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 Hexahedral Elements Developed by the ASME Special Working Group on Computational Modeling

Chi-Fung Tso
Arup, London, UK

David P. Molitoris
Westinghouse Electric Company LLC

Michael Yaksh
NAC International

Spencer Snow
Idaho National Laboratory

Doug Ammerman
Sandia National Laboratories

Gordon S. Bjorkman
U.S. Nuclear Regulatory Commission

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 hexahedral elements using both reduced and selectively reduced integration. Three loading levels are considered; the first maintains strains within the elastic range, the second induces moderate plastic strains, and the third produces large deformations and large plastic strains.

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.

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 [5].

In this paper, the authors present the convergence study results for a propped cantilever beam constructed of hexahedral elements. The beam is loaded by a uniformly distributed load that is ramped up over a finite time to a constant value. The beam convergence problem is illustrated in Figure 1.

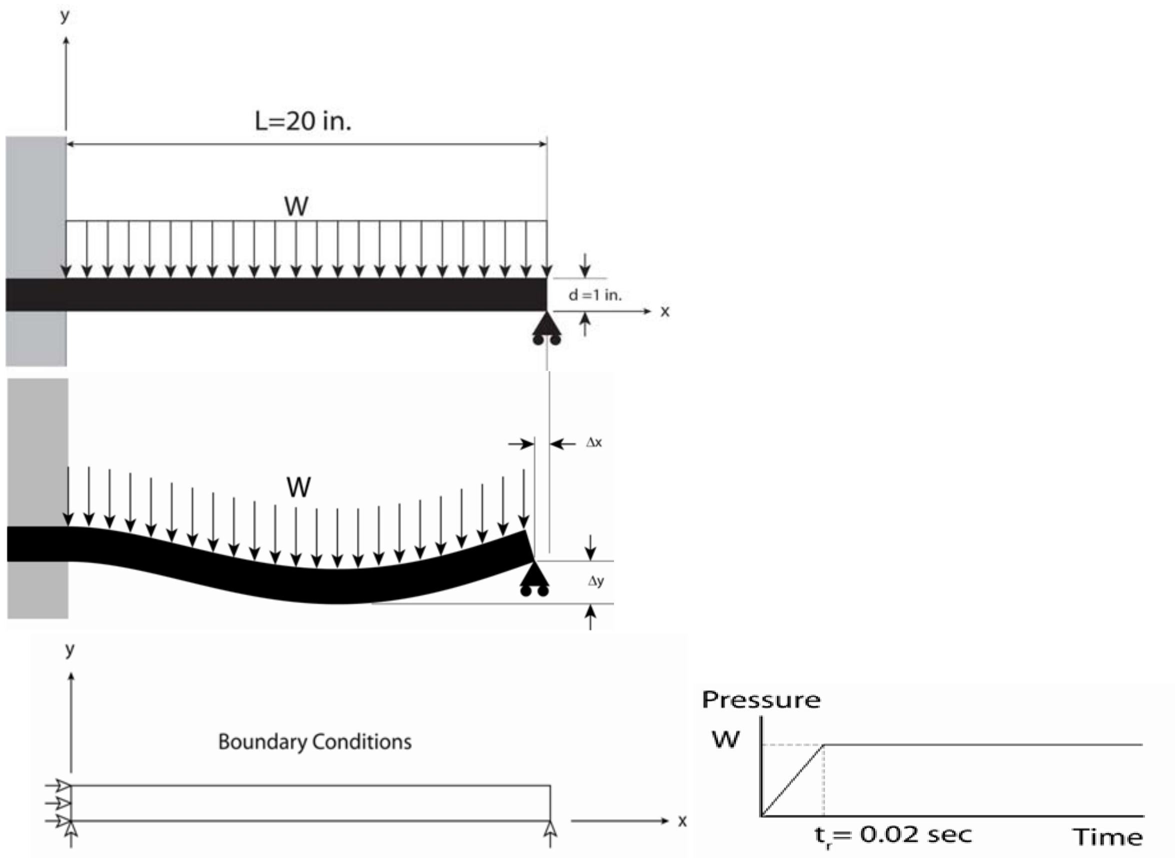


Figure 1: Propped cantilever convergence problem

Three difference load cases were defined in order to assess solution convergence under entirely elastic stresses, moderate plasticity, and large plastic deformation using both single-point integration elements and full selectively-reduced integration elements. Several members of the SWG analysed the problem using LS-DYNA and ABAQUS. The results were collated and compared, and the results are presented in this paper.

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=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 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.

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 ν = 0.3, and a mass density of ρ = 7.39e-4 lb/in^3.

Three levels of loading are defined: W = 100 psi, 240 psi and 500 psi. The smallest load case produces an elastic response while the other two loads produce different levels of plasticity. Plastic hinges will initiate under the 240 psi loading while large plastic deformation will occur under the 500 psi loading.

The problem is evaluated with the range of mesh densities shown in Table 1. The beam is meshed with 2, 3, 5, 7 and 9 elements through the beam thickness using element aspect ratios of 10, 2, 1, and 0.5. Hence, the coarsest mesh has 2 elements over the beam height and 4 elements along the beam length (i.e. 2x4 elements) with an element aspect ratio of 10, and the finest mesh has 9 elements over the beam height and 360 elements along the beam length (i.e. 9x360 elements) with element aspect ratio of 0.5. In total, 5 x 4 = 20 different mesh densities were used for the solution convergence study. (NOTE: it is recognized that aspect ratio of 10 is significantly beyond the aspect ratio commensurate with good practice. Such elements are included in the study to demonstrate that fact.)

The problem is to be analysed using two different element formulations for the 8-noded brick element: fully-reduced (single-point) integration and full selectively-reduced integration. The hourglass control settings for the single-point integration formulation are to be set so as to minimize hourglassing and hourglass energy.

Table 1: Beam Mesh Densities used in Study

Elements through Beam Thickness	Element Aspect Ratios			
	10	**2**	**1**	**0.5**
2	2x4	2x20	2x40	2x80
3	3x6	3x30	3x60	3x120
5	5x10	5x50	5x100	5x200
7	7x14	7x70	7x140	7x280
9	9x18	9x90	9x180	9x360

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 lower surface of the beam,
- X-deflection at the propped edge of the beam,
- X-direction stress, von Mises stress, and effective plastic strain at the top and bottom elements at the built-in end of the beam.

MODEL SETTINGS AND ASSUMPTIONS

The problem was analyzed in LS-DYNA by Arup and Westinghouse Electric Company (WE), and in ABAQUS/Explicit by Idaho National Laboratory (INL). The following identifies some unique modeling assumptions used in some of the analyses:

Load Application:

The vertical pressure loading was applied in two different ways:

- A distributed nodal load, calculated from the pressure load on the initial area of the top surface of the beam, was applied to the nodes on the top beam surface.
- A segment-based "traction" load, which is recalculated at each time-step based on the current area of the top surface of the beam, was applied to the top beam surface. The traction load remains vertical throughout the beam displacement.

Bottom Corner Elastic Elements:

The beam model is supported vertically at the end corner nodes as shown in Figure 1. The "point" type support will cause excessive distortion of the corner support elements under the 500 psi loading. To mitigate the excessive element distortion, some of the analyses were run with elastic only material properties at the bottom corner support elements.

Table 2 summarizes the analyses carried out by three members of the SWG and presented within this paper.

Table 2: Summary of Presented Analyses

SWG Member	FEA Code	Single-Point Int.		Full Selectively-Reduced Int.
		Nodal Load	Traction Load	Nodal Load
Arup	LS-DYNA	All loads All meshes		All loads All meshes
WE(1)	LS-DYNA	All loads All meshes		500 psi load All meshes
INL(2)	ABAQUS		All loads All meshes(3)	

(1) WE = Westinghouse Electric Company
(2) INL = Idaho National Laboratory
(3) 500 psi load case used elastic elements at the bottom corners.

RESULTS

Figure 2 shows the final deformed geometry and plastic strain of the propped cantilever from the three loadings using the finest mesh (9x360 mesh), single-point integration elements, all elastic-plastic elements, and distributed nodal loading as analyzed by Arup using SL-DYNA. The results show the following:

- Under the 100 psi loading, the behavior is entirely elastic,
- Under the 240 psi loading, some plasticity is found at the built-in end,
- Under the 500 psi loading, two plastic hinges have formed resulting in large deformation.

CONVERGENCE – SINGLE-POINT INTEGRATION

Figure 3 shows the final deformation of all 20 beam mesh densities identified in Table 1 under the 500 psi loading, as analyzed by Arup with LS-DYNA, using single-point integration point elements. The beam models with element aspect ratio of 10, especially those with only few elements through the thickness, gave distinctly different results from the other models.

A selection of results from the analyses with single-point integration elements are shown in Figure 4, Figure 5, and Figure 6 for the 100 psi, 240 psi and 500 psi loadings, respectively. These plots show the solution convergence behavior of the various mesh densities as determined by Arup, WE, and INL using both LS-DYNA and ABAQUS/Explicit. The following is noted and observed:

- The results for the 100 psi and 240 psi loadings, as shown in Figure 4 and Figure 5 respectively, were taken from the analyses which do not have the elastic element at the corner supports. The pressure loads were applied as a distributed nodal load in the LS-DYNA analyses and as a traction load in the ABAQUS analyses. At these loading levels, the difference in the way the loads were applied makes negligible difference to the results.
- All results for the 500 psi loading, as shown in Figure 6, were taken from the analyses which applied the pressure load as a distributed nodal load. Due to the large deflections, nodal and traction loads would produce different results. The LS-DYNA results were taken from analyses which do not have the elastic element at the corner supports, while the ABAQUS results were taken from analyses which have elastic elements at the corner supports.
- In terms of displacements, convergence appears reasonably achieved with 7 or more elements through the thickness, and aspect ratios of 2, 1 and 0.5.
- The models with an element aspect ratio of 10, even with a large number of elements through the thickness, gave significantly different results from the others, confirming that such extreme aspect should not be used in bending dominant problems.
- The elemental stress and strain at corner location (0,1,0) continues to increase as the number of elements through the thickness increases. This apparent non-convergence is because stresses are calculated at the integration point of the elements, and not extrapolated to the node at (0,1,0). Hence, the integration point gets closer and closer to the corner (0,1,0) as the number of elements increases, but never actually reaches (0,1,0). With 9 elements through the thickness, the outermost integration point is still 11% of half the beam depth away from the outer surface.
- Results from the two LS-DYNA users are extremely close, except for the results from the model with element aspect ratio of 10 where more variability is present. Results from LS-DYNA and ABAQUS also compare very well for 100 psi and 240 psi loadings, except for the results from the model with element aspect ratio of 10. For the 500 psi loading, the ABAQUS results from the model with

element aspect ratio of 2 are also distinctly different from the LS-DYNA results, but results for the finest meshes agree.

In general, agreement between the codes and users improves with mesh refinement, with the best agreement from the most refined models.

CONVERGENCE – FULL SELECTIVELY-REDUCED INTEGRATION

Figure 3 shows the final deformation of all 20 beam mesh densities identified in Table 1 under the 500 psi loading, as analyzed by Arup with LS-DYNA, using full selectively-reduced integration elements. The beam models with element aspect ratio of 10, especially those with only few elements through the thickness, gave distinctly different results from the other models. At large aspect ratios, the results are observed to be less accurate than those obtained from the single-point integration elements shown in Figure 3. This is likely due to the fact the full selectively-reduced integration elements are susceptible to shear locking at large element aspect ratios under bending.

Results from the single-point integration models and full selectively-reduced integration models are compared in Figure 8 and Figure 9 for the 100 psi and 500 psi loadings, respectively. The comparison shows similar convergence of results for element aspect ratios of 2, 1, and 0.5. At larger element aspect ratios, the full selectively-reduced integration elements are much stiffer in bending due to shear locking than the single-point integration elements.

CONCLUSIONS

The propped cantilever beam problem is a bending dominant loading condition. From the results presented herein, the following conclusions are drawn for this bending dominant condition:

- At least 7 elements through the thickness of the beam and element aspect ratios of 2 or less will provide reasonably converged solutions for deformation, stress, and strain (within 5 % of the converged solution).
- An element aspect ratio of 10 is inadequate in bending and should not be used.
- The single-point integration elements (with appropriate hourglass control) are less sensitive to elements with poor aspect ratios than the full selectively-reduced integration elements.

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.

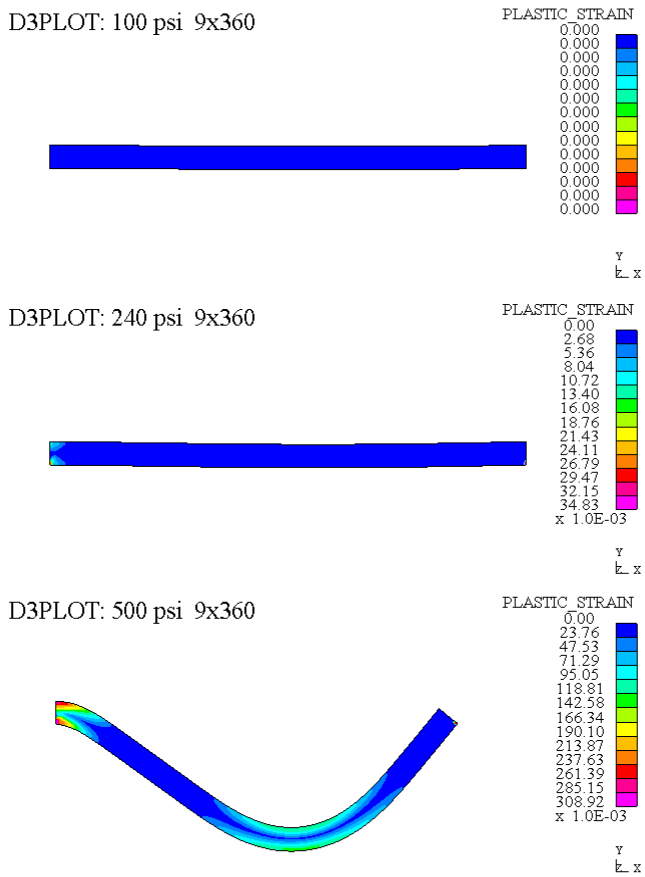


Figure 2: 9x360 Mesh, Final Deformation and Plastic Strain

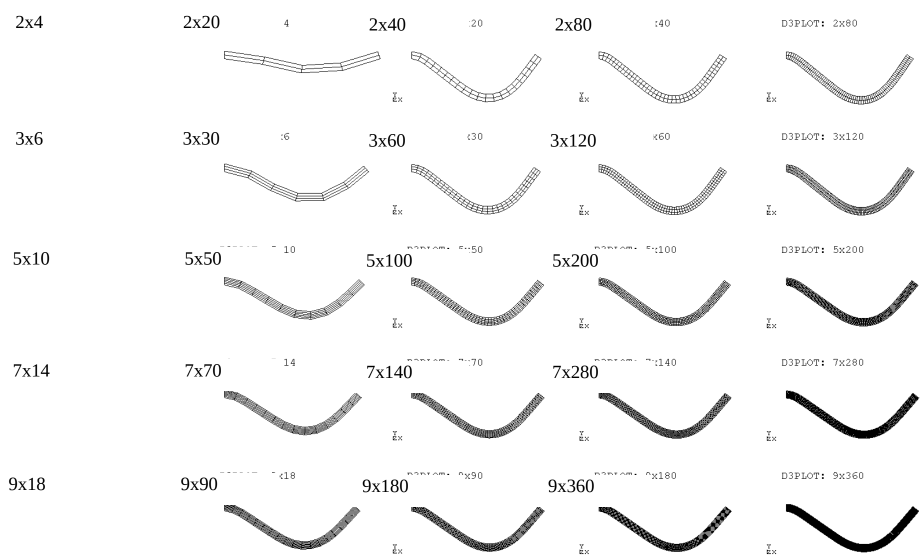
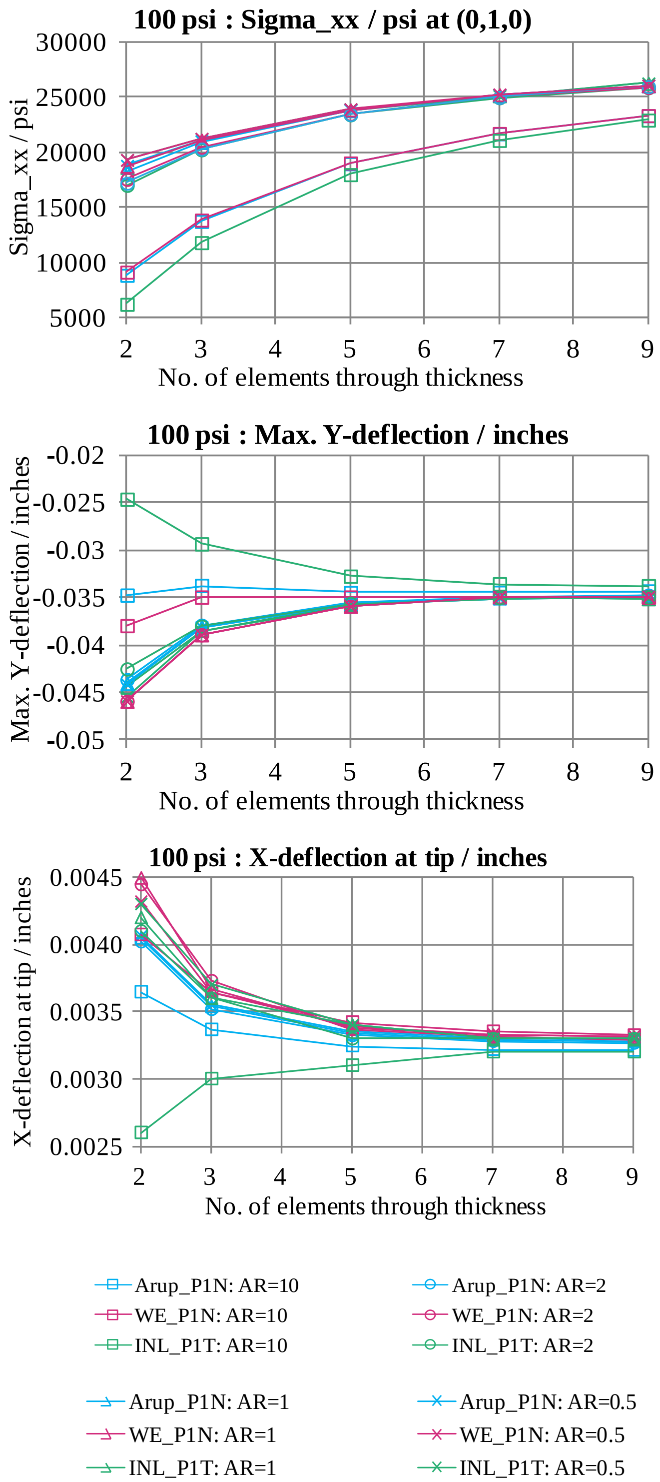
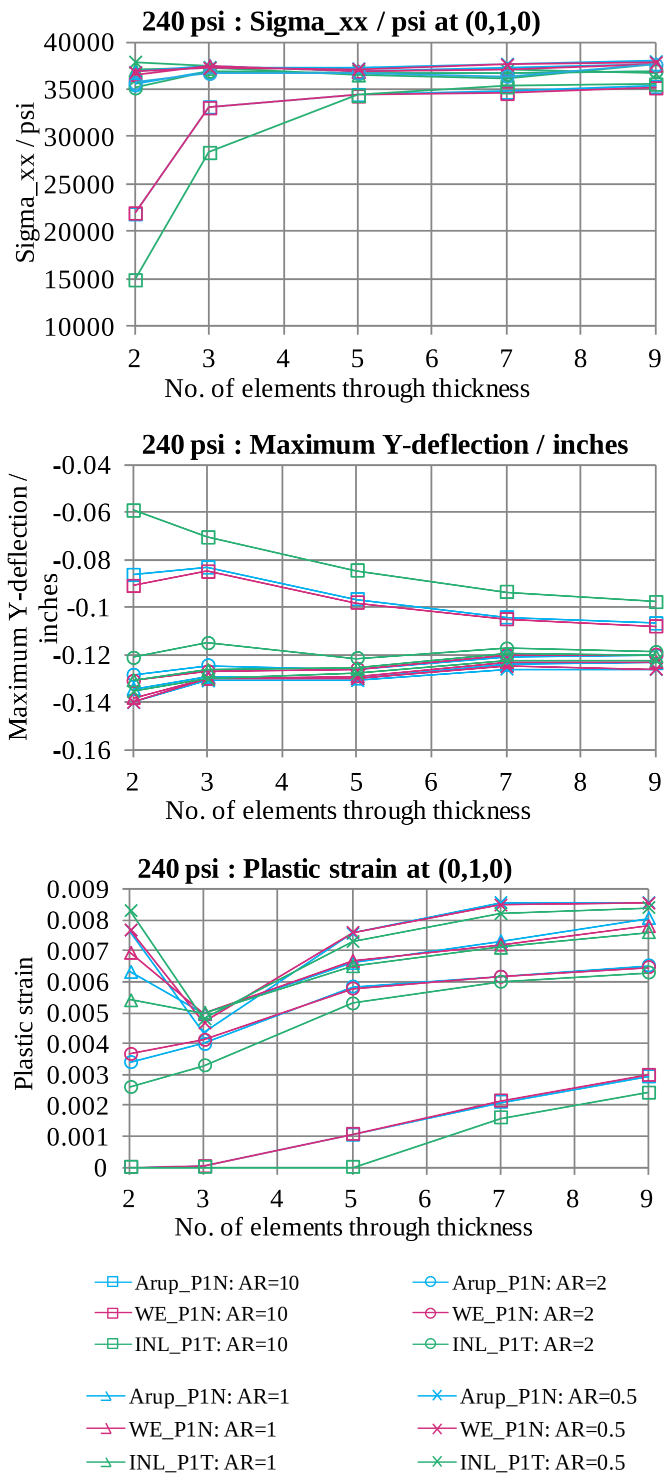


Figure 3: Deformed propped cantilevers, single-point integration elements under 500 psi nodal load



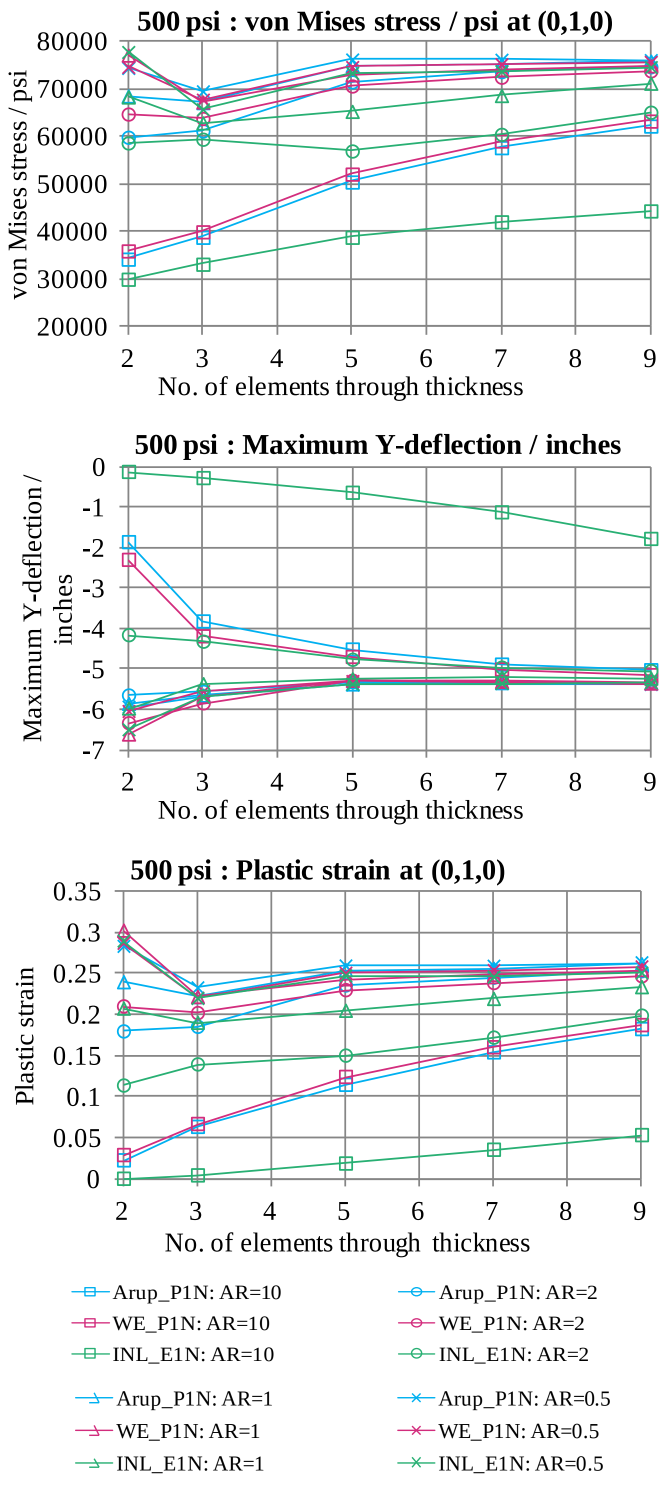
P1N = single-point int., distributed nodal load
P1T = single-point int., traction load

Figure 4: Convergence results, single-point integration elements under 100 psi load



P1N = single-point int., distributed nodal load
P1T = single-point int., traction load

Figure 5: Convergence results, single point integration elements under 240 psi load



P1N = single-point int., distributed nodal load
E1N = elastic elements at support corners, single-point int., distributed nodal load

Figure 6: Convergence results, single-point integration elements under 500 psi load

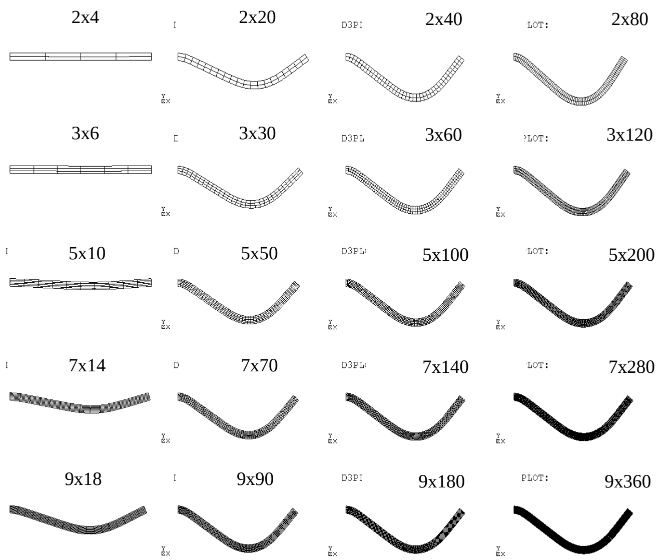
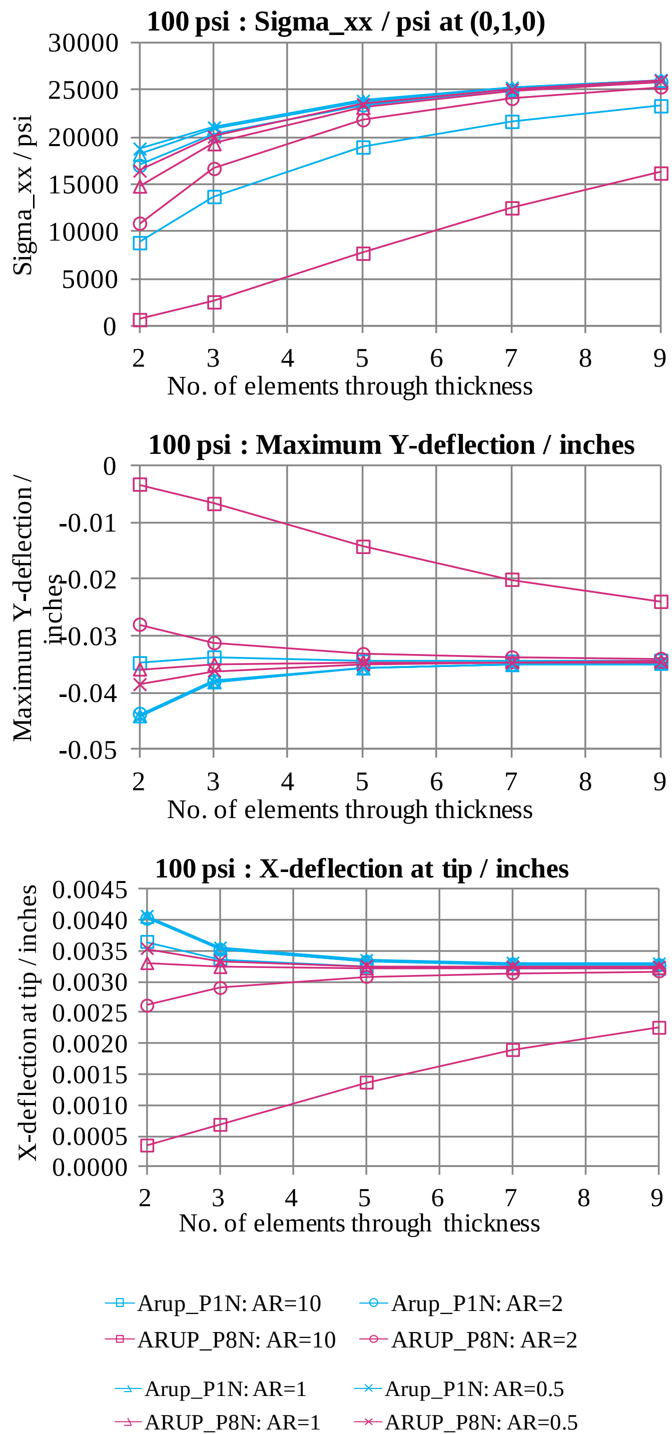
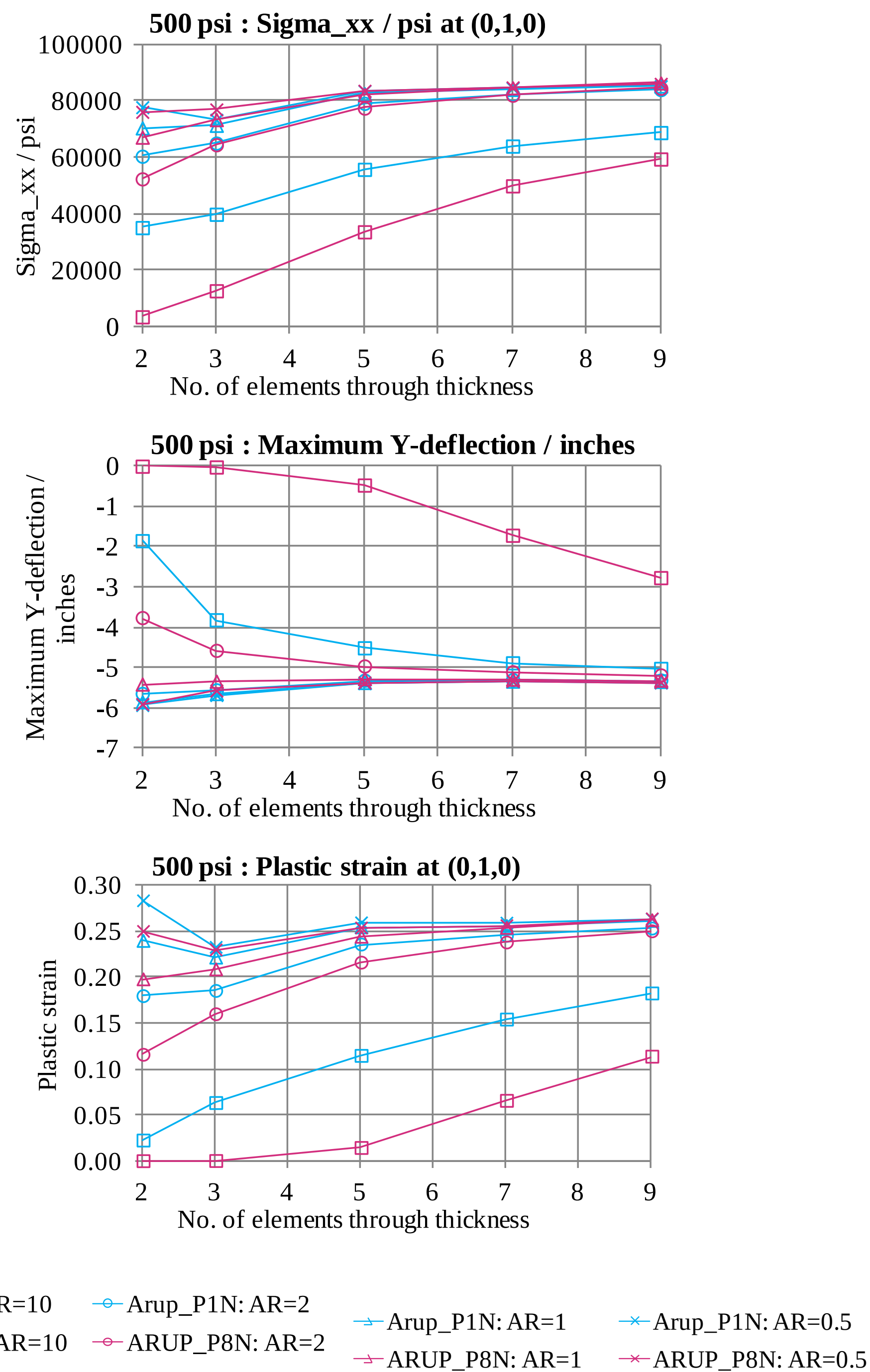


Figure 7: Deformed propped cantilevers, full selectively-reduced integration elements under 500 psi nodal load



P1N = single-point int., distributed nodal load
P8N= full selectively-reduced int., distributed nodal load

Figure 8: Convergence results, single-point integration elements vs. full selectively-reduced integration elements under 100 psi load



P1N = single-point int., distributed nodal load
P8N= full selectively-reduced int., distributed nodal load

Figure 9: Convergence results, single-point integration elements vs. full selectively-reduced integration elements under 500 psi load