.25. Hence, the coarsest mesh has 1 element over the beam height and 4 elements along the beam length (i.e. 1x4 elements) with an element aspect ratio of 5, and the finest mesh has 3 elements over the beam height and 240 elements along the beam length (i.e. 3x240 elements) with element aspect ratio of 0.25. Each beam mesh is evaluated using 3, 5, 7, and 9 in-plane integration points through the element thickness. Therefore, a total of 5 x 3 x 4 = 60 different cases were evaluated in the convergence study.

The problem is to be analysed using two different element formulations: fully-reduced (single-point) integration and full selectively-reduced integration. The hourglass control settings for the reduced integration formulation are to be set so as to minimize hourglassing and hourglass energy.

Results will be obtained after the full load has been applied and the beam has come to rest. To accomplish this, damping may be applied well after full load application. The following outputs will be obtained.

- Maximum y-deflection of the beam,
- Effective plastic strain at the top built-in end of the beam,

- Maximum effective plastic strain in the center region of the beam.

The deformation and plastic strain results will also be compared to similar results obtained using thin shell elements and hexahedral elements for the beam mesh.

Table 1: Beam Mesh Densities used in Study

Elements through Beam Thickness	Element Aspect Ratios[1]				
	5	2	1	0.5	0.25
1	1x4	1x10	1x20	1x40	1x80
2	2x8	2x20	2x40	2x80	2x160
3	3x12	3x30	3x60	3x120	3x240

(1) Each beam mesh is evaluated with 3, 5, 7, and 9 in-plane integration points through the element thickness.

MODEL SETTINGS AND ASSUMPTIONS

The problem was analyzed using LS-DYNA with the following modeling settings and assumptions:

- The power-law hardening material is modeled using *MAT_SIMPLIFIED_JOHNSON_COOK, which includes the power law relationship ($a + b \cdot \varepsilon^n$).
- The load is applied using distributed nodal load, calculated from the pressure load on the initial area of the top surface of the beam.
- Mass damping is applied well after full load application to damp out oscillations.
- Viscous hourglass control (IHQ=1) with a very small coefficient (QH=0.001) was applied.

RESULTS – REDUCED INTEGRATION

Figure 2 shows the final deformation for all 15 beam mesh densities identified in Table 1 under 500 psi loading using reduced integration elements with 3 in-plane integration points. The meshes with an element aspect ratio of 5, especially with 1 element through the thickness, gave distinctly different results from the other meshes. This was also observed for the other meshes using 5, 7, and 9 in-plane integration points. No hourglassing was observed in any beam models. Figure 3 and Figure 4 show the final deformation and maximum effective plastic strain in the 3x60 beam with 3 in-pane integration points, respectively.

Figure 5 is a plot of the maximum vertical displacement of the beam versus element aspect ratio for the beams with 1, 2, and 3 elements through the thickness using 3, 5, 7, and 9 in-plane integration points. The results appear to converge to a maximum displacement value of about -5.25 inches. The results with 1 element through the beam thickness vary significantly with element aspect ratio. At a large aspect ratio of 5, the beam is too stiff and significantly under predicts displacement while at an aspect ratio of 0.25 (NIP 3 and NIP5), the beam is too soft and over predicts displacement. The results are better behaved with additional elements through the beam thickness at smaller element aspect ratios. At 2 or more elements through the beam thickness, the change in the number of in-plane integration points has a small effect on final displacement results. The displacement results appear reasonably converged using 2 or more elements through the thickness, an element aspect ratio of 1 or less, and 3 in-plane integration points.

Figure 6 is a plot of the effective plastic strain at the top built-in end of the beam versus element aspect ratio for the beams with 1, 2, and 3 elements through the thickness using 3, 5, 7, and 9 in-plane integration points. The results are from the top integration point of the top corner element. The results continue to increase as the number of elements through the thickness and the number of in-plane integration points increase. This apparent non-convergence is expected because the integration points are located in the center of the element. Hence, the integration point gets closer and closer to the top corner as the number of elements and in-plane integration points increase. The maximum effective plastic strain obtained is about 0.25.

Figure 7 is a plot of the maximum effective plastic strain in the center region of the beam versus element aspect ratio for the beams with 1, 2, and 3 elements through the thickness using 3, 5, 7, and 9 in-plane integration points. The results appear to converge to a maximum effective plastic strain value of about 0.14. The results mirror the maximum displacement results shown in Figure 5. The results with 1 element through the thickness vary significantly with element aspect ratio. The results are better behaved with additional elements through the beam thickness and at smaller aspect ratios. In this case, the number of in-plane integration points has a noticeable effect on results. The results appear reasonably converged using 2 or more elements through the thickness, an element aspect ratio of 1 or less, and 7 or more in-plane integration points.

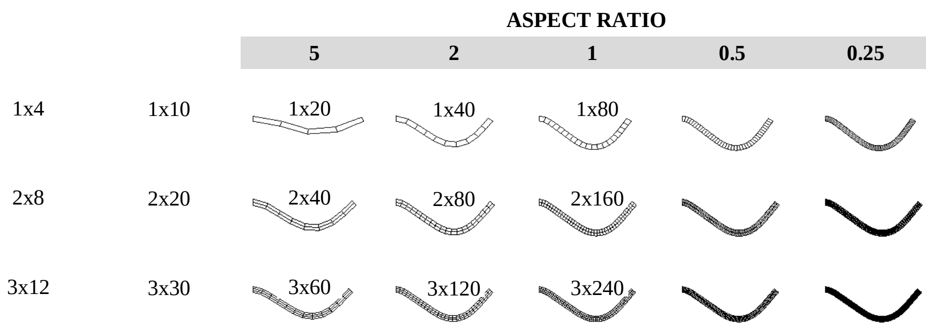

Figure 2: Final Deformation, Reduced Integration with 3 In-Plane Integration Points (NIP3)

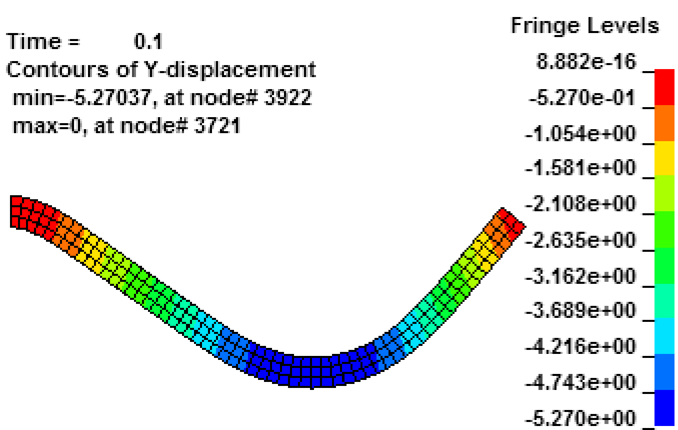

Figure 3: 3x60 Mesh, Final Deformation, Reduced Integration with 3 In-Plane Integration Points (NIP3)

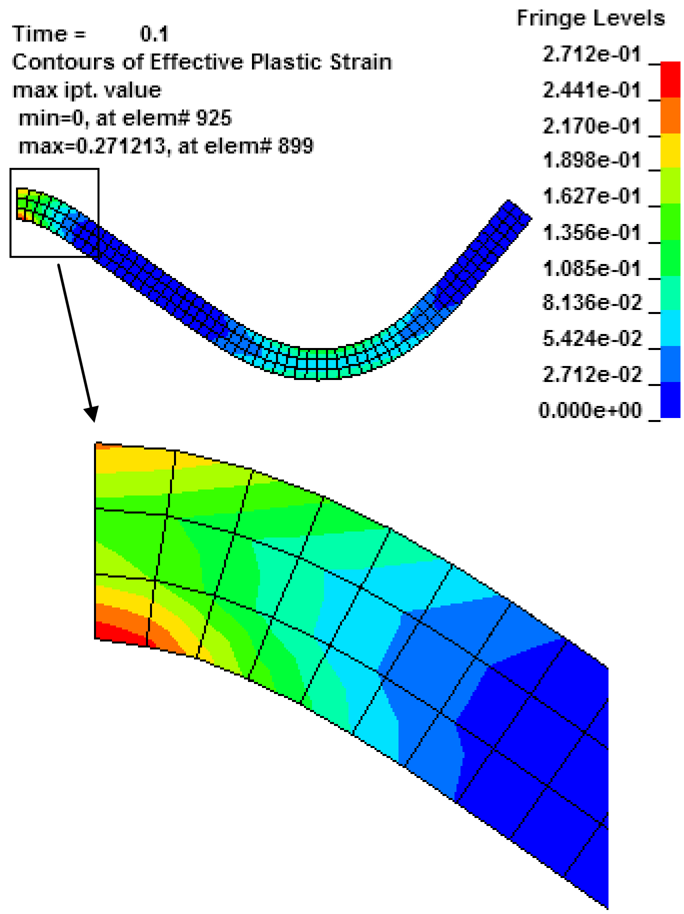

Figure 4: 3x60 Mesh, Maximum Effective Plastic Strain, Reduced Integration with 3 In-Plane Integration Points (NIP3)

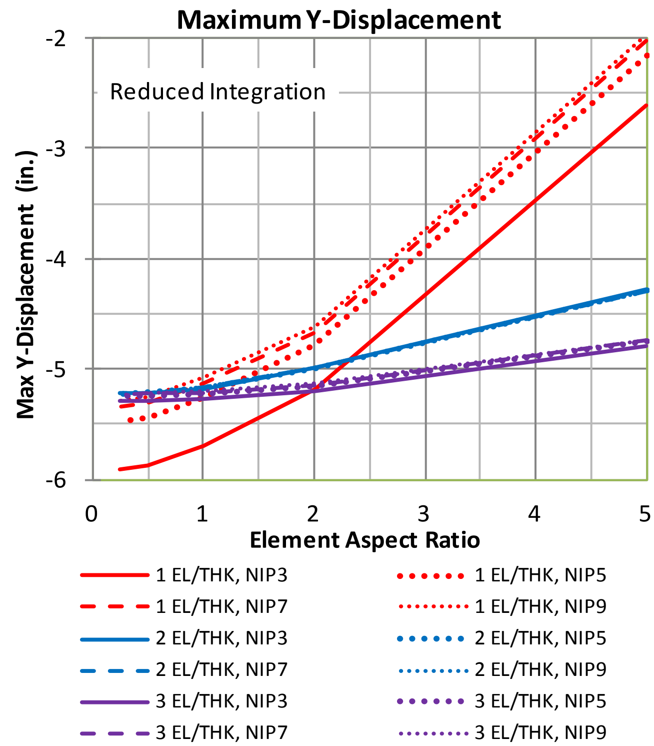

Figure 5: Convergence Results, Reduced Integration

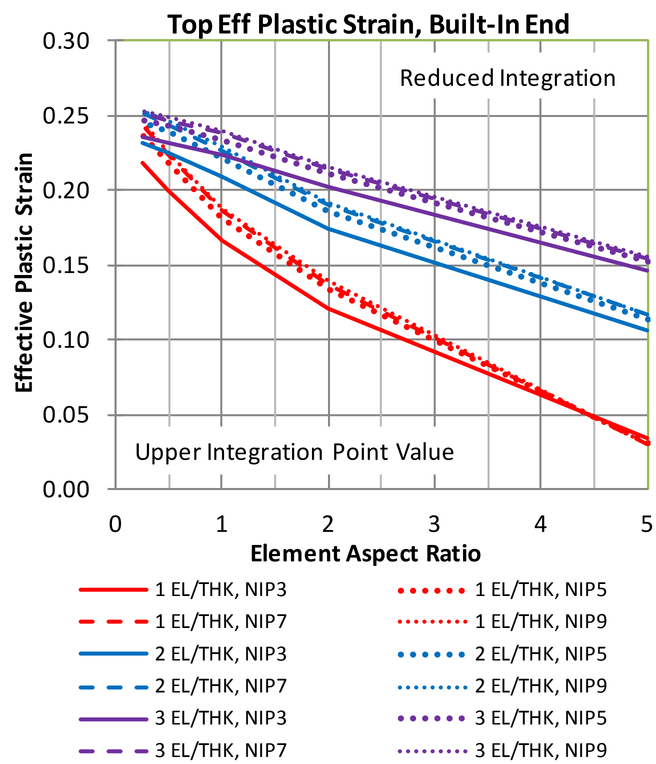

Figure 6: Convergence Results, Reduced Integration

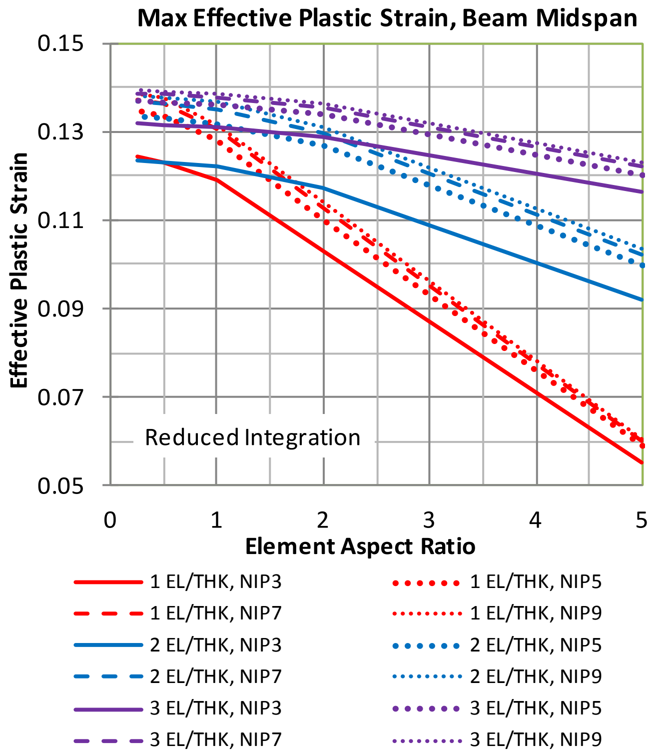

Figure 7: Convergence Results, Reduced Integration

RESULTS – FULL SELECTIVELY-REDUCED INTEGRATION

Figure 8 shows the final deformation for all 15 beam mesh densities identified in Table 1 under 500 psi loading using full selectively-reduced integration elements with 3 in-plane integration points. The results are similar to the reduced integration results; however, these elements appear more sensitive to large aspect ratios. The meshes with an element aspect ratio of 5, especially with 1 element through the thickness, gave distinctly different results from the other meshes. This was also observed for the other meshes using 5, 7, and 9 in-plane integration points. Figure 9 and Figure 10 show the final deformation and maximum effective plastic strain in the 3x60 beam with 3 in-pane integration points, respectively.

Figure 11 is a plot of the maximum vertical displacement of the beam versus element aspect ratio for the beams with 1, 2, and 3 elements through the thickness using 3, 5, 7, and 9 in-plane integration points. The results appear to converge to a maximum displacement value of about -5.25 inches, same as the reduced integration results. These results are less accurate at large aspect ratios than the reduced integration results shown in Figure 5. Similarly, the results with 1 element through the beam thickness vary significantly with element aspect ratio. At a large aspect ratio of 5, the beam is too stiff while at an aspect ratio of 0.25 (NIP 3 and NIP5), the beam is too soft. The results are better behaved with additional elements through the beam thickness at smaller element aspect ratios. At 2 or more elements through the beam thickness, the change in the number of in-plane integration points has a small effect on final displacement results. The displacement results appear reasonably converged using 2 or more elements through the beam thickness, an element aspect ratio of 1 or less, and 3 in-plane integration points.

Figure 12 is a plot of the effective plastic strain at the top built-in end of the beam versus element aspect ratio for the beams with 1, 2, and 3 elements through the thickness using 3, 5, 7, and 9 in-plane integration points. For the full selectively-reduced elements, the results are the average from the top, 4 in-plane integration points in top corner element. The results continue to increase as the integration point gets closer and closer to the top corner. The convergence behavior for 1 element through the thickness is noticeably worse than that observed for the reduced integration solution in Figure 6. The maximum effective plastic strain value obtained is about 0.25, same as the reduced integration results.

Figure 13 is a plot of the maximum effective plastic strain in the center region of the beam versus element aspect ratio for the beams with 1, 2, and 3 elements through the thickness using 3, 5, 7, and 9 in-plane integration points. For the full selectively-reduced elements, the results are the average of 4 in-plane integration points. The results appear to converge to a maximum effective plastic strain value of about 0.14, same as the reduced integration results. The results mirror the maximum displacement results shown in Figure 11. The results with 1 element through the thickness vary significantly with element aspect ratio. The results are better behaved with additional elements through the beam thickness and at smaller aspect ratios. In this case, the number of in-plane integration points has a noticeable effect on results. The results appear reasonably converged using 2 or more elements through the thickness, an element aspect ratio of 1 or less, and 7 or more in-plane integration points.

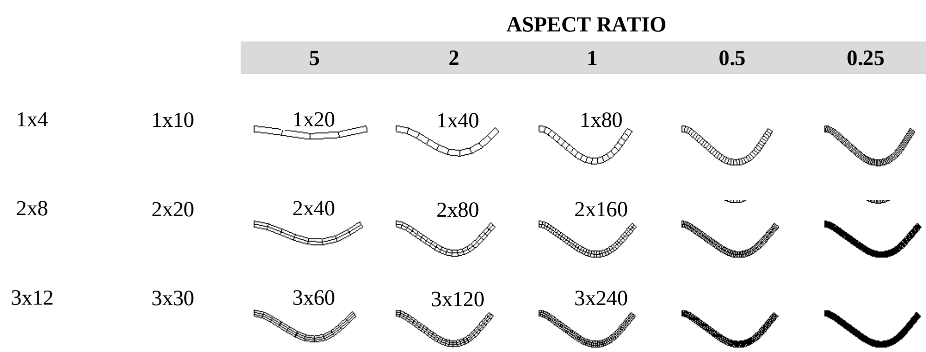

Figure 8: Final Deformation, Full Selectively-Reduced Integration with 3 In-Plane Integration Points

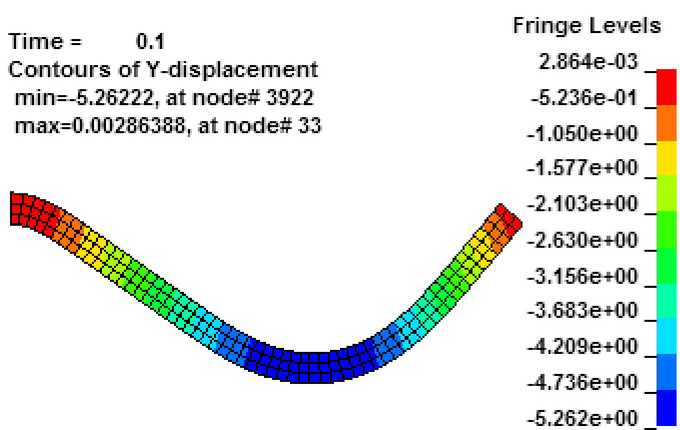

Figure 9: 3x60 Mesh, Final Deformation, Full Selectively-Reduced Integration with 3 In-Plane Integration Points (NIP3)

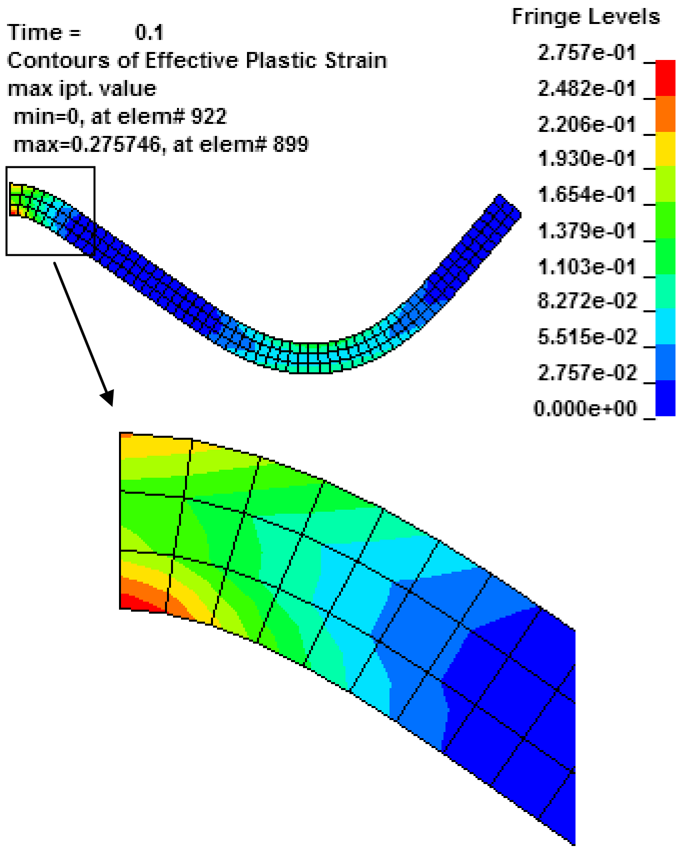

Figure 10: 3x60 Mesh, Maximum Effective Plastic Strain, Full Selectively-Reduced Integration with 3 In-Plane Integration Points (NIP3)

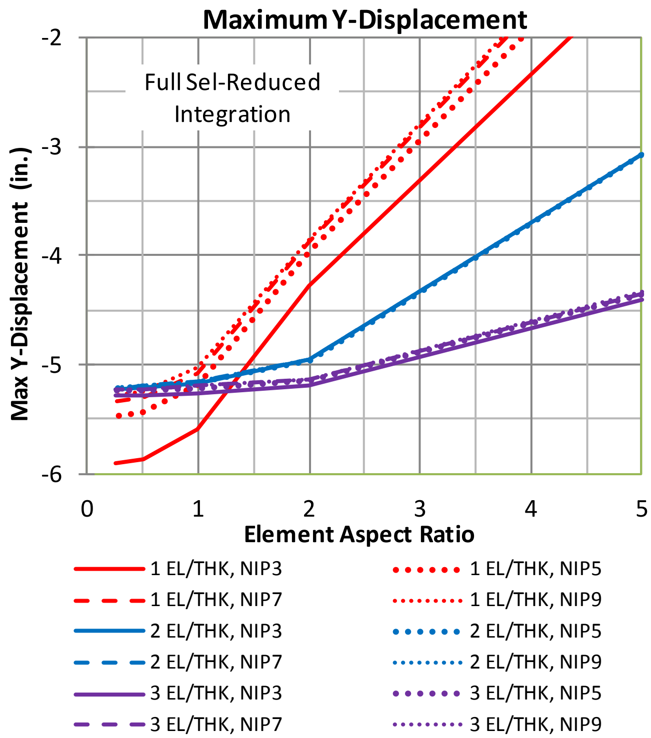

Figure 11: Convergence Results, Full Selectively-Reduced Integration

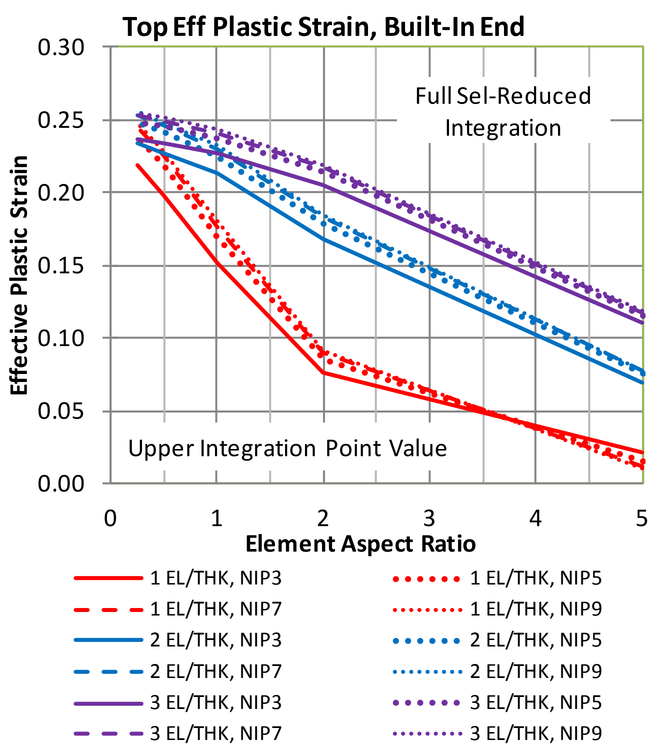

Figure 12: Convergence Results, Full Selectively-Reduced Integration

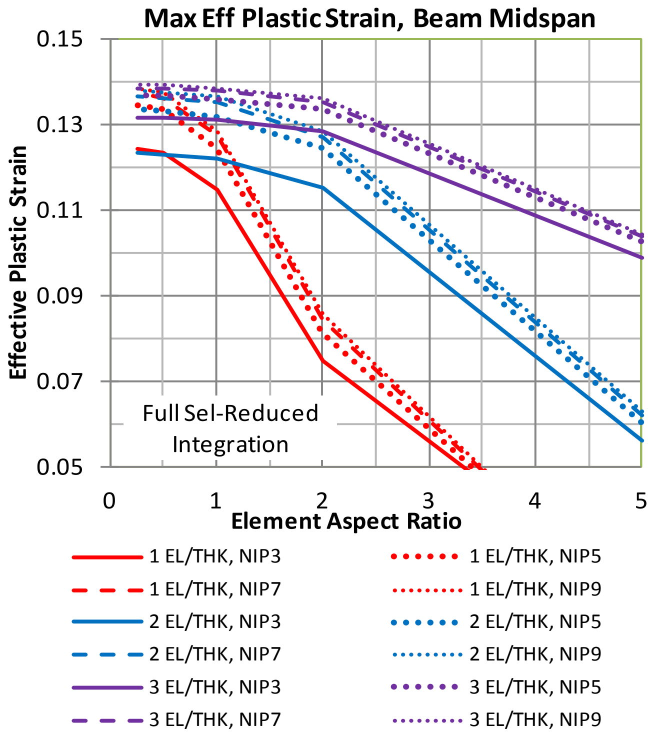

Figure 13: Convergence Results, Full Selectively-Reduced Integration

COMPARISON OF RESULTS – THICK SHELL VS. HEXAHEDRAL VS. THIN SHELL

The beam problem was solved using thick shells, hexahedral, and thin shells with varying mesh densities and in-plane integration points. In this section, the convergence results from each element type are compared. Select beam meshes from each element type at varying element aspect ratios were selected for the comparison. Table 2 summarizes the through thickness beam meshes selected for the comparison as well as the number of in-plane integration points selected for the shell elements. This table also identifies the resulting number of integration points through the beam thickness to give a sense on how close an integration point will be to the beam surface. The selected beam meshes and number of in-plane integration points were chosen based on the expectation that these values will produce reasonably converged results.

Two thin shell models are used in the comparison. For the “Midplane Loading” thin shell model (Figure 14), the loading and the prop support is applied to the beam shells, which is effectively located at the beam midplane surface. For the “Offset Loading” thin shell model (Figure 15), a thin shell (0.01-inch thick) was tied to the top offset surface of the beam shell elements (0.99-inches thick) to offset the load application. Additionally, a thin, vertical rigid shell was attached to the end of the beam to offset the prop support to the actual bottom corner of the beam. These modifications were made to better simulate the actual loading and boundary conditions on the beam.

Table 2: Beam Elements and Mesh Densities

Element				Total Int Pts Through Beam
Type	**Form**	**Number Through Thickness**	**Number In-Plane Int Pts**	
Thick Shell	Reduced	2	7	14
		3	7	21
Hex	Reduced	7	-	7
		9	-	9
	Full SR	9	-	9
Thin Shell	Reduced	1 Midplane Loading(1)	7	7
	Reduced	2 Offset Loading(2)	7	Effectively 8

(1) The loading and the prop support is located at the midplane surface of the beam cross section (see Figure 14). The results are taken from [6].

(2) A thin shell was tied to the top offset surface of the beam shell elements to offset the load application. A thin, vertical rigid shell was attached to the end of the beam to offset the prop support. These modifications make the applied loading and boundary conditions more realistic (see Figure 15).

Figure 16 is a plot of the maximum vertical displacement of the beams versus element aspect ratio. The results appear to converge around a maximum displacement value of about -5.25 inches. For the “Midplane Loading” thin shell model, the maximum displacement of the beam (-4.56 inches) is much less than that of the other beams. This is to be expected because the loading and prop support boundary condition is applied at the midplane surface of the beam. The “Offset Loading” thin shell model provides more accurate results because it better simulates the actual loading and boundary conditions on the beam. This demonstrates the necessity to choose the appropriate element type for this particular beam problem; using a thin shell element without surface offset for loading and boundary conditions will not produce accurate results. The reduced integration hexahedral meshes provide more accurate results at larger element aspect ratios due to more elements through the thickness.

Figure 17 is a plot of effective plastic strain versus element aspect ratio at the top, built-in corner of the beam. The results are from the top integration point of the top corner element, except for beam model “Hex 9 EL/THK Full SR”, where extrapolated nodal results are reported. The results are in good agreement at small element aspect ratios (exception being the “Midplane Loading” thin shell model). The results appear to converge around an effective plastic strain of about 0.26. The extrapolated nodal result from the full selectively-reduced hexahedral mesh appears to have leveled off at an effective plastic strain of about 0.27. The hexahedral meshes provide more accurate results at larger element aspect ratios due to more elements through the thickness.

Figure 18 is a plot of maximum effective plastic strain versus element aspect ratio in the center region of the beam. All results are integration point values, except for the “Hex 9 EL/THK Full SR” beam mesh, which are extrapolated nodal results. The convergence results are more spread out, ranging between 0.11 and 0.14 effective plastic strain. The “Midplane Loading” thin shell model has the smallest value (0.11), which is to be expected given that it deflects the least. The hexahedral meshes provide the next set of higher results. These results increase as the number of elements through the beam thickness increases. The results top out at about 0.126 effective plastic strain, which is an extrapolated nodal value. The highest set of results is provided by the thick and "Offset Loading" thin shell meshes. These element meshes converge around an effective plastic strain of about 0.139 at small aspect ratios.

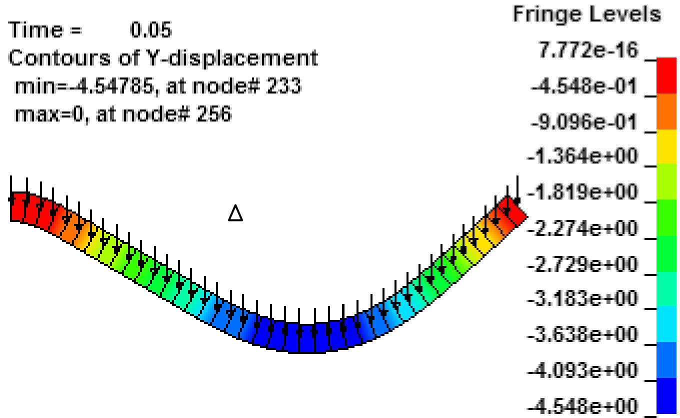

Figure 14: "Midplane Loading" Thin Shell Model

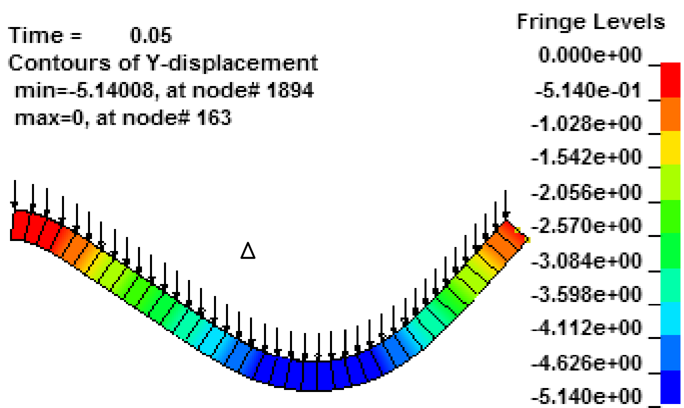

Figure 15: "Offset Loading" Thin Shell Model

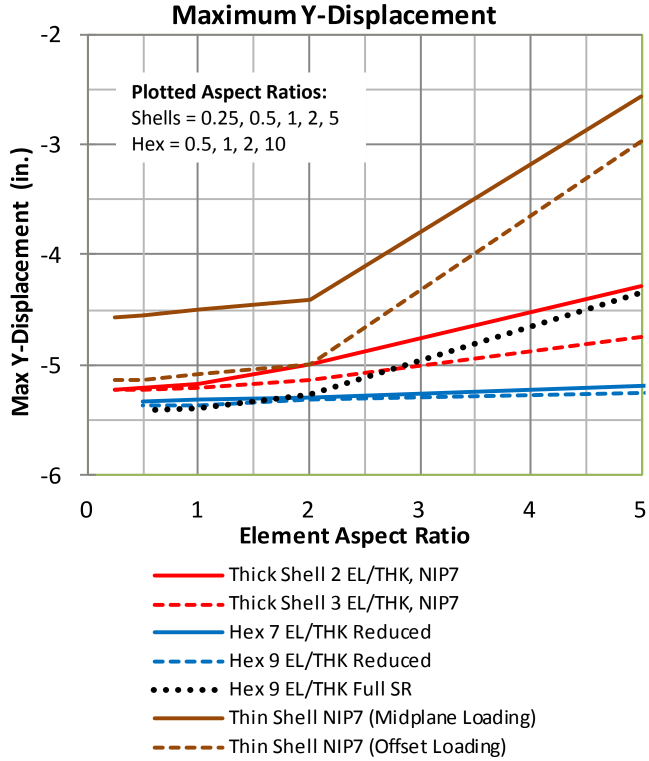

Figure 16: Convergence Results, Shells and Solids

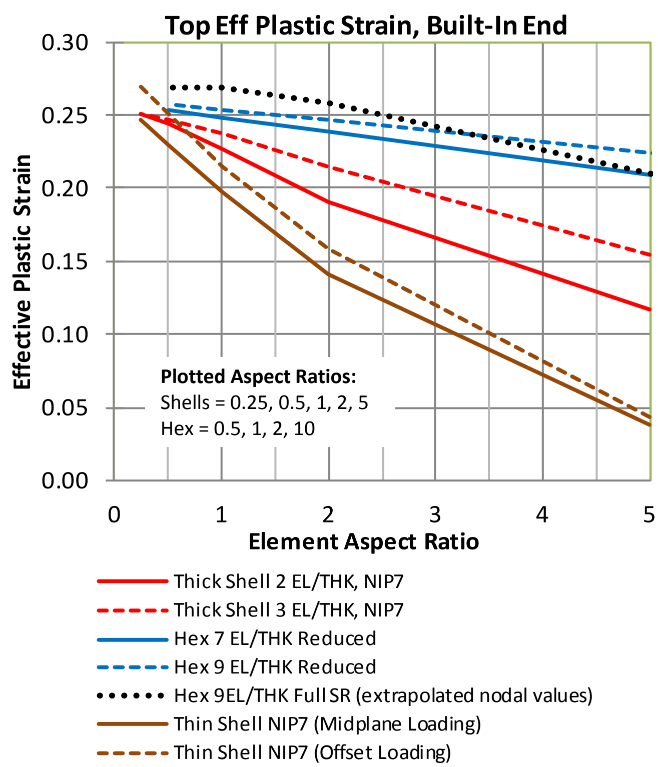

Figure 17: Convergence Results, Shells and Solids

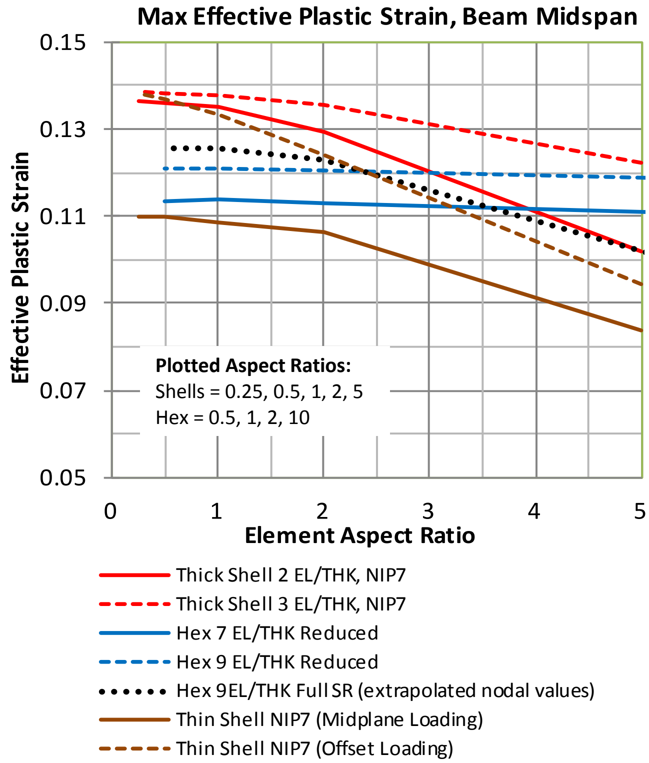

Figure 18: Convergence Results, Shells and Solids

CONCLUSIONS

A propped cantilever beam problem was used to perform a mesh convergence study for LS-DYNA thick shell elements with varying in-plane integration points. From the results presented herein, the following conclusions are drawn relative to this large plastic bending problem:

- The results with 1 element through the beam thickness vary significantly with element aspect ratio. The results are better behaved with additional elements through the beam thickness at smaller element aspect ratios.
- At large aspect ratios, the reduced integration elements produce more accurate results than the full selectively-reduced integration elements. Both element formulations produce essentially identical results at small element aspect ratios. Therefore, there is no advantage to using full selectively-reduced elements for this problem
- The results (deformation, midspan effective plastic strain) appear reasonably converged using 2 or more elements through the thickness, an element aspect ratio of 1 or less, and 7 or more in-plane integration points.

For comparison purposes, the beam problem was also solved using hexahedral and thin shell elements with varying mesh densities and in-plane integration points. Select beam meshes from each element type at varying element aspect ratios were selected for the comparison of convergence results (see Table 2). From the results presented herein, the following conclusions are drawn relative to this large plastic bending problem:

- The thin shell model with the offset loading and boundary conditions provided more realistic results than the midplane loading model. This demonstrates the necessity to choose the appropriate element type for this particular beam problem.
- The results for maximum displacement and the effective plastic strain in the top, built-in corner of the beam were generally in good agreement for all elements types at small element aspect ratios (the exception being the "Midplane Loading" thin shell model).
- The results for maximum effective plastic strain in the center region of the beam are more spread out. The thick and "Offset Loading" thin shell meshes provide the highest set of results, which converge to around 0.139 effective plastic strain at the smallest element aspect ratios. The results from the hexahedral meshes do not converge; the results increase with more elements through the thickness. The results top out at about 0.126 effective plastic strain, which is an extrapolated nodal value.
- The reduced integration hexahedral meshes provided more accurate results at larger element aspect ratios due to more elements through the thickness

REFERENCES

1. LS-DYNA, Livermore Software Technology Corporation, Livermore, CA.
2. ABAQUS/Explicit, Dassault Systemes Simulia Corporation, Providence, RI.
3. Snow S. D., Morton D. K., Pleins E. L., Keating R., "Strain-Based Acceptance Criteria for Energy-Limited Events," PVP2009-77122, ASME PVP Conference, July 26-30, 2009, Prague, Czech Republic.
4. ASME Boiler & Pressure Vessel Code, Section III, Division 3, "Containments for Transportation and Storage of Spent Nuclear Fuel and High Level Radioactive Material and Waste," 2007.
5. Regulatory Guide 7.6, Revision 1, "Design Criteria for the Structural Analysis of Shipping Cask Containment Vessels," U.S. Nuclear Regulatory Commission, March 1978.
6. Bjorkman G. S., Molitoris D. P., "Mesh Convergence Studies for Thin Shell Elements Developed by the ASME Task Group on Computational Modeling," PVP2011-57705, ASME PVP Conference, July 17-21, 2011, Baltimore, Maryland, USA.