VECTRA
GROUP LIMITED

Memorandum

To	Croft Associates
Copies	
From	Gareth Jones
Department	
Reference	925-3272/M2
Date	15/03/10

Safkeg LS 3979A- Mesh Sensitivity Analyses

In response to a comment by the NRC on the FEA of the Safkeg LS 3979A, Vectra has carried out a mesh sensitivity study. The study used the model of the body of the Safkeg LS loaded under internal pressure only. This simplified model allowed many cases to be run quickly but still gives results which can be related to the main FEA study.

The region circled in Figure 1, at the join between the body and the top flange, was the part of the model where the mesh was refined. The number of elements through the thickness of the body was increased from 4, as used in the original study, to 6, 8, 10, 12, 14 and 16. In all cases, the number of elements along the length of this section was adjusted so that the aspect ratio of the elements (length of one side of an element to the other side of the element) was maintained. All of these cases used first-order elements, as used in the original study. Two additional cases used second-order elements. Both of these cases used 16 elements through the wall thickness, but one used 4 elements around the fillet radius, as used in all the other cases, and one used 8 elements around the radius. The results using second order elements are considered to be closest to the actual stresses, however for reasons of computation power and model size, it is not always possible to use these elements because the contact algorithms, which were required in the main study, are not efficient with second order elements, hence the need, as in this case, for the use of fewer, single-order elements.

Results

Figure 2 shows the stress profile, from the inside to the outside, at the junction between the body and top flange for all the cases. The stress shown is Tresca, which is the same as the stress intensity as required by NRC Regulatory Guide 7.6. The model with 4 elements through the thickness produces a profile which, whilst a fair representation of the actual stress profile, does under predict the peak stresses; however, the results are sufficient to give a good approximation to the deformations of the container, as described below, and hence their use for most flask impact analyses. The accuracy of prediction of the peak stresses improves the greater the number of elements through the thickness, as might be expected. As stated above, the results using second order elements should give an accurate profile. The case with 8 elements around the radius had a slightly smaller peak stress and the case with 4 elements around the radius.

Registered Office
Europa House, 310 Europa Boulevard, Gemini Business Park,
Westbrook, WARRINGTON WA5 7YQ
Tel: +44 (0) 1925 444648 Fax: +44 (0) 1925 444701
Email: info@vectragroup.co.uk Website: www.vectragroup.co.uk
Registered in England and Wales No. 01803667

For the assessment of the containment vessel, membrane, bending and peak stresses were calculated and compared with allowable values. The membrane, bending and peak stresses for this study are given in Table 1 and figures 3-5. The variation in membrane stress was only 6%, the variation in bending stress was 14% and the variation in peak stress was 92%. If we take the final case, second order elements and mesh refinement around the radius, as being closest to the correct answer, the difference between this and the first case with 4 elements was -4%, 11% and 86% for the membrane, bending and peak stress respectively. For the HAC cases, the peak stress is not assessed as it does not affect the predicted deformations of the container during impact.

For the NCT cases, the peak stress result is more important as it affects the fatigue life. The influence of peak stress vs profile is not straightforward, due to the restraining influence of the lower stressed material within the bulk of the section, however as a conservative approach Vectra has reviewed all the stresses results from the original study and has applied an 86% increase to all of the peak stresses. The design margins were still above zero in every case.

Table 1: Stress linearization

Mesh	**Membrane stress (MPa)**	**Bending stress (MPa)**	**Peak stress (MPa)**
4 elements	7.4	16.5	17.0
6 elements	7.3	17.0	19.0
8 elements	7.3	17.9	22.0
10 elements	7.3	17.9	23.0
12 elements	7.3	18.1	23.8
14 elements	7.4	18.6	25.5
16 elements	7.5	18.8	27.4
16 elements, 2nd order	7.1	18.4	32.7
16 elements, 2nd order, 8 elements around radius	7.1	18.3	31.7

Conclusions

1. The mesh used in the original study was adequate for the HAC cases as it produced reasonable estimates of the membrane and bending stresses.
2. The mesh used in the original study was less able to accurately capture peak stresses. A very refined mesh, using second-order elements is required to accurately capture the peak stresses at some locations. However, for reasons of model size and computational capacity, it is not generally practical to use this level of refinement in the main study.
3. All the stresses from the original study have been reviewed and peak stresses have been scaled by a factor of 1.86 based on the results of this sensitivity study. All of the design margins were still above zero, thus confirming that, even with a very conservative assessment, the design is still acceptable.

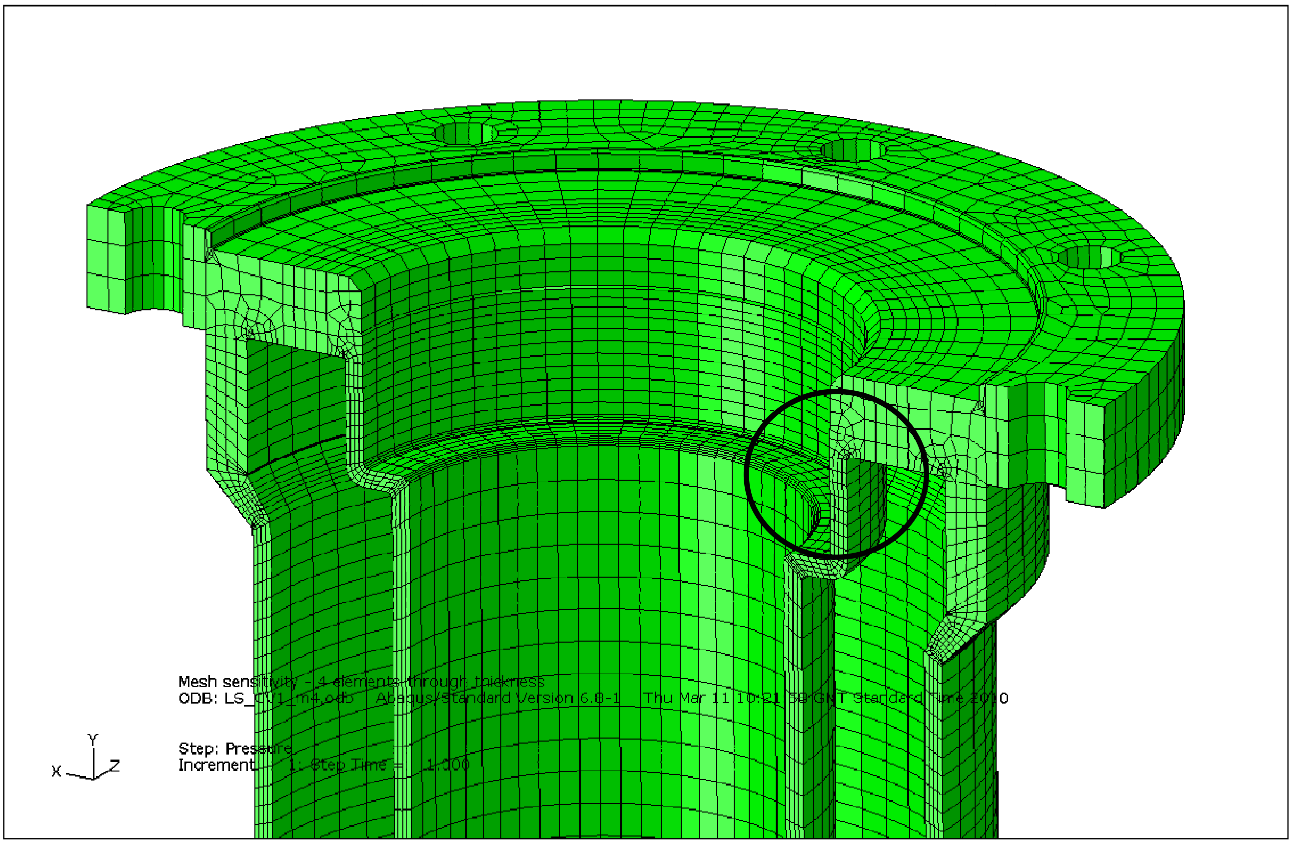

Figure 1: Finite element mesh

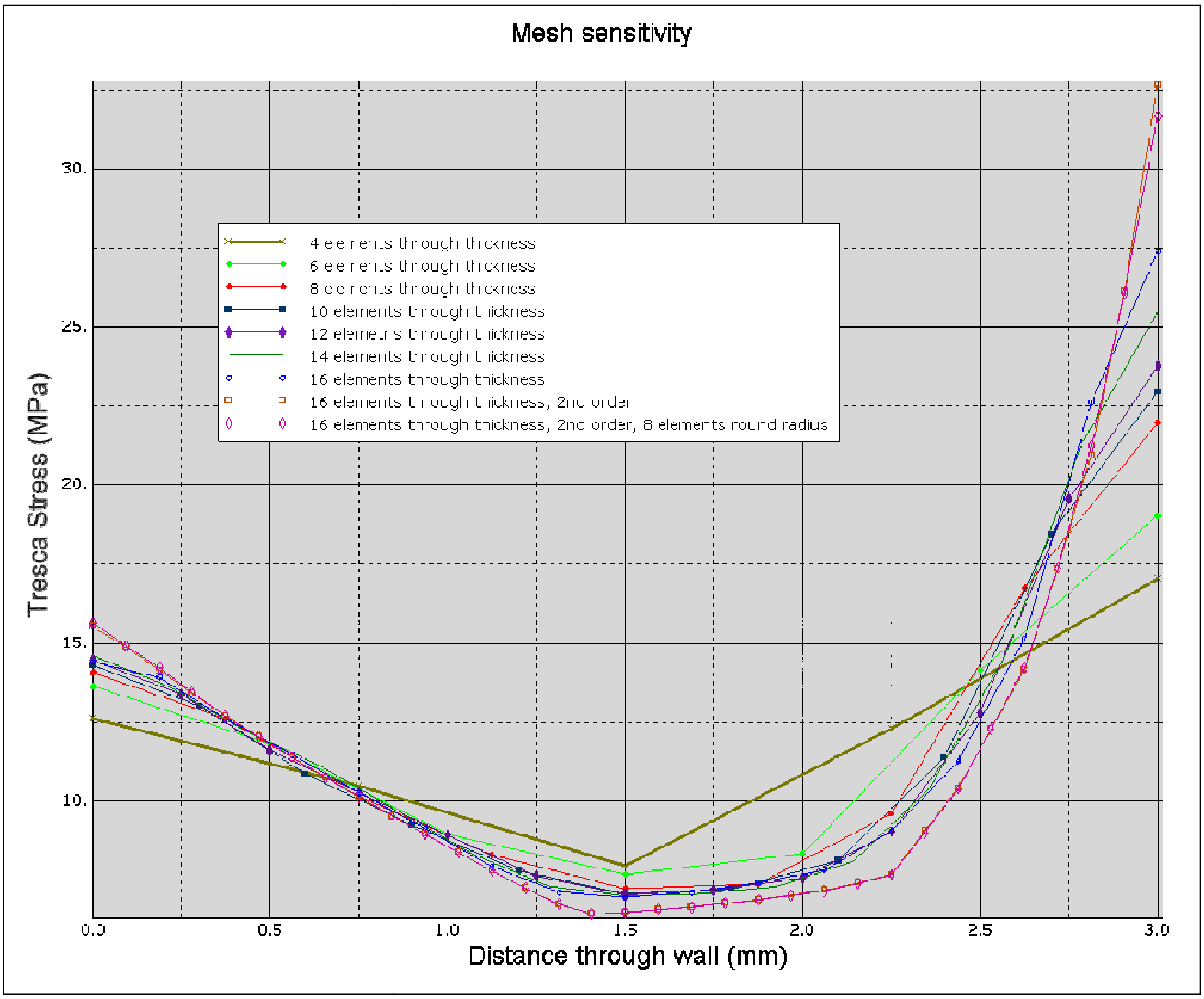

Figure 2: Stress profile through wall

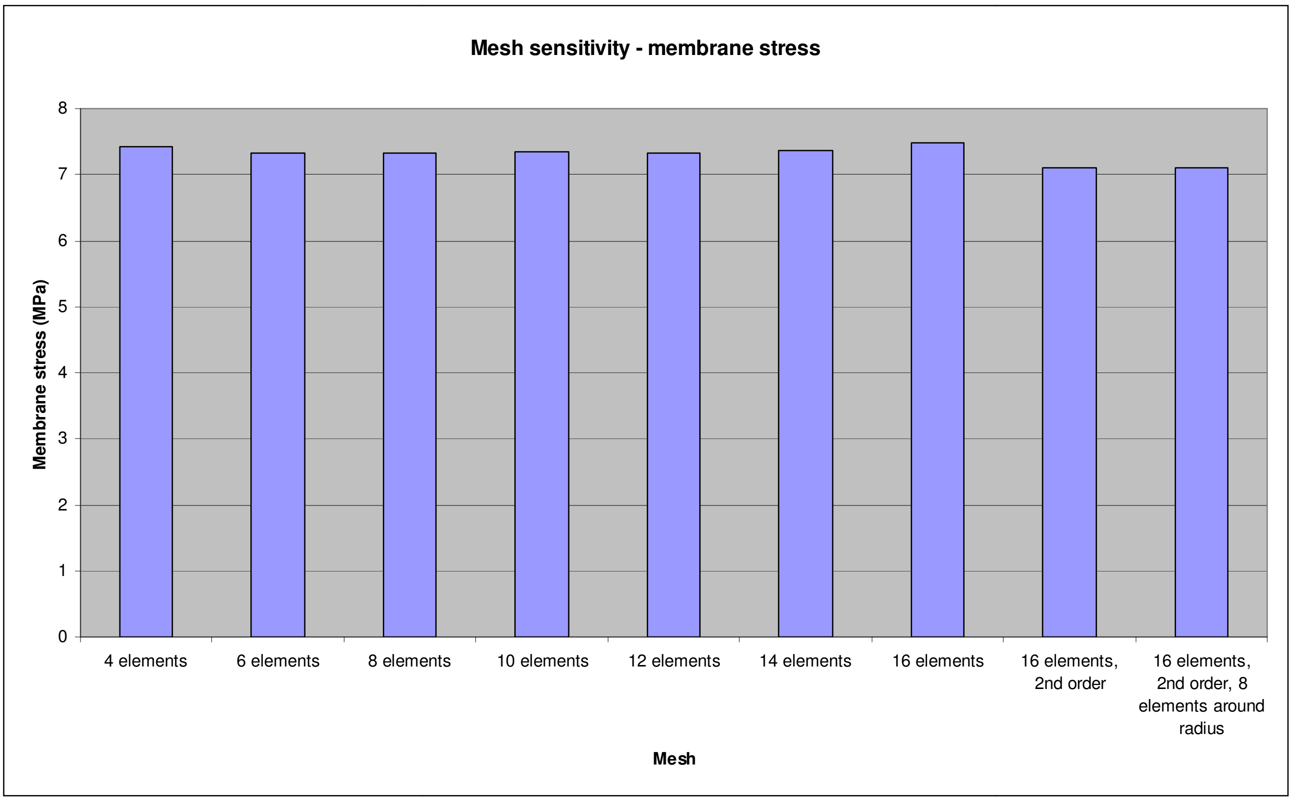

Figure 3: Membrane stress

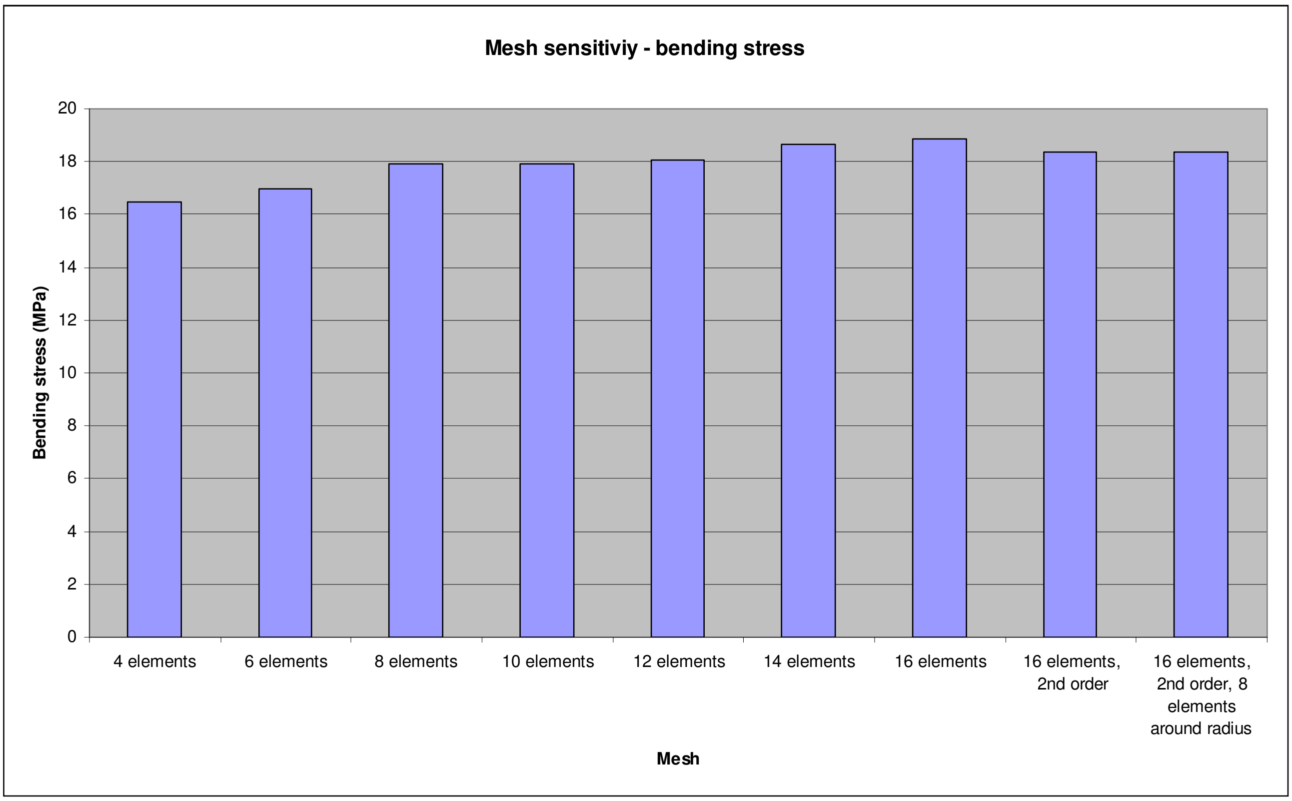

Figure 4: Bending stress

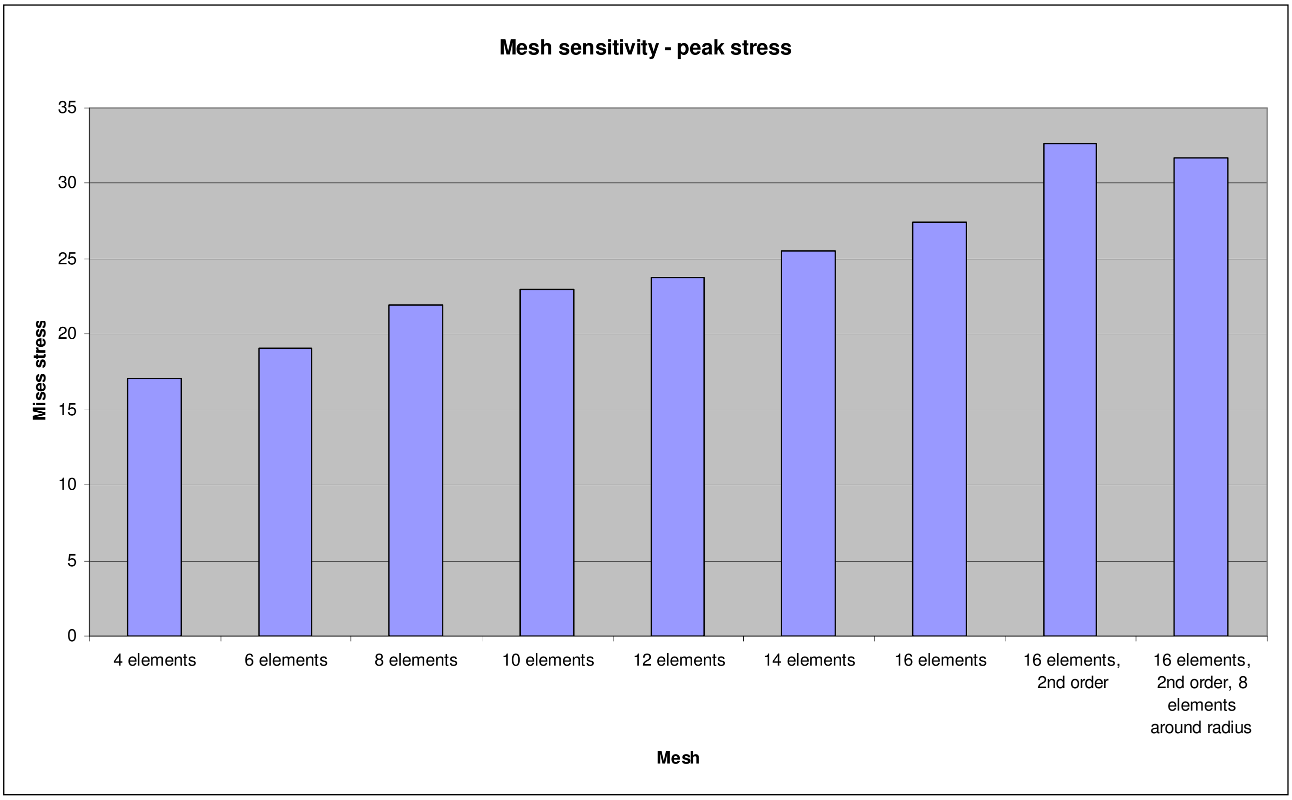

Figure 5: Outside peak stress