Validation of blast simulation models via drop-tower tests


Author
Joakim Rydman

June 16, 2018
Supervisor: Jörgen Vedin, BAE Systems Hägglunds
Examiner: Staffan Schedin, Umeå University
Abstract

This study aims to validate a screw joint simulation model used by BAE Systems in LS-DYNA during blast simulations. It is important that the screw joint simulation model is physically correct, since the simulation results can influence major design decisions. The study provides a short overview on the subject of bolts and screws, material deformation and stress and strain in materials, of the finite element method (FEM) and on some specific numerical methods used in this study. BAE Systems started a validation project of the screw joint simulation model in 2015, but it was not finished due to other priorities. In this older project some drop-tower tests measuring the axial force in a screw joint were conducted. These old tests can now serve as validation data for the screw joint simulation model. The screw joint simulation model used by BAE Systems is dependent on a special kind of finite element formulation; a so called beam element. This study provides a finite element analysis on this simulation model, which is implemented through an established industry FEM solver called LS-DYNA.

The validation of the screw joint simulation model is done against three drop-tower experiments performed at 900, 1000 and 1100mm drop height respectively. The drop-tower experiments were replicated in LS-DYNA, with a prescribed velocity on the falling parts rather than simulating a free fall and non-elastic impact. A comparison between the simulation model using beam elements, that is used by BAE Systems, and a similar simulation model using solid elements is presented as part of the validation. To make sure that the result of the study is confident, a local mesh convergence study and a study of the mass scaling numerical method in LS-DYNA is also presented. The results show that the screw joint simulation model using beam elements is valid according to the available experimental data. In one of the experiments, where the drop-test was performed twice, an average maximum force on the screw was measured to be 33.5±4.8 kN. Simulations of the same case, under the same conditions, using beam elements resulted in a maximum force on the screw of 35.4 kN, well within the experimental result range. In the other two drop-tower experiments, the simulated results showed correlation considering the error sources in the simulation model and the statistical spread that is present in the experimental results. The simulation model using beam elements is also similar to the results using solid elements, which also indicates that the beam model is valid. All in all, it is shown that the beam model can be used to produce safe results that either overestimate or place the simulations of the axial force in the screw in the upper spread of the measurements.

Keywords: Screw, bolt, screw joint, bolted connection, simulation, model, FEM, FEA, finite element, LS-DYNA, drop-tower tests, validation
# Contents

1 Introduction ........................................... 1  
  1.1 Background to the study ............................... 1 
  1.2 Specific goals ...................................... 2 
  1.3 Limitations .......................................... 2 

2 Theory and definitions ................................. 3  
  2.1 Material deformation ................................. 3 
      2.1.1 Stress and strain ............................... 3 
      2.1.2 3-dimensional stress and Von Mises-stress .......... 4 
      2.1.3 Stress and strain relationship .................... 6 
  2.2 Bolts and screws .................................... 7 
  2.3 Finite Element Analysis ............................. 9 
      2.3.1 The Finite Element Method ....................... 9 
      2.3.2 Richardson Extrapolation ....................... 11 
  2.4 LS-DYNA ............................................ 12 
      2.4.1 Governing equations for LS-DYNA ................. 13 
      2.4.2 Timestep control with mass scaling method in LS-DYNA .... 13 

3 Methodology ........................................... 15 
  3.1 Experimental set up .................................. 15 
  3.2 Simulation set up .................................... 17 
      3.2.1 Velocity tracking ............................... 17 
      3.2.2 FE Model ....................................... 20 
  3.3 Screw joint models .................................. 23 
      3.3.1 Beam simulation model ......................... 23 
      3.3.2 Solid simulation model ......................... 24 

4 Pre-study ............................................. 26 
  4.1 Mesh convergence study ............................... 26 
  4.2 Study of the mass scale factor ....................... 28 
  4.3 Conclusion .......................................... 30 

5 Results ................................................ 32 
  5.1 Case 28 and 31, measurements and simulations ....... 32 
  5.2 Case 29, measurements and simulations ................ 34 
  5.3 Case 30, measurements and simulations ................ 35 
  5.4 Plastic strain ...................................... 36 

6 Discussion ............................................ 37 

7 References ............................................ 40
1 Introduction

1.1 Background to the study

Vehicles designed to transport personnel in war zones need to mitigate a possible mine blast so that no injury occurs on the passengers. An important factor to consider is how screw joints or bolted connections in the vehicle behave during a blast. If the joints or connections fail, crucial components of the vehicle may be destroyed, equipment fastened inside might come loose and contribute to passenger injury and the structural integrity of the vehicle might be compromised. Physically, a blast is a fast, dynamic event that occurs during a short duration of time (<15 ms). To test how a vehicle withstands a blast scenario, real explosives may be used and then the results after the explosion may be studied, however the results may be hard to interpret due to the chaotic nature of the explosion. It is hard to repeat the same blast experiment twice, due to a number of factors such a new vehicle is needed and there is an uncertainty regarding the loading of the explosives. Therefore, drop-tower tests are often used as a substitute to explosions, since they also subject objects to transient shock forces that are easier to repeat. An example where a drop-tower test is used instead of an explicit blast scenario is to test vehicle seat blast mitigation capability, which has been done by the US Army. Drop-tower tests are also frequently used by the automobile industry to make comparisons with car crashes. [1]

Even though drop-tower tests are a common substitution for blast scenarios, it is still an expensive experiment to perform, since the whole vehicle can be destroyed during the drop. It is also inconvenient from a design viewpoint to have to produce a new model every time you want to investigate whether a new solution improves the design or not in regards to, for example, blast mitigation or crash safety. This is why blast simulations are conducted instead. Simulations have the added benefit that any modeled part of the vehicle can be monitored, at any given time during the simulated blast. The simulations are often performed by the means of finite element analysis software, abbreviated as FEA.

A finite element, FE, model of a vehicle requires a FE formulation of many different vehicle parts that have different mechanical properties and therefore behave differently. Plates for example, are often approximated as 2d-objects, since they are so thin in one dimension compared to the other two. Likewise, beams such as screws or bolts can be approximated as 1d-objects stretching from one node to another in the FE-model. When removing the other degrees of freedom, one often substitutes their behavior with some governing equation derived from solid mechanics. Because of the multitude of solid mechanical models, this has led to several different element formulations that can be applied for plate like or beam like elements that aim to preserve the real solid mechanical behavior in the finite element. [2] Formulating these elements often results in fewer elements for a part than if they would have been modeled solidly, resulting in a smaller mass matrix, and also allows for bigger time steps, all factors which decreases computation time.
At BAE Systems Hägglunds, LS-DYNA is one of the FE-solvers used to conduct blast simulations on vehicles. To simplify their simulations they have developed a simulation model for screw joints and bolted connections based upon the beam element instead of solid elements. This speeds up the modeling process since a vehicle can require hundreds, if not thousands, of screw joints to be modeled. The screw joint model with the beam element allows for a mostly automated process of adding the screws and applying pretension to the screw, a very important process for fastening the screw. The physical validity of the beam simulation model is still unclear, as no attempt to validate the model has been finished. A couple of years ago, in 2015, the company performed some tensile tests, to establish the yield strength of the screws, and some tests with a drop-tower rig, to monitor shock forces. The results were quite lackluster in the later instance as much time was spent developing the measurement method and then the project was canceled due to other priorities. This resulted in that further development of the simulation model and the validation of the model was abandoned.

### 1.2 Specific goals

The aim of this study is to validate the screw joint simulation model with beam elements used by BAE Systems. This is done via simulations of existing drop-tower tests that measured axial forces in screws. The old drop-tower tests were performed at 3 different drop-heights: 900, 1000 and 1100mm. All were performed with an attached external load of 18.7 kg. To perform a validation of the screw joint simulation model, a complete model of the drop-tower tests must be developed. Then simulations, run in LS-DYNA, must be conducted of the drop-tower tests with the screw joint simulation model using beam elements. Additionally, another screw joint simulation model including solid finite elements instead of beam elements is also presented for comparison with the experimental results and the simulation model we should validate.

### 1.3 Limitations

This study limits itself to investigate how the screw joint performs during transient tensile stresses, that is shock forces in the axial direction of the screw. The company has earlier performed drop-tower tests measuring the axial force in the screw, which will be compared with simulations. An analysis on the validity of the screw joint simulation model using beam elements is included in the discussion.
2 Theory and definitions

2.1 Material deformation

An object, such as a screw, which is not influenced by any external force may be considered to be in its rest shape. As long as no external force is applied, it will continue to look as it does forever. If the objects geometry remains unchanged while subjected to an external pressuring force, the object is rigid. A deformation that is temporary while the force is applied to the object is called elastic. If the deformation is permanent, it is called plastic. [3] Ductile materials, including most common steels for example, undergo both elastic and plastic deformation before rupturing. [4] The deformation is elastic up until the elastic limit, after which any deformation is permanent. [5]

2.1.1 Stress and strain

While analyzing material deformation due to forces, the terms stress and strain are frequently used to describe the force on and deformation of an object, respectively.

Consider a steel bar, that is loaded by two tensile forces at each of its ends and therefore is elongated:

![Diagram of a steel bar](image)

Figure 1: A steel bar of initial length $L_0$ is elongated to length $L$ by a force $F$ (red arrows) acting on the initial cross-sectional area $A_0$ at both ends of the bar. After the elongation, the cross-sectional area of the steel bar is $A$. The initial bar colored in gray and the elongated state in black.
The elongation $\delta$ is the difference between the deformed length, $L$, and the rest shape length, $L_0$. From this the *engineering strain* can be defined:

$$\epsilon_e = \frac{\delta}{L_0},$$

which is valid for small deformations $\delta \ll L_0$.

From the force on the end of the bar we can define the *engineering stress*:

$$\sigma_e = \frac{F}{A_0}$$

where $A_0$ is the cross-sectional area of the steel bar in its rest shape.

This is different from the *true stress*

$$\sigma_t = \frac{F}{A}$$

that is defined using the current force on the steel bar and its current cross-sectional area.

There is a *true strain* as well, which is defined as

$$\epsilon_t = \ln \frac{L}{L_0}$$

and is non-linear and valid for large deformations. [6] [7]

One might wonder why there is an engineering stress and a true stress. The engineering stress is a more pragmatic quantity than the true stress, since doing stress measurements on objects would be very hard if you also had to measure the cross-sectional area of the object simultaneously as it is deforming. The only parameter changing in the engineering stress is the applied force during a tensile test, which makes it much easier to measure. It is also worth noting that during small deformations the stresses should roughly be the same since $A \approx A_0$. We should also note that strain is itself a dimensionless unit. In the simple tensile-test it is a measure of the current length relative to the original length of the steel bar.

### 2.1.2 3-dimensional stress and Von Mises-stress

The earlier definitions of stress and strain comes from considering the tensile test, which is a special case when the deforming force acts along the main axis of the object. [3] Strain and stress are actually not defined only along the main axis of the deforming object, they are vectors dependent on the cross-sectional area that is considered. The orthogonal components of the stress vector are defined using the bending moments instead...
of the normal acting force on the area. [7] Of course, in Cartesian space we can create three independent cross-sectional planes through an object. This means that for each cross-sectional area, we can define three different stress vectors. If we would consider an infinitesimal cube, this would lead to 9-stress components for three different planes cutting the cube. This is the prelude that leads up to the Cauchy stress tensor theory. [8] The stress and strain in 3d is denoted as $\sigma_{ij}$ and $\epsilon_{ij}$, respectively. As both are tensor quantities, instead of looking at each of the components alone one creates a scalar quantity of the tensor. The von Mises-stress is defined as:

$$\sigma_v = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}},$$

where the indices 1,2,3 indicate stress along the principal axis. A physical interpretation of the von Mises-stress is the distortion energy failure theorem, that says that failure in the material occurs when the distortion energy in the actual case is more than the distortion energy in a simple uni-axial loaded case at the time of failure. [9] To determine whether an object undergo structural failure or not, one often monitors the von Mises-stress in the object. If the von Mises-stress exceeds a particular stress the object has started to deform permanently. This stress is called the yield stress, and is explained more in the next section. That the object starts to deform permanently does not necessarily mean that the object has failed or broken. Usually an object can undergo some plastic deformation before rupturing so therefore the plastic strain, $\epsilon_p$, can be monitored instead. It is calculated as the von Mises-stress, but with the stress components $\sigma_{ij}$ exchanged for the strain components $\epsilon_{ij}$. 
2.1.3 Stress and strain relationship

When describing a ductile material, for example the steel in screws, it is usually done by means of a stress-strain graph such as the one below in figure 2.

Figure 2: A stress-strain graph of a ductile material, showing a linear elastic region and two different stress-strain curves for the true stress and the engineering stress respectively in the plastic region (here the engineering stress is denoted as "conventional" or "nominal" stress). The elastic region ends at point B, equivalent to the elastic limit that is reached when the yield stress is applied to the material, where the material starts to yield and afterwards undergo plastic deformations for any further applied stress. The figure was originally found in [10].

Up until the elastic limit, it is very common that steels behave linearly elastic similar to Hooke’s law:

$$\epsilon_e = \frac{\sigma_e}{E}$$  (6)

which introduces the elastic modulus $E$, also known as Young’s modulus, that defines the stress-strain relation of the material in the elastic region. [7] The elastic limit is reached when the yield stress is applied to the material. After the yield stress, denoted as point B in figure 2 above, the material deforms permanently until rupture at point F in figure 2.

Materials may even be non-linear in the elastic region and therefore require a function that describes the elasticity. [3]
In the plastic region, it is not uncommon to make an approximation of the stress-strain relationship as a linear fit between the yield stress (point B in figure 2) and the ultimate stress (point E in figure 2). The slope of this linear fit is called the *tangent modulus*. When defining materials, it is usually referenced by giving the Young’s modulus, the yield stress, the ultimate stress and how much plastic strain the material withstands until rupture, which can be seen as the distance from point B to point F along the strain axis in figure 2.

### 2.2 Bolts and screws

The general consensus on the difference between a screw and a bolt is that a screw is *screwed* into something and a bolt is used to *bolt* something together. The main difference between them is the use of a nut for the bolt and a threaded hole for the screw. A schematic, showing the relevant parts of the screw and the difference between a screw and a bolt can be seen below in figure 3.

![Schematics showing screw and bolt](image)

**Figure 3**: Schematics showing a (a) screw and a (b) bolt, outlining the diameter \(d\) of the screw/bolt shank and the clamping length \(L_k\), which is shorter than the whole length of the shank (or the screw/bolt for that matter). The screw is inserted into a threaded hole, and the bolt is inserted through a hole and then fastened with a nut. The upper part of the bolt and screw is called the head. Washers are applied between the plates and the head and nut.

The *axial force* in the screw or the bolt, is the force along the main axis, outlined in figure 4. The shear force, would be the force orthogonal to the main axis that the screw experiences due to friction between the screw head and the plate, and the contact between the shank and the plate if the materials are forced into each other.
There is an established international standard for screws which introduce notation for
different sizes; M4, M6, M8, M10, M12 and so on with incrementing numbers that signify
larger diameters of the bolt shank. There are also different types of screws, 8.8 and 10.9
predominantly, that signify different material strengths.

A screw joint has a special function, as either a fastener or a sealer in case of a leak,
among other things, and when the screw is tightened the applied torque results in a static
force that remains. This force is called **pre-tension**, or **pre-load**, and is the reason the
screw is kept in place and that a bolt actually *bolts* two things together. [11] [12]

In this study, the specific screw studied is an M8 screw of type 8.8. Some material and
dimensional properties of the screw is given in table 1:
Table 1: Table of material and dimensional properties of an M8 screw of type 8.8

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield stress [MPa]</td>
<td>640</td>
</tr>
<tr>
<td>Ultimate stress [MPa]</td>
<td>800</td>
</tr>
<tr>
<td>Elongation at rupture $A_5$ [%]</td>
<td>12</td>
</tr>
<tr>
<td>Tangent modulus [MPa]</td>
<td>2347</td>
</tr>
<tr>
<td>Pre-tension force [N]</td>
<td>17100</td>
</tr>
<tr>
<td>Rupture force axial/shear [N]</td>
<td>29280</td>
</tr>
<tr>
<td>Screw diameter [mm]</td>
<td>6.83</td>
</tr>
</tbody>
</table>

2.3 Finite Element Analysis

Finite element analysis (FEA), also known as the finite element method (FEM), is a general numerical method used to solve partial differential equations approximately. The method is characterized by both the chosen solver algorithms and whatever post-processing procedures there are, but also by more fundamental mathematical choices as for how to create a variational formulation and how the discretization is performed. Although the method can be used analytically, it is most prominently used to simulate problems related to heat transfer, fluid dynamics, electromagnetism, structural analysis or other things in the field of computational physics. [2] [13] In this study a FEM solver called LS-DYNA will be used to simulate a drop-tower test. The full finite element formulation used in LS-DYNA will not be derived in this section (I humbly refer to the LS-DYNA Theory manual instead), but the governing equations of the LS-DYNA formulations are presented. Also a quick mathematical overview of a finite element method is given, intended to give some basic understanding of the method for the uninitiated reader.

2.3.1 The Finite Element Method

The following section is a derivation of the finite element method, referenced from the book *The Finite Element Method* [2]. Consider Poisson’s equation

\[-\nabla^2 u = f, \text{ in } \Omega\]

\[u = 0, \text{ on } \partial \Omega\]  \hspace{1cm} (7)

where $\Omega$ is the domain, $\partial \Omega$ is the boundary of the domain, $\nabla$ is the operator $(\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z})$, and $u$ is the unknown solution to the problem. To create the finite element approximation of $u$, one must first create the variational formulation (also known as the weak form) of equations (7) and (8). After this, a triangulation needs to be introduced so that the continuous problem only is solved in discrete places of the domain. To derive a variational formulation, a test function $v$ that that is assumed to vanish on the boundary $\partial \Omega$ and Green’s theorem is used. First Poisson’s equation is multiplied with the test function.
and then an integration on the whole domain is coupled with Green’s theorem:

\[
\int_{\Omega} f \, dx = -\int_{\Omega} \nabla^2 uv \, dx \\
= \int_{\Omega} \nabla u \nabla v \, dx - \int_{\partial \Omega} n \cdot \nabla uv \, ds \\
= \int_{\Omega} \nabla u \nabla v \, dx.
\]

This removes the second derivative of \( u \) from the problem formulation. In this specific case, the contribution from the boundary condition also disappears because of the assumption on the test function \( v = 0 \) on \( \partial \Omega \). After this, a suitable space that the variational form is valid on is introduced:

\[
V = \{ v : (\int_{\Omega} v^2 \, dx)^{1/2} + (\int_{\Omega} (\nabla v)^2 \, dx)^{1/2} < \infty \} \\
V_0 = \{ v \in V : v|_{\partial \Omega} = 0 \}.
\]

\( V \) is a so called *Hilbert space* and \( V_0 \) is the same space where all the functions are zero on the boundary.

The problem, first described by equations (7) and (8), can now be described by the variational form: find \( u \in V_0 \) such that

\[
\int_{\Omega} \nabla u \nabla v \, dx = \int_{\Omega} f \, v \, dx, \forall v \in V_0.
\]

The problem posited by (14) is still continuous, which is a problem when trying to solve something by means of a computer. The variational formulation (14) has a notable difference from the original Poisson’s equation (7): It only holds in the space \( V_0 \) where the test function \( v \) is defined. From here, the formulation of the finite element approximation of (14) can be made. A triangulation on the domain \( \Omega \), otherwise called a mesh, needs to be introduced. The triangulation defines the discrete subspace \( V_h \in V \), that contains some of, but not all, nodes in \( V \) and the points contained depend upon the kind of triangulation. An example of a triangulation would be a collection of triangles on a 2d surface: The surface has infinitely many points but there is only a discrete number of points contained by the corners of the triangles. Whether triangles, squares or in the 3d-case tetrahedrons, cubes or some other shape is used, defines the subspace \( V_h \) and therefore the finite element method. To also satisfy the boundary conditions, the subspace \( V_{h,0} \subset V_h \) is defined:

\[
V_{h,0} = \{ v \in V_h : v|_{\partial \Omega} = 0 \}
\]

To obtain the finite element approximation, the space \( V_0 \) is replaced with \( V_{h,0} \) in the variational formulation (14): find \( u_h \in V_{h,0} \) such that

\[
\int_{\Omega} \nabla u_h \nabla v \, dx = \int_{\Omega} f \, v \, dx, \forall v \in V_{h,0}.
\]
This is discrete in theory, but to perform the discretization we need to choose some kind of interpolation functions to define the values in the discrete points. A common choice are the so called hat or tent functions, that span the discrete space $V_{h,0}$. The basis of $V_{h,0}$ is $\{\phi_i\}_{i=1}^{n}$ that consists of the hat or tent functions associated with the $n_i$ interior nodes provided by the triangulation such that $\phi_i \phi_j = \delta_{ij}$. The functions $\phi_i$ fulfill the requirements of the test function $v \in V_{h,0}$, and therefore the finite element approximation (16) is valid to write as

$$\int_{\Omega} \nabla u_h \nabla \phi_i dx = \int_{\Omega} f \phi_i dx, \ i = 1, 2, ..., n_i. \quad (17)$$

The function $u_h$ can then be represented by the linear combination

$$u_h = \sum_{j=1}^{n_i} \xi_j \phi_j \quad (18)$$

where $\xi_j$ are the unknown nodal values of $u$ in the interior nodes $n_i$.

Using the linear combination given by (18) in (17), the following linear system written in matrix form can be derived:

$$A \xi = b, \quad (19)$$

where

$$A_{ij} = \int_{\Omega} \nabla \phi_j \cdot \nabla \phi_i, \ i, j = 1, 2, ..., n_i \quad (20)$$

and

$$b_i = \int_{\Omega} f \phi_i, \ i = 1, 2, ..., n_i. \quad (21)$$

When solved, this linear system results in the finite element solution of the Poission problem stated in (7) and (8).

The error, between the finite element solution and the real solution, is dependent on the mesh size $h$.

### 2.3.2 Richardson Extrapolation

Many numerical methods, such as schemes for numerical differentiation and integration and also the FEM itself, are dependant on a "step" size $h$. In general for such methods, we can say that they have an infinite sequence of computed values $\{A_n\} = A(h_n)$ that converge to $A$, the limit, when $n \to \infty$ or the step size $h \to 0$. If the limit $A$ exists, we can say that the error is $A(h) - A = O(h^p)$. In a numerical differentiation scheme, letting the step size $h$ approach 0 is equivalent to approaching the real derivative, however there are some numerical problems with letting $h$ tend to zero in a computer (because of the truncation error). [14] In the FEM, if the mesh size $h$ was allowed to approach
0 we would have a basically "continuous element", which should be equivalent to the analytical solution.

The Richardson Extrapolation

\[
R(h, p) = \frac{p^k A(h) - A(p, h)}{p^k - 1}
\]  \hspace{1cm} (22)

has an error on the order of \(O(h^{p+1})\), which means that the error is smaller for the Richardson Extrapolation than for the initial approximation \(A(h)\). Together with two approximated values at different step sizes \(h\), the limit \(A\) can be determined:

\[
A = \frac{t^{p_0} A \left( \frac{h}{t} \right) - A(h)}{t^{p_0} - 1} + O(h^{p_1}).
\]  \hspace{1cm} (23)

This expression can yield an approximation of \(A\) with an error order \(O(h^{p_1})\), if the convergence order \(p_0\) is known for the method. If the convergence order is not known, it can itself be approximated if the approximations \(A(h), A(h/t)\) and \(A(h/s)\) are known, since:

\[
A = \frac{t^{p_0} A \left( \frac{h}{t} \right) - A(h)}{t^{p_0} - 1} = \frac{s^{p_0} A \left( \frac{h}{s} \right) - A(h)}{s^{p_0} - 1} + O(h^{p_1})
\]  \hspace{1cm} (24)

leads to the approximate relationship

\[
A \left( \frac{h}{t} \right) + \frac{A \left( \frac{h}{t} \right) - A(h)}{t^{p_0} - 1} \approx A \left( \frac{h}{s} \right) + \frac{A \left( \frac{h}{s} \right) - A(h)}{s^{p_0} - 1}.
\]  \hspace{1cm} (25)

This can be solved numerically to find \(p_0\), and then the Richardson Extrapolated value can be calculated by equation (23). \[15\] An easy way to do this, would be to choose a sufficiently big initial step size \(h\) and then compute \(A(h), A(h/t)\) and \(A(h/s)\) using \(t = 2\) and \(s = 4\) in the above equations.

2.4 LS-DYNA

LS-DYNA is a terminal based FE-solver with origins back to the 1970s. Nowadays assembly is most often done through the use of computer aided engineering programs. An example is ANSA, that have 3d modeling properties and can apply different conditions on a geometric model. The output to LS-DYNA comes in notepad editable text files with the .k or .key extension. These files contains the instructions to the solver in the form of keywords and cards, which hold conditions of the physical conditions applied and the information about the meshed geometry. It is possible to create modular parts in separate text files of a FE-model that later are assembled together in another, main, text file.
2.4.1 Governing equations for LS-DYNA

From the LS-DYNA Theory manual the governing equation of LS-DYNA is the momentum equation

$$\sigma_{ij,j} + \rho f_i = \rho \ddot{x}_i$$  \hspace{1cm} (26)

where $\sigma_{ij}$ is the Cauchy stress tensor, $\rho$ is the current density, $f$ is the body force density, $\ddot{x}$ is acceleration. Since $\sigma_{ij}$ is a tensor indicial notation is used where the comma denotes covariant differentiation. The derivation of equation (26) to a variational formulation and then finite element approximation can be found in the LS-DYNA Theory manual, together with the boundary conditions for equation (26).

2.4.2 Timestep control with mass scaling method in LS-DYNA

LS-DYNA uses an explicit time integration scheme (although it can use implicit methods as well), and thus the minimal time step in each iteration for a stable solution is dictated by the so called Courant-Lewy-Friedrich or CFL condition. \[16\] The creators have introduced their own, CFL-like, conditions (I guess they are bounded in the lower regime) that apply for their specific code and can be found in the LS-DYNA Theory manual.

For a solid shell element, the CFL condition in LS-DYNA is given by

$$\Delta t_e = \frac{v_e}{c A_{e\text{max}}}$$  \hspace{1cm} (27)

where $v_e$ is the volume of the element, $A_{e\text{max}}$ is the area of the largest side of the element and $c$ is the plane stress sound speed given by

$$c = \sqrt{\frac{E}{\rho (1 - v^2)}}$$  \hspace{1cm} (28)

determined from Young’s modulus $E$, the material density $\rho$ and the characteristic velocity $v$.

Similar to equation (27), the critical time step of a shell element is given by

$$\Delta t_e = \frac{L_s}{c}$$  \hspace{1cm} (29)

where $L_s$ is the characteristic length of the shell element. What is important in both equation (27) and (29) is the factor $1/c$. This factor effectively puts $\rho$ in the numerator and thus the material density will determine the critical time step.

A method in LS-DYNA to deal with long computation time is to simply mass scale critical elements that otherwise result in very small time steps: we call this mass scaling and it is used in this study. In LS-DYNA this is implemented via the *CONTROL_TIMESTEP-keyword and choosing a mass scale factor, which is the critical
time step that we allow in the simulation. In any element where the critical time step
given by equation (27) or (29) is smaller than the time step we allow, the density is scaled
appropriately to make $\Delta t_e$ larger. This of course adds mass to the model.
3 Methodology

3.1 Experimental set up

As the experiments already have been performed we can’t modify the experimental setup in this work. The setup is however important for the development of the complete FE model, so a short overview of the drop-tower rig is presented in this section.

The drop-tower is a structure with a big metal plate, called a sled, with attached wheels that allows it to move vertically along two supporting metal rails. It has a lower support structure consisting of metal beams that allow it to stand steadily on the ground, where the metal rails are attached. The two metal rails are connected with a metal beam at the very top. The sled is lifted to the desired drop height with an overhead crane. In figure 5 below, a close view of the drop-tower rig from another experiment can be seen.

Figure 5: A close view of the drop-tower rig from former tests performed to investigate the shock mitigation of vehicle seats. A crash test dummy is seated in a vehicle seat which in turn is attached to the drop-towers sled. The lower part and the rails of the support structure can be seen.
The same drop-tower was used for the axial shock measurements on the screws. To attach the measurement equipment to the drop tower, a holder was constructed that would allow for measurement of the axial and shear force on the screw. The measurements themselves were performed with strain gauges on a console, that could be attached with the screw specific for the study to the holder. The measurement setup can be seen below in figure 6, outlining the holder, console and the strain gauges used for the measurement, as well as the position of the screw that was the subject of the experiments.

Figure 6: The measurement setup for the drop-tower tests. The holder is attached to the sled in the background. The holder and console are attached to each other by one M8 8.8 screw that is the subject of the experiments. On either side of the console there are strain gauges attached that are calibrated to measure the force in the screw. In the middle of the console is an accelerometer which unfortunately did not work during their testing, and below the console are the external weights attached which were used to increase the load on the screw.
3.2 Simulation set up

To simulate the conditions of the axial shock test in the drop-tower rig, a complete FE model of the drop-tower rig was initially implemented, where the ground support structure and some small edges on the holder was ignored. Contact forces were implemented to make the parts interact with each other and the sled was given an initial velocity dependent on the drop height as $v_{\text{ini}} = \sqrt{2gh}$. The results from this model showed that almost none of the kinetic energy had been absorbed in the impact that basically was modeled like an elastic collision, even though friction was included in the contact forces. This was probably due to that most of the ground support structure and the floor beneath the rig was ignored in the FE model. These parts, that were not included in the model, would have absorbed more energy in the impact. It soon became apparent however, that the free-fall velocity model already had been discarded in the previous simulation work by the company. Instead they had approximated the sleds velocity from one high speed camera film of one singular drop test. This approximation was also fitted to other drop heights, by comparing the initial velocity from the high speed camera film to the initial velocity given by $v_{\text{ini}} = \sqrt{2gh}$. This approximated velocity curve was then prescribed to the bottom of the sled, yielding more realistic accelerations on the sled and screw than when simulating an impact with contact forces and a free-fall velocity model.

To improve on this model that uses velocity curves, new high speed camera films were created for several drop-tests in the drop height range of 700-1200mm with 50mm increments. In the new films, there were tracking markers attached to the bottom of the sled. The sled velocity, which could be found from the tracking of these markers in the high speed camera film, were then used as simulation input on the bottom of the sled.

3.2.1 Velocity tracking

The production of the velocity curves used as simulation input took three steps. First the sled was dropped from a specific height and the impact was filmed with a high speed camera. Next the markers on the bottom of the sled was tracked with a program called META from the film. The result of the tracking was then differentiated to produce velocity curves, and lastly the curves were trimmed and shifted in time so that the first value corresponded to $t = 0$ in ANSA. In figure 7, three example stills from the high speed camera film of a 1000mm drop can be seen.
Figure 7: Frames from the high speed camera film of the 1000mm drop. The frames encapsulate the bottom of the sled, showing one frame (a) right before the impact, one (b) during the impact and one (c) right after the impact. Three tracking markers are visible, where the lower one is stationary and the two upper ones move during the drop.

In figure 8 a screenshot of the last video frame after the markers had been tracked in ANSA can be seen. Note that the tracking procedure produces position data of the markers on the available bitmap per frame, not their velocity.

Figure 8: Screenshot showing the markers (red squares) and the positions (green lines) tracked on the bitmap by META.

From the bitmap positions of both markers in figure 8, an average position of the bitmap position can be created. The average bitmap position is then scaled with the known distance between the markers. Using the FPS of the high speed camera, the time for each position can be determined. Forward differentiation of this position data results in velocity data. The data is then exported as curves for LS-DYNA with the *DEFINE_CURVE-keyword in separate files, as seen in figure 9.
Figure 9: Complete velocity curve produced from the high speed camera film of the 1000mm drop.

Before use, the complete curves were imported to ANSA and trimmed to only be defined between their peak-to-peak values as shown in figure 10.
Figure 10: Trimmed velocity curve produced from the high speed camera film of the 1000mm drop. Any velocities before the maximum negative velocity, at \( t = 67.2 \) ms, and after the maximum positive velocity, at \( t = 80 \) ms, has been removed. The curve has also been shifted to begin at at \( t = 0 \) ms.

### 3.2.2 FE Model

In figure 11 the complete FE-model of the drop-tower can be seen, including the sled and the support structure. The drop-tower rig is modeled with shell elements, and stationary boundary conditions are used on the bottom of the drop-tower rig.
Figure 11: The full FE-model of the drop-tower.

A closer view of the holder and the console can be seen in figure 12, as they are the most relevant parts of the model. The holder is modeled with shell elements and the console is modeled with solid elements. The studs (with a threaded hole for the screw joint) are modeled as rigid bodies and are attached to the holder shell via rigid connections.

Figure 12: A closer view, showing the console (orange), the holder (yellow), the studs on the holder (transparent) and the sled of the drop-tower (green).

The joint between the holder and the console, is described in the next section as this is where the different screw joint simulation models are used. Some geometric simplifications were made on the holder model, where small edges were removed that otherwise would result in small elements. The whole holder was presumed to be of the same material when in reality it is made up of several different steel types with different material strengths due to their thicknesses. This simplification is done under the assumption that
the parts do not start to yield due to plasticity during the simulation and the experiment. The shell elements used to model the holder uses the combined thicknesses of the different steel types, but disregards the welding between them, so two plates welded together are modeled as one thicker plate. The connection between the holder and the sled are made with singular beam elements of the same kind that is used in the screw joint beam simulation model. This allows for some flexibility in the holders movement relative to the sled. The alternative would be rigid connections between nodes on the sled and the holder, which would overestimate the load transfer between the parts.

Contact forces were defined with a static coefficient of friction $FS = 0.15$ with the keyword *GENERAL_AUTOMATIC_SINGLE_SURFACE*. This seemed to be a general friction coefficient that the department used for steel-steel friction. They were defined on sets of parts, so that the contact force only was defined between parts in the same set. One set included the drop-tower support structure, the sled and the holder. The other set included the studs on the holder, where the threaded hole for insertion of the screw was, the console and the screw head.

The extra weight used as applied load on the screw, was modeled via the *ELEMENT_MASS_NODE_SET*-keycard defined on the lower face of the console as shown below in figure 13.

![Figure 13: Figure of the console (green) attached to the holder (transparent blue) showing where the external load was applied on the bottom surface of the console. The orange dots are the nodes on the bottom of the surface where the extra weight is applied.](image)
3.3 Screw joint models

The screw joint simulation model to be validated, is the first one presented in this section. The second one is a variant of that simulation model using several solid elements for the screw shank instead of a beam element. The beam simulation model used at the computations department of BAE is very similar to the model presented by Uwe Sonnenschein in his article *Modeling of bolts under dynamic loads* [18]. It also has similarities with the models proposed by Shailesh Narkhede et al [19] and by Michalis Hadijoannou et al [20]. All their models use different kinds of beam elements in their simulation models, and they also perform simulations with solid element models of the screw joint for comparison. The biggest difference is that these authors investigate how their simulation models using beam elements behave during slow tensile tests and if the modeled application of pre-tension is physically correct. They do not investigate how the models behave during shock forces, or if there is any difference of the model behavior between shock forces and slow forces. They get good correlation between their simulated results, both between beam and solid models, and also between their measured tensile tests and their simulations. All in all, it should be expected that the BAE model behaves similar to these authors models during slow tensile tests but there can be a difference in how the model behaves during shock forces since the contact algorithms may behave different during short time impulses.

The only contact force defined for the screw joint simulation models is the one between the so-called null beams and the screw shank, either modeled as one beam element or several solid elements. The null beams are defined on the perimeters of the console hole where the screw is inserted through. They are supposed to deal with shear forces that should appear due to contact interaction between the screw shank and the console hole walls. This is not defined explicitly since the model with the null beams is used in the simulation model. Instead the interaction between the null beams and the screw shank models the shear forces. This is defined with a static friction coefficient $FS=0.15$ and the keyword *CONTACT_AUTOMATIC_GENERAL_ID*. This shear force doesn’t matter much in this work though, since we are performing simulations of drop-tower tests that mainly subject the screw to axial and not shear forces.

3.3.1 Beam simulation model

Our beam simulation model consists of three parts; the screw shank, the screw head and the null beams. The material properties such as the Young’s modulus, tangent modulus and rupture point of the screw can be derived from table 1 together with the relations spoken about in section 2.2. The screw shank was modeled as a beam element of type **ELFORM 9 S-W**, which only is defined for the material **MAT_SPOTWELD**. The **ELFORM 9 S-W** was originally implemented in LS-DYNA to model spot-welds. The material **MAT_SPOTWELD** is a plastic-kinematic material, which means that it has a defined elastic region and plastic region, given by a bi-linear approximation. The screw
head is just a circular thin disc, modeled with shell elements and of the \texttt{MAT\_RIGID} material. The material \texttt{MAT\_RIGID} means the whole head will be treated as a rigid body. The screw head can’t deform and any force applied to the head anywhere is applied directly to the whole rigid body. The null beams are placed along the perimeter of the hole on the console where the screw is inserted through. They are made of the \texttt{ELFORM 1} type beam elements and of material \texttt{MAT\_NULL}, where the later means that they have no effective mass in the simulation.

The beam simulation model is shown in figure 14, outlining the different parts of the simulation model and showing where the screw is inserted in the full FE-model.

![Figure 14: The screw joint simulation model using beam elements (a) alone and (b) inserted with the surrounding (transparent) parts.](image)

The beam element is simply a line between two nodes, connecting the screw head with the stud on top in figure 14b with a rigid connection. It has a set diameter, equal to the one given in table 1, which defines the circumference around the beam element that the beam element can interact with.

An upside to the \texttt{ELFORM 9 S-W} type elements is that pre-tension on the screw can easily be applied with the \texttt{*INITIAL\_AXIAL\_FORCE\_BEAM}-keyword, which takes the desired pre-tension force and a load curve (which allows for ramping the pretension slowly if that is necessary). The pre-tension is applied in the initialization stage of the solver. The force in the screw body can be analyzed through the \texttt{*DATABASE\_SWFORC}-keyword, that enables us to monitor reaction forces, spotweld length, moment and torsion for all beam elements of type \texttt{ELFORM 9 S-W}.

### 3.3.2 Solid simulation model

A similar screw joint simulation model that instead of a beam element just models the whole screw shank solidly was also used. The same materials are used in both models. The solid simulation model is illustrated in figure 15.
The pretension for the solid model is a bit trickier to apply. In this case we made use of the *INITIAL_STRESS_SECTION-keyword. This approach requires us to define a cross section plane in the middle of the screw body using the *DATABASE_CROSS_SECTION_PLANE-keyword. The pretension is applied as stress, that is $[kN/mm^2]$ in our case, with a load curve that as well allow for ramping of the pretension if that is necessary. The force in the screw body can be analyzed through the *DATABASE_SECFORC-keyword, that uses the same cross section defined to apply the pretension. A drawback to this method, is that the pre-tension is only applied to the elements closest to the defined cross-section plane, not the whole screw shank. In the beam element case, the pre-tension is defined inside a single element.
4 Pre-study

Before the final simulations are run, the model is investigated in a pre-study in which a mesh convergence study and a study of the mass scaling factor are performed. It is important to know whether the quality of the mesh or the choice of the mass scale factor affects the end results.

4.1 Mesh convergence study

Because the majority of the model only consists of the drop-towers sled and the support structure, the mesh was investigated by refining the mesh locally around the screw joint. The surrounding mesh, on the sled and the support structure, was deemed to be precise enough to produce confident results. Instead the focus was how the local mesh around the screw joint including the holder and the console influenced the result, as these are the parts that intimately interact with the screw joint simulation model. An example of the refined mesh can be seen in figure 16 below.

![Figure 16: An example of the mesh surrounding the holder, console and the part of the sled closest to the holder. The finer mesh is the area where the local mesh was refined in the local mesh convergence study.](image)

To study the local mesh convergence in this area we choose to monitor the maximum axial force in the screw during a drop-tower test, both for simulation model using beam and solid elements. A velocity curve from a 900mm drop was prescribed as motion to the sled, and an external load of 18.7 kg was added on the bottom of the console.

The mesh was created by setting limits on the edges of the model, edges of the holder and console, that the mesh had to form after. The limits were set to 20, 10 and 5 mm
(mesh size $h$, $h/2$ and $h/4$). Otherwise than setting limits to the mesh, the mesh was automatically created by the program meshing tools in ANSA. Initially, a further mesh size refinement was considered at 2.5mm ($h/8$), however this resulted in an estimated computation time of about 84 hours and was abandoned. Instead of a fourth data point, Richardson extrapolation was used to extrapolate the maximum force in the screw if the mesh size approached the zero limit. The convergence order of the method was approximated for the beam and solid results respectively with equation (25) and the extrapolation with equation (23). The results, with the Richardson extrapolated values, is presented in figure 17.

Figure 17: The maximum force in the screw for the different mesh sizes corresponding to $h = 20mm$, $h = 10mm$, $h = 5mm$ and the extrapolated value for $h \approx 0mm$. The upper curve shows the result of the screw joint simulation model using beam elements, the lower curve shows the result using solid elements.

From the relation $A - A(h) = O(h^p)$, explained in section 2.3.2, we can calculate the error between the extrapolated value and the simulated values:
Table 2: The relative local mesh dependent error $\frac{O(h)}{A(h)}$, calculated for the screw joint simulation model using beam elements and solid elements.

<table>
<thead>
<tr>
<th>Mesh size [mm]</th>
<th>$h = 5$</th>
<th>$h = 10$</th>
<th>$h = 20$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\frac{O(h)_{Beam}}{A(h)}$ [%]</td>
<td>1.7</td>
<td>3.7</td>
<td>8.3</td>
</tr>
<tr>
<td>$\frac{O(h)_{Solid}}{A(h)}$ [%]</td>
<td>0.6</td>
<td>1.5</td>
<td>4</td>
</tr>
</tbody>
</table>

Only the maximum force in the screw was considered for the convergence study. However if we study the axial force in the screw over time as in figure 18 below, we can see that the time span of the shock force pulse changed slightly for the different mesh sizes.

![Figure 18](a) (b)

Figure 18: The axial force vs time in the screw joint, for the simulation model using beam elements in (a) and using solid elements in (b). For both the simulation cases, the pulses become wider in time and have a lower peak force for smaller meshes.

This is just a very slight change. It could have been interesting to study a parameter as the impulse of the shock instead of just the maximum force, since that would take into account the change in time as well.

Interestingly enough, the computation time did not vary much due to the local mesh difference (other than for the one abandoned), all being at around 50 minutes.

4.2 Study of the mass scale factor

The mass scale factor is given in a dimension of time and decides which elements that are scaled to a higher density. The same drop-test was used as in the local mesh convergence study. Simulations were run for factor values of $[1e-4, 3e-4, 6e-4, 8e-4, 1e-3]$ and we used the local mesh with size $h = 5$mm. The factor affects computation time,
which can be seen in figure 19 below, and the decreasing trend can be seen for both the simulation model using beam and solid elements, respectively.

![Computation Time vs Mass Scale Factor](image)

**Figure 19:** Computation time vs Mass scale factor, ie the smallest allowed time step in the model. The computation time decreases as the mass scale factor increases for both cases.

Clearly the factor affects computation time, it even seems to converge to some ultimate computational time (probably bounded by computational power). Unfortunately, contact forces implemented in LS-DYNA start to behave unstable for slightly bigger time steps than the minimal we allowed for now (1e-3), so we will not try to decrease the computation time further. In the figure, the minimum computation time was 9 and 12 minutes for the beam and solid simulation model, respectively.

As the time step control method adds more mass for bigger factors it can affect the result. Most obviously this happens when the mass is added to the console and thus the load on the screw increases, which in turn results in an increasing maximum force on the screw. This can be seen in figure 20 below.
Figure 20: The maximum force in the screw versus the mass scale factor, for the screw joint simulation model using beam elements and solid elements. The maximum force on the screw increases as the mass scale factor increases for both cases.

The fact that the force does not increase with the lowest mass scale factors for the beam simulation model is probably due to that the beam elements density itself is not increased. For the solid model on the other hand, the elements that make up the screw shank have their density scaled. Overall we can see an increasing trend, with a maximum difference of about 2.5 kN between the minimum and maximum scale factor. For small mass scale factors the only elements that will be mass scaled are the ones surrounding the screw joint; where the most level of detail is on the mesh (remember that a quite coarse mesh is used elsewhere). When we increase the mass scale factor, more and more elements around the screw joint (most importantly, below the screw joint) are mass scaled resulting in an increased load applied to the screw itself. Thus the force on the screw increases.

4.3 Conclusion

We choose the finest of the meshes we tried, with mesh size $h = 5mm$. The monitored maximum force doesn’t differ that much from the Richardson extrapolated values, for both simulation models, and there wasn’t any significant change in computation time that could outweigh the use of the finest mesh. The error due to the mesh is different for the beam model and the solid model. Using the biggest of the errors, 1.7 %, gives the biggest error bars to use in our model due to the mesh.

The optimal smallest time step seems to be to bound them at $\Delta t = 6e^{-4}$. The maximum
force does not change more than 0.5 kN for both simulation models there, which is less than 1.5% of the force computed when we allow for the smallest time step in the study. It also has a short computation time, at around 15-16 minutes.

All together, one should expect that the error in the model is approximately 3.5% of the simulation values, although it should be lower for the solid model.
5 Results

From the experiments performed two years ago, only four cases were usable for this study. Most of the drop-tower tests tried to measure the axial force in the screw with an accelerometer attached to the console, but it was broken. Several other tests were also slow tensile tests used to calibrate the strain gauges. The other drop-tower tests were only used to document which cases led to screw joint failure or not. The four usable cases come from the drop-tower tests that were performed with the calibrated strain gauges, as explained in section 3.1. The tests were all performed on a screw of size M8 type 8.8, with an external load of 18.7 kg, and performed from different drop heights:

- **Case 28**: dropped from a height of 900mm
- **Case 29**: dropped from a height of 1100mm, resulted in screw failure
- **Case 30**: dropped from a height of 1000mm
- **Case 31**: dropped from a height of 900mm.

The two cases 28 and 31 have the same drop height, so they are modeled in the same way. This permits a comparison between these two experimental results and we can calculate a standard deviation. However, as the standard deviation is calculated only from two values, we can not establish any confidence bands on the deviation. During case 29, for 1100mm, the screw broke during the drop so the data have been cut where measurement noise appeared (due to the console being released and then bouncing). For both case 29 and 30, the results are simply presented and compared to the simulated results.

5.1 Case 28 and 31, measurements and simulations

In figure 21 the experimental measurements of case 28 and case 31 can be seen together with the simulations of a drop-tower test performed from a drop height of 900mm.
Figure 21: Results figure for the 900mm height drop-tower test case. Experimental data correspond to case 28 (dashed line) and case 31 (dot-dashed line), and the simulated data is using the beam simulation model (diamond markers) and solid simulation model (circular markers). The maximum force of each case can be seen in table 3.

The simulated values for the beam and solid model are similar to each other, differing about 3% from each other, and are in between the two different measurements for this case. They are above the mean value of the maximum axial force, as can be seen in table 3. Note the big difference between the simulations and measurements in the beginning: This is because the pre-tension is clearly visible in the simulations but not visible in the measurements at all, since the strain gauges are not calibrated to measure the pre-tension. Remember that the pre-tension is a static force in the screw, so the measurements only measure the impulse force during the impact.

The maximum axial force on the screw in figure 21 was determined for each curve, and are presented below in table 3 together with the average of the measured maximum force and its standard deviation.
Table 3: Maximum axial force in the screw for the simulated and experimental cases for the 900mm drop case, including the average of the two experimental values and its standard deviation.

<table>
<thead>
<tr>
<th>$F_{max}$ for</th>
<th>Beam</th>
<th>Solid</th>
<th>Case 28</th>
<th>Case 31</th>
<th>Average ± std</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value [kN]</td>
<td>35.4</td>
<td>34.5</td>
<td>30.2</td>
<td>36.9</td>
<td>33.5 ± 4.8</td>
</tr>
</tbody>
</table>

5.2 Case 29, measurements and simulations

The case for the drop height of 1100mm only had one experimental measurement, in which the screw also broke during testing so that high frequency noise appeared in the measurement. The part with high frequency noise has been discarded in the figure below, that presents the simulated and experimental shock force on the screw.

![Figure 22: Axial shock force on the screw, measured and simulated for the 1100mm drop case. The measured values (dashed line) correspond to case 29, where screw failure was observed. Therefore, the end of the measurement containing high frequency noise has been discarded. The simulated cases are for the screw joint simulation model using beam elements (diamond markers) and solid elements (circle markers). The maximum force of each case can be seen in table 4.](image-url)
Once again, the simulations are similar to each other and this time they are also on par with the measurements. If one looks at the maximum axial force in the screw, tabulated in table 4, the measured maximum axial force is between the two simulations.

Table 4: Maximum axial force in the screw in the simulated and experimental cases for the 1100mm drop case.

<table>
<thead>
<tr>
<th>$F_{\text{max}}$ for</th>
<th>Beam</th>
<th>Solid</th>
<th>Case 29</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value [kN]</td>
<td>36.8</td>
<td>35.6</td>
<td>35.9</td>
</tr>
</tbody>
</table>

5.3 Case 30, measurements and simulations

The final validation case was a drop-test from 1000mm drop height. The results can be seen below in figure 23.

![Figure 23: Axial shock force on the screw, measured and simulated for the 1000mm drop case. The measured values (dashed line) correspond to the measured force in case 30. The simulated cases are for the screw joint simulation model using beam elements (diamond markers) and solid elements (circle markers). The maximum force of each case can be seen in table 5.](image)
Again, the simulations are similar to each other but for this case the measured force is almost always lower than the simulations. The maximum axial force in the screw for the measurement and the simulations can be seen below in table 5.

Table 5: Axial shock force on the screw, measured and simulated for the 1000mm drop case.

<table>
<thead>
<tr>
<th>$F_{max}$ for</th>
<th>Beam</th>
<th>Solid</th>
<th>Case 30</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value [kN]</td>
<td>36.8</td>
<td>35.6</td>
<td>32.5</td>
</tr>
</tbody>
</table>

5.4 Plastic strain

If a screw ruptures or not is determined by monitoring the plastic strain in the simulated screw. From the datasheet, table 1, an M8 type 8.8 screw ruptures at 12 % engineering plastic strain ($A_3$). This can be converted to 11% $\epsilon_p$. The plastic strain in the simulations is summarized in table 6 below:

Table 6: Table of the simulation time when $\epsilon_p$ in the screw exceeds 11 % and the maximum $\epsilon_p$ in the screws.

<table>
<thead>
<tr>
<th>Time when $\epsilon_p&gt;$11%</th>
<th>Max $\epsilon_p$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Beam</td>
</tr>
<tr>
<td>Case 28/31</td>
<td>4.5 ms (11.7%)</td>
</tr>
<tr>
<td>Case 29</td>
<td>3.5 ms (11.8%)</td>
</tr>
<tr>
<td>Case 30</td>
<td>5 ms (11.4%)</td>
</tr>
</tbody>
</table>

All the beam simulations of the plastic strain seem reasonable, but the plastic strains from the solid model are way out of line. This is probably because of a fault in the analysis method. When this analysis method is used for the solid model the plastic strain in the element with the maximum plastic strain will be reported. For the beam model this is not the case, since there only is one element - the beam element. In the case of the solid model there will instead be local elements that reach high plastic strain and therefore skew the analysis. If one element is "broken" does not mean that the whole screw has broken. Further, that the plastic strains are so high might be because of a problem with the time stepping. If an external force is applied to the whole screw, it is absorbed in the first solid elements closest to the application of the force. When the time step is not small enough, the force will not propagate to the next element in a realistic manner. Even though the added mass through the mass scaling is not very relevant for the maximum axial force in the screw, maybe it can influence the development of plastic strain in the solid model in a negative way (as big time steps will make the propagation of the force through the elements less realistic).
6 Discussion

To begin we should consider the simulation model error derived in the pre-study, sec 4. The model error was approximately 3.5% of the simulation values, and if one takes this to 4% instead none of the simulation results have an error greater than 1.5 kN. The difference between the solid and beam model is between 0.9-1.2 kN in all simulations, which is approximately on par with model error. To know whether this model error influences the validation one has to consider the other error sources with the model, and of what magnitude they might be.

One problem with the simulation model is that we neglect the torque on the screw, coming from that the applied load is attached to a long steel rod at the bottom of the console in reality (see figure 6 in section 3, Methodology). The prescribed velocity is only vertical, not horizontal. The sled actually moves a bit from left to right and vice versa during the drop, which barely can be seen from the still images from the high speed camera film in figure 7. This horizontal movement could have been tracked as well and prescribed to the sled, but it wasn’t. Even with the horizontal velocity, the torque on the screw would still not be modeled realistically since we don’t model the long steel rod where the weights were attached. It’s hard to determine whether this would influence our results very much, since if the sled is skewed and a bit of the horizontal velocity is made to contribute to the axial force then the vertical velocity will lose some of it’s contribution. The vertical velocity is much higher than the horizontal though, so this has probably led to an overestimation of the axial force in the screw joint.

Regarding our simulations, the beam model seem to calculate a slightly higher force on the screw than the solid model in all cases, as can be seen in figure 21, 22 and 23. This may be because of the different pre-tensions for the models, that probably comes from the different methods used to apply the pre-tension in the simulation model. The INITIAL_AXIAL_FORCE_BEAM-keyword actually applies a specific force on one beam element whilst the INITIAL_STRESS_SECTION-keyword only applies the equivalent stress in the elements neighboring the cross-sectional plane; thus the pre-tension is not defined in the whole screw shank for the solid model but it is for the beam model.

Comparing our measurements with the simulated results, and taking our simulation error into account, case 29 (figure 22) suggests that the simulations are on par with the measurements whilst case 30 (figure 23) suggests the simulations are overestimated (approximately by 10%). If we observe case 28 and 31 together, the 900mm drop case, we can see that there is a big difference between the two measurements and that the simulations are in between them. The average measured maximum force on the screw is $33.5 \pm 4.8$ kN, and the simulation results are in range of that spread, even considering the simulation model error of about 4%. Also, the fact that case 30, a 1000mm drop-test, resulted in a smaller maximum axial force (32.5 kN) than the maximum axial force for case 31 (36.9 kN), a 900mm drop-test, implies that there is a big spread in the
measurement results that is not quite established. It seems logical that the force on the screw should increase with the drop-height, and not decrease. This probably has to do with a variance in the sled velocity during the actual drop. The sled might fall a bit skewed, which results in different forces on the screw (as the right side of the sled hits the ground first and the left later, or vice versa). That the sled is skewed might also impact the friction force between the metal rails and the sled, which slows down the sled.

This leads us to a significant error source in the simulation model: the velocity curves were produced from one drop from each drop-height respectively. There is probably a spread in these velocities as well, which we haven’t observed. To remedy this error we could measure many velocities from the same drop-test and establish the statistical spread in the velocity, which would lead to a statistical spread in our simulation results. An easier way would be to perform velocity tracking and force measurements simultaneously of the same drop-test. This error in our simulation input of the velocity curves can actually be quite big, but it’s unclear how big it is.

The beam simulation model has one advantage to the solid simulation model: The ability to easily monitor the plastic strain $\epsilon_p$ in the screw. When a simulation of a vehicle with many screws are performed, $\epsilon_p$ is monitored in every screw and if it is higher than the plastic strain at rupture, 11% in the M8 type 8.8 case, screw failure is determined. If we look at table 6, we can see that the screw in case 29 breaks at 3.5 ms. Depending on how you view the measurement results in figure 22 of the same case, that also shows that the screw broke at around 5 ms, the time the screw held during the force pulse might be correlated to this simulation time. After all, the measurement results does not change in the first 1.5 ms and we don’t know whether this is because the whole drop-tower rig is static or if the sled has started to drop. We see the first clear changes in the force on the screw at around 1.5 ms for the measurement values, so the drop time might begin at 1.5 ms and the rupture time might be $5 - 1.5 = 3.5$ ms after that. The simulation input should start right before the time of the impact, as the maximum negative velocity is used as the first value in the velocity curve and it is the impact that slows it down. This is a very vague correlation but it is worth mentioning.

In conclusion, the model is valid according to the measurements available, however we can’t conclude how confident we are in those measurements. Overall, the simulation model is in range of the measured results or produces higher forces. This is better since a model that indicates screw failure when it doesn’t break at least doesn’t indicate failure when the screw actually breaks, so we can use it to make safe predictions. The beam model also produces simulation results very similar to the solid model, which would have been the alternative simulation model but unpractical when a large number of screws are included in the model. This might actually be one of the biggest validation factors we have, since the FEM is very established in the industry and the solid model is in a way the default way for the FEM.

It is still unclear whether shear forces on the screw is modeled correctly with the beam model, since it hasn’t been validated due to lack of experimental data. If one wants to
investigate this, or strengthen the validation of the axial forces in the screw, I would suggest that new drop-tower tests should be performed and that the sleds velocity is measured together with the force measurements. The sleds horizontal velocity should also be included as simulation input in this case. In the axial experiments, one could also measure the elongation of the screws, or indicate that they ruptured if so is the case. The measured elongation of the screws can also be compared to the simulated elongation of the beam elements, giving another parameter to validate against. One could also look at how big the impulse is to the screw up until rupture, which would be determined by the plastic strain in the simulations, and see if this correlates between the measurements and the simulations.
7 References


