A finite element study of shell and solid element performance in crash-box simulations

Mahdi Bari
Abstract
This thesis comprehends a series of nonlinear numerical studies with the finite element software’s LS-Dyna and Impetus AFEA. The main focus lies on a comparative crash analysis of an aluminium beam profile which the company Sapa technology has used during their crash analysis.

The aluminium profile has the characteristic of having different thickness over span ratios within the profile. This characteristic provided the opportunity to conduct a performance investigation of shell and solid elements with finite element analysis.

Numerical comparisons were made between shell and solid elements where measurable parameters such as internal energy, simulation times, buckling patterns and material failures were compared to physical tests conducted prior to this thesis by Sapa technology.

The performance investigation of shell and solid elements was initiated by creating models of the aluminium profile for general visualization and to facilitate the meshing of surfaces. The meshing procedure was considered to be an important factor of the analysis. The mesh quality and element orientations were carefully monitored in order to achieve acceptable results when the models were compared to physical tests.

Preliminary simulations were further conducted in order to obtain a clear understanding of software parameters when performing crash simulations in LS-Dyna and Impetus AFEA. The investigated parameters were element formulations and material models. A general parameter understanding facilitated in the selection of parameters for actual simulations, where material failure and damage models were used.

In conclusion, LS-Dyna was observed to provide a bigger internal energy absorption during the crushing of the beam with longer simulation times for solid elements when compared to shell elements. Impetus AFEA did on the other hand provide results close to physical test data with acceptable simulation times when compared to physical tests.

The result difference obtained from the FE-software’s in relation to physical crash experiments were considered to be varied but did indicate that shell elements were efficient enough for the specific profile during simulations with LS-Dyna. Impetus AFEA proved that the same time to be numerically efficient for energy approximations with solid elements refined with the third polynomial.
Preface

This thesis was made with great help from the supervisor Mats Larsson at University West. It is also important to acknowledge Sapa technology for inviting me to discuss the thesis in person and showing me the facility where some of the tests mentioned in this thesis were performed.

Finally, I would like to thank the Company supervisor Björn Olsson for providing guidance throughout the thesis work.
Contents

Abstract ........................................................................................................... i
Preface ............................................................................................................. ii
Symbols and glossary....................................................................................... v
1 Introduction ................................................................................................. 1
   1.1 Company description........................................................................... 1
   1.2 Problem background........................................................................... 2
2 Overview of previous work .......................................................................... 2
   2.1 Aim and limitations........................................................................... 3
3 Methodology ............................................................................................... 3
   3.1 External search................................................................................... 4
   3.2 Design and simulations.................................................................... 4
   3.3 Comparison of data.......................................................................... 4
4 Theory ........................................................................................................ 5
   4.1 Crash analysis................................................................................... 5
   4.2 Element theory................................................................................ 6
5 Profile design ........................................................................................... 8
   5.1 Basic design..................................................................................... 8
   5.2 Meshing.......................................................................................... 10
6 Simulations ............................................................................................... 11
   6.1 Preparatory simulations.................................................................... 11
      6.1.1 Element formulation.................................................................. 13
      6.1.2 Simulation efficiency............................................................... 16
   6.2 Choice of parameters....................................................................... 18
   6.3 Simulations with shell elements...................................................... 19
   6.4 Simulations with solid elements...................................................... 22
7 Simulations with Impetus AFEA............................................................... 23
8 Further measurements.............................................................................. 25
9 Results ....................................................................................................... 27
10 Analysis and discussion.......................................................................... 28
11 Conclusions and future work................................................................. 29
References .................................................................................................... 30
Appendices

A. Sapa beam
B. Timestep equations
C. Shell and solid CAD models
D. Mesh differences
E. Initial buckling and folding observation
F. Time steps calculated by LS-Dyna
G. Material data
H. Material failure and GISSMO (Visual, Times and Debris)
I. Mesh differences in Impetus AFEA
J. Energy measurements of material failure for shell and solid elements in relation to physical tests
K. Energy comparisons for Impetus AFEA and physical tests
L. Final data comparisons
Symbols and glossary

CAD – Computer Aided Engineering
Mesh – Grid of nodes and elements
FEA/FEM- Finite Element Analysis/Method
GISSMO – Generalized Incremental Stress State dependent damage Model
NIP – Number of Integration Points
CPU – Central processing unit
GPU- Graphics processing unit
NDA- Non-disclosure agreement


1 Introduction

Understanding how material failure is simulated in a reliable way is considered to be crucial for the advancing technology in the automotive industry. To be able to safely predict deformation in a crash analysis, an understanding of how different parameters such as element formulation and its implications on simulation results is needed.

The analysis was performed on a crash box where the main purpose of the part is to prevent the adjacent car structure from absorbing even minimal amounts of damage during low velocity crashes. To be able to measure the amount of damage that the car structure absorbs, the crash box is analysed by measuring the amount of energy that can be absorbed by the part.

The choice of whether to use shell or solid elements is often experience based even though there are some guidelines. Using wrong element types can produce non-acceptable results [1].

One main inquiry to be investigated is the question whether solid elements produce more accurate results in relation to shell elements with acceptable simulation times. The results are implied as the absorbed internal energy during a crash. The main finite element software which is used during this study is LS-Dyna. A minor comparative study was also made with the FE-software Impetus AFEA.

This inquiry was based on the fact that the geometry of the crash box is a thin walled structure with a varying body thickness. The varying body thickness provides different thickness to span ratios which are to be considered, see Appendix A.

Results from this thesis will provide with an overview of simulation results in LS-Dyna and Impetus AFEA in relation to physical tests.

1.1 Company description

Sapa Group is the world leader in aluminium solutions with worldwide manufacturing and distribution of aluminium profiles. Sapa group currently has 23,000 employees in more than 40 countries with the headquarters located in Oslo, Norway.

Sapa Technology is Sapa’s internal research and development resource located in Finspång, Sweden. Sapa Technology takes an active part in business development and technical development of products and processes throughout the group. Sapa technology conducts development projects in collaboration with Sapa companies and their customers as well as universities and research institutes [2]. This thesis is made in collaboration with Sapa Technology and is performed at University West in Trollhättan, Sweden.
1.2 Problem background

Simulations have been performed by Sapa Technology in which the FE-software LS-Dyna was used. The simulated beam profile consisted of shell elements and was simulated with different material models. Simulation results showed some concern regarding deformation when compared to physical tests with the use of shell elements [2].

Result comparisons with shell and solid elements were requested since a varying thickness exists within the profile. The purpose was to investigate if more accurate results could be achieved. Sapa Technology has been using the FE-software Impetus AFEA for other simulations and has for further comparison requested a result comparison between the two FE-software’s.

2 Overview of previous work

During previously performed crash tests by Sapa technology, a material model comparison in LS-Dyna was conducted in order to see which model presents the most reliable results. Indoor crush tests were performed and compared to simulated data in order to measure result discrepancy in absorbed energy. The analyzed geometry was a company specific aluminum profile, see Appendix 11.A.

Following material models were compared:

- Isotropic linear plasticity material model without a material failure criteria
- Anisotropic material model with material failure criteria named Generalized Incremental Stress State dependent damage Model (GISSMO).
- Company specific material model developed by Simlab in Norway with a failure criteria and a hardening rule optimized after necking.

The GISSMO model showed the most realistic results in the crush tests due to same crack behaviours in the simulations as in the indoor tests, see Figure 1.

Figure 1, Indoor crush test and simulation with the GISSMO material model
Sapa Technology has also performed an FE-analysis of a three point bending test where GISSMO showed a slightly more accurate behaviour compared to the other material models by having a more realistic force-displacement ratio.

Physical crush-tests did in some cases result in a statistical anomaly where the middle-wall ruptured. This phenomenon was considered to be stochastic.

The conclusion of the performed tests was that the commercial material model GISSMO is recommended for future simulations over the non-commercial Simlab model since it is faster and maintains reliable simulation results [2].

2.1 Aim and limitations

The aim of this thesis is to answer the question whether shell elements are good enough for the crash analysis performed by the company. Conducting a numerical analysis with solid elements and comparing simulation results with previous work can produce a general understanding regarding the choice of element type. The numerical study can further explain if and why one certain type of element produces more reliable results than the other.

The analysis will solely be focused on crush simulations. The simulation results will be compared to previous work done by Sapa Technology. The analysis will not include any physical crush tests since they have already been performed by Sapa Technology. Simulations will be performed in the FE-software’s LS-Dyna and Impetus AFEA.

3 Methodology

The chosen approach to achieve a progressive yet measurable result span during the thesis work is divided into three subcategories; Internal and external search, design & simulations and result comparison. An iterative strategy was used to maintain a good overview of results achieved in each step, see Figure 2.

![Figure 2, Methodology overview](image)

Each of the above mentioned subcategories were executed in a sequence and were not to be executed in a random order due to the risk of time loss. As seen in Figure 2, an iterative procedure is combined with the task sequence. The purpose of such procedure was to maintain an effective work progress.
It has been noted by previous experience that it is important to take a step back and compare the results with already conducted research. In most cases, additional research is required since surprising anomalies might occur, often caused by initial software parameters.

Comparing achieved results with company data is the final step in the sequence and requires thorough understanding of previous steps in order to draw adequate conclusions.

### 3.1 External search

Relative information gathered in the courses HFC400 and FMB300 [5, 6] were used as a starting point when creating a theory basis for the project. Information such as material data and previous research was provided by Sapa Technology and was used throughout the thesis work.

The search step was to be continuous throughout the project work. Time efficiency was considered to be an important factor during the work, hence the search scope was constrained and adapted by each planned activity. The information search did expand for each proceeding step in the methodology sequence presented in Figure 2.

### 3.2 Design and simulations

The computer aided engineering software Siemens PLM NX was used to design necessary simulation profiles. A finite element pre-processor, in this case Hypermesh was further used to create different mesh combinations on the designed surfaces.

Simulations with LS-Dyna were executed through an external calculation cluster located at PTC, Production Technology Centre in Trollhättan, Sweden. Simulations in Impetus AFEA were executed by the use of the graphical processing unit residing within the computer. All simulations with LS-Dyna were performed by the use of 12 CPU’s in the calculation cluster.

### 3.3 Comparison of data

Comparisons of company data in relation to simulated data were executed in order to validate performed simulations in a reliable way. Company data contained simulation results as well as physical test results. Provided data from the company was considered to be reference information during the thesis work.

The reference information was used as a discrepancy measurement tool when measuring variations in unit magnitudes obtained from different simulations. The comparisons were made on the absorbed internal energy and simulation time values obtained from simulations with shell or solid elements.
4 Theory

Performing nonlinear numerical analysis requires a good knowledge basis before the actual work. This section covers fundamental theory required to perform the analysis. Covered topics describe important factors of a crash analysis and the importance of having a good mesh.

4.1 Crash analysis

In order to achieve as reliable numerical results as possible, a good approach is needed. One of the more distinctive characteristics of a crash is that the events happen during a short period of time which opens up the possibility of using an explicit numerical method. Another possibility is to use an implicit approach which is more hardware demanding since more variables are added in the calculation [7].

Crash analysis is very contact dependent since the ability to simulate an actual crash relies on multiple factors and can have fatal results if not done correctly [4]. The contact definition has two main categories to choose from, nodal based contact and segment based contact. These are further divided into sub categories consisting of one way respectively two way contact types.

Using a nodal based contact requires more work if different mesh constellations are to be used since the node identification numbers change depending on the mesh constellation. Another disadvantage is that the contact search locates the closest nodes on different segments, which makes this contact definition unstable for meshes consisting of elements with varying aspect ratios.

The segment based contact searches for node segments during contact which is preferred during crash analysis. The choice of one way or two way definition lies in whether both the slave and master part are checked for penetration or if only the slave part is checked.

To ensure that no penetration occurs before the intended contact, an offset of colliding parts is recommended, see Figure 3.

![Figure 3, Offset for contacts](image)

The offset is recommended to be at least the sum of thickness divided by two [8].
4.2 Element theory

The choice of element type is in most cases dependent on the geometry which is to be analysed. The analysis is being made on a thin walled structure, which provides two different possibilities to approach the problem. The approaches are whether to use shell or solid elements. General guidelines mention the length over thickness ratio as a good reference point where the ratio presents a boundary between the choice of shell and solid elements [1].

Working with shell and solid elements yields in many different element types to choose from [10]. The efficiency of an element type is dependent on the surface geometry and application. In crash analysis, folding or buckling of elements often occur so the chosen element type must be suitable in definition to withstand and take advantage of such conditions.

For shell elements, a quadrilateral element type with sufficient amount of integration points in the element and through the thickness is considered to be efficient when simulating a crash. The same formulation of integration points is suited for solid brick elements. Choosing the amount of integration points in an element is considered to be part the element formulation [11].

There are some well used element formulations in LS-Dyna when simulations are performed with quadrilateral shell elements. One of the recommended formulations is the Belytschko-Lin-Tsay formulation, denoted as “ELFORM=2” which provides one integration point on the element surface. Another element formulation is the fully integrated shell formulation, denoted as “ELFORM=16” which provides four integration points on the element surface, see Figure 4.

![Figure 4, Element formulations for quadrilateral shell elements](image)

The amount of integration points on the surface can provide different results in a crash simulation. Four integration points on the element surface do in most cases provide a more accurate surface buckling during crash simulations when compared to the one integration point formulation. More integration points on an element surface can however facilitate in warping of the element, so hourglass stiffness must be considered [10].
For solid elements, the corresponding element formulations exist with more options if shear locking is for example present. If eight-noded brick elements are to be used, the recommended element formulations are either the default formulation with one integration point, denoted as “ELFORM=1” or the fully integrated solid formulation, denoted as “ELFORM=2” which provides eight integration points on the element surface, see Figure 5. [10]

![Figure 5, Element formulations for solid brick elements](image)

The described element formulations for shell and solid elements are considered to be analogous due to the same integration point formulation of the element surface.

The number of integration points (NIP) through the thickness is an important variable. To achieve a good approximation of the material behaviour during nonlinear deformation, a higher amount of integration points is recommended. It is important to note that a higher number of NIP’s results in a longer simulation time but yields theoretically more correct results. During a crash analysis it is common to have at least five integration points [11].

Element formulations are further dependent on the element construction where the element’s skewness, warping and angular deviation are defined with respect to recommended values. Creating a mesh with poor elements can yield in undesired simulation results [9].

Another part of the element formulation is the definition of the element’s orientation. This is done by firstly defining the element orientation with respect to the global coordinate system. By doing so, a normal vector of an element is defined. The purpose of defining the element normal lies in keeping track of the NIP’s and simply to keep track of which side is top or bottom, left or right [11].
5 Profile design

Visualizing the beam from different aspects was considered to be an efficient starting point. Different methods were used to design the profile where the main purpose was to achieve a model design with different design options for the meshing procedure.

5.1 Basic design

The basic design was constructed by an assembly of two sheet bodies for simulations with shell elements. For simulations with solid elements, a full thickness body was made, see Appendix C.

All dimensions were provided by Sapa technology, see Appendix 11.A. The dimensions were adapted to post performed physical tests, where the crash impact was in the need of being controlled, hence the angles. The geometry was slightly modified by changing the starting location of the angles to the starting point of the middle wall radius instead of the centre of the beam, see Figure 6.

![Figure 6, Modified geometry](image)

To create a good base model for the meshing step, some preparations of the designed model were made. It was desired to have as many quadrilateral elements as possible for the shell mesh, and brick elements for the solid element mesh. Applied angles in the beam model were considered to possibly create an uneven distribution of elements in the meshing process. In order to obtain a homogenous mesh throughout the model, boundary constrains were needed to be defined in case of mesh editing.
It was known that the crushing of the beam had a fixed distance, so creating equal lengths on the geometry would push the uneven distribution of elements further to the bottom where no crushing is performed.

To construct usable boundary constraints, a projection of the angles on the top of the structure was applied 220 mm from the top plane. The projection was used to subdivide the model into three parts consisting of a top, bottom and a middle part, see Figure 7.

As seen in figure, the design has clear boundary conditions which would if necessary be used for later mesh editing. The same procedure was performed on the solid model. By performing this design precaution, the meshing step would later on be performed in a more efficient manner since eventual failures in the mesh would be reduced.
5.2 Meshing

The designed models were at this point ready to be meshed. Two types of mesh strategies were used. The first type was a free mesh over the whole surface and the second type was a boundary controlled mesh over subdivided surfaces. The differences can be seen in Appendix D.

The element size for shell elements was 2 x 2 [mm] with smaller element ratios towards the middle of the beam where the element size was 1.5 x 2 [mm]. The latter element size was also applied to the connecting edges of the middle wall.

It was known that the beam would fold at the sides during the crushing, so putting three elements over the thickness for the brick mesh was considered to be sufficient in order to capture the folds. With three elements over the thickness, the element size was 2 x 2 x 0.9 [mm] and with the element dimensions of 1.5 x 2 x 0.9 [mm] at the middle wall.

To ensure that the mesh quality was adequate before simulations, an element check was performed in Hypermesh. Parameters such as element skewness, warping and angular deviation were checked and compared to recommended values specified in the software. The values are explained in the literature Basics about FEA [9].

The free meshed surfaces and the boundary controlled mesh surfaces were quality controlled before they could be approved for simulation; the quality check is displayed in Table 1.

<table>
<thead>
<tr>
<th>Shell elements</th>
<th>Recommended values</th>
<th>Obtained Max values Type 1</th>
<th>Obtained Max values Type 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Skewness</td>
<td>&lt;60°</td>
<td>20,85°</td>
<td>34°</td>
</tr>
<tr>
<td>Warping</td>
<td>&lt;5°</td>
<td>1,99°</td>
<td>18,78°</td>
</tr>
<tr>
<td>Angular deviation</td>
<td>&gt;45° and &lt;135°</td>
<td>68° and 111,5°</td>
<td>49,2° and 134°</td>
</tr>
<tr>
<td>Jacobian</td>
<td>&gt;0,6</td>
<td>0,96</td>
<td>0,6</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Solid elements</th>
<th>Recommended values</th>
<th>Obtained Max values Type 1</th>
<th>Obtained Max values Type 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Skewness</td>
<td>&lt;60°</td>
<td>24,63°</td>
<td>50°</td>
</tr>
<tr>
<td>Warping</td>
<td>&lt;5°</td>
<td>0,24°</td>
<td>25,72°</td>
</tr>
<tr>
<td>Angular deviation</td>
<td>&gt;45° and &lt;135°</td>
<td>56,44° and 120,25°</td>
<td>35° and 157°</td>
</tr>
<tr>
<td>Jacobian</td>
<td>&gt;0,6</td>
<td>0,66</td>
<td>0,43</td>
</tr>
</tbody>
</table>

The main reason for the presented quality differences was that the free mesh did not have any boundaries within the surface body so it was cumbersome to edit a specific area in the mesh without altering the whole mesh body. Having a boundary constrained mesh created an improved control over the mesh and resulted in better element quality.
6 Simulations

This section covers the simulation process of the beam profile. Preparatory simulations were made before actual simulations which included simulations with company parameters and different element formulations. Efficiency focused simulations were also performed during the preparatory step where the influence of time step was investigated. No material damage or failure models were used during the preparatory simulations for the sake of simplicity.

Results from the preparatory simulations were used as indication for which parameters to use during actual simulations where material damage and material failure models would be used. The preparatory step was only performed in LS-Dyna since Sapa technology used the FE-software as the main tool during their simulations. Simulations with Impetus AF EA are discussed later on. Measurements of obtained results were performed throughout the simulation procedure and were further analysed in section 8, further measurements.

6.1 Preparatory simulations

The purpose during the preparatory step was merely to investigate performance of different software parameters as well as understanding the crash procedure. Simulations were compared to company data with the purpose of understanding parameter implications on the resulting data. Following software parameters were investigated:

- Element formulations for shell and solid elements
- Time step parameters

The preparatory step facilitated actual simulations by reducing simulation failures.

Given data from the company facilitated simulations by knowing how the crushing procedure was to be made and which physical parameters to use. The crushing was performed in a vertical motion by a rigid top plate moving at 2 m/s in the negative direction for 200 mm, see Appendix 11.A for a graphical description. Material specific parameters such as the stress-strain curve and Young’s modulus were also provided by the company and can be found in Appendix G.

A geometry with minor adjustments compared to the company geometry was as previously mentioned created see section 5.1, basic design. Initial simulations were in the preparatory step performed where the only difference was the geometry. The provided software parameters were used during the mentioned initial simulations.

This procedure was conducted in order to spot any differences in the internal energy for both geometries when compared to each other, see Figure 8 on the next page.
Obtained results were observed to display a linear behaviour of the internal energy curvatures for both geometries in the graph. The energy curves had a negligible difference at the beginning of the crush and did deviate from each other towards the final crushing destination. The difference in absorbed internal energy was 800 J. Simulation times were also monitored and compared, see Table 2, Simulation efficiency.

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Company</th>
<th>University West</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elements</td>
<td>16120</td>
<td>35904</td>
</tr>
<tr>
<td>Simulation Time</td>
<td>02:34:09</td>
<td>03:55:04</td>
</tr>
</tbody>
</table>

Simulation times were as seen in the table proportional to the number of elements in the geometry. The created geometry had approximately twice the amount of elements compared to the geometry provided by the company.

Since the results did not show completely different magnitudes of the internal energy, it was assumed that the main reason for the discrepancy was the altered geometry. To supplement the initial analysis, an ocular buckling comparison was made of the crashed beams, see Appendix 11.E.
6.1.1 Element formulation

Element formulations were investigated in order to obtain as accurate results as possible. For shell elements, both “ELFORM=2” and “ELFORM=16” were used during simulations. The simulation results were compared to company values where the only difference in the software parameters was the element formulation. The physical difference was the geometry. Figure 9 illustrates the difference in internal energy for both geometries.

The company simulation was performed with “ELFORM=16”. As seen in the figure, the internal energy had a slightly different curvature behaviour but the same final magnitude for both “ELFORM=2” and “ELFORM=16” for the new geometry.

With a negligible difference in internal energy for the two element formulations, an ocular analysis of the buckling and folding was performed, see Figure 10.
The buckling and folding was observed to show a rather coarse behaviour for “ELFORM=2” which could be explained by the lesser amount of integration points on the element surface. Comparing “ELFORM=2” to “ELFORM=16” did illustrate a more symmetrical buckling and folding of the surfaces. Buckling and folding comparisons to physical experiments were at this point not yet performed since the simulations did not include a material failure which physical tests are exerted to.

A bigger variety of element formulations were analysed for solid elements. Simulation results are presented in Figure 11.

![Figure 11, Internal energy difference with different solid element formulations](image)

The internal energy was observed to have big energy discrepancies depending on the chosen element formulation. Looking at ELFORM=3” in the figure, it is seen that the element formulation suffered from negative energy values (black curve). This was fixed by adjusting the hourglass stiffness of the elements (red curve). An ocular buckling analysis was also made for solid element simulations see Figure 12.

![Figure 12, Buckling with different solid element formulations](image)

The ocular analysis did as for the shell elements show discrepancies in the buckling and folding symmetry when analogous element formulations were compared. It was as
previously stated that the buckling and folding symmetry was dependent on the amount of integration points on the element surface.

A result comparison was made for the element formulations to conclude the investigation of element formulations for both shell and solid elements, see Figure 13.

![Figure 13, Internal energy for shell and solid elements](image1)

It was observed that the internal energy values were grouped by a higher and a lower value cluster. The difference in values was quite significant with a 6.4 kJ difference between the maximum and minimum values in the graph. For the analogous element formulations the differences were 320 J for one point integration formulation and 490 J for the fully integrated element formulation. The results were also analysed by comparing simulation time values, see Figure 14.

![Figure 14, Simulation times for shell and solid element formulations](image2)

It was observed that the simulation times were significantly longer for solid elements in comparison to shell elements. It was however not expected that “ELFORM=16” would be faster to simulate than “ELFORM=2” with respect to element theory, see section...
4.2. It is on the other hand important to note that the simulation times might vary due to different processor efficiencies in the calculation cluster.

### 6.1.2 Simulation efficiency

Performing simulations with solid elements proved to not be time efficient so simulations with different time steps were performed to investigate if the time efficiency could be increased. The time steps were controlled by adding mass to the elements in order to speed up the simulation times, see Appendix B for more information. Software parameters provided by Sapa technology did include a time step value which had the magnitude of 1e-7.

Performing simulations with different time step were considered to take too long time when simulating solid elements, so shell elements were used to get the idea of the time step implications on the results. With the help of equations presented in Appendix B, a time step of the value 2.75e-7 could be achieved.

Simulations with the calculated time step were performed with increased and decreased time steps by the factor of 10. Simulation times were observed to increase with smaller time steps and decrease with a bigger time step, see Figure 15.

![Figure 15](image)

**Figure 15, Simulation times with different time step values**

Obtained time values below the calculated time step (marked in red) was considered to be unstable since the calculation time did not show a steady decrease while values above the calculated value were observed to have a steady increase.
The calculated time step was further observed to yield in a decreased simulation time for the new geometry when compared to the default time step.

Internal energies were for each time step value noted and compared, see Figure 16.

![Figure 16, Internal energy difference for each time step](image)

Energy magnitudes were observed to be consistent for the calculated time step and smaller values. Smaller time steps did not show any discrepancies in obtained energy values. Bigger time step values were on the other hand observed to absorb more energy. It was considered that values close to the company provided time step value of 1e-7 were time efficient with respect to produced energy values.

A time step value could be automatically calculated in LS-Dyna by setting the time step to 0 in LS-Dyna. The automatic procedure recorded the hundred smallest obtained time steps. A default value was for that reason considered for further simulations with solid elements. The results from this analysis showed that the manually calculated time step did not differ too much from the time step calculated by LS-Dyna, see 11.FAppendix F.
6.2 Choice of parameters

Software parameters were with the previous step concluded chosen in LS-Dyna for simulations with shell and solid elements. Chosen element formulations for both shell and solid elements were the analogous formulations which are the one integration point formulation and the fully integration formulation.

The contact type was chosen to be of the type segment based search with a master and slave logic. The slave part was the crashed part to be crashed and the master part was the tool.

Simulations were at this point performed with the company provided anisotropic damage model “GISSMO” and a software integrated material failure parameter in the isotropic material model. The damage parameters for GISSMO will not be revealed due to a non-disclosure agreement (NDA) with Sapa technology. Simulations with material failure were applied by setting the failure parameter to 70%, 80% and 90% of the plastic strain. Elements that reached specified values were deleted by LS-Dyna.

The calculated time step value was chosen from the previous section in order to obtain time efficient simulations with shell elements. A default time step value was further chosen for simulations solid elements in order to achieve results safe from any time step manipulations.

Internal energies and simulation times were to be measured as previously. Simulations with the chosen software parameters were performed by the following scheme:

- Simulations with shell elements in combination with GISSMO as well as a material failure model
- Simulations with solid elements in combination with a material failure model

Simulations with solid elements were only performed with a material failure model since GISSMO was not adapted for solid elements. The result comparison for shell and solid elements with the material failure model had the purpose to merely indicate the difference in absorbed energy magnitudes.
6.3 Simulations with shell elements

Simulations with shell elements were performed with both the new geometry and the geometry provided by the company. The material failure model was used during simulations of both geometries with the purpose of seeing eventual result discrepancies. Obtained results were not observed to differ in energy behaviour when compared to each other but did differ in the energy magnitude for the new geometry, see Figure 17.

![Figure 17, Simulations with material failure model for the new geometry](image1)

The highest absorbed energy occurred when most of the material was intact, in this case with the specified value of 90% failure to plastic strain so the results were considered to be valid. The absorbed energy was for the company provided geometry observed to have the same energy behaviour as for the new geometry. The energy magnitudes were further observed to be slightly larger in magnitude when compared to the new geometry, see Figure 18 on the next page.

![Figure 18, Simulations with material failure model for company geometry](image2)

Analysing both graph shows that more energy was absorbed with the mesh provided by the company. The difference in magnitude for both geometries was 1.6 kJ for 90%, 700 J for 80% and 800 J for 70% fail to plastic strain.
Simulations with GISSMO in combination with the new geometry displayed an energy behavior which was not by a linear nature as previous simulations were. The geometry provided by the company did on the other hand show a consistent energy behavior with previous simulations, see Figure 19.

The internal energy was observed to have a linear behavior for the new geometry until approximately 70 [mm] of displacement before starting to increase in magnitude. The increase was further noted to accelerate at a higher rate when exceeding approximately 120 [mm] of displacement.

A visual comparison was made for both material models. The material failure model showed that fewer elements were deleted with a larger failure parameter, see Figure 20.

![Figure 19, Absorbed energy with GISSMO for both geometries](image)

The deleted elements showed consistency with energy magnitudes presented in Figure 18 since deleted elements implied that eventual energy stored in the failed elements was deleted.

GISSMO was at first sight observed to show that a large amount of elements was deleted and was compared to previously performed simulations with a material failure model, see Figure 21.
The visual presentation showed that GISSMO provided more wrecked models in comparison to the material failure simulations.

Investigating the simulated GISSMO models from a farther distance displayed that not all elements were deleted. Debris in different sizes was observed to be scattered around the model where energy still was absorbed, see Figure 22.

Simulations with the company provided geometry did not provide debris of the same magnitude as shown in Figure 22. The debris was further observed not to absorb any energy for the company simulations. It was not determined if the absorbed energy in the debris for the new geometry was excluded from the final energy values in LS-Dyna. See Appendix H for more illustrations from this section.
6.4 Simulations with solid elements

Simulations with solid elements were as previously mentioned only performed with a material failure model. The beam was observed to absorb more internal energy when simulated with solid elements in comparison to shell elements, see Figure 23.

![Figure 23, absorbed energy with Solid vs. Shell elements](image)

It was further noted that the two analysed element types did create an upper and lower value cluster. A visual comparison was also made of the simulated beams. The results were as for shell elements observed to relate to the energy graph as previously by showing that more elements were saved when a larger fail parameter was set, see Figure 24.

![Figure 24, Solid elements with material failure model](image)

Simulation times were also noted and compared for all simulations in this chapter and can be found with other more visually detailed figures and comparisons from this entire section in Appendix H.
Simulations with Impetus AFEA

Simulations with impetus AFEA were done in a different manner since the FE-software offers a different set of software parameters in comparison to LS-Dyna. The element formulations were automatically selected by Impetus with an option of changing the polynomial order in the elements, which means that the mesh would be divided and refined into more elements.

Simulations were for consistency reasons performed with and without the polynomial option to analyse the differences. Sapa Technology provided parameters for a damage model to use when performing simulations with Impetus AFEA. The damage models in Impetus AFEA and LS-Dyna are software wise formulated in different ways. As for GISSMO simulations in the previous chapter, the damage parameters for Impetus AFEA will not be revealed due to the NDA. The simulations were done by the following scheme:

- Simulations with the same fine solid mesh as used in LS-Dyna performed with and without a damage model
- Simulations with a coarse mesh to be refined with a polynomial of the third degree, with and without a damage model

It was considered to be important to create similar input files for both FE-software’s in order to have a good understanding of eventual discrepancies. A series of preparatory simulations were made with the fine mesh used in LS-Dyna where the tool failed to reach the crushing destination so a coarser mesh was created with an element size increased by the factor of two. The differences in the meshes used during simulations with Impetus can be seen in Appendix 0.

Obtained energies were observed to differ depending on the choice of mesh. A fine mesh absorbed more internal energy than a polynomial refined mesh, see Figure 25.

![Figure 25, Internal energies for solid elements in Impetus AFEA](image-url)
It was further observed that the damage model did not affect the results for the fine mesh, which indicated that no elements were deleted. The refined mesh did show more reliable results by illustrating that the damage model affected the results.

Simulations with the damage model in combination with the refined mesh did provide with the lowest internal energy value of 20.3 kJ compared to the highest noted value of 28.7 kJ for the fine mesh.

An ocular analysis of the buckling and folding was for further comparison performed, see Figure 26.

Simulations with the fine mesh were considered to be failed since the damage model did not affect the simulation results both visually and energy wise.

Simulation times varied since different GPU’s were used during the thesis. The highest noted simulation time was 14 hours for the fine mesh while the simulation time for the refined mesh was 3.5 hours.

Figure 26, Visual differences in simulations
8 Further measurements

Results from simulations with the two FE-software’s were at this point collected and measured with reference to each other and to physical tests. Energy comparisons will be referred to the Appendix due to graph sizes.

Comparing results from simulations with a material failure model for both shell and solid elements did show that solid elements were inclined to yield in larger energy values compared to physical tests. Shell values were noted on the other hand to yield in values a bit lower than physical tests, see Appendix J for more information.

Visual comparisons showed that beams simulated with material failure were exerted to shearing at the top middle surface. This characteristic was not found in any of the physical test samples. Similarities were on the other hand found in the edges of the beam where the physical tests displayed cracks and simulated beams displayed empty element spaces which indicate that elements had been deleted, see Figure 27.

![Shell 70% strain, Solid 70% strain, Physical test](image)

Figure 27, Shell and solid model in comparison to physical sample

The simulated beams did seem to have the same surface behaviour at the bottom where the sides were pushed in. The folds in the shell model illustrated better visual approximation than solid element which were exerted to more shear. Energy values obtained from simulations with Impetus AFEA where the third polynomial option was used were observed to show similar magnitudes and behaviours as for the physical tests. See Appendix K for comparisons.
It was observed when comparing simulation models from Impetus to physical test samples that many visual similarities could be found. Simulated models showed minimal shear at the middle top surface and did further show cracks in the centre and edges of the crashed beam. Mentioned visual characteristics could also be found in physical samples, see Figure 28.

For final comparison the obtained energy curves from all simulations were cross referenced to physical tests. Comparisons to more physical samples were also performed, the final measurement data can be found in Appendix L.
9 Results

Obtained energy values from simulations with solid elements were in general higher than values obtained from simulations with shell elements and physical test data. Shell elements did on the other hand absorb less energy in comparison to physical tests. The simulation times were also noted to be larger when performing simulations with solid elements.

Results obtained from simulations with GISSMO in combination with the new geometry were observed to have different energy behaviours during the crash when compared to simulations with company geometry. The energy magnitude was further noted to be larger for GISSMO when simulated with the new geometry. A characteristic energy behaviour was noted where the energy suffered from a rapid increase in magnitude when exceeding approximately half of the crushing distance towards the end of the crushing destination. Simulations with material failure in combination with shell elements did yield in energy values close to the company simulated GISSMO values but with lower simulation times.

The internal energy was for the new geometry further observed to be larger in magnitude during simulations without any material failure or damage model in comparison to simulations with the company provided geometry.

Solid elements were seen to be consistent in the high magnitudes of absorbed energy when simulated with and without a material failure model. The magnitudes did decrease with an applied failure model.

Simulations with Impetus AFEA were not only time efficient but also provided both visually and energy accurate results. The accuracy was as previously measured by comparisons to physical tests. Energy behaviours did in Impetus show consistency throughout simulations when compared to both FE-software’s. The absorbed energy in Impetus did illustrate a linear behaviour and was as in most simulations with LS-Dyna observed to decrease in magnitude when a damage model was applied.

Visual observations showed a shear at the top of the beam when simulated with LS-Dyna while no visual shear was present in Impetus.
10 Analysis and discussion

Simulations with the new geometry in LS-Dyna were at some points cumbersome by displaying higher absorbed energies for lower material failure parameters than for higher failure parameters. This was considered to be odd. Applying a warping stiffness of the elements fixed the issue but indicated at the same time that elements with different aspect ratios in the same mesh can prove to be troublesome.

Solving the issue by applying a warping stiffness was also considered to be logical since elongated elements are not considered to be as robust as square ones. The different aspect ratios of the elements were also considered to be a factor for the achieved energy behaviours when simulating GISSMO in combination with the new geometry. The results from GISSMO simulation did also present an odd behaviour since the amount of absorbed energy increased rapidly and exceeded other noted energy values. The debris that was noted in the GISSMO simulation did show that energy was stored at some locations which indicates that some optimizations might be needed in GISSMO with the purpose of removing the excessive debris.

Simulations with solid elements showed that the absorbed energy was somewhat overestimated by LS-Dyna. The solid elements did also take long time to simulate and were not considered to be efficient neither in energy estimation nor in time efficiency.

Performing simulations with Impetus AFEA was considered to be much simpler due to an updated interface and greatly reduced list of software parameters in comparison to LS-Dyna. The input code was by the reduced software options much smaller and could because of that be easily be altered, so eventual changes could be performed in a fast manner.

The software provided two contact formulations where the difference wasn’t clear. Both formulations were thereby investigated and showed quite different outcomes. The contact formulation “Contact_super” made the beam too stiff for some reason so it exploded. The contact formulation “Contact” resembled the formulation in LS-Dyna by presenting the option of choosing a master and slave part. It was also the latter contact formulation that resulted in a full simulation.

By using the third polynomial option for setting up an element formulation, a coarse mesh could be used as input data. The FE-software recoded the mesh automatically and simulation results were achieved fast. The simulation results did overall present a better quality in both visual and energy aspects.

Visual comparisons did show that the element formulation in both FE-software’s is different. The same basic geometry was used in both software’s and yet different outcomes were achieved in which a noted shear could be present in LS-Dyna but not in Impetus AFEA.
11 Conclusions and future work

The results proved that shell elements were efficient in order to provide a good estimation of energy values with not too costly simulation times. Simulations with solid elements were not observed to provide similar results so it is not recommend to use solid elements for simulations with this type of profile where the walls are very thin.

Even though obtained energy magnitudes with shell elements in combination with material failure were somewhat lower than the physical tests, the obtained results were considered to be a good approximation indicator for absorbed energy values. Simulation times with a material failure model were also noted to be significantly shorter than for GISSMO.

Simulations with Impetus AFEA were observed to provide different results compared to simulations with solid elements in LS-Dyna. The refined third polynomial elements did not only provide good visual presentations of the crashed beams but also energy values in the range of physical tests. It was in the end of the comparison step seen that the visual presentations often mirrored energy data. For example, the material failure simulations showed a direct correspondence by illustrating that more energy had been absorbed when more elements were present. Another example would be the result comparison between Impetus and physical tests. The visual comparison showed more similarities to physical test samples than simulations with LS-Dyna and mirrored the similarities to observed energy behaviours and magnitudes.

The results indicate that it is recommended to use shell elements when simulating this geometrical type of profile with LS-Dyna. If solid elements are to be used, the polynomial option in Impetus AFEA is recommended.

It was further considered to be important to investigate result implications when manipulating the time step parameters even though no major discoveries were made. The main purpose of this minor investigation was to see if differences in internal energy values and simulation times could be obtained with larger or smaller time steps and by doing so, decrease the simulation times. Obtained data proved that same magnitudes in the internal energy could be achieved with smaller time steps but with the cost of increased simulation times. The change in absorbed energy would first occur when larger time steps were applied. This analysis was considered as an additional investigation during the thesis work.

For future work, more simulations could be performed with different aspect ratios of the elements in the same mesh to observe the differences for shell and solid elements, this thesis only presented one example of different aspect ratios.

Simulations with shell elements could also be further investigated by analysing more element formulations in LS-Dyna.
References


A finite element study of shell and solid element performance in crash-box simulations

A. Sapa beam

Axial loading

Rigid top wall

Rigid bottom wall

h = 300mm

Velocity = 2 m/s

Crushing depth = 200mm

Height = 300mm
B. Time step equation

Critical time step = \frac{\text{Element Area}}{\text{Speed of sound} \times \text{Largest element length}}

\text{Speed of sound} = \sqrt{\frac{E}{\rho(1 + v^2)}}

E = \text{Young’s modulus}
\rho = \text{Specific mass density}
v = \text{Poisson}

Formulas can be found in references [11] and [9].

The mass scaling manipulates the speed of sound parameters when applied. A simple example is to understand that Newton’s second law in crash analysis is mass dependent, and if the mass is scaled too much, the result changes.
C. Shell and solid CAD models

Shell surface to the left. Solid surface to the right.
D. Mesh differences

Difference in mesh between the free mesh and a boundary constrained mesh. The boundary constrained mesh consist only of quadrilateral elements for the shell mesh and brick elements for the solid mesh. The free mesh suffers from an uneven distribution of elements with triangular elements filling the gaps.
E. Initial buckling and folding observation

Comparison of buckling and folding on the crashed beams. The left beam presents the company geometry.
F. Time steps calculated by LS-Dyna

Comparison of time steps between Sapa’s geometry and the new geometry.

![Graph of 100 smallest time steps for Sapa geometry](image)

- **DT2MS=1e-7, 2h 35 min**
- **DT2MS=0, 1h 17min**

![Graph of 100 smallest time steps for the new geometry](image)

- **DT2MS=0, 4h simtime**
- **DT2MS=-2.75e-7, 2h simtime**
G. Material data

Given material properties:

- Mass density = 2.7e-9
- Young’s modulus = 70 000 MPa
- Poisson’s ratio = 0.33
- Yield stress= 320 MPa

With the stress-strain curve,
H. Material failure and GISSMO (Visual, Times and Debris)

Shell
GISSMO

Material failure at 70% to plastic strain

GISSMO with the new geometry

GISSMO with company geometry

Appendix E:5
Appendix E:6

A finite element study of shell and solid element performance in crash-box simulations

Solid

90% strain

80% strain

70% strain
Fringe level comparison for both geometries (GISSMO)
Closer look on where the internal energy was absorbed (GISSMO)
Debris difference for the geometries (GISSMO)
Simulation times with material failure and GISSMO (New geometry only)
I. Mesh differences in Impetus AFEA

Fine mesh  
Coarse mesh  
Refined mesh with the third polynomial
J. Energy measurements of material failure for shell and solid elements in relation to physical tests
A finite element study of shell and solid element performance in crash box simulations

K. Energy comparisons for Impetus AFEA and physical tests

Internal energy for solid elements in Impetus AFEA

Energy comparisons for Impetus AFEA and physical tests

Test 2-1
Test 2-2
Test 2-3
Test 2-4
Test 2-5
Test 2-6
Test 2-7
Test 2-8
Test 2-9
Test 2-10
Test 1-5
Test 1-6
Test 1-7
Test 1-8
Test 1-9
Test 1-10
Test 1-11

- Fine mesh with damage model
- Fine mesh without damage model
- Refined mesh without damage model
- Refined mesh with damage model

(kJ)

Displacement [mm]
A finite element study of shell and solid element performance in crash-box simulations

Appendix

Final data comparisons

Internal energy for solid elements in Impetus AFEA

- Highest noted value with fine mesh in Impetus AFEA: 28.7 kJ (T1_10,000)
- Solid elements (High range): 35 kJ
- Physical tests and Impetus AFEA with polynomial (Right in the middle): 22.8 kJ (GISSMO VIN)
- Shell elements (Low range): 20.9 kJ (IA Fetish 0,000)

Displacement [mm] vs. Internal energy [KJ] graph
Model comparison between FE-software’s
Physical test samples and simulated models with material failure

Sapa geometry (shell)  New geometry (shell)  New geometry (solid)
Impetus third polynomial model with physical test samples