Looking to the future PATRAM 2010

PROPPED CANTILEVER MESH CONVERGENCE STUDY USING HEXAHEDRAL ELEMENTS

Chi-Fung Tso
Arup

David P. Molitoris
Westinghouse Electric Company

Spencer Snow
Idaho National Laboratory

Alex Norman
Arup

ABSTRACT

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 modelling 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.

The first convergence study consists of an elegantly simple problem of a cantilevering beam, simply supported at one end and built in at the other, loaded by a uniformly-distributed load that is ramped up over a finite time to a constant value. Three different loads were defined, with the smallest load to cause stresses that are entirely elastic and the largest load to cause large plastic deformations. Material properties, loading rates and boundary conditions were also defined.

A number of the members of the Task Group analysed the problem. The results were collated and compared, and this paper presents some preliminary results of this study.

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 modelling guidance document for the explicit dynamic analysis of energy-limited events.

The 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 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.

The first convergence study consists of an elegantly simple problem of a cantilevering beam, simply supported at one end and built in at the other, loaded by a uniformly-distributed load that is ramped up over a finite time to a constant value. Three different loads were defined, with the smallest load to cause stresses that are entirely elastic and the largest load to cause large plastic deformations. Material properties, loading rates and boundary conditions were also defined.

A number of the members of the Task Group analysed the problem. The results were collated and compared, and this paper presents some preliminary results of this study.

CONVERGENCE PROBLEM DEFINITION

The first convergence problem is a propped cantilever under a uniformly distributed load. It is shown in Figure 1.

The cantilever is 20 inches long with a 1-inch square cross section. It is simply supported at the right end and built-in at the left end. It is loaded with a uniformly distributed load on its top surface which ramps up from 0 at t=0s to W at t=0.02s, and held constant thereafter. The ramp time was chosen to be an order of magnitude greater than the lowest natural frequency of the propped cantilever. The load is to remain oriented vertically regardless of beam deflection.

Three levels of loading are defined: W = 100 psi, 240 psi and 500 psi. The cantilever is expected to remain purely elastic under the 100 psi loading. Plastic hinges are expected to form in the 240 psi loading, and there will be high plasticity under the 500 psi loading.

The problem is to be modeled as a plane strain problem with one element in the Z direction and with all the nodes in the model restrained from displacement in the Z direction. The other boundary conditions are defined as follows:

- The nodes at X=0 and Y=0, and at X=20 and Y=0 to be restrained from Y displacements
- All the nodes on the built-in end restrained from X displacements

The problem is to be analysed with a range of mesh densities, as follows:

- In the Y direction: 2, 3, 5, 7 and 9 elements through the beam thickness
- In the X direction: element sizes to achieve aspect ratio - defined as the ratio of [length in X direction] to [length in Y direction] - of 10, 2, 1 and 0.5

Hence, the coarsest mesh has 2 elements over the height and 4 elements along the length (i.e. 2x4 elements) with an element aspect ratio of 10, and the finest mesh has 9 elements over the height and 360 elements along the length (i.e. 9x360 elements) with element aspect ratio of 0.5. There are, in total, 5 x 4 = 20 different meshes.

(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 material is assumed to be stainless steel, and its stress strain behaviour is to be modelled with a power-law hardening model:

$$\sigma = \sigma_y + A\varepsilon_p^n$$

with σ_y = 30ksi, A = 192ksi, n = 0.74819

Other material properties are: E=28000 ksi, ν = 0.3 and $\rho = 7.385\text{x}10^{-4}$ lbf s^2 in^{-4}.

The problem is to be analysed with fully-reduced single-integration-point 8-noded brick elements. Parameters for hourglass control and damping control are to be defined by the analyst.

Required outputs, all steady-state values at the end of the analysis, are as follows:

- Maximum Y-deflection of the lower surface of the cantilever
- Maximum X-deflection at the bottom edge of the simply-supported end of the cantilever
- X direct stress, von Mises stress and effective plastic strain at the top and bottom nodes at the built-in end of the cantilever

A propped cantilever under a uniformly-distributed load in purly elastic condition is a well-defined problem with a well-known solution. The highest stress in the lengthwise direction is found at the top and bottom faces of the built-in end, with a magnitude of

$$\sigma_{max} = W L^2 c / 8 I \quad \text{per unit beam width}$$

where c is the Y-distance from the neutral axis to the top or bottom surface and I is the second moment of area of the section. The maximum vertical deflection should occur 11.6 in from the built-in end, and is expected to be

$$y_{max} = W L^4 / 185E I \quad \text{per unit beam width}$$

ANALYSES

The problem was analysed in LS-DYNA by Arup and Westinghouse Electric Company (WE), and in ABAQUS/Explicit by Idaho National Laboratory (INL).

Although the key variable in the analyses, and the focus of the study, was mesh refinement to achieve convergence, the following additional variables were included in the analysis matrix:

- Element formulation – although fully-reduced single-integration-point elements were defined to be used for the analyses, fully-integrated selectively-reduced elements were also used by Arup and WE for comparison. Hourglass controls are needed with single-integration-point elements to control zero-stiffness modes of deformation. Selecting appropriate parameters for these controls requires good judgement and experience. An advantage of using fully-integrated selectively-reduced elements is that hourglass control is not required.
- Methods of applying the loads – the 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
  - a segment-based 'traction' load, which is recalculated each time-step based on the current area of the top surface of the beam.
- Using elastic element at the supports – the models are supported vertically at single nodes at the two ends, which could encourage excessive deformation of the corner support elements. Some of the analyses were run with elastic elements at the corner supports to limit this deformation.

Table 1 shows the analyses carried out by the three organizations.

	Single-point Fully-plastic Nodal load	Single-point Fully-plastic Traction load	Single-point Elastic corners Nodal load	Single-point Elastic corners Traction load	Fully-integrated Fully-plastic Nodal load
LS-DYNA (Arup)	All loads All meshes	All loads All meshes	All loads All meshes	All loads All meshes	All loads All meshes
LS-DYNA (WE)	All loads All meshes	-	500 psi only 9x360 only	-	500 psi only All meshes
ABAQUS/Explicit (INL)	-	All loads All meshes	500 psi only All meshes	500 psi only All meshes	-

Table 1. Summary of analyses performed

RESULTS

Figure 2 shows the final deformed geometry of the cantilever with plastic strains and X-direction stresses (in MPa) from the three loadings as obtained from the models with the finest mesh (9x360 mesh) of fully-reduced single-integration point elements, with nodal loads and with all elastic-plastic elements, as analysed by Arup using LS-DYNA.

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

CONVERGENCE – SINGLE INTEGRATION POINT ELEMENTS

Figure 3 shows the deformation of the cantilever under the 500 psi loading at the end of the analysis from all the models, as analysed by Arup with LS-DYNA, using fully-reduced single-integration-point elements. The 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-integration-point elements to show the convergence behavior is shown in Figures 4, 5 and 6 for the 100 psi, 240 psi and 500 psi loadings respectively.

The results of the 100 psi and 240 psi loads, as shown in Figures 4 and 5 respectively, were taken from the analyses which do not have the elastic element at the supports. The loads were applied as nodal loads in the LS-DYNA analyses and as 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 the results of the 500 psi load as shown in Figure 6 were taken from the analyses which applied the load as 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 supports, while the ABAQUS results were taken from analyses which elastic elements at the 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 never be used.

The elemental stresses and strains at (0,1,0) continue to rise 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 (0,1,0). Hence, the integration point gets closer and closer to (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. 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.

In the analyses with the 500 psi loading, 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.

CONCLUSIONS

The results show that 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 and should not be used.

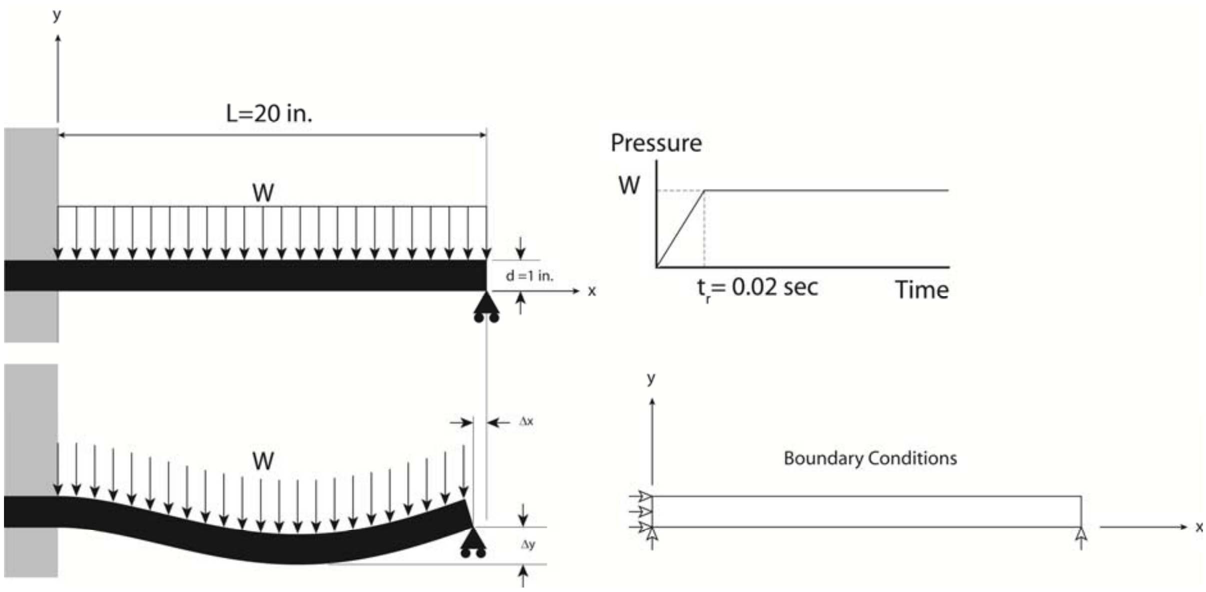

Figure 1. Propped cantilever convergence problem

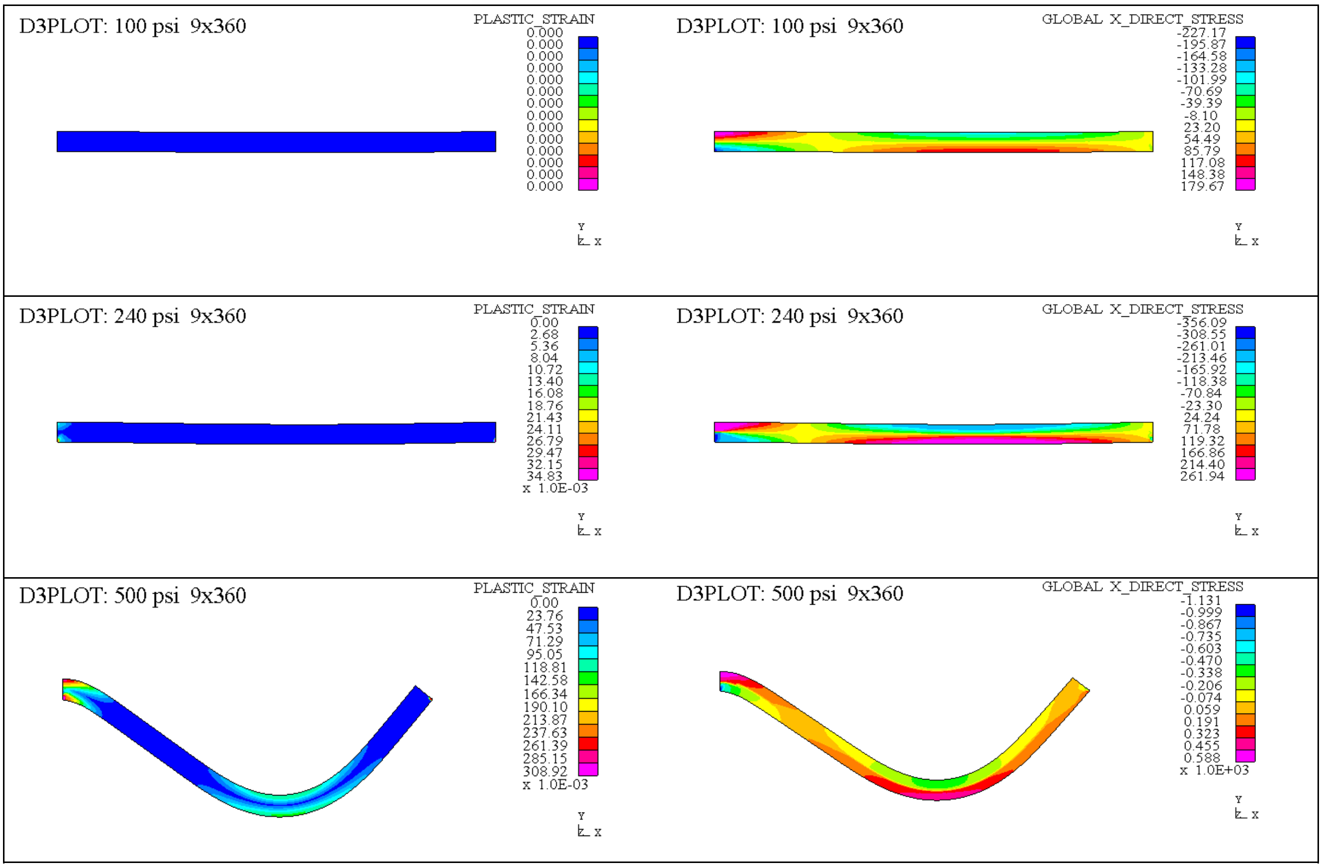

Figure 2. Stress and strain at the end of the analysis using the 9x360 mesh

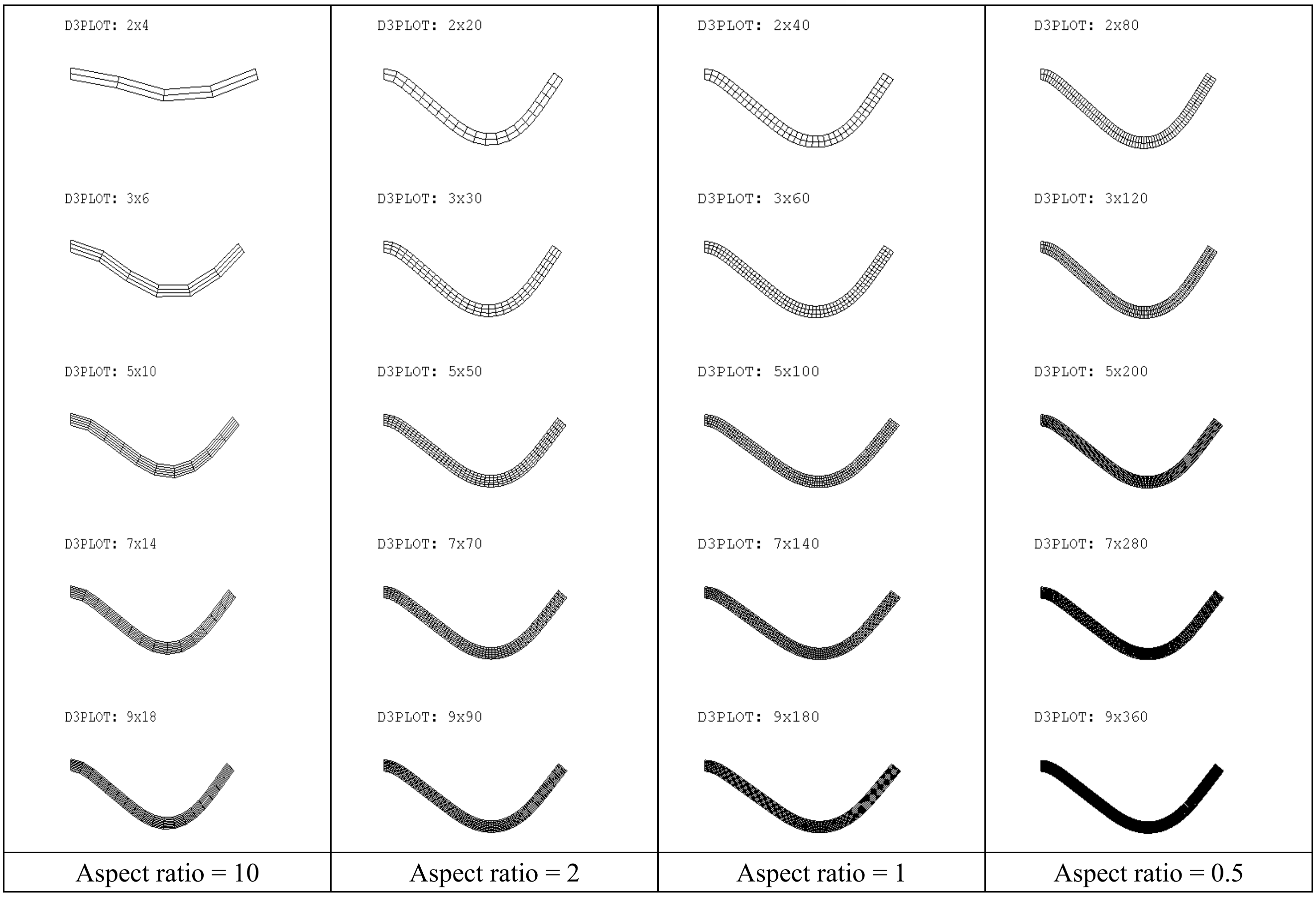

Figure 3. Deformed propped cantilever under the 500 psi nodal load from the different meshes

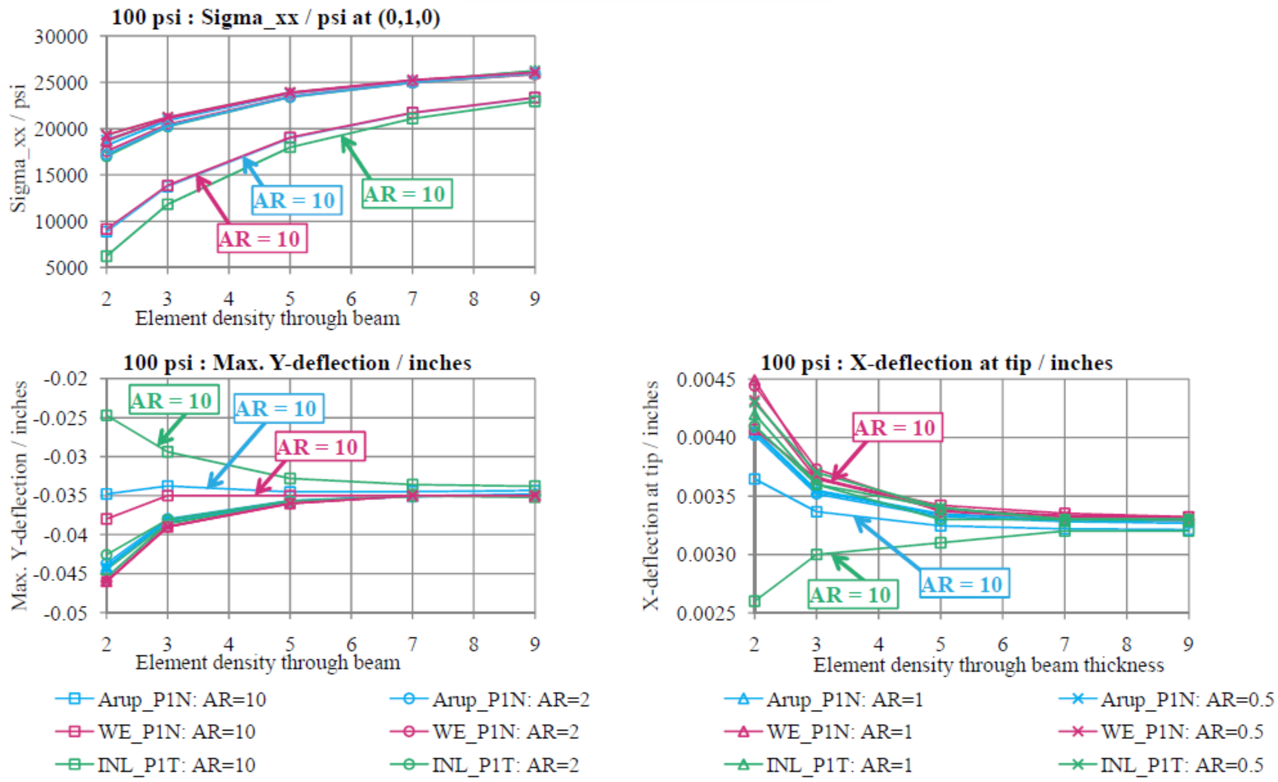

Figure 4. Selection of results from 100 psi loading

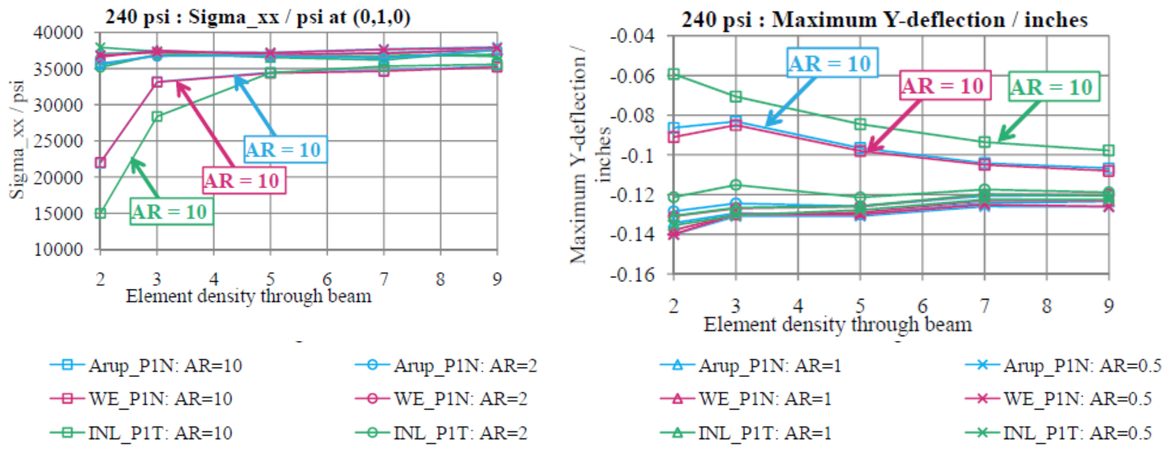

Figure 5. Selection of results from 240 psi loading

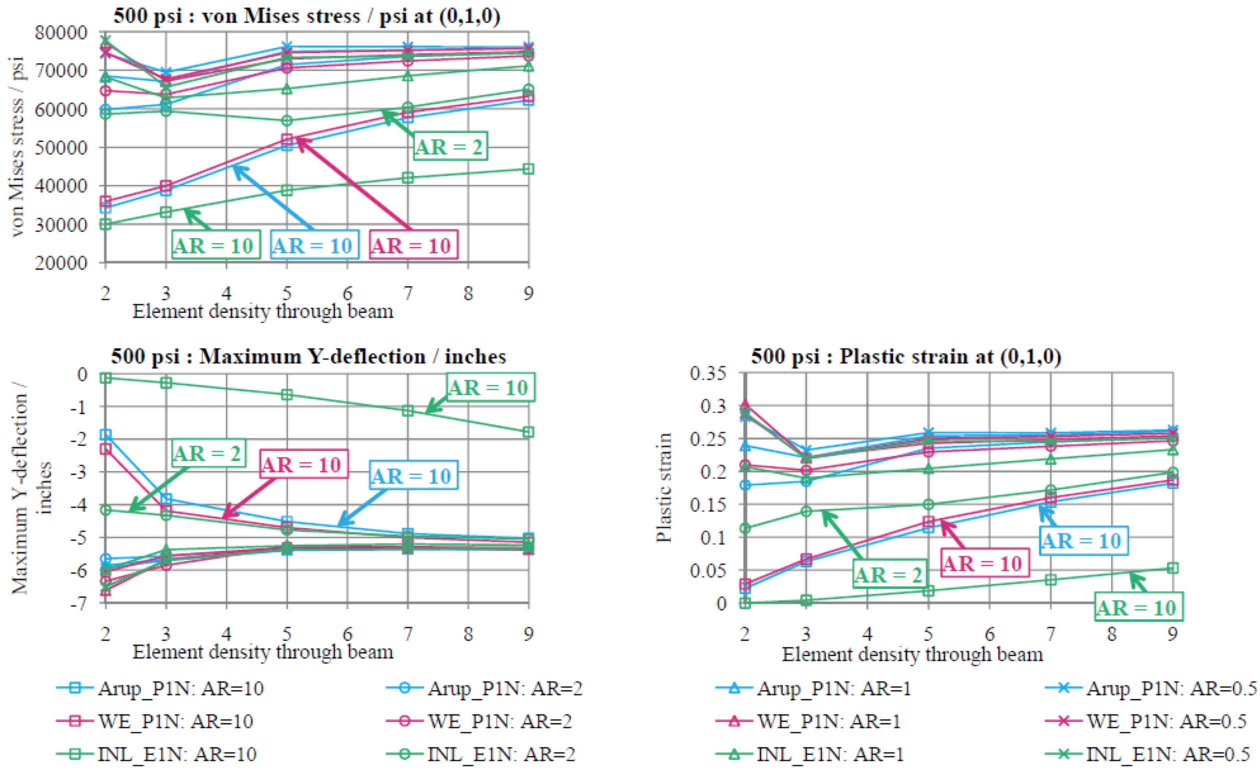

Figure 6. Selection of results from 500 psi loading