Evaluation of Advanced Element Formulations for Failure Prediction of Highly Complex 3D-Printed Parts

Subhransu Mohapatra

SABIC Research & Technology Pvt Ltd
Plot# 81 to 85, Chikkadunasandra, Sarjapura, Bengaluru, India-562125
Subhransu.mohapatra@sabic.com

ABSTRACT-
Additive manufacturing has moved beyond rapid prototyping and is now starting to be used for manufacturing of functional parts such as in aircraft interior ducting systems and healthcare applications. This has resulted in a paradigm shift in design requirements for 3D printed parts from simply aiding in rapid physical visualization of designs to sustaining operating loads. Furthermore, along with the functional load bearing requirements, 3D printed designs should demonstrate benefits of lower weight and cost which are possible through innovative shapes often arrived at through detailed topology optimization. However, these innovative shapes often offer significant challenges in predictive performance evaluation due to their geometric complexity. This is amplified for structures where failure prediction under operating environment is a part of the design process such as in automotive applications.

Typically, explicit solvers such as LS-DYNA® are used to predict highly non-linear events such as component failures. However, the highly complex shapes arrived through topology optimization and manufactured through 3D-printing process are extremely cumbersome to model with traditional elements used in explicit solvers which are hexahedral in shape. On the other hand, tetrahedron elements can model such shapes with considerable ease, but are avoided in classical explicit analysis methodology due to loss of simulation accuracy. Livermore Software Technology Corporation (LSTC) has recently introduced higher-order tetrahedron elements in LS-DYNA® which has the potential to reduce the loss of accuracy while holding onto the ease of modeling. But this benefit comes with a significant increase in simulation time due to the higher order integration approach. This paper deals with evaluation of the suitability of higher order tetrahedron elements to model highly complex shapes typical of additive manufacturing process for failure prediction of a highly complex 3D-printed specimen part representing a commonly used structural element. The predictions are compared with actual physical test results which point to satisfactory performance of the element formulation for such use. Additionally, recommendations are made for further study to improve the simulation fidelity.
***Keywords*— 3D Printing, Additive Manufacturing, Numerical Prediction, Stiffness, Strength, Topology Optimization, Geometry Complexity, LS-DYNA®, element formulations**

1. Introduction

Additive manufacturing has moved beyond rapid prototyping and is now starting to be used for manufacturing of functional parts such as in aircraft interior ducting systems and healthcare applications. This has resulted in a paradigm shift in design requirements for 3D printed parts from simply aiding in rapid physical visualization of designs to sustaining operating loads. Furthermore, along with the functional load bearing requirements, 3D printed designs should demonstrate benefits of lower weight and cost which are possible through innovative shapes often arrived at through detailed topology optimization. However, these innovative shapes often offer significant challenges in predictive performance evaluation due to their geometric complexity. This is amplified for structures where failure prediction under operating environment is a part of the design process such as in automotive applications.

Typically, explicit solvers such as LS-DYNA® are used to predict highly non-linear events such as component failures. However, the highly complex shapes arrived through topology optimization and manufactured through 3D-printing process are extremely cumbersome to model with traditional elements used in explicit solvers which are hexahedral in shape. On the other hand, tetrahedron elements can model such shapes with considerable ease with automatic algorithms available in commercial discretization software such as HyperMesh®, but are avoided in classical explicit analysis methodology due to loss of simulation accuracy.

LS-DYNA® has been introducing various tetrahedral element formulations since 1991-92. However, these formulations are yet to gain acceptance in the industry. Numerous papers have studied the advantages and limitations of tetrahedral element formulations [1], [2], [3]. These papers have compared tetrahedral formulations against standard hexahedral elements for simple shapes where analytical solutions are available. The papers conclude that the lower order tetrahedral elements are not suitable due to loss of accuracy, whereas higher order tetrahedral elements are accurate, but ~ 6 times costlier than traditional hexahedral elements.

There is currently little literature in public domain which evaluates the different tetrahedral elements for complex shapes for which closed form solutions are not possible and validation can be done only through physical testing. Topology optimized shapes preferred to be manufactured through additive manufacturing are highly complex and only feasible element types that can be used for performance evaluation are tetrahedral elements. This article deals with evaluating various tetrahedral element formulations for failure prediction of highly complex topology optimized parts typically manufactured through additive manufacturing.

2. Topology Optimization to Leverage Benefits of 3d-printing

As mentioned in previous section, highly complex, but optimized designs are preferred for 3D printed designs. This optimizes the material distribution in exact areas where it is needed. Topology optimization is a numerical technique which is used to arrive at an optimal distribution of materials for improving performance objectives such as maximizing stiffness while minimizing mass.

In this paper, a standard 3-point bend specimen used for material characterization has been taken through the topology optimization process without any manufacturing constraints to find out the most optimal distribution of materials for given load conditions. The specimen geometry is as per the ASTM D790 M specification (ASTM D790, 1993) [4].

Production-grade 3D-printable thermoplastic ABS-M30 is chosen as the material for 3D printing. For any 3D printable material, mechanical properties are sensitive to the direction of printing. Between horizontal (along the direction of printing) and vertical (perpendicular to the direction of printing) moduli, horizontal modulus is used for optimization and subsequent simulations since the three point bend specimen is printed in this direction. Only Young's Modulus and yield stress are required at this stage. Poisson's ratio is assumed to be 0.4.

Before proceeding with the optimization, baseline performance is evaluated. A point load of 430 N is applied at the centre of the 3-point bend specimen. Maximum vertical displacement and maximum stress for the rectangular specimen under this bending load is used for benchmarking of the optimized solution.

During the topology optimization, the objective was to minimize mass with the stress constrained to be lower than 25 MPa in the design space based on the yield stress of 31 MPa for ABS-M30™. This ensures that the optimization results in the lowest weight part which can take given load without failure.

The optimized topology with threshold element density of 0.3 is shown in Figure 1. Element density is the percentage volume of the element which is active, e.g., 0.0 density means completely absent, 0.3 implies 30% present and 1.0 implies completely present.

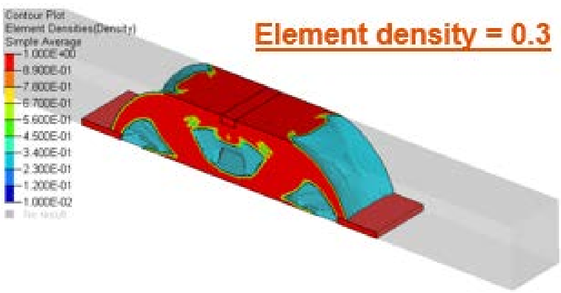

Figure 1 Optimized topology with threshold element density = 0.3

The internal topology (visualized by making the exterior transparent) for the geometry is shown in Figure 2. As can be seen, the geometry is quite complicated.

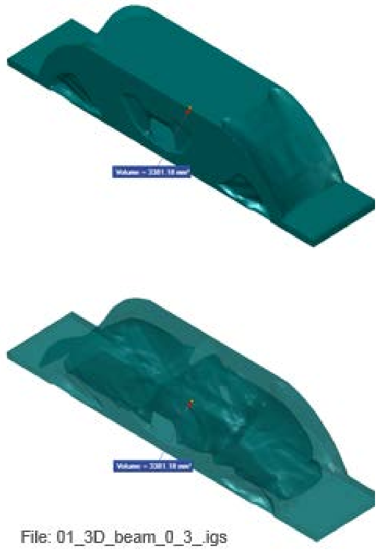

Figure 2 Internal geometrical complexities after topology optimization

Deflection and stress comparisons between baseline and optimized geometries are compared to ensure that while deflection is higher by 2.08 times, the maximum stress is within material yield limit. This ensures that we take full advantage of the material distribution while being within the safe design zone.

The weight savings from the topology optimization can be quantified in two ways. In the first scheme, the volume of the optimized geometry can be compared against the baseline geometry to indicate weight saving. In other scheme, optimized volume can be compared with the volume of the original beam after removing the overhanging ends. The second scheme gives a more accurate estimate of weight saving since it is trivial to deduce that the overhanging portions are not needed for the current loading condition. These three configurations are illustrated in Figure 3. The weight savings for the two schemes are 83.5% and 68.3% respectively.

Figure 3 Illustration of lightweight configuration with optimized topology

The topology optimized shapes usually have rough surfaces to show the portion of the element volume which is needed as part of the optimized load path. This needs to be smoothed before printing the part. Additionally some minor geometry clean-up is usually required to generate a printable volume.

3. Failure Prediction using LS-DYNA® Numerical Simulations

1. Model Set-up

In any numerical simulation, it can be expected that high local stresses would develop at the support points and point of load application. This can cause excessive deformation of the local elements around supports and loading points, which results in non-physical behaviour. These purely numerical manifestations can be eliminated in two ways.

The first method is to model load application and support rollers with actual geometry as well as using a much finer mesh discretization. The second approach is to model small local areas near the loading with elastic material model without failure to prevent unrealistic high local deformation.

The latter approach is used in the current study for the reasons of simplicity with very low levels of expected loss of accuracy. The model is shown in Figure 4. The local yellow areas have been excluded from plasticity and failure modelling.

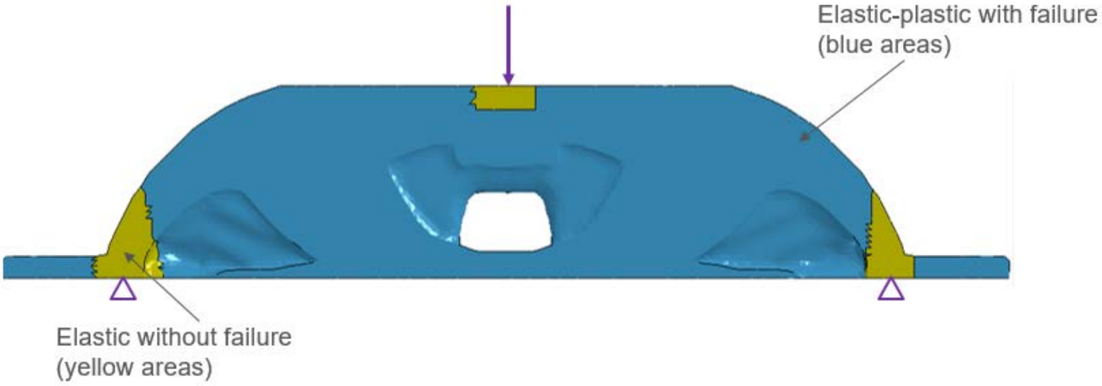

Figure 4 Model Set-up

Also, the actual 3D-printed part is made up of individual interface layers that result from the printing process and is really not a homogeneous solid form. However, the current model doesn't take that level of detail into account and models the beam as a homogeneous solid body. Material properties were obtained from the material datasheet supplied by the material supplier [5]. The summary of the relevant material properties is shown in Table 1. These material properties are measured as bulk material properties and hence the material properties can be used in the type of model used.

Mechanical Property	Test Method	Value	
		Along printing direction	Lateral to printing direction
Young's Modulus	ASTM D638	2230MPa	2180MPa
Yield Stress	ASTM D638	31 MPa	26MPa
Ultimate Stress	ASTM D638	32 MPa	28MPa
Tensile elongation at yield	ASTM D638	2%	1%
Tensile elongation at break	ASTM D638	7%	2%

Table 1 Mechanical properties of the 3D-printing material

The complete stress-strain curve is not provided in the datasheet. However, noting that the yield and ultimate stresses are very close, a linear assumption between the yield point and ultimate point has been assumed. Between along the direction of printing and perpendicular to the direction of printing properties, along the direction of printing values are used since the three point bend specimen is printed in this direction.

2. *Selection of Element Types*

Element type selection is one of the most important steps in a finite element analysis. Observing the complex and solid type of geometry from Figure 6, it can be easily derived that mid-surface can neither be extracted nor will be representative of the geometry. Hence, neither the thin or thick shell element formulations were selected for simulation.

From among the solid element formulations, hexahedral and pentahedral formulations are more accurate and used as standard element types in explicit simulations. However, they need to be manually created by splitting the complete geometry into manageable domains. With the highly complex shapes of topology optimized geometries, this is ruled out since the time taken will be prohibitively long.

The only feasible option to discretise the geometry is to use tetrahedral elements which can capture any arbitrary shape. However, traditional tetrahedral elements aren't accurate enough and hence not usually used. However, recently LSTC (owner of LS-DYNA® tool) has come up with new formulations of tetrahedral elements which are claimed to be accurate for explicit simulations as well for medium level of deformations [2]. Hence it was decided to try out the traditional as well as newer tetrahedral element formulations to check the robustness, accuracy and speed of different tetrahedral formulations and decide on the most suitable option for finite element analysis.

3. *Comparison of Various Tetrahedral Element Formulations in LS-DYNA®*

There are basically 5 different types of tetrahedral element formulations available in LS-DYNA. The features of these formulations are compared in Table 2 below.

Tetrahedral element formulation	Integration Type	No of Nodes	Mid-side Nodes included or not	Additional Remarks
4	Selectively reduced integration	4	No	Rotation degrees of freedom (d.o.f) included
10	Reduced integration	4	No	-
13	Reduced integration	4	No	Average Nodal pressure scheme used
16	Selectively reduced integration	10	Yes	Nodal weighting factors are unequal
17	Selectively reduced integration	10	Yes	Nodal weighting factors are equal

Table 2 Comparison of various tetrahedral element formulations in LS-DYNA®

4. *Evaluations of Various Tetrahedral Element Formulations*

The simulation was carried out with all the above tetrahedral element formulations to compare their performance in terms of robustness, accuracy and speed. Robustness was measured in terms of ability to complete the run with normal termination without numerical issues. Run was performed until 1.5ms by which time the beam reaches its highest load level (strength). All the runs were made using 2 CPUs with SMP 971 version revision R7.1.1 single precision solver. The first level of comparison for robustness and speed is shown in Table 3.

Element Formulation	Normal Termination	Simulation Time (hrs)
4	Yes	8
10	Yes	1.5
13	Yes	2.5
16	Yes	17
17	Yes	28

Table 3 Comparison of robustness and speed between various tetrahedral element formulations

Based on the above comparison, it is clear that all the element formulations are quite robust for the given simulation. From simulation speed, it is very evident that element formulation 10 is the fastest. This is as per expectation since it is a classical reduced integration element. The more involved reduced integration elements are 2 to 5 times more costly. The newer second order element formulations are even more costly (10-20 times).

Checking the failure pattern, all the runs give very similar failure patterns in the middle whereas the higher order elements also point to an additional failure point near one support end (Figure 5). This may be due to better capture of stress concentration near to the supports for the higher order elements and hence point to the need for more elements being added to the elastic-no failure zone for those cases.

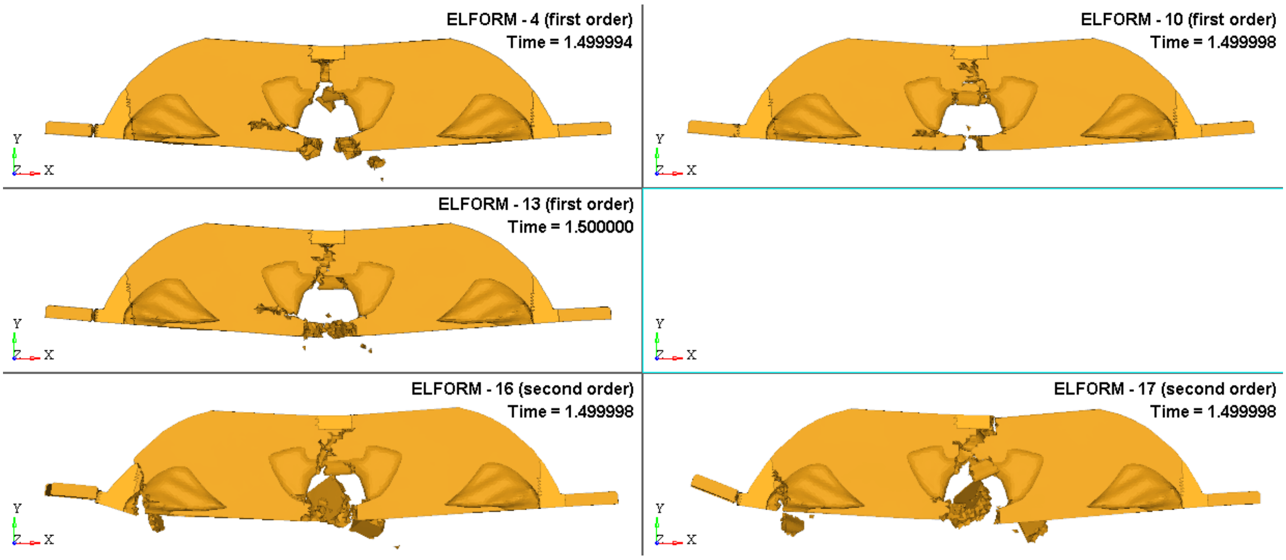

Figure 5 Comparison of failure patterns for different element formulations

Comparison of force vs. deflection curves can be seen in Figure 6. As expected, the responses are far smoother for second order elements compared to first order elements. It can be observed that in spite of large differences in simulation speed, the quantitative results are very similar except that the higher order elements fail a little bit earlier in displacement due to the additional failure near the supports. Other than that, the responses are almost identical.

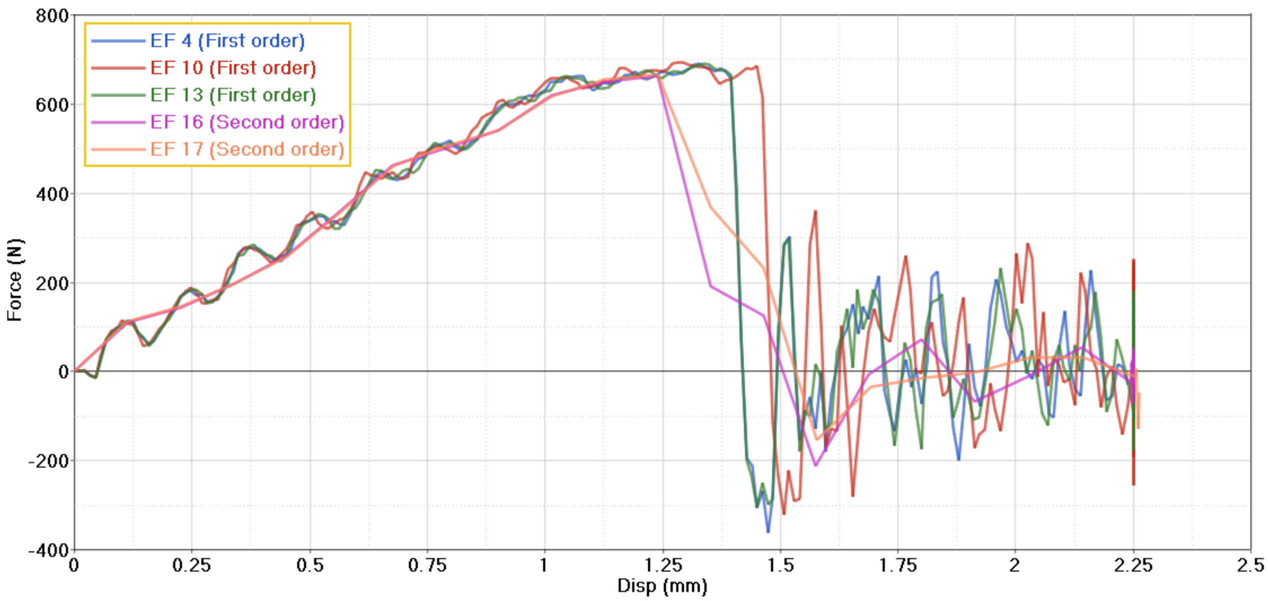

Figure 6 Comparison of force vs. deflection curves for different element formulations

5. Evaluations of Mesh Density

It's important to make sure that mesh densities don't play a significant role in simulation results. Hence, a mesh convergence study was carried out. Simplest element type, type 10, was selected for the convergence study. Simulations were carried out with three different mesh densities with average element lengths at 0.2 mm, 0.4 mm and 0.8 mm. The failure patterns and force vs. deflection curves are compared in Figure 7 and Figure 8. From the figures, it's quite clear that overall results are similar for all mesh densities except for initiation of crack near to one support as mesh density increases which also manifests itself in the force dropping off at an earlier time. This is similar to the effect we have seen for first order elements with more degrees of freedom (e.g., element formulation 4) and second order elements. This is as per expectation since either by increasing the mesh density or by using higher order elements, we are increasing the degree of freedom (d.o.f.) count which should result in the same behaviour.

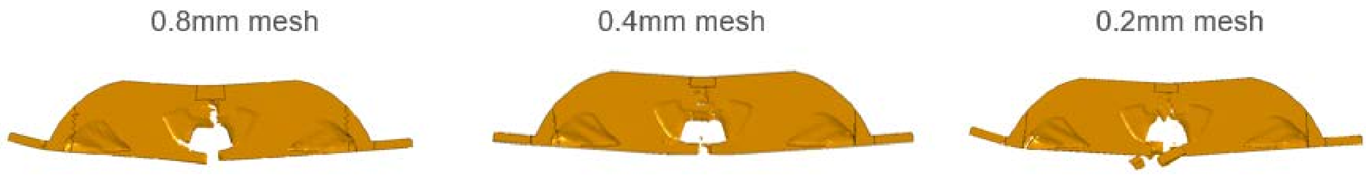

Figure 7 Comparison of failure patterns for various mesh densities

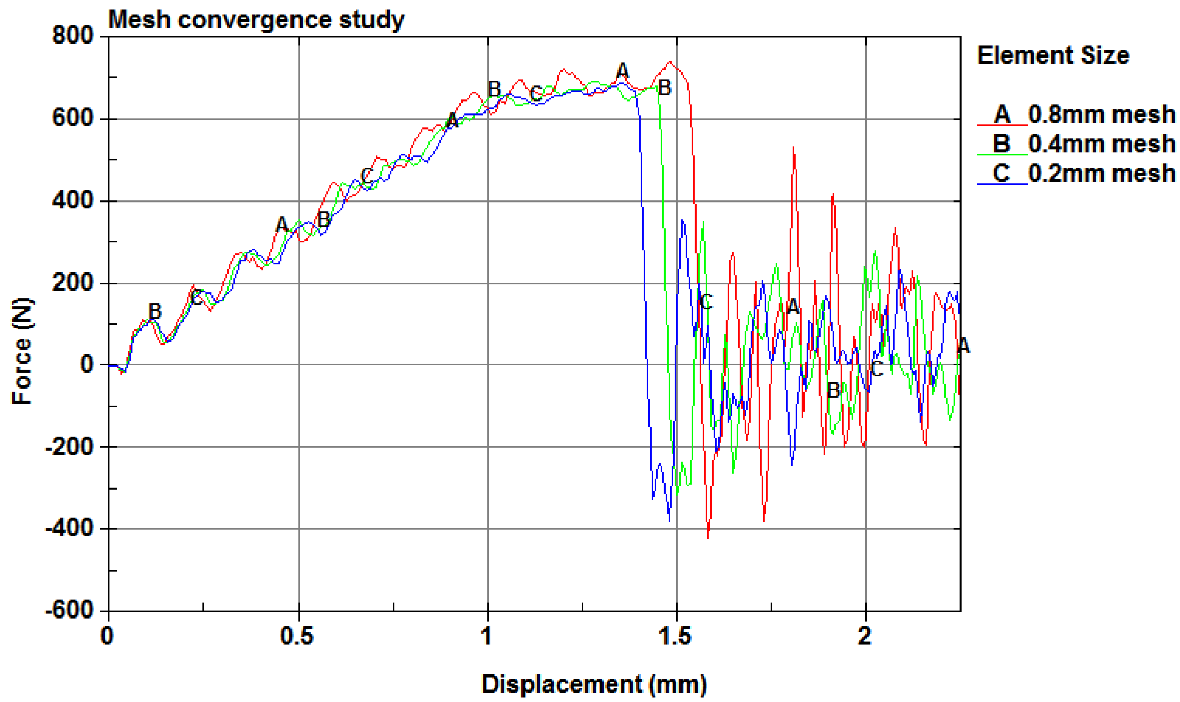

Figure 8 Comparison of force-deflection curves for various mesh densities

4. 3D Printing of Optimized Geometry and Physical Tests

The topology optimized shape which was simulated above was printed with ABS-M30 material. This material allows use of a dissolvable support material which helps in printing the complex geometry coming out of the topology optimization process.

The experimental flexural load-displacement curve obtained with the 3-D printed coupon of ABS-M30 material conforming to the optimized topology described in the foregoing section is shown in Figure 9. The failed specimen is shown in Figure 10. As can be seen from Figu**re 6**, 3-point flexure tests are repeatable for the 3-D printed coupon.

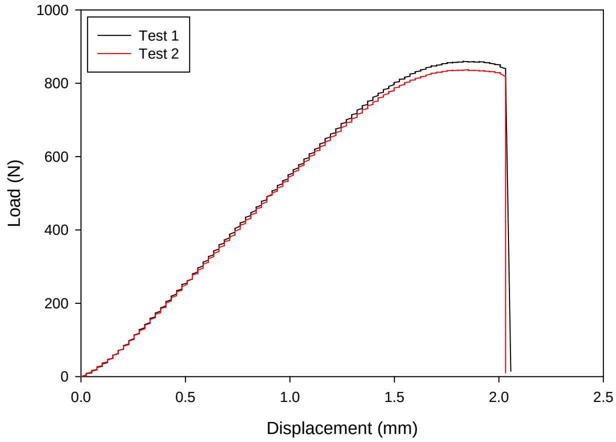

Figure 9 Force vs. deflection curves form physical tests for topology optimized geometry

Figure 10 Broken specimens after test

5. Comparison of Simulation Results with Test Data

The comparison of simulation results with physical test data are shown in Figure 11. As can be seen from the figures, LS-DYNA® predicts stiffness very well whereas the prediction of failure strength is somewhat further from test results. The load plateauing effect similar to test is more visible for lower order elements compared to higher order elements due to the support condition effect as described earlier. Component failure is predicted at a lower load level (850N in test vs. 700N in simulation) and at a significantly earlier displacement. This results in a conservative estimate of the strength which may be an adequate estimate for design purposes currently.

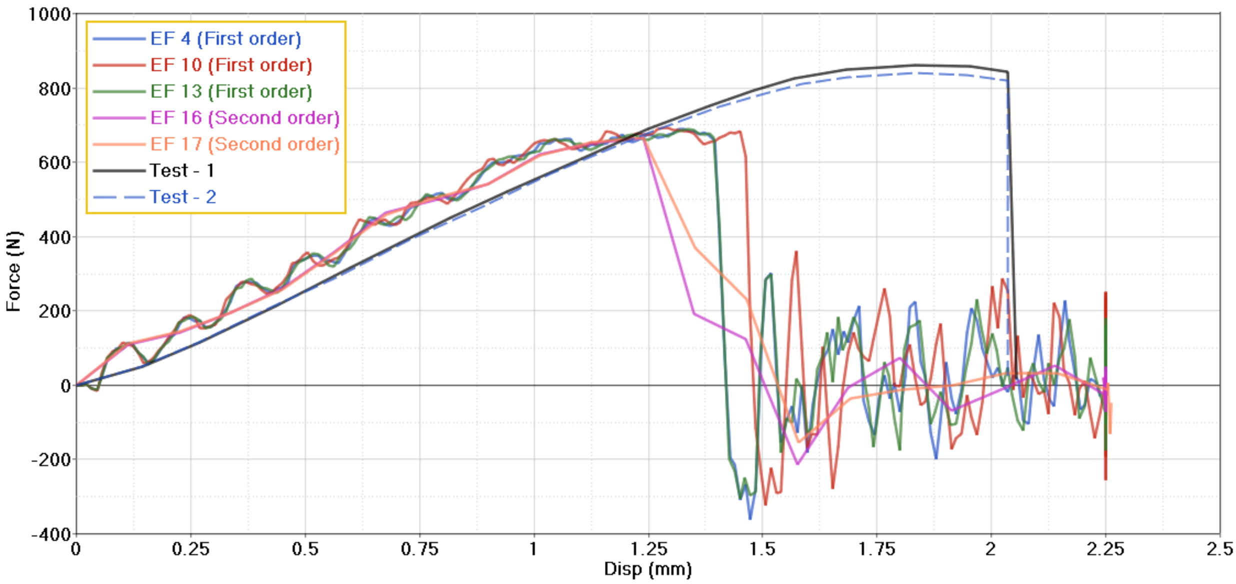

Figure 11 Comparison of Simulation Results with Test Data

However, the prediction of lower levels of load carrying capacity in the simulation needs to be investigated to further improve simulation accuracy. One of the most important areas to investigate is the datasheet mechanical values. Actual mechanical properties may significantly vary from the printed data sheet values for 3D printed material. This is due to a significant number of printing parameters other than print direction such as nozzle size, printing speed, temperature etc. which can potentially affect the mechanical properties. This would need a further focused effort which is out-of-scope for this work.

6. Comparisons of Various Element Formulations from Robustness, Accuracy and Speed Aspects

With the above study, the following conclusions about accuracy, robustness and speed for different element formulations can be derived.

1. All tetrahedral element formulations available in LS-DYNA® are equally robust to solve slow loading scenarios such as 3-point bending
2. All formulations predict similar stiffness and failure strength even though the failure happens a little earlier for second order elements due to boundary condition effects.
3. Element formulation 10 is best in terms of speed. The more involved reduced integration elements are 2 to 5 times costlier. The newer second order element formulations are even more costly (10-20 times).

7. Summary

Additive manufacturing is gaining rapid ground in the world of manufacturing. The full benefits of additive manufacturing can be obtained when complex, but fully optimized shapes until now impossible to produce through other manufacturing methods, are brought into design. The performance prediction of these complex shapes, particularly failure prediction, is always a challenge. Tetrahedral element formulations are suitable to discretise these complex shapes, but they are not standard element types used in explicit simulations for failure prediction. However, the current study shows various tetrahedral element formulations in LS-DYNA® are very robust and reasonably accurate for predicting part performance. The stiffness is very accurately predicted while the prediction of strength is conservative. Among the various tetrahedral element formulations, formulation 10 shows the best performance in terms of speed while being robust and reasonably accurate at the same time.

8. Acknowledgment

I would like to acknowledge the support extended by my colleague Dr. Prasad Dasappa in performing the topology optimization and Mr. Matt Burrato for help in printing the topology optimized geometry.

9. Literature

[1] Elsäßer K., Keding B., Müllerschön H. Pedrazzi C., Comparison of different element types in structural analysis, www.dynalook.com/Personally/af02_v10_elsaesser_trw_old.pdf
[2] Hale S., "Should Tetrahedral Elements Be Avoided In Explicit Dynamics Analysis?" CAE associates, https://caeai.com/blog/should-tetrahedral-elements-be-avoided-explicit-dynamics-analysis
[3] Erhart, T., Review of solid element formulations in LS-DYNA®. LS-DYNA® Forum, 2011
[4] ASTM D790, M.-9. (1993). Standard test methods for flexural properties of unreinforced and reinforced plastics and electrical insulating materials. In A. S. Materials, Annual book of ASTM Standards (p. Vol.08.01).
[5] Material Datasheet for ABS M-30 grade,
http://usglobalimages.stratasys.com/Main/Files/Material_Spec_Sheets/MSS_FDM_ABSM30.pdf