Elastomer Analysis, a case study in element selection and mesh discretization

Russell Moerland[*], Tod Dalrymple[**], Jaehwan Choi[**]

* Trelleborg Automotive, Grand Haven, Michigan, USA
** ABAQUS, Inc., Great Lakes Region, Northville, Michigan, USA

Abstract: While nonlinear FEA of rubber or elastomer components is quite common today, there is still some uncertainty in modelling choices and representations. Material modelling choices, curve fitting from material test data, etc., have been discussed in many papers, but some fundamental issues of element selection and mesh discretization for nearly incompressible and fully incompressible materials are less discussed. Our initial motivation was solving structural-acoustic STL problems in 3D using Tet elements. This STL problem was simplified to a stress analysis problem to investigate some anomalous behaviours. We use a thick rubber disk example problem and create a case study examining element selection in ABAQUS/Standard and mesh discretization. Both nearly incompressible and fully incompressible material behaviour is investigated and conclusions drawn about element selection and accuracy of the displacement and stress solution under various mesh discretizations. SI units are used for all analyses. Most of these analyses were run in an early pre-release, ABAQUS V6.7pr2

Keywords: STL, Elastomer, Rubber, Hyperelastic, Incompressible, Element Selection, Mesh Size

1. Introduction

While Trelleborg has been performing nonlinear stress analysis of elastomer components for many years, this recent study was motivated as we began to use ABAQUS/Standard for a new class of problem. This new class of problem was the simulation of the sound transmission loss (STL) through a rubber component. For example, the STL performance of a rubber boot that seals the steering linkage at the firewall. One of the design performance criteria for this sealing is the boot's ability to block engine compartment noise from being transmitted into the passenger compartment. This performance criterion is referred to as the boot's sound transmission loss, or "STL". Currently, Trelleborg investigates this performance through acoustic testing of each proposed design and Trelleborg wants to use the ABAQUS software to predict the STL performance using a coupled structural-acoustic simulation. With the overall goal of developing elastomer component simulation methods for the STL class of problem, we first developed several smaller test models that compared the STL simulation of a one-seal versus two-seal simple flat elastomer seals. One of our goals was to use these small STL models to successfully establish a modeling protocol and explore issues surrounding element choices and element sizing. For the general 3D component STL analyses we were concerned about meshing with Hexahedral (Hex) elements versus the use of Tetrahedral (Tet) elements. And if we were to use Tet elements, what element size would be required. These concerns arose naturally because the component geometry, like boot geometries, is complex and Tet meshing would save considerable man-time in building models. There was a general traditional belief that Tet meshes might be sufficient to capture deformations in stress analysis, but how accurate would they be for predictions of sound transmission loss in a coupled structural –acoustic analysis? Eventually, these STL models were simplified even further to remove the acoustics aspect and just look at a nonlinear stress analysis. These further simplifications were driven by some anomalous behavior(s) observed in the structural-acoustic results. Eventually we broadened the scope of these low level investigations to cover a wide range of both axisymmetric and three-dimensional element types, including linear and quadratic formulations and Hex and Tet topologies. In the following sections of this paper we will explain the initial simple one-seal and two-seal STL structural –acoustic analyses that motivated this study and progress to the ensuing simple stress analysis models in which we studied element selection and element sizing issues. We will draw some conclusions regarding element choice(s) that are pertinent for both elastomer stress analysis and elastomer STL simulations.

2. Motivation - Initial STL Test Models

We developed two relatively small structural-acoustic models in order to study the STL class of problem and guide us in establishing a modeling protocol for future 3D component STL simulations. These two models are shown in Figure 1 and Figure 2 in their 2D form. These two models were also revolved into 3D to be able to compare the use of Hex versus Tet element formulations.

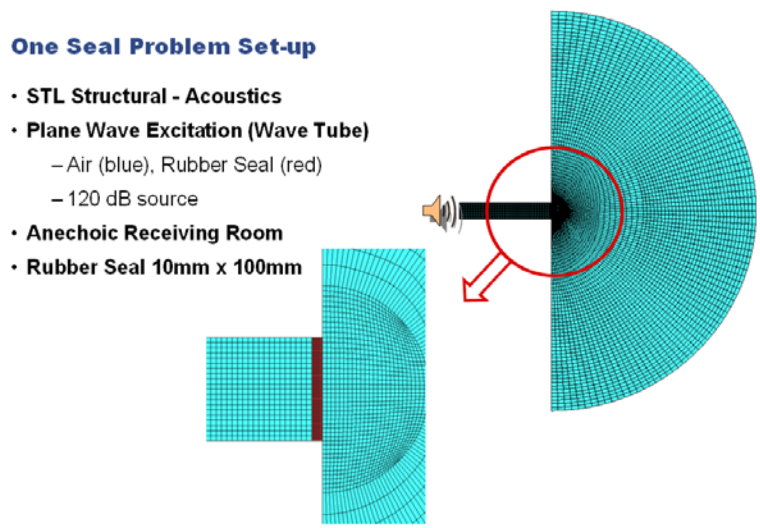

Figure 1. Simple one-seal STL analysis model.

The only difference between these two models is that the two-seal model uses two layers of elastomer material separated by a 20mm air gap. This was done because qualitatively we know that the two-seal configuration (with air gap separation) will have a significantly better noise blocking capability. This means that the two-seal arrangement will have a higher sound transmission loss. Both of these models use the same hyperelastic material model for the flat elastomer seal and typical air properties.

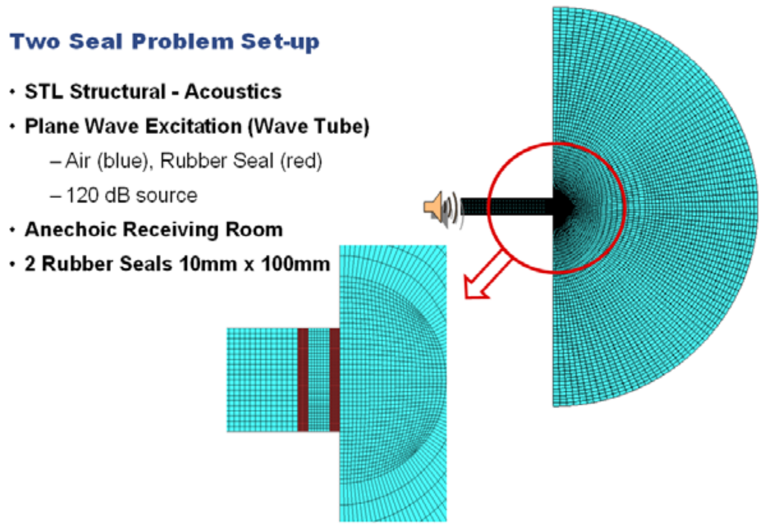

Figure 2. Simple two-seal STL analysis model.

Here are some more specific details on the model parameters:

- **120 dB magnitude excitation at left side of the wave tube**
- **Wave Tube is 540 mm in length, 100mm wide (high)**
- **Typical air element size = 5 mm x 5 mm**
- **Typical air properties**
  - Density = 1.2e-12 (N-s2)/mm4
  - Bulk Modulus = 0.142 MPa
  - Thus wavespeed, c = 344e3 mm/s
- **Anechoic Receiving Room**
  - Circular shape, typical air properties, radius=1200 mm,
  - Non-reflecting boundary at outer edge
- **Rubber Seal 10mm (thickness) x 100m (height)**
  - Hyperelastic material model, Mooney-Rivlin (MR) Material
    - C10=0.8 MPa, C01=0.2 MPa
    - Density = 1e-9 (N-s2)/mm4
- **Analysis Procedure is Steady State Dynamics, Direct**
  - Keyword line is: *Steady State Dynamics, direct, frequency scale=LINEAR
  - Frequency Range is 100 to 2000 Hz in increments of 10 Hz

These small models in their 2D form run in just a few minutes and require very little machine resources (memory, disk space). They are also simple enough to make many model variations and begin to understand issues in the STL class of problems. We will now present some typical results for this type of problem. The most important results are the STL plot across the frequency range of interest. From experiments this result is typically shown as a third-octave band plot. For these simulation results we will show the STL results across a continuous frequency, or discrete frequency (no third-octave integration). As expected the two-seal geometry results in significantly higher STL values (noise blocked better). From this plot you can also see how the STL has "drop-outs" – a low STL value means the seal is not blocking the noise, the seal is allowing a good deal of noise to be transmitted (bad). The "drop-outs" in STL happen at frequencies where the seal is in resonance. These STL plots required the use of some python scripts generated by the ABAQUS Great Lakes office to extend the functionality of ABAQUS/Viewer.

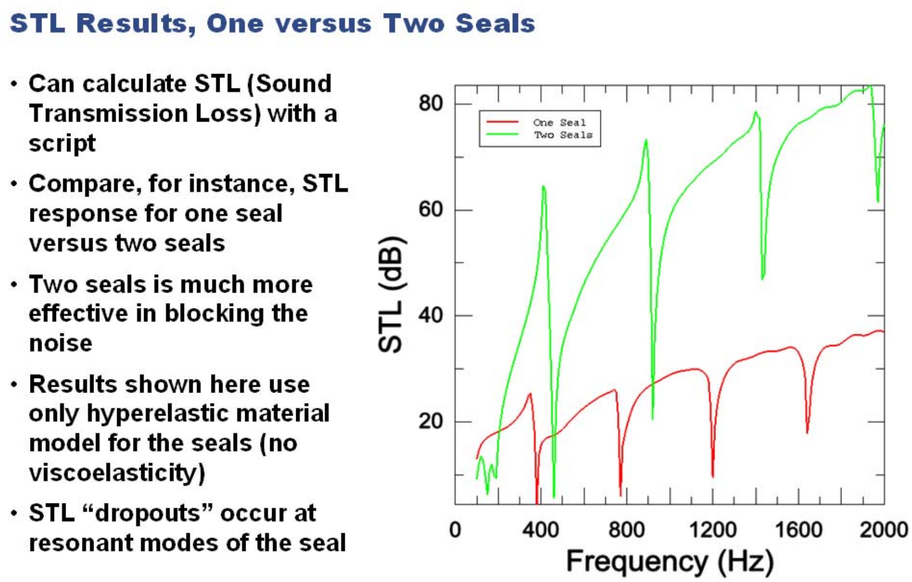

Figure 3. STL results comparison for one-seal (red) versus two-seal (green) geometry.

Another aspect of these results is that we can look at the sound pressure levels and the seal deformed shapes in ABAQUS/Viewer. Figure 4 shows a typical contour plot of SPL and seal deformation for the one-seal configuration at a frequency of 100Hz. We can skip across the span of frequencies and understand the resonant modes of the seal and perhaps gain some insight into how the design might be modified to shift resonances, improve STL, etc.

Using this simulation process to predict the STL performance, one can quickly evaluate design alternatives such as material thickness changes, air gap changes, general shape changes, etc. Since these results in Figure 3 were for a perfectly elastic (nonlinear elastic, but perfectly elastic) material we wanted to investigate the effect of viscoelasticity on the STL result. This should add physical realism since the elastomer material is clearly viscoelastic. We modified our STL models to include some material viscoelasticity using a Prony series definition. In Figure 5 we compare the purely elastic STL results to that for the hyperelastic + viscoelastic material model. Qualitatively these results are very intuitive. The damping provided by the viscoelasticity (red curve in Figure 5) damps out the resonant vibration of the elastomer and this greatly increases the STL values at and near resonances ("drop-outs" are minimized). Also notice how the resonant frequency is shifted to a slightly higher frequency which is to be expected.

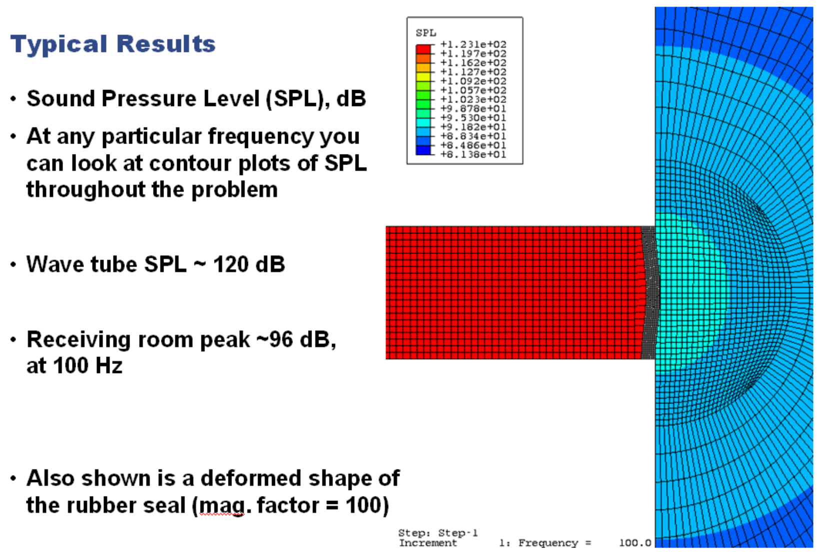

Figure 4, Typical sound pressure level and deformed shape result.

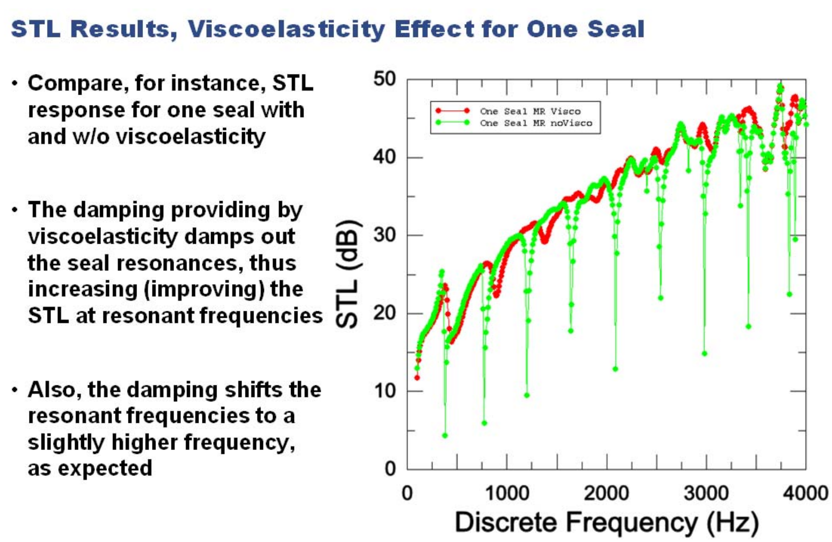

Figure 5, STL results changed by adding Viscoelasticity.

3. Simplification to Stress Analysis

While we gained a lot of insight into the STL problem using the models shown above, we discovered some anomalous behavior especially as we investigated the 3D versions of the STL models. Again, our motivation was to pave the way for using real world components, meaning complex geometries. For meshing complex geometries we would prefer to use a Tet element topology in building STL models, but we wanted to gain some confidence in their use. The Tet elements we investigated were the C3D10, C3D10H, C3D10M and C3D10MH elements. We compared these solutions to reference solutions generated with a densely meshed axisymmetric analysis and compared as well to various Hex element discretizations. We wanted to investigate the hybrid version versus the non-hybrid version elements since we were already aware of ABAQUS Answer 2056, titled *Performance of hybrid tetrahedral elements with fully incompressible materials.* The conclusion from this ABAQUS Answer is that the use of fully incompressible hyperelastic materials and hybrid Tet elements in ABAQUS Version 6.6 will lead to significantly larger wavefronts and analysis run-times 20 times longer (slower) than if a small amount of compressibility is specified for the elastomer. This results because Lagrange Multipliers are used in the fully incompressible case (D=0.0), but are not used when any small value of D (bulk compliance) is used.

We determined that the root of the anomalous behavior seen in the 3D STL models was arising from the seal behavior. Thus we set out to simplify our coupled structural-acoustic STL analysis model to an even simpler stress analysis model. Our first course of action was to simply extract the seal geometry from the one-seal model, retain the existing displacement boundary conditions around the perimeter of the flat seal and replace the acoustic load with a pressure load. The hyperelastic model (no viscoelasticity) was retained from the prior STL model. The hyperelastic material parameter "D", the bulk compliance, was varied from 0.0 (fully incompressible) to 0.001 (slightly compressible, equivalent to a realistic bulk modulus of 2000 MPa). These simplifications led to a model as shown in Figure 6.

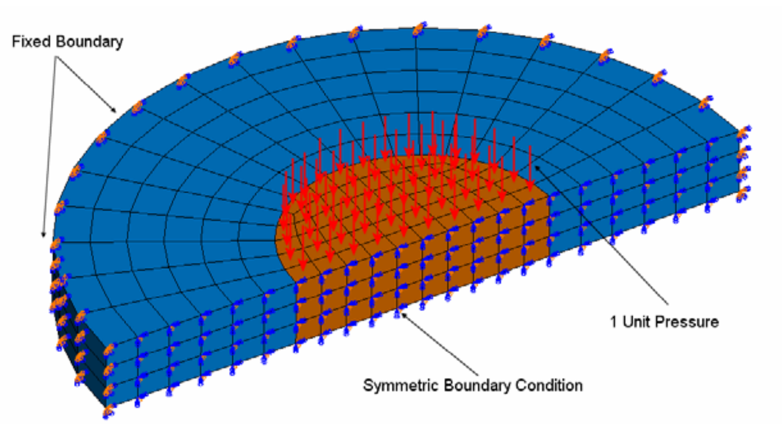

Figure 6, Simple Stress Analysis model.

As can be imagined the workhorse Hex elements such as the C3D8R, C3D8H and C3D8RH gave good results that agreed well with our reference axisymmetric solution for both maximum center displacement and S11 stress results. Also, as can be expected, since this elastomer problem has plenty of room to shear (not highly confined) the effect of D=0.0 versus D=0.001 was negligible. And using a non-hybrid Hex element such as C3D8 or C3D8R (must use D=0.001) gave consistently good results as well. We began to realize with this simple model that the C3D10H element showed anomalous behavior in both the slightly compressible and the fully incompressible case. The C3D10H element with D=0.001 (slightly compressible) material gave a reasonable center displacement, but showed anomalous stress results. The C3D10H element with D=0.0 (fully incompressible) failed to converge at only 7% of the applied pressure load. As we began to communicate these findings to the ABAQUS QA and Development organizations, there was an objection raised that this test problem showed a significant stress gradient, and perhaps singular solution, near the fixed displacement boundary condition around the perimeter. We tried several modifications to the test problem in an effort to avoid this issue.

4. A Second Simplification for Stress Analysis

After several trial and error modifications, we were able to modify the geometry of the problem to avoid the perimeter gradient issue. This resulted in the model geometry as shown below in Figure 7. All dimensions shown in Figure 7 are in millimeters. Using this modified geometry we created both axisymmetric and 3D models. The 2D and 3D boundary conditions and loading conditions are shown schematically in Figure 8 and Figure 9 respectively.

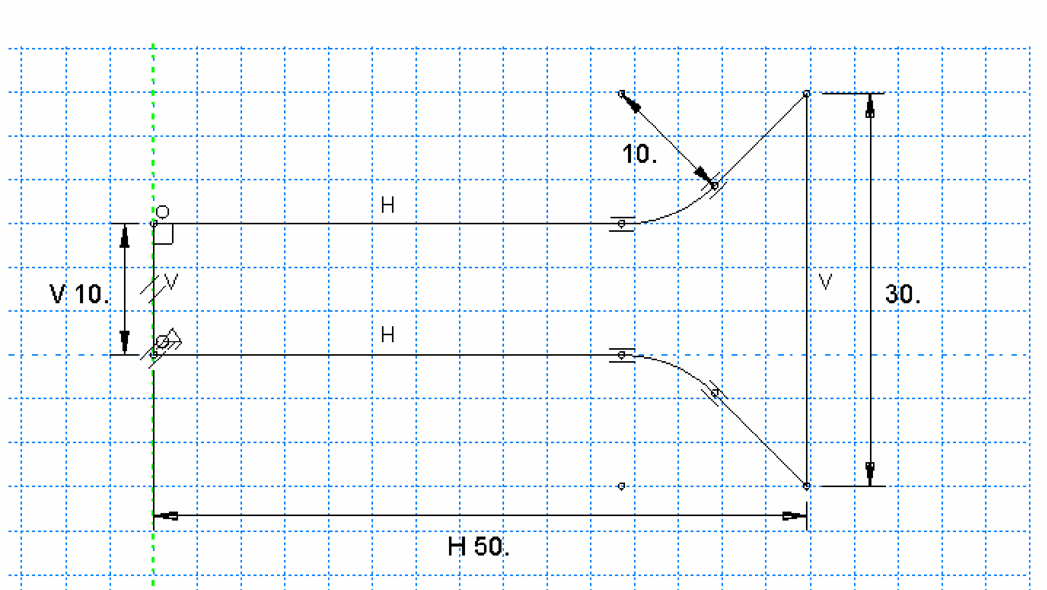

Figure 7. Simpler thick rubber disk geometry.

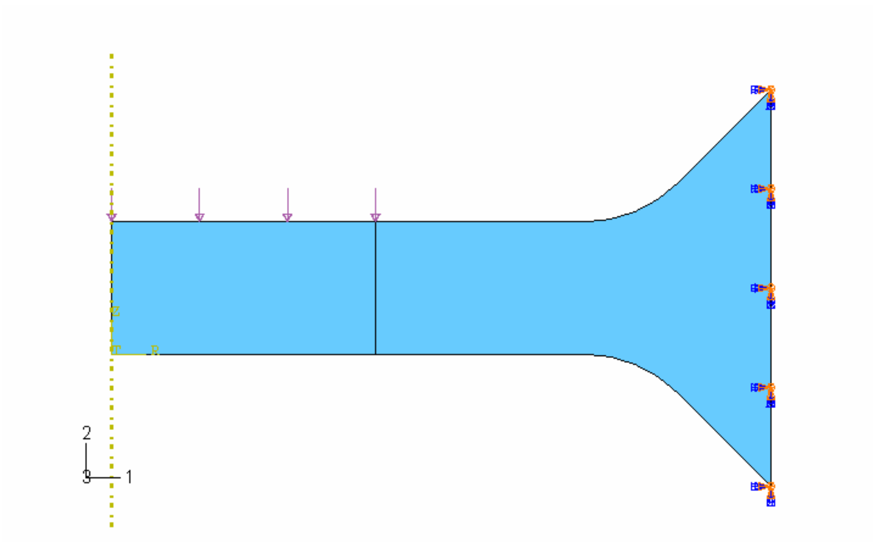

Figure 8. Axisymmetric Model Pressure Load and Fixed Boundary Condition.

.

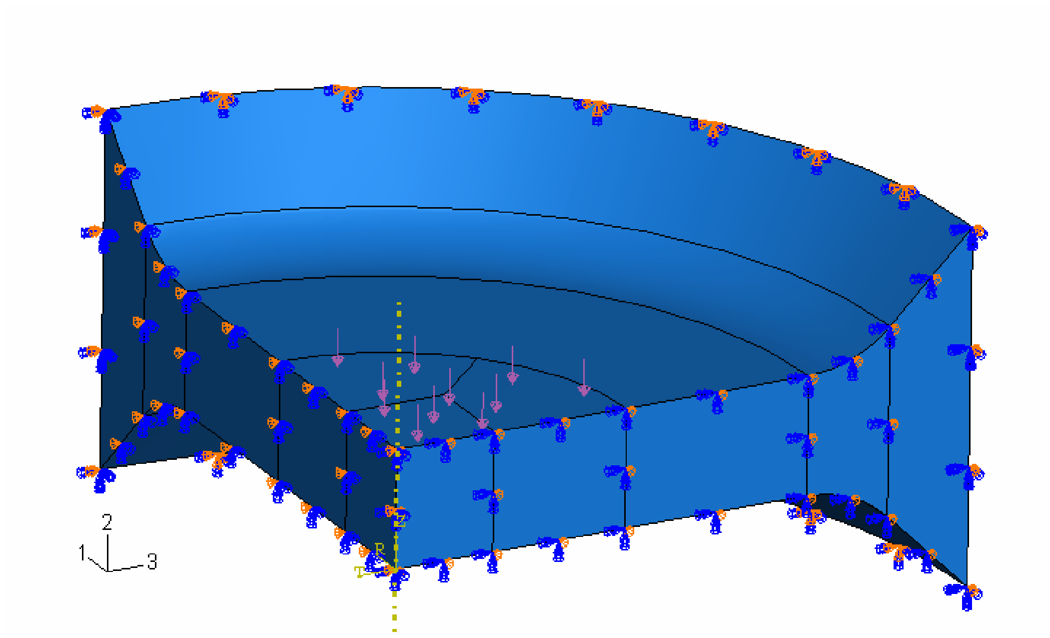

Figure 9. 3D Quarter-symmetric model with Pressure Load, symmetry BC and Fixed BC.

Our first step using this new geometry was to establish a reference solution using both CAX4 and CAX8R element types. These reference solutions will be used later for comparing to the 3D solutions. A mesh refinement study was used with element side lengths of 2mm, 1mm, 0.5mm, 0.2 mm and 0.1mm. Both of these studies used the slightly compressible material model with D=0.001. At the 0.2mm and 0.1mm element side lengths the mesh is so dense that an undeformed shape, deformed shape, or contour plot with element edges displayed is simply a black blob. All displacement results for these models are gathered at a single node at the bottom center of the rubber disk by using the ABAQUS/Viewer probe value feature. The vertical displacement result, U2, for the CAX4 element mesh study is shown below in Table 1. We will call the solution from the 0.1mm element size the reference solution for this element study. As can be seen from Table 1, the percent difference for all the solutions is less than 0.35%. So a 2mm mesh size is adequate to capture the correct displacement for any analysis work. (All percent "higher" or "lower" remarks are with respect to the magnitude).

Table 1, CAX4 element mesh refinement study, slightly compressible material

Mesh size	0.1mm	0.2mm	0.5mm	1mm	2mm
U2	-2.40035	-2.40038	-2.40063	-2.40116	-2.40883
	Ref. solution				0.35% higher

A similar mesh refinement study was performed using CAX8R elements and the slightly compressible material model. The peak displacement results for this mesh refinement study are shown below in Table 2.

Table 2, CAX8R element refinement study, slightly compressible material

Mesh size	0.1mm	0.2mm	0.5mm	1mm	2mm
U2	-2.40033	-2.40440	-2.40033	-2.40030	-2.39976
	Ref. solution				0.02% lower

The stress results from the CAX4 element study are shown below in Table 3. All stress results are taken from the legend display in ABAQUS/Viewer using default settings. We show the maximum S11 stress, minimum S11 stress and the maximum pressure (hydrostatic) stress. The results from the finest mesh size, 0.1mm, are considered the reference solution. Contour plots showing the spatial distribution of S11 stress for the CAX4 element refinement study are shown in Figure 10. The maximum (tensile) S11 stress is consistently at the bottom center of the rubber disk and the minimum (compressive) S11 is consistently at the top center of the disk.

Table 3, Stress Results from the CAX4 element refinement study

Mesh size	0.1mm	0.2mm	0.5mm	1mm	2mm
S11 Maximum	0.2692	0.2681	0.2644	0.2586	0.2445
	Ref. solution		1.8% lower	3.9% lower	9.2% lower
S11 Minimum	-0.2667	-0.2652	-0.2609	-0.2541	-0.2373
	Ref. solution		2.2% lower	4.7% lower	11% lower
Max Pressure	0.1902	0.1892	0.1860	0.1810	0.1682
	Ref. Solution				11% lower

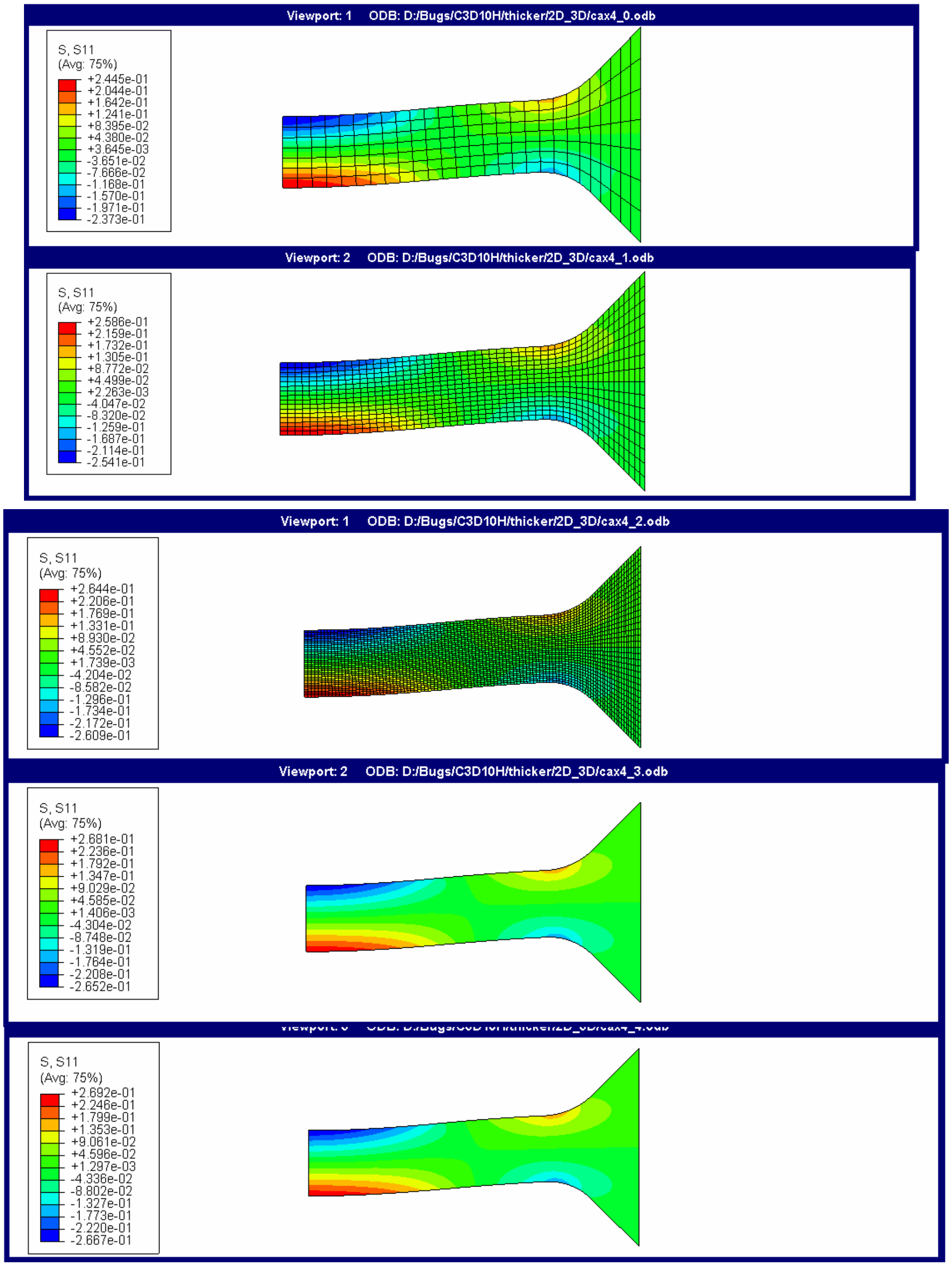

Figure 10. S11 Stress Distribution Comparison for the CAX4 study.

From the CAX4 element refinement study (with slightly compressible material) we see that the displacement field is quite accurate for all choices of element size, while the peak stress values differ by about 11% from the reference solution. This gives us a good sense of the accuracy of the linear quadrilateral element for this type of problem.

We have already shown the CAX8R displacement results and we show below in Table 4 the S11 stress and pressure stress variations. Notice that the reference solution now changes only slightly (about 0.5%) compared to the CAX4 reference solution. And the CAX8R stress variations are very small. For all practical purposes you could conclude that all of the mesh discretizations with the CAX8R element give the same S11 minimum and maximum answers. To three significant digits the U2 displacement reference solution is -2.40mm, the max. S11 stress reference solution is 0.270 MPa, the min. S11 stress reference solution is -0.268 MPa, the max. Pressure reference is 0.191 MPa and min. Pressure reference is -0.180 MPa. These reference solutions will be used later in evaluating the accuracy of the 3D solutions.

Table 4, Stress Results from the CAX8R element refinement study

Mesh size	0.1mm	0.2mm	0.5mm	1mm	2mm
S11 Maximum	0.2704	0.2705	0.2705	0.2706	0.2711
	Ref. solution				0.26% higher
Max. Pressure	0.1912				0.1913
	Ref. Solution				
S11 Minimum	-0.2681	-0.2681	-0.2681	0.2682	-0.2684
	Ref. solution				0.11% higher
Min. Pressure	-0.1803				-0.1805
	Ref. Solution				

In order to compare the slightly compressible case to the fully incompressible case, we have extended this axisymmetric study to include the use of the CAX8H element, using the D=0.0 value. The displacement results as well as the stress results are extremely close to the slightly compressible case for the CAX8R element, proving our point that this test problem is unaffected by the change from D=0.001 to D=0.0. The rubber disk has plenty of room to shear and the shear modulus dictates the solution. Notice that to three significant digits the reference solutions stated above (element size = 0.1mm) for the CAX8R / slightly compressible case are exactly the same for the CAX8H / incompressible case.

Table 5, CAX8H element refinement study, incompressible material

Mesh size	0.1mm	0.2mm	0.5mm	1mm	2mm
U2	-2.39857	-2.39857	-2.39858	-2.39857	-2.39840
S11 Maximum	0.2705	0.2705	0.2706	0.2707	0.2710
Max. Pressure	0.1913	0.1913	0.1913	0.1913	0.1913
S11 Minimum	-0.2682	-0.2682	-0.2682	-0.2683	-0.2683
Min. Pressure	-0.1803	-0.1804	-0.1804	-0.1804	-0.1805

While the main purpose of these axisymmetric analyses and element refinement studies was to establish a baseline or reference solution, we can already start drawing some conclusions from the 2D work. From the CAX4 analysis with slightly compressible material we concluded that: (i) the test problem suffers from no singularity near the perimeter, (ii) the displacement variation across the range of element size is near zero, (iii) the stress variation is about 11% and (iv) there is a consistency of stress distribution regardless of element size choice. From the study using CAX8R and D=0.001 versus the study using CAX8H and D=0.0 we can conclude: (i) for either case of D=0.001 or D=0.0 the solutions are the same for all mesh sizes, (ii) results are completely consistent with and a bit more accurate than the CAX4 case, and (iii) we have obtained a clear reference solution unchanged to three significant digits. It is worth noting at this point that each of these successful element types under integrates the pressure field. Also, it may not be obvious, but the CAX8 element should not be used for incompressible or nearly incompressible problems. The ABAQUS intro course notes (Version 6.6) in appendix A, slide A1.51 state that fully integrated quadratic elements should not be used for slightly compressible (nearly incompressible) materials. We tested the CAX8 element and the C3D20 element in this work and found very poor solution results that we attributed to volume locking.

5. 3D Stress Analysis Solutions, Hex Elements

We revolved this axisymmetric geometry into 3D to test various element types and element sizes. Table 6 below shows the stress results for the C3D8 element with the slightly compressible material. Figure 11 below shows the spatial distribution of S11 stress for the 2mm mesh seed. The 2mm mesh seed C3D8 model has 3,420 nodes, 2,635 elements and 10,260 DOF. Run-time for the 2mm problem is on the order of 40 seconds.

Table 6, C3D8 element study, slightly compressible material

Mesh size	Ref Solution		0.5mm	1mm	2mm
U2	-2.40			-2.39735	-2.40185
				0.11% low	0.08% high
S11 Maximum	0.270		0.2607	0.2513	0.2270
				7% low	16% low
Max. Pressure	0.191		0.1827	0.1744	0.1528
				9% low	20% low
S11 Minimum	-0.268		-0.2565	-0.2455	-0.2171
				8% low	19% low
Min. Pressure	-0.180		-0.1728	-0.1655	-0.1464
				8% low	19% low

We also studied the C3D8H element with both D=0.001 and D=0.0. For each of these cases the accuracy results were almost identical to the C3D8 element results shown in Table 6. This matches our experiences with the axisymmetric work with the CAX8R and CAX8H element types. For comparison, the 2mm mesh seed C3D8H model with D=0.001 has 3,420 user nodes, 2,635 elements and 12,895 variables (dof + LM) and took about 24 seconds to run. The 2mm mesh seed C3D8H model with D=0.0 has the same model size and run-time. All of the C3D8 element variations and compressible / incompressible material variations show consistent spatial variations of the stresses as well as reasonable values of these stresses for all of the mesh discretizations.

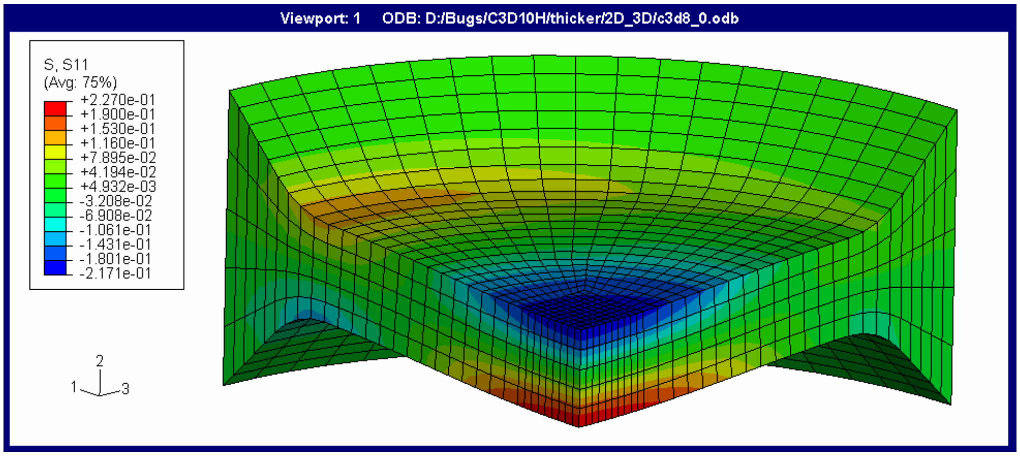

Figure 11. S11 Stress result for the C3D8 model, slightly compressible material.

6. 3D Stress Analysis Solutions, Tet Elements

As we built 3D models for investigating Tet elements, we knew that the classic linear Tri and Tet elements in ABAQUS Version 6.6 and earlier should not be used for elastomer problems. This is clearly stated in the ABAQUS Users Manual. It is worth noting that for Version 6.7 the C3D4H element has been reformulated to provide reasonable solutions for elastomer problems. Our focus in this study is on the use of the C3D10 family of elements. We focused on the C3D10H element using either the D=0.001 or D=0.0 material. The earliest meshes used tried to keep the DOF count equivalent to the C3D8 meshes. These meshes showed poor results so we re-meshed keeping the element side length equal. Keeping the edge length equal meant building C3D10H element meshes with far greater DOF's than the Hex meshes, but we did this to avoid any argument about what was causing the poor results. Table 7 below shows the results for the C3D10H element with the slightly compressible material. Figure 12 shows the spatial distribution of S11stress for the 2mm mesh seed. The 2mm mesh seed C3D10H model has 35,007 nodes, 23,508 elements and 199,053 variables (dof + LM). While the U2 displacement result seemed quite reasonable, the stress results are in error by over 100%.

Table 7, C3D10H Results

Mesh size	Ref. Solution		2mm
U2	-2.40		-2.39
S11 Maximum	0.270		0.5873
Max. Pressure	0.191		0.8495
S11 Minimum	-0.268		-0.8469
Min. Pressure	-0.180		-0.5836

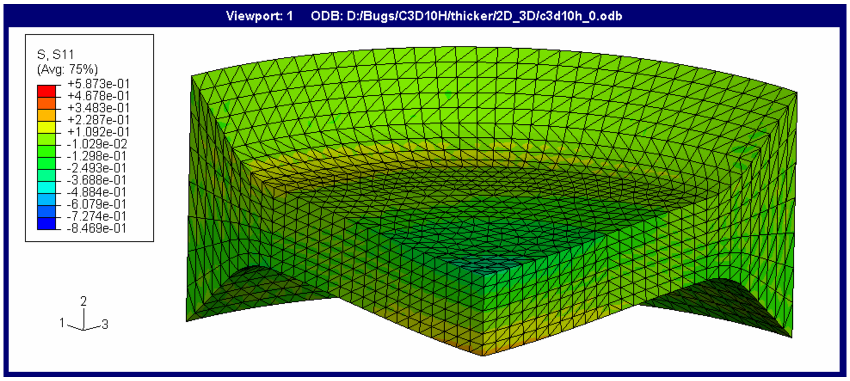

Figure 12. S11 Stress result for the C3D10H model, slightly compressible material.

Looking into this deeper one sees very wide variations in the stress results within a single element. Using the probe value feature shows, for instance, a S11 variation from -0.257 to -0.172 in element 12901. This is a very large variation that does not happen in the CAX4, CAX8H, C3D8, or C3D8H elements.

We also investigated the C3D10H element with fully incompressible material and things looked even worse. The U2 result was -2.30 (versus ref. solution of -2.40) and the default min and max stresses shown in ABAQUS/Viewer were something like -4.1E+4 to 4.3E+4! To better understand the C3D10H element with incompressible material results, we have now set the contour plot stress limits to the reference solution. For S11 this is taken as +-.27 to -0.27. The results, shown below in Figure 13, are surprising to say the least.

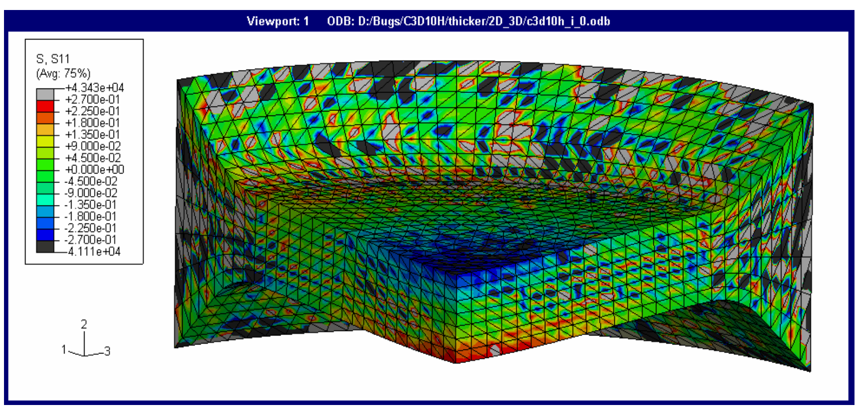

Figure 13. S11 Stress result for the C3D10H model, incompressible material.

We suspected this phenomenon was due to element locking. Working with the ABAQUS QA and Development organizations this was identified as a volume locking issue. The V6.6 and earlier C3D10H formulation was deficient in the under integration of the pressure stress field. A bug report was filed and this element formulation was corrected for the Version 6.7 release. We have run some preliminary C3D10H models in Version 6.7pr4 which show this bad behavior to be corrected.

7. Conclusions and Observations

Our element selection and element size studies using the simple thick rubber disk have set some expectations for results accuracy. In keeping with traditional beliefs, the quadrilateral and hexahedral elements are to be preferred both for their accuracy and keeping the problem size minimized. However, for complex 3D geometries the man-time savings in the use of Tet elements may be the overriding concern. A significant result of this work is that a volume locking deficiency in the C3D10H element was identified and this element behavior was corrected in ABAQUS Version 6.7. Here are some specific conclusions and guidelines in element selection:

1. Fully integrated quadratic elements should not be used for slightly compressible (nearly incompressible) materials, this is a known issue and documented (see intro course note appendix A, slide A1.51). This includes CAX8 and C3D20 elements
2. CAX4 element models used in mesh refinement study prove:
    a. No singularity exists
    b. Provides reference result (from finest mesh)
    c. Consistency of S11 stress distribution with mesh refinement
3. CAX8R versus CAX8H element study proves:
    a. For either slightly compressible or incompressible material the solutions are the same for all mesh densities.
    b. For slightly compressible case:
        i. Same Reference solution as for CAX4 elements
    c. For slightly compressible versus incompressible case:
        i. No difference in solution
4. C3D8 element models show a 3D solution that matches the CAX4 results.
5. For C3D8H elements, the slightly incompressible material shows the same results as the incompressible material. This verifies conclusion 3c above.
6. The C3D10M and C3D10MH elements gave reasonable results, but were more expensive than the C3D10 and C3D10H elements.
7. For C3D10H element models, the slightly incompressible material shows errors in stress plots. Relatively large stress variations occur within a single element.
8. For C3D10H with incompressible material, the stress results are very bad and they appear to exhibit a locking phenomenon.

Items 6 & 7 were behaviors seen in V6.6 of ABAQUS and corrected in Version 6.7

Afterwards

Version 6.7 of ABAQUS also has reformulated the C3D4H element and we were able to perform some very preliminary work showing good results. It would be desirable to revisit this work and mesh the Tet models more closely matching the DOF count of the Hex models. Also, we would like to return to the STL subject matter and prove the usefulness of the Version 6.7 C3D10H element and the re-formulated C3D4H element for meshing complex 3D geometries.