



**ČESKÉ
VYSOKÉ
UČENÍ
TECHNICKÉ
V PRAZE**

**Ústav mechaniky, biomechaniky
a mechatroniky**

**Porovnání MKP simulace s experimentem
při materiálových zkouškách**

**Comparison of FEM simulation with experiment in
material tests**

BAKALÁŘSKÁ PRÁCE

2019

David Janata

Studijní program: Teoretický základ strojního inženýrství

Studijní obor: bez oboru

Vedoucí práce: Ing. Petr Beneš, Ph.D.

PROHLÁŠENÍ

Prohlašuji, že jsem bakalářskou práci s názvem: „Porovnání MKP simulace s experimentem při materiálových zkouškách“ vypracoval samostatně pod vedením Ing. Petra Beneše, Ph.D. a s použitím literatury uvedené na konci mé bakalářské práce v seznamu použité literatury.

DECLARATION

I declare that this thesis, by the name “Comparison of FEM simulation with experiment in material tests” was created solely by me, with the help of my thesis supervisor Ing. Petr Beneš, Ph.D. and the literature provided at the end of this thesis in the list of references.

Prague 16.08.2019

David Janata

PODĚKOVÁNÍ

Na tomto místě bych velmi rád poděkoval svému vedoucímu mé bakalářské práce Ing. Petru Benešovi, Ph.D. za odborný dohled, konzultace, vedení a čas, který mi věnoval, a které mi značně pomohly k vypracování této bakalářské práce. Dále bych rád poděkoval svým kolegům. Primárně Ing. Lukáši Horáčkovi za jeho odborné rady a pomoc. Také mému vedoucímu Ing. Josefu Kuklovi za jeho vřelý a ochotný přístup. V neposlední řadě také Ing. Eleně Čižmárové Ph.D. za výpomoc při provádění Charpyho zkoušky.

ACKNOWLEDGEMENT

I would like to take this opportunity to thank my thesis supervisor Ing. Petr Beneš, Ph.D. for his guidance, supervision, and time he dedicated to helping me in writing this bachelor thesis. Further I would like to thank my colleagues. Primarily Ing. Lukáš Horáček for his professional advice and aid. Also, my team lead Ing. Josef Kukla for his willing and complaisant approach. Last but not least Ing. Elena Čižmárová Ph.D. for her help in performing the Charpy test.

ANOTAČNÍ LIST

Jméno autora: **David Janata**

Název BP: **Porovnání MKP simulace s experimentem při materiálových zkouškách**

Anglický název: **Comparison of FEM simulation with experiment in material tests**

Rok: 2019

Studijní program: TZSI

Obor studia: bez oboru

Ústav: Ústav mechaniky, biomechaniky a mechatroniky

Vedoucí BP: Ing. Petr Beneš, Ph.D.

Bibliografické údaje: Počet stran 33

Počet obrázků 18

Počet tabulek 4

Počet příloh 2

Klíčová slova: MKP, Metoda Konečných Prvků

Keywords: FEM, Finite Element Method, FEA, Finite Element Analysis

Anotace: This bachelor thesis deals with the topic of fracture simulations within LS-DYNA. Another point in this thesis is the application of GISSMO and performing two separate experiments.

Annotation: Tato bakalářská práce se zabývá šířením prasklin a jeho simulováním v LS-DYNA. Jako další bod této práce je aplikace GISSMO modelu a provedení dvou nezávislých experimentů.

OBSAH

Prohlášení	4
Declaration.....	4
Poděkování	5
Acknowledgement.....	5
Anotační list	7
Obsah.....	8
1. INTRODUCTION.....	1
1.1. OBJECTIVE:	1
1.2. RESULTS	1
1.3. EXPERIMENT.....	1
1.4. LS-DYNA	2
2. THEORY.....	3
2.1. ELEMENT EROSION	3
2.1.1. GISSMO – DESCRIPTION	3
2.2. MATERIAL TESTS.....	4
2.2.1. UNIAXIAL TENSILE TEST	5
2.2.2. CHARPY IMPACT TEST.....	6
2.2.3. CUSTOM DROP WEIGHT IMPACT TEST	8
2.3. MATHEMATICAL ANALYSIS.....	9
2.3.1. IMPLICIT VS. EXPLICIT	9
3. PRACTICAL PART	11
3.1. BASIC STRUCTURE OF CODE	11
3.1.1. LS-PrePost.....	12
3.2. MATERIAL MODELLING	13
3.2.1. *MAT_ADD_EROSION AND *MAT_024	13
3.2.1.1. *MAT_024	13
3.2.1.2. *MAT_ADD_EROSION	13
3.3. MODEL PARAMETRIZATION	14
3.4. EXPERIMENT RESULTS.....	15
3.4.1. CHARPY IMPACT TEST – THE EXPERIMENT	15
3.4.2. CUSTOM DROP WEIGHT TEST – THE EXPERIMENT	17
3.4.3. CHARPY IMPACT TEST – THE SIMULATION.....	19
3.4.3.1. MODEL PREPARATION.....	19
3.4.3.2. ANALYTICAL PART	20
3.4.4. CUSTOM WEIGHT DROP TEST – THE SIMULATION	21
3.4.4.1. MODEL PREPARATION.....	21
3.4.4.2. ANALYTICAL PART	23

4. THE RESULTS	26
4.1. CHARPY IMPACT TEST – RESULTS.....	26
4.2. CUSTOM DROP WEIGHT TEST – RESULTS	27
5. CONCLUSION.....	30
List of abbreviations	31
List of references	32
List of figures	33
List of tables	33
List of appendices	33

1. INTRODUCTION

1.1. OBJECTIVE:

The aim of this thesis is to research theory and practice behind material failure simulation in LS-DYNA, apply gained knowledge in simulating two experiments.

One of the experiments is a standardized material test and the other is a custom drop-weight test. Results of the experiments and their counterpart simulations will then be compared. Should any appear, the differences in between them will prove useful in understanding how simulation parameters should be set. With newly gained understanding, I will then attempt to mitigate any and all differences between the simulation and the real-world experiments. Hopefully setting the failure criteria more accurately with each iteration of the simulation, finally simulating the experiment precisely with no apparent differences from the experiments. Optimally I will attain new knowledge which I, as well as my colleagues, will then be able to use henceforth.

1.2. RESULTS

This thesis is subject to my Non-disclosure agreement (NDA) signed with AKKA therefore most results that appear in this thesis are somehow altered or omitted as to fulfill the contract. Real results from my research will be handed only to my direct professional superior. Thank you for your understanding.

1.3. EXPERIMENT

In order to evaluate the reliability of said simulation, experimental data correlating with the model is required. Out of the many possible material tests I chose to perform the Charpy Impact Test as well as a custom drop-weight test that is described in more detail further.

1.4. LS-DYNA

LS-DYNA is a finite element program, developed by LSTC (Livermore Software Technology Corporation) used for simulation of real-world physics and engineering problems. Its application spans across many fields of engineering expertise, such as but not limited to: automobile, aerospace, construction, military, manufacturing, and bioengineering industries. Along with LS-Dyna, LSTC develops and maintains support software such as LS-PrePost, a pre- and post- processing software which can be used to build models and evaluate results in a lean and efficient way. LS-Dyna itself uses “.k” files (also sometimes referred to as “.key” files) in order to run the simulation. This “.k” file contains information about the model itself (e.g. x, y, z coordinates of each node of the mesh) as well as keywords. These keywords are what tells the solver how nodes should interact with one another, what, if any boundary conditions are set and prescribes the way load is applied among other things.

2. THEORY

2.1. ELEMENT EROSION

In order to input the correct conditions for LS-DYNA to evaluate material failure with, first we need to understand how LS-DYNA operates with said conditions. The keyword that is used to define failure conditions for elements is `*MATERIAL_ADD_EROSION`. This keyword allows me to choose among different failure models and fracture criteria. These criteria include but are not limited to: Maximum pressure in the element at failure, maximum principal strain at failure, maximum principal stress at failure, maximum shear strain at failure or failure time. The user then chooses a minimum of two conditions that need to have been met before the element erodes. The failure model I choose to operate with will be GISSMO (Generalized Incremental Stress-State dependent damage model).

2.1.1. GISSMO – DESCRIPTION

An in-depth understanding of GISSMO is unnecessary for the purpose of this thesis and has been explained in works such as [2] as well as the LS-DYNA User's manual [1].

However, the basic and important rule for GISSMO is the damage accumulation rule which, according to F. Andrade, M. Feucht and A. Haufe, is as follows [3]:

$$\Delta D = \frac{n}{\epsilon_f(\eta)} D^{(1-\frac{1}{n})} \Delta \epsilon_p \quad (1)$$

Where D is current amount of damage, n is a damage exponent, η is triaxiality, $\epsilon_f(\eta)$ is the fracture strain as a function of triaxiality $\Delta \epsilon_p$ is an increment of plastic strain. Upon reaching $D = 1.0$, a tear in material is assumed.

Triaxiality η is a parameter defining a materials stress-state. It is important to incorporate stress-state into simulations since it heavily impacts results.

Triaxiality is defined as such:

$$\eta = \frac{I_1}{\sigma_{vm}} \quad (2)$$

Where I_1 is the first stress invariant

$$I_1 = \frac{\sigma_1 + \sigma_2 + \sigma_3}{3} \quad (3)$$

σ_{vm} is of course the von Mises stress and σ_1 through σ_3 are the principal stresses.

Von Mises stress can be calculated as:

$$\sigma_{vm} = \sqrt{\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - \sigma_1\sigma_2 - \sigma_1\sigma_3 - \sigma_2\sigma_3} \quad (4)$$

From equations (2), (3) and (4) we get:

$$\eta = \frac{\sigma_1 + \sigma_2 + \sigma_3}{3\sqrt{\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - \sigma_1\sigma_2 - \sigma_1\sigma_3 - \sigma_2\sigma_3}} \quad (5)$$

Equation (5) makes it quite clear that the value of triaxiality ranges from -1 to 1.

$$-1 \leq \eta \leq 1$$

2.2. MATERIAL TESTS

Understanding the properties of a material is necessary before considering applications of said material in engineering problems. To determine properties of a material standardized tests have been developed and are used.

2.2.1. UNIAXIAL TENSILE TEST

One of the basic material tests is the uniaxial tensile test. In this test a standardized geometry made of the studied material is stretched until failure by a known load. Having knowledge of the geometry as well as the load applied to it we can easily calculate the stress strain dependency known as Young's modulus using this formula:

$$\sigma = \epsilon E \quad (6)$$

Where:

$$\sigma = \frac{F}{A_0} \quad (7)$$

And:

$$\epsilon = \frac{\Delta L}{L_0} \quad (8)$$

F – Force applied, A_0 – Initial cross-sectional area of the geometry normal to the load direction, ϵ – Strain, ΔL – Change in length of the specimen, L_0 – Initial length of the specimen and E – Young's modulus.

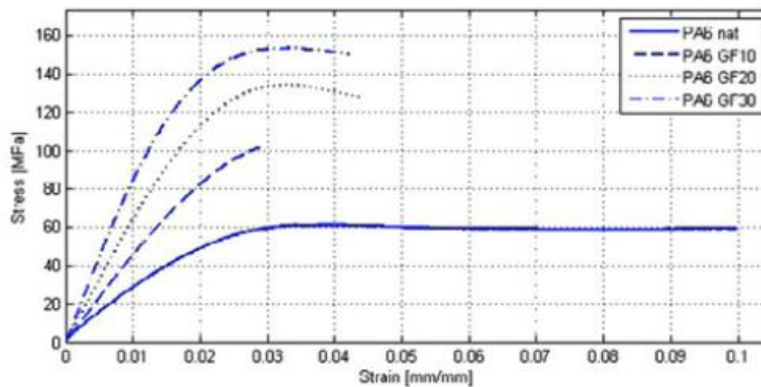


Figure 1 – Tensile stress-strain curves for four grades of reinforced polyamide [5]

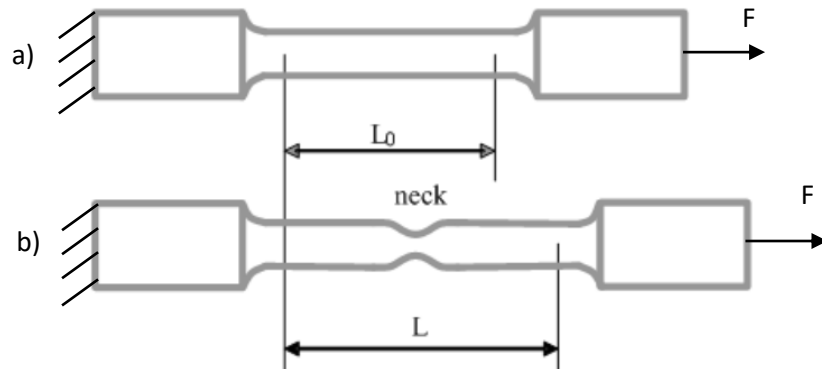


Figure 2 – Diagram of the uniaxial tensile test

Figure 2 a) shows the initial state of the tested geometry and Figure 2 b) shows the geometry after load is applied and a neck has formed. Figure 1 also illustrates the boundary conditions of the uniaxial tensile experiment as well as the direction of the load applied.

The uniaxial tensile test will not be performed as a part of this thesis, for the information about the material that would be found (i.e. Young's modulus, ultimate tensile strength, etc.) as a result of this test were generously provided for me by my employer.

Table 1 – Material properties of PA6-GF30-I

E	μ	ρ	R_m	A_5
5100-9200 ¹ MPa	0.35	1.36 g/cm ³	100 MPa	7.3%

¹ – Young's modulus is dependent on the material's humidity

2.2.2. CHARPY IMPACT TEST

The Charpy impact test is a standardized material test. The purpose of this test is to evaluate the amount of energy absorbed by a material during fracture. The testing machine consists of a hammer of a known mass that is dropped from a known height. The hammer impacts a notched specimen (Fig. 3) and the difference in the initial height and the maximum height after striking the specimen is measured.

Strain rate dependency is also a factor in this test and may be studied through a quantitative form of this experiment, however for the purposes of this thesis, strain rate dependency will be omitted.

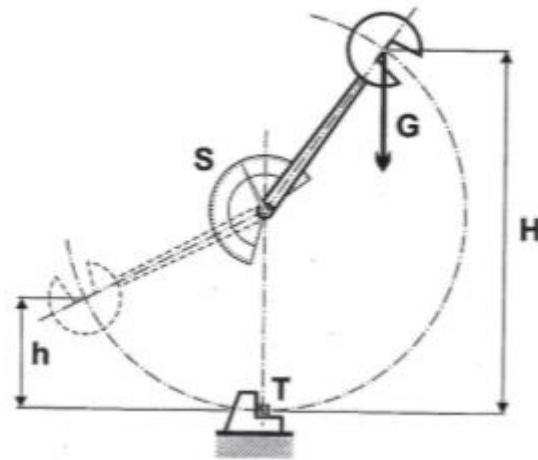
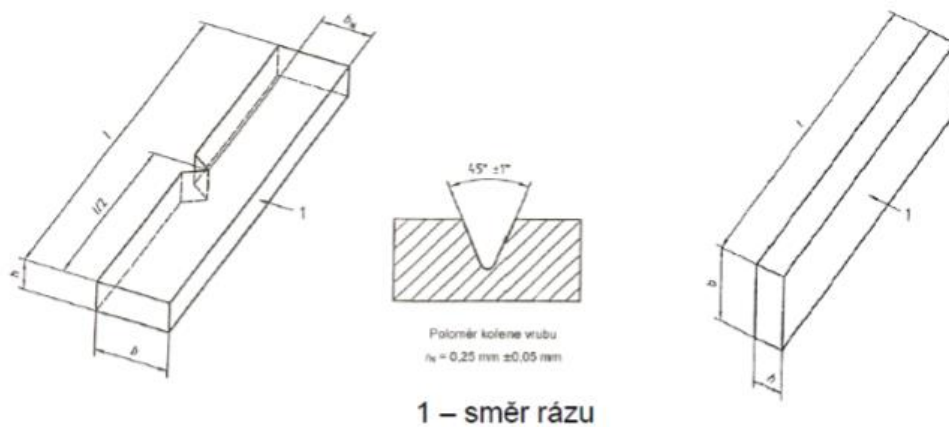


Figure 3 – Charpy testing machine [4]



Rozměry zkušebního tělesa

Typ zkušebního tělesa	Délka ^a <i>l</i>	Šířka ^a <i>b</i>	Tloušťka ^a <i>h</i>	Rozpětí <i>L</i>
1	80 ± 2	10,0 ± 0,2	4,0 ± 0,2	62 ^{+0,5} _{-0,0}

Figure 4 – Standardized specimen for Charpy impact test

l stands for length of the specimen, *b* stands for width of the specimen, *h* stands for depth of the specimen and *L* stands for the distance between holders in which the specimen sits.

Figure 4 is excerpt from the Czech alternative to [6].

2.2.3. CUSTOM DROP WEIGHT IMPACT TEST

For the purpose of visualizing fracture forming through simulation and experiment I've decided to come up with a custom experiment. This experiment will be analogous to an already existing standardized test, described in-depth in [7].

In this custom experiment a weight of spherical shape will be dropped onto the chainwheel cover from incrementally increasing height with the intend to find such drop height from which on a fracture can be safely assumed. A simulation will then be run with the exact weight geometry and drop height, expecting similar results. In the event of results from the simulation and experiment differing, minor changes to the fracture criteria within will be incrementally performed and the simulation ran again. Attempting at simulating identical fractures to the chain sprocket cover, as formed in the experiment.

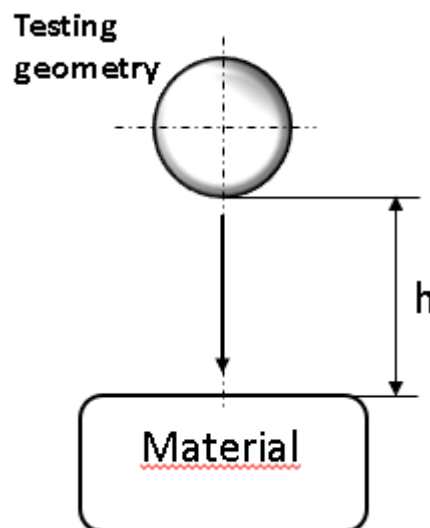


Figure 5 – Diagram of the custom drop weight test

Where h – Drop height and the testing geometry is a steel sphere with rounded edges.

A video record will also be a part of this experiment. This video will serve as a tool for comparison with the simulation. This experiment will play a major part of the practical part of this thesis. It will serve to show the progress, when adjusting simulation parameters to fit the real-world, physical experiment.

2.3. MATHEMATICAL ANALYSIS

It is self-evident that solving FEM problems by hand can be considered impossible. Since calculating the motion and interaction of each node with one another, of which there can be millions in a simulation, would take an immense amount of time and manhours. Taking into consideration that this calculation would have to be done for each time-step separately. Computational tools have been created to ease the work of engineers.

“To solve time dependent ordinary differential equations (ODE) and partial differential equations (PDE) with finite element analysis, either an explicit or an implicit method is used.” Writes H. Sandberg and O. Rydholm in [8]. In the case of these simple material tests and the custom experiment, I will be dealing with the explicit method exclusively. To gain an understanding of the implicit method, which will not be described in this thesis, the reader is refer to [1] and [9].

2.3.1. IMPLICIT VS. EXPLICIT

In this sub-chapter, I will attempt to explain some of the core differences between explicit and implicit analyses, as well as their respective pros and cons.

The **explicit** method of simulating a problem is reasonably fast and it doesn't require quite as much computational power as its counterpart. However, errors in results accumulate with each time-step, so the answer will diverge over time from the one that's looked for. Explicit analysis also requires for the set time-step to be smaller than the time that it takes for the shock wave, that arises from loading, to travel across the smallest element in the meshed model (Courant–Friedrichs–Lewy condition). Such as:

$$\Delta t = \frac{d_{min}}{c} \quad (9)$$

Where Δt is the maximum time-step possible for the simulation to finish its run without errors, d_{min} is the minimum distance between any two nodes belonging to the same element in the model and c is the speed of sound in the material through which the shock wave is travelling. The speed of sound in a material can be calculated like so:

$$c = \sqrt{\frac{E}{\rho(1 - \mu)}} \quad (10)$$

Where E is Young's modulus of said material, ρ is its density and μ is its Poisson's ratio.

The **implicit** method of simulating a problem is much more demanding on computational power, therefore more expensive and more time consuming, but produce less error-prone results. It is important to keep in mind that implicit analysis can introduce an artificial phase-shift when responding to external forces (i.e. the model responds slower in simulation than in reality).

FEM problems can be categorized into time-dependent and time-independent groups. We speak about 'time-dependent' problems when acceleration of the studied model/simulation is non-negligible. Some of my previously mentioned experiments and standardized material test can prove to be useful examples of this. Take a simple Charpy Impact test. When the hammer hits the test specimen in the lower-most position of the apparatus, the deceleration of the hammer happens in just a few milliseconds upon the hammer's first contact with the notched specimen. Such problem would undoubtedly call for an explicit simulation. In other words: explicit simulation aims to solve for the acceleration (x'').

On the other hand, 'time-independent' implicit analysis is sought after when trying to simulate problems that have no significant acceleration. Current quantities calculated in a time-step are based on quantities calculated in previous steps. This is called Euler Time Integration Scheme which is deemed the simplest numerical method of solving ODEs with set boundary conditions. Here are some examples of implicit analysis usage. Simulating a bolt being screwed into material, simulating lifting a chainsaw slowly by the handle or even placing a monitor slowly onto a table. All of these are problems in which acceleration plays no major role therefore implicit analysis is preferable. In other words: explicit simulation aims to solve for displacement (x).

Explicit	Implicit
$Ma_n = f_n^{ext} - f_n^{int}$	$M\Delta a_{n+1} + K\Delta u_{n+1} = f_{n+1}^{ext} - f_n^{int} - Ma_n$
Direct solution	Iterative solution
Conditionally stable (CFL condition)	Unconditionally stable
Efficient, fast	Linearization necessary
Impact, crash	Strength, durability

Table 2 – Easy to read comparison of explicit and implicit

Table 2 shows the differences between explicit and implicit analysis in an easy to understand fashion.

3. PRACTICAL PART

The main concern of this part of the thesis will be fitting the experimental data to correlate with the simulation. The video recording of the cylinder drop test and subsequent tear-forming, as well as the shape and length of the tear, will hopefully closely resemble the simulated drop test.

3.1. BASIC STRUCTURE OF CODE

It is obvious that for running a simulation in LS-DYNA solver a code is required. This code can be quite long and confusing so standard practice at AKKA is to split the code up into 3 different files for easier maintenance and debugging. The 3 files are named the **element** file, which carries information about the geometry of the model, the coordinates of each node in the mesh, and in certain cases some boundary constraint information. The element file also splits up the model into parts. For instance, a chainsaw would be split into parts such as, but not limited to: The handle, the chain, the trigger and of course the chain sprocket cover. The element file then feeds this information to the **baugruppe** file (note

that the names of this file is in German since the sole customer, with whom I work, is a German company. Baugruppe loosely translates to Assembly). In the baugruppe file, information about material properties are assigned to each part. Last but not least comes the **Analyse** file, into which both the element and baugruppe files feed. In the analyse file keywords are used to prescribe the simulation behaviour. The term “simulation behaviour” means, for example: which, and in what format, results should be exported; what time-step is set; how contacts should behave; if any of the objects are in motion, the analyse file prescribes that motion. These were just a few examples of the uses for the analyse file, for further information, reader is referred to the code provided.

Figure 6 shows a diagram of the files in order. Note that the alters the formatting of the code when saving a file. Which would hinder the calculations engineer’s job, were it used on the baugruppe or analyse files which should be easy to read and easy to understand.



Figure 6 – File diagram

3.1.1. LS-PrePost

“LS-PrePost is an advanced pre and post-processor that is delivered free with LS-DYNA. The user interface is designed to be both efficient and intuitive.” [12] Writes the developer, Livermore Software Technology Corporation, on their official site.

The pre-processor (shown in figure 6) is a software tool used to develop, build and prepare a model for a simulation. Even though the pre-processor is used for altering the element file exclusively, it is regularly used to read-only the analyse file as well, which is useful for debugging the simulation.

3.2. MATERIAL MODELLING

In this chapter of the thesis I will go into how the material is modelled, what properties are prescribed to it, and last but not least, the iterative process of simulating the drop test will be described in detail.

3.2.1. *MAT_ADD_EROSION AND *MAT_024

As previously stated, almost everything that the LS-DYNA solver works with comes in the form of keywords. Two of these keywords that are important to note in this part of the bachelor thesis are the *MAT_024 (*MAT_PIECEWISE_LIENAR_PLASTICITY) and *MAT_ADD_EROSION.

3.2.1.1. *MAT_024

This keyword is the same as the keyword *MAT_PIECEWISE_LIENAR_PLASTICITY but the numbering serves computation engineers as a tool of convenience because it shortens the keyword. The reason I choose this as the material definition to work with is for its simplicity and experience I have working with it. Its defining features are Young's modulus, mass density and Poisson's ratio, all of which are material properties that are known to me in regard to PA6-GF30-I.

3.2.1.2. *MAT_ADD_EROSION

This keyword (mentioned in 2.1.) is an add-on keyword for material properties. It allows me to choose among different failure models and fracture criteria. These criteria include but are not limited to: Maximum pressure in the element at failure, maximum principal strain at failure, maximum principal stress at failure, maximum shear strain at failure or failure time. The user then chooses a minimum of two conditions that need to have been met before the element erodes. The two criteria I choose to operate with are maximum principal stress at failure and maximum principal strain at failure. Their values are subject to change with each iteration of the simulation. The reason I chose to operate with these criteria is simple: they are most straightforward, easy to understand and, in my opinion, are the criteria that best describe the conditions under which material fails in this scenario.

3.3. MODEL PARAMETRIZATION

In any and all FEM simulations a parametrization of the tested model is required. This parametrization is achieved through meshing the geometry of the simulated specimen. The model geometry is either acquired through CAD modelling or directly created in a FEM Pre-processor. In this thesis, both methods are used.

It is obvious that for more complex specimen, modelling of the part in a CAD program is beneficial, as CAD programs, such as Catia or Solidworks, are specifically created for designing geometries. On the other hand, for simple parts, like the notched Charpy testing specimen, a pre-processor is more than sufficient for the task of creating, and subsequently meshing, the geometry.

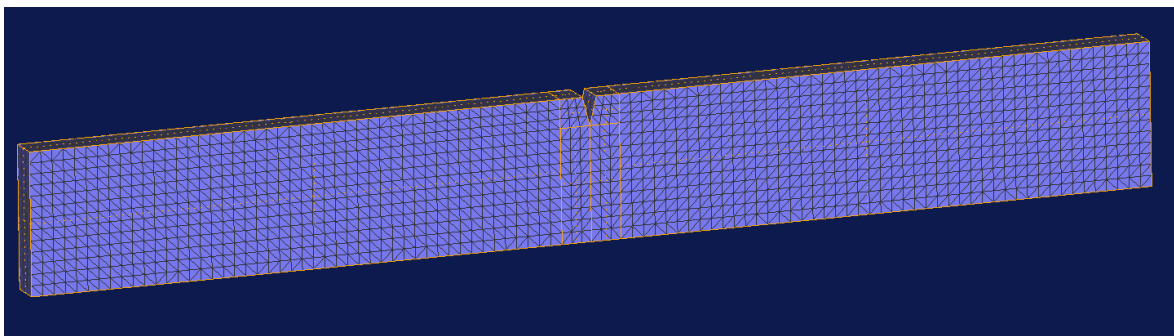


Figure 7 – Notched Charpy testing specimen - meshed

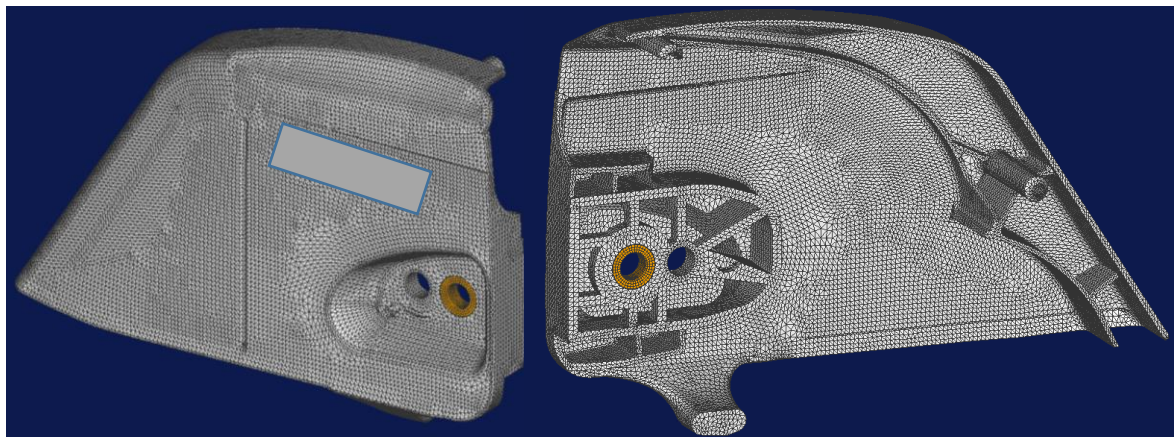


Figure 8 – The chain sprocket cover mesh

Figure 7 depicts the mesh used for the Charpy testing specimen. Note that the mesh size is quite small, meaning the average distance of two nodes belonging to the same

element, is insignificant when compared to the size of the entire model. This is beneficial for a number of reasons. The most important one being that a larger sized mesh could yield unreliable results. This mesh was created exclusively in the pre-processing program MEDINA.

To illustrate the differences between creating a geometry in a pre-processor and in a CAD program figure 8 shows the chain sprocket cover mesh.

A steadfast reader will undoubtedly understand that creating such a complex geometry in a pre-processing program would be bothersome to say the least. The CAD geometry of this part was created in Creo and later exported into MEDINA pre-processor to be meshed.

3.4. EXPERIMENT RESULTS

In this chapter I will go over the results of the two experiments that took place for the purposes of this bachelor thesis.

3.4.1. CHARPY IMPACT TEST – THE EXPERIMENT

The Charpy impact test took place on the 1st of August 2019 in the CTU material testing laboratory on Charles square. I would like to take this opportunity to thank Ing. Elena Čižmárová, Ph.D. for allowing me to use the laboratory and for her time, advice and tutelage during the experiment.

I tested 5 V-notched specimens. The tested samples didn't quite fit the norm with their geometry since they have been cut out of the chain sprocket cover. This means that some irregularities can be found on the specimen surface (e.g. slight thickness alterations, residue of the logo, etc.). The specimen also didn't have the recommended thickness of 4 mm but rather 2.4 mm on average.

The testing machine also suffers from offset and hasn't been calibrated prior to the experiment taking place so I will be subtracting 0.09J from each individual result since that is the value of the offset that appears when no test has yet taken place.

Specimen No.	Measured value	Real value
1	0.39 J	0.30 J
2	0.66 J	0.57 J
3	0.46 J	0.37 J
4	0.48 J	0.39 J
5	0.49 J	0.40 J

Table 3 – Charpy impact test results

It is plain to see that specimen 3 through 5 tested at very similar values of fracture energy. Specimen 1 and 2 did not. This might be due to the unreliable nature of their specific geometries. Another explanation could be offered by the machine samples 1 and 2 were the first samples to be tested and this could be the reason why the machine was off on their values. Due to long-term inactivity, the machine could have suffered from buildup of dust and corrosion. Alternatively, the measuring mechanism, which presumably consists a frictionless block, that is pushed by the hammer to a certain height, could've had its stiffness altered, due to corrosion or dust. See figure 10 for tested Charpy specimen.

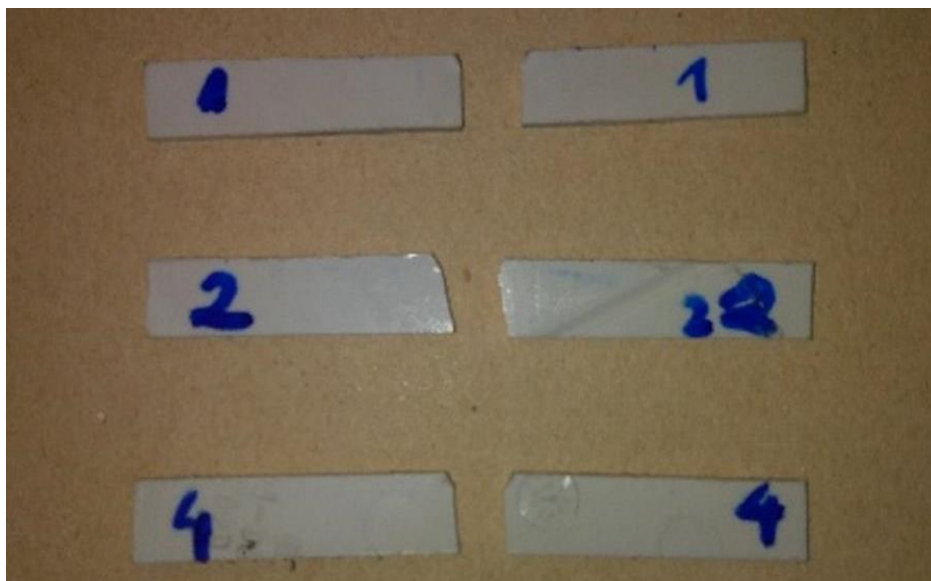


Figure 10 – Charpy test specimens

3.4.2. CUSTOM DROP WEIGHT TEST – THE EXPERIMENT

As previously described, the custom drop weight test consists of a spherical weight being dropped at the chain sprocket cover from a certain height, calculating the impact energy of the weight and seeing if the chain sprocket cover had cracked from the impact. Consequently, examining the shape and length of the crack that, hopefully, had formed.

This experiment took place on the 6th of August 2019 in my hometown of Býšť, Pardubický kraj. I used a shot-putting shot as the weight, for its a heavy, metal and spherical object. For the unaware shot-putting is a track and field discipline involving pushing a heavy ball as far as possible in a single push. To know the parameters of the ball I needed to measure its circumference.

As for the anchor point for the chain sprocket cover, I opted for a piece of train rail. The dimensions fit almost perfectly. Photographic documentation of the alignment can be found in appendix 1. I tried various drop heights, each time on a separate chain sprocket cover of course, and the results of each one, including photographs, can be found also in appendix 1.

I would like to note here that before starting the experimental process I had already created one “dummy” simulation. This simulation proved useful in estimating the first drop height. This was necessary since the number of chain sprocket covers, I was provided with, was not infinite and I had already spent some of them on the Charpy impact test. For a simple overview of the results, see table 4. You might notice that the first two experiments were not described in the theoretical part of the thesis. They were omitted since they proved to be unsuccessful and their geometries later iterated upon to end up with a simple sphere. These first two experiments took place in the CTU Material laboratories on the same day as the Charpy impact test.

Experiment No.	Drop height	Testing geometry	Result
1	1 200 mm	Hexagonal rod Sharp edges 2 400 g	Unsatisfactory
2	1 500 mm	Cylindrical rod Rounded edges 4 000 g	Unsatisfactory
3	2 400 mm	Sphere 4 000 g	Semi-satisfactory
4	1 800 mm	Sphere 4 000 g	Satisfactory
5	1 500 mm	Sphere 4 000 g	Satisfactory

Table 4 – Evaluation of the custom drop weight test results



Figure 11 – Custom drop weight test - experiment setup

Figure 11 shows the experiment setup.

3.4.3. CHARPY IMPACT TEST – THE SIMULATION

3.4.3.1. MODEL PREPARATION

The simulation part of the Charpy impact test began with measuring the testing equipment and tested specimen. The measurements of the hammer made it easy to calibrate the hammer's geometry and parameters for simulating. First, I made the geometry for the hammer. The nature of this experiment makes it so that its only necessary for the hammer to strike the specimen with 7.5 Joules of energy. This was achieved by measuring the simulated hammer's moment of inertia. MEDINA proved useful for this. Since I needed to prescribe motion to the hammer, I had to find the rotational velocity upon impacting the specimen. This was calculated through the energy equation:

$$E = \frac{1}{2}I\omega^2 \quad (12)$$

Separating out for ω :

$$\omega = \sqrt{2\frac{E}{I}} \quad (13)$$

Inputting the values as follows: $E=7.5\text{J}$, $I = 2.69 \text{ t}\cdot\text{mm}^2$ we get $\omega = 74.56 \text{ s}^{-1}$

With the hammer set up I needed to model the test specimen. The dimensions of the specimen, though previously mentioned, are repeated here for reader's convenience/ The length of the specimen is 80 mm, the width of the specimen is 10 mm and the thickness of the specimen is 2,4 mm. The notch in the middle of the specimen, lengthwise, is 2 mm deep at an 60° angle.

Having created and meshed the geometries of the hammer and the specimen, all that was left at this point was deciding on how to constrain the specimen so that it would best replicate the real-world physics. I decided that constraining the specimen through

boundary conditions or mathematical constraints would not only be bothersome, but might yield unreliable results, so I opted for replicating the holders for the specimen and constraining all degrees of freedom for those. Figure 12 shows the model.

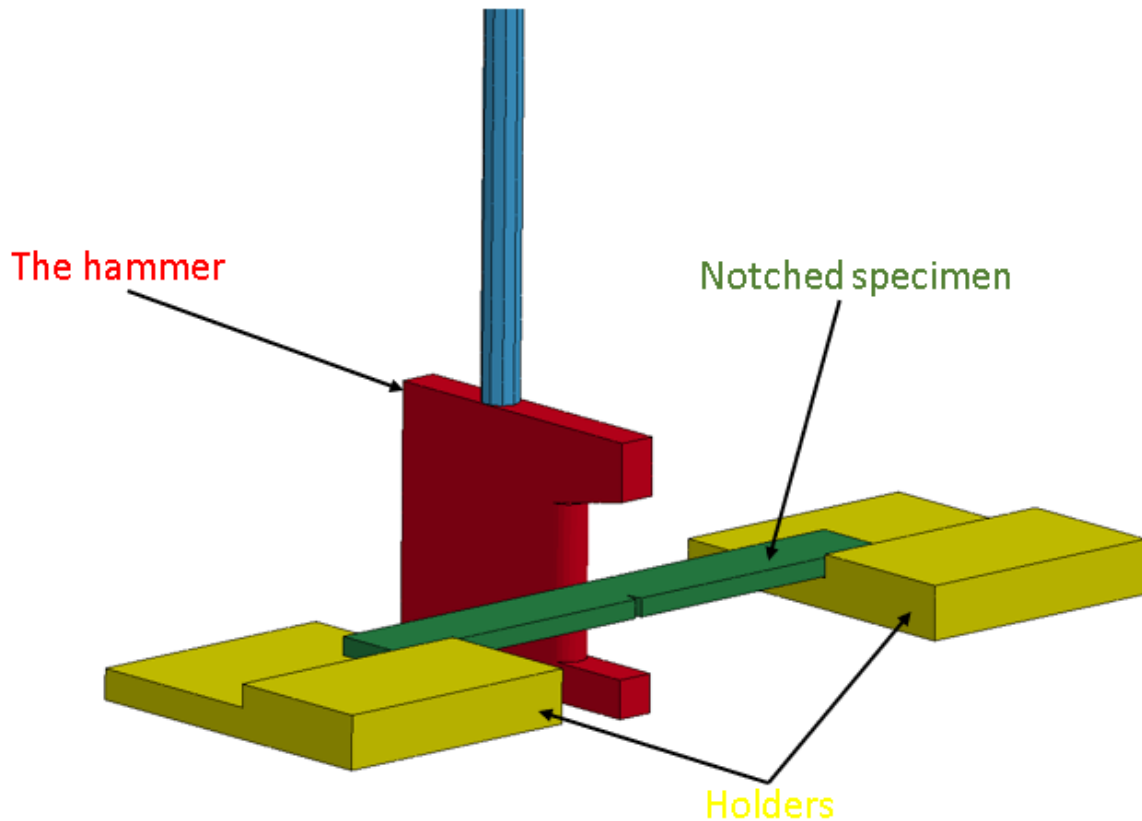


Figure 12 – Charpy test - simulation setup

3.4.3.2. ANALYTICAL PART

With the physical model set-up, it was time to assign materials. The notched specimen was obviously assigned the properties of PA6-GF30-I. The material assigned to the holders is unimportant since they're modeled as completely RIGID, I opted for a rigid steel. The same goes for the hammer. The only difference is that the hammer's head is modelled as a RIGID steel but the arm holding the hammer head needed to be modeled as elastic for the purposes of the constrains at the top end of the arm. Since the keyword that is used to constrain the top end of the hammer arm is *CONSTRAINED_NODAL_RIGID_BODY (CNRB) and this keyword only works with elastic materials.

Having assigned the materials to each part in the simulation, it was now necessary to define how contacts would work and give the simulation its boundary and initial conditions. In this case, the only initial condition was defined through the keyword *INITIAL_VELOCITY_GENERATION which tells the hammer how fast, and around which global axis it should rotate. The boundary conditions were the constrains of the holders and the top end of the hammer. The holders were forbidden any type of movement in their material card by restraining all degrees of freedom. The hammer CNRB was forbidden all displacement degrees of freedom and two rotational degrees of freedom so it would best mimic the real-world testing machine. Note that the first time-step of the simulation is just before the hammer hits the specimen. Having set up the simulation this way saves on computational time and aids in controlling the outcome.

3.4.4. CUSTOM WEIGHT DROP TEST – THE SIMULATION

3.4.4.1. MODEL PREPARATION

As was the case in the previous chapter of this thesis, this experiment will also begin with measurements. It is necessary to know the dimensions and mass of the spherical weight, that will be used as the drop weight geometry. From these two parameters I can calculate the density of the sphere, which will be necessary since the material card for *MAT_024 (*MAT_PIECEWISE_LINEAR_PLASTICITY) requires for density to be input. We can calculate the density from this simple equation:

$$\rho = \frac{m}{V} \quad (14)$$

Where ρ is density, m is mass and V is the volume of the sphere, simply calculated as:

$$V_{sphere} = \frac{4}{3} \pi r^3 \quad (15)$$

After calculating the material's density, I could easily create a material card for the sphere.

The material I chose to run with was a RIGID steel material.

Figure 10 in chapter 3.3 shows the mesh on the chain sprocket cover already, so I find it unnecessary to show it here again. Please take into consideration that the chain sprocket cover isn't a single part, there is also a steel ring (colored orange in figure 10). I used this ring to show the reader the various possibilities of meshing. The chain sprocket wheel is meshed using TETRA4 elements whereas the ring is meshed using HEXA8 elements exclusively. It is standard practice to mesh simple geometries (e.g. rings, cubes, springs) with HEXA elements, since they often yield more trustworthy results. For more information on different mesh attributes, the reader is referred to [12].

Using equation (15) I could calculate the volume of the sphere. The measured circumference is 350 mm. That gives us the volume of $7.24 \times 10^5 \text{ mm}^3$. From this value I could then go on and calculate the density, since the mass of a standard shot is known to be 4kg (4000g). I weighed the ball of course and it showed that it indeed weighs 4kg. Using equation (14) I calculated that the density of the material the shot is made from comes up to $5.52486 \times 10^{-6} \text{ kg/mm}^3$. Using [13] I wasn't precisely able to determine the material of the sphere. The material itself isn't an important piece of information. Since the sphere is modeled as rigid, all that I needed to know was its density, since that is the determining factor of mass.

Back to the topic at hand. Having assigned the materials to each part. Rigid steel for the spherical weight, elastic steel to the hexahedrally meshed ring and, of course, PA6-GF30-I to the chain sprocket cover, I aligned the geometries properly and started writing the analyse file.

Please note that I'm not using classic SI units like meters or kilograms. This is due to the fact that in mechanical engineering, it is best practice to use millimeters as units of length. This leads to a conversion of all other to best fit their use.

3.4.4.2. ANALYTICAL PART

This part of the thesis is where the reader is bound to properly understand the simulation process in LS-DYNA and FEA (Finite Element Analysis) in general. Having written the baugruppe file in the model preparation chapter, this chapter is almost solely focused on the analyse file. As opposed to chapter 3.4.3.2 – the Charpy simulation, this chapter will go more in-depth into the entire process, since simulating this drop test is a more challenging and nuanced. The Charpy simulation was useful only for understanding and fine-tuning *MAT_ADD_EROSION and GISSMO, as well as some experimental insight.

The whole analyse and baugruppe files (with some information omitted due to the NDA) is available in appendix 2.

The analyse file can be broken into several parts, which I will go over one-by-one. These parts are: 1) PARAMETERS – these serve the same function as variables do in any sort of programming. 2) CONTROL OUTPUT – Control keywords are used to change the default way that ls-dyna calculates problems. It is the first and foremost part of the simulation that needs writing. It allows me to set the simulation time, the time-step length, etc. 3) DATABASE – Settings for output of results. Telling the solver which results to print and in what format. 4) CONTACT – There are many different contact definitions to choose from, it is up to the engineer to decide which contacts are best-suited for the simulation at hand. Contacts prescribe how surfaces or nodes are to interact with one another when close to each other. 5) CONSTRAINS – Basically just allowing/disallowing degrees of freedom for parts or sets. And last but not least 6) INITIAL – setting initial conditions for the model.

These are of course just some of the options that are used in day to day simulations, but I chose these specifically because they play their part in this simulation and are a great medium to explain the process through.

In order to achieve results similar to the experimental it was necessary to run many iterations of the simulation, tweaking and changing settings for a progressively better result. Most of the changes were made to the *MAT_ADD_EROSION keyword as that's the keyword that is responsible for fracture forming. Still, some of the changes needed to be made to the time-step size, the simulation termination time and even contact definitions. At first the time step was too large and had to be made smaller, so that the fracture forming

would be better visible. Contact definitions were a problem because one of the settings for contacts defines static and dynamic friction, these friction coefficients were also altered through-out to yield better results. Initially the friction coefficient was set to 0.2 for all part interactions. This, however, is obviously not the case in the real world. So, I iterated upon them through trial and error. For the contact between the steel drop weight and the chain sprocket cover, I opted for a very low coefficient of friction in hope that it would solve the problem with cracks forming in places they didn't during the experimental part. For the contact between the chain sprocket cover and the rigid rail, I chose a very high coefficient of 0.4 to better simulate the fact that I used duct-tape to hold the cover in place.

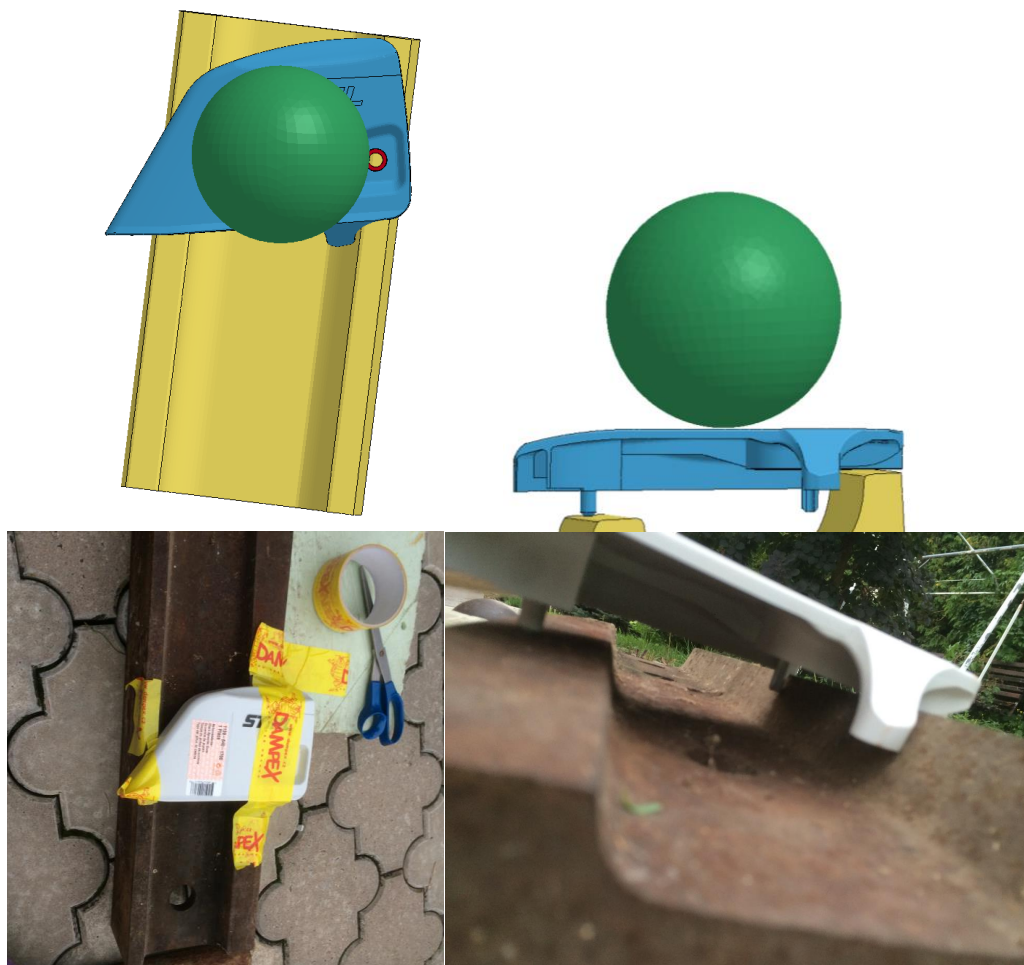


Figure 13 – Comparison of simulation model with experiment

Figure 14 shows a comparison of the experiment setup and the simulation model setup.

From the first few iterations of the simulation it became apparent that GISSMO can play a vital role in fracture forming mechanics if properly calibrated. This calibration process still was no easy task. But after the first few iterations, the results started to look alike.



Figure 14 – Comparison of results and simulation of the custom drop weight test

Figure 13 shows the comparison of the simulation result and the experimental result.

However, problems would soon arise and show that the simulation was still improperly calibrated, since some cracks would form in places that they hadn't in the experiment.

4. THE RESULTS

In this chapter of the thesis, I go over the results of both the experiments and the simulations. The results will be compared and some of the methods explained.

4.1. CHARPY IMPACT TEST – RESULTS

Trying to fine-tune the *MAT_ADD_EROSION keyword, as well as its GISSMO properties was successful. The simulation showed some interesting results. The aim was to calibrate the hammer energy to fit with the experiment i.e. for the hammer to have 7.5J of initial energy and to lose roughly the average of the real values from table 3. I deemed it necessary to not take into account attempts 1 and 2, since their values differed by quite a large margin. Calculating the average of attempts 3 through 5 gives us an average energy loss of $0.3866\text{J} \sim 0.39\text{J}$. I was able to achieve this after only a few iterations, since this is a simple simulation. Most work went into altering element erosion criteria and observing what results each alteration would provide.

Figure 15 shows the dependency of select energies on time. The energies I think illustrate the problem best are: Total energy of the model, kinetic energy, internal energy and sliding energy.

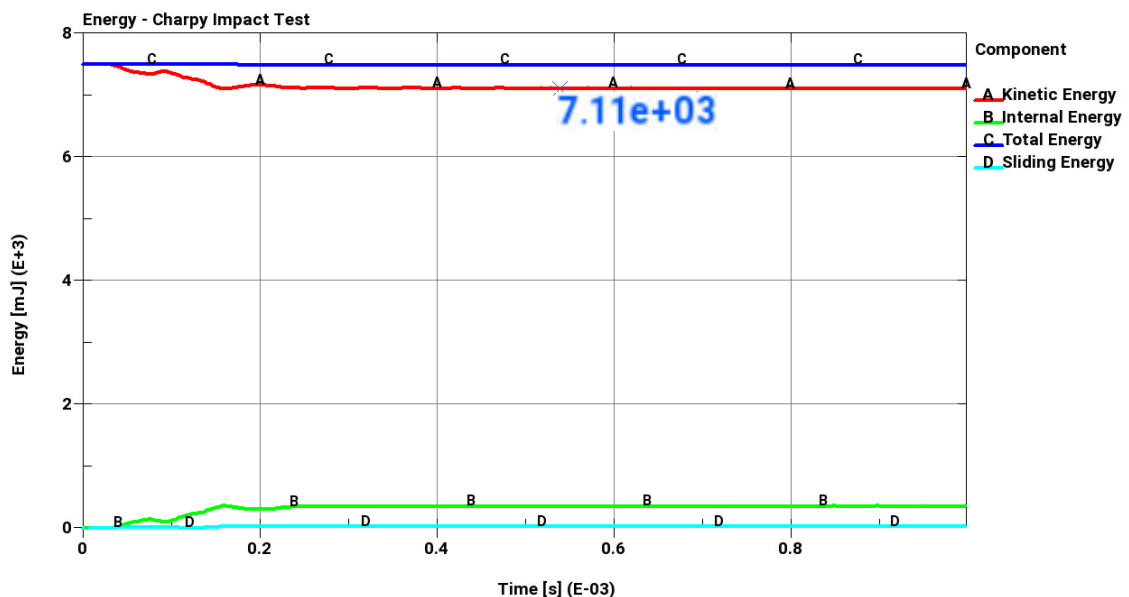


Figure 15 – Energy graph for the Charpy test

Figure 16 shows 1st principle stress as it progresses through the fracture.

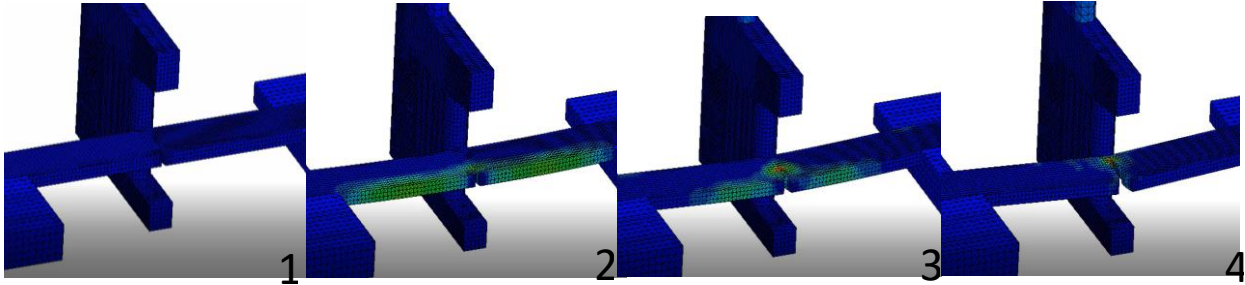


Figure 16 – Progression of first principal stress in the Charpy simulation

As obvious from figure 13, the simulation was successful. After obtaining positive results I ran one more simulation without GISSMO active, to see the difference that GISSMO makes to such a small-scale problem. The difference in simulations with GISSMO on/off was there, but not very noticeable. The main difference it made was the internal shape of the tear. Without GISSMO, the tear was very straight and linear. With GISSMO active, the tear looked much more believable from a real-world standpoint. GISSMO activation seemed to make little to no difference in energy consumption by the material. This can however be explained. This experiment is a very one-dimensional problem. Triaxiality doesn't really play a big part in this specific fracture forming problem. And as was covered by chapter 2.1.1. stress-state is one of the major defining factors for GISSMO fracture modeling.

4.2. CUSTOM DROP WEIGHT TEST – RESULTS

The custom drop weight test results aren't one-hundred percent satisfactory. Though the experiment seemed to have yielded good results i.e. the fractures would form in the same, expected, place in experiments 3 through 5, there still was and is a major difference between the experiment and the simulation.

The biggest problem I was facing when simulating this experiment was that the chain sprocket cover would fracture not only in the area that it should. A major crack would also appear in an area where no fracture formed during the experiment. See figure 17.

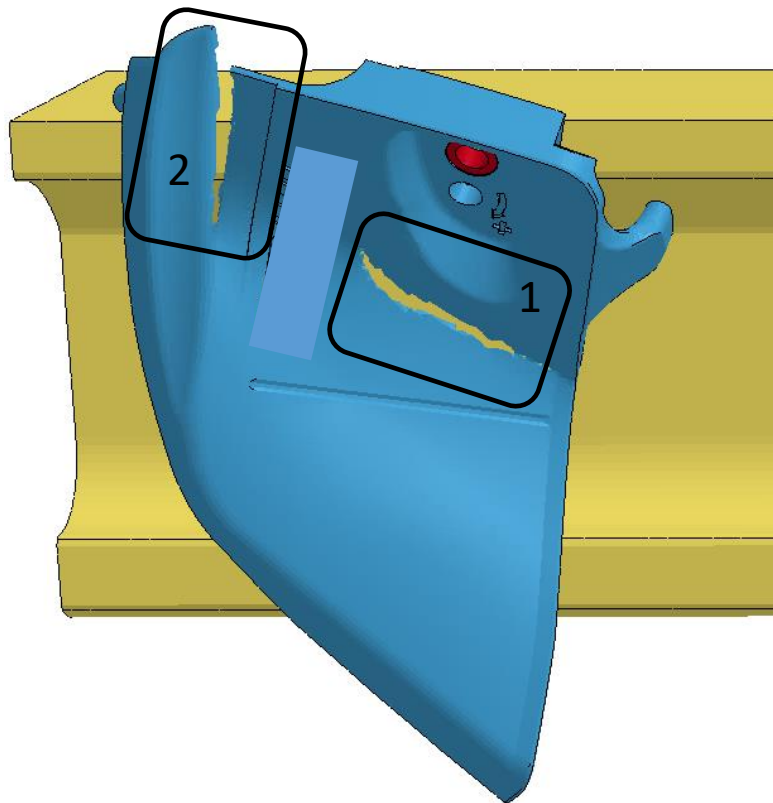


Figure 17 – Custom drop weight test - simulation result

Area labeled 1 is the area where, during the experiment, the chain sprocket cover **did** crack. Area labeled 2, on the other hand, is where **no fracture** formed in any of the experiments.

A possible explanation of this could be, that I used duct-tape during the experiment, to hold the chain sprocket cover in place upon sphere impact. The duct-tape held the chain sprocket cover all the way through area 2. I attempted to simulate the chain sprocket cover's limited movement, due to the duct-tape, by setting the friction coefficient between the chain sprocket cover and the rail to 1 in places, where the duct-tape went over it. Obviously, the contact, as a whole, couldn't have its friction coefficient set to 1, as that would alter the results massively. See figure 18 for the areas with friction coefficient set to 1. This approach didn't change the results much. The fracture in area 2 would still appear. After more iterations and attempts I concluded that the fracture should've indeed appeared during the experiment as well. Due to a lack of resources and equipment, the experiment was haphazardly performed and thus the results vary from the simulation.

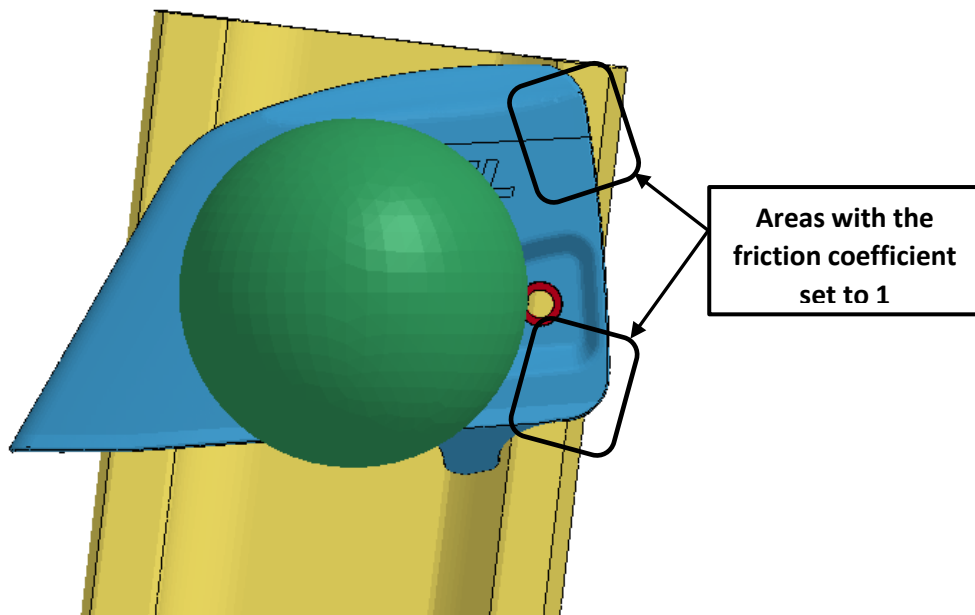


Figure 18 – Custom drop weight test friction setup

The fracture that repeatedly forms in area 1 also suggests that *MAT_ADD_EROSION and GISSMO were setup properly. Since the shape of the curve closely resembles measured data. See appendix I.

5. CONCLUSION

The aim of this thesis was to research the theory and practice behind material failure simulations in LS-DYNA, namely GISSMO (Generalized Incremental Stress State dependent damage MOdel), as well as fracture forming and initiation. Researching how fractures form and under what conditions they begin to sprout is a wide-spread engineering discipline that's used in many branches of industry. Spanning from aerospace through manufacturing to biomechanical engineering.

Another goal of this thesis was to perform experiments in which fractures formed, the Charpy impact test and a custom drop weight test and attempt to simulate these experiments in LS-DYNA, using element erosion and GISSMO. Both these experiments were carried out and their results used as a tool in understanding the subject matter.

Both goals were, to a certain degree, accomplished. The Charpy impact test was performed successfully and its counterpart simulation calibrated properly. The energy consumed by the specimen during fracture in simulation, correlates exactly with measured data during the experiment. As does the size and shape of the fracture. This leads me to believe that the element erosion model with GISSMO was accurately set up.

The custom drop weight test, on the other hand, was a more difficult simulation to fine-tune. Since there are many more variables to take into account, the simulation had to be iterated on significantly more. With simulation results approaching measured data more closely with each iteration. Progress unfortunately also declined with each iteration. The last few simulations I ran seemed to show almost the same result, one that didn't quite match measured data. A fracture would repeatedly form, in simulation, not only in areas where it had during the experiment, but also where the experiment showed no fractures. After thorough thought, this leads me to believe that the experiment was executed inadequately and the data it provided is unreliable.

New insight I gained through research on this topic is undoubtedly valuable. This newly gained knowledge on the inner workings of GISSMO and element erosion is sure to prove useful when simulating problems similar in nature.

LIST OF ABBREVIATIONS

<i>PA6-GF30-I</i>	<i>PolyAmide – Glass Fibre - Impact</i>
<i>FEM</i>	<i>Finite Element Method</i>
<i>FEA</i>	<i>Finite Element Analysis</i>
<i>CAD</i>	<i>Computer-Aided Design</i>
<i>ODE</i>	<i>Ordinary Differential Equation</i>
<i>PDE</i>	<i>Partial Differential Equation</i>
<i>GISSMO</i>	<i>Generalized Incremental Stress State dependent damage MOdel</i>

List of references

- [1] LS-DYNA User's manual
- [2] J. Effelsberg, A. Haufe, M. Feucht, F. Neukamm, P. DuBois. 2012. On the Parameter Identification for the GISSMO Damage Model. 12th International LS-DYNA Users Conference, Detroit.
- [3] F. Andrade, M. Feucht, A. Haufe. 2014. On the prediction of material failure in LS-Dyna: A comparison between Gissmo and Diem, In 13th International LS-DYna Users Conference.
- [4] ČIŽMÁROVÁ, Elena a Jana SOBOTOVÁ. *Nauka o materiálu I. a II.: cvičení*. V Praze: České vysoké učení technické, 2014. ISBN 978-80-01-05550-2.
- [5] Francesca Cosmi, Tensile-properties-of-the-different-grades-of-polyamide https://www.researchgate.net/figure/Tensile-properties-of-the-different-grades-of-polyamide_tbl1_235005131
- [6] ISO 179-2:1997 Plastics -- Determination of Charpy impact properties -- Part 2: Instrumented impact test
- [7] ISO 6603-1:2000 Plastics -- Determination of puncture impact behaviour of rigid plastics -- Part 1: Non-instrumented impact testing.
- [8]. H. Sandberg, O. Rydhol; Evaluation of Material Models to Predict Material Failure in LS-DYNA; 2016
- [9] LS-DYNA Theory Manual
- [10] Ajay Harish; <https://www.simscale.com/blog/2019/01/implicit-vs-explicit-fem/>; 2019
- [11] Torbjörn Johansen, et al. Implicit analyses in LS-DYNA; 2016
- [12] <http://www.lstc.com/products/lstc-prepost>
- [13] *Matematické, fyzikální a chemické tabulky: pro střední školy*. Praha: Ottovo nakladatelství, 2015. ISBN 978-80-7451-472-2.
- [14] A comparison of all-Hexahedra and all Tetrahedral Finite Element Meshes for elastic & elastoplastic analysis. Proceedings 4th International Meshing Round table Sandia National Labs, pp 179-181, Oct. 199

List of figures

Figure 1 – Tensile stress-strain curves for four grades of reinforced polyamide [5]	5
Figure 2 – Diagram of the uniaxial tensile test	6
Figure 3 – Charpy testing machine [4]	7
Figure 4 – Standardized specimen for Charpy impact test.....	7
Figure 5 – Diagram of the custom drop weight test.....	8
Figure 6 – File diagram.....	12
Figure 7 – Notched Charpy testing specimen - meshed.....	14
Figure 8 – The chain sprocket cover mesh	14
Figure 9 – mesh of the tested chain sprocket cover.....	14
Figure 10 – Charpy test specimens	16
Figure 11 – Custom drop weight test - experiment setup.....	18
Figure 12 – Charpy test - simulation setup.....	20
Figure 13 – Comparison of simulation model with experiment.....	24
Figure 14 – Comparison of results and simulation of the custom drop weight test	25
Figure 15 – Energy graph for the Charpy test.....	26
Figure 16 – Progression of first principal stress in the Charpy simulation	27
Figure 17 – Custom drop weight test - simulation result	28
Figure 18 – Custom drop weight test friction setup.....	29

List of tables

Table 1 – Material properties of PA6-GF30-I.....	6
Table 2 – Easy to read comparison of explicit and implicit	11
Table 3 – Charpy impact test results.....	16
Table 4 – Evaluation of the custom drop weight test results.....	18

List of appendices

Appendix I – Photographs of the custom drop weight experiment

Appendix II – Analyse and Baugruppe files of the custom drop weight experiment

APPENDIX I:







APPENDIX II:

Custom drop weight test – Analyse file



```
*KEYWORD
$
*TITLE
9999A_0110_kft0021_x02
$
*INCLUDE_PATH
../../../../k-modell/
$
*INCLUDE_PATH
../../../../G0000/material/lsdyna/
$
$=====
=====
$
$-----
$   !!! Einheiten: t, mm, s, N, MPa !!!
$-----
$
$=====
=====
$                                     Parameter:
$
$=====
=====
$
*Parameter
$ Time History
$--->---1---->---2
$   PRMR1|    VAL1|
$     rTHIST 3.500E-06
$
*Parameter
$ Time step
$--->---1---->---2
$   PRMR1|    VAL1|
$     rTSTEP -0.5E-08
$
*Parameter
$ End Time
$--->---1---->---2
$   PRMR1|    VAL1|
$     rTEND  12.0e-3
$
$=====
=====
$                                     Control Output:
$
$=====
=====
```



```

$
*CONTROL_DEBUG
$
$
*CONTROL_ACCURACY
$-->---1--->---2--->---3--->---4--->---5--->---6--->---7---
->---8
$   osu|      inn|   pidosu|
      4
$
*CONTROL_CONTACT
$-->---1--->---2--->---3--->---4--->---5--->---6--->---7---
->---8
$  slsfac|  rwpnal|  islchk|  shlthk|  penopt|  thkchg|  orien|
enmass|
      1.00      2      2      1      2
$  usrstr|  usrfrc|  nsbcs|  interm|  xpene|  ssthk|  ecdt|
tiedprj|

$  sfric|  dfric|    edc|    vfc|    th|  th_sf|  pen_sf|

$  ignore|  frceng|  skiprwg|  outseg|  spotstp|  spotdel|  spothin|
      2      1
$
$
*CONTROL_ENERGY
$-->---1--->---2--->---3--->---4--->---5--->---6--->---7---
->---8
$   hgen|    rwen|  slnten|    rylen|
      2      2      2
$
$
*CONTROL_OUTPUT
$-->---1--->---2--->---3--->---4--->---5--->---6--->---7---
->---8
$  npopt|  neecho|  nrefup|  iaccop|  opifs|  ipnint|  ikedit|
iflush|
      1      1000
$  iprtf|  ierode|  tet10|  msgmax|  ipcurv|  gmdt|
      1      1      500      0
$
$
*CONTROL_SOLID
$-->---1--->---2--->---3--->---4--->---5--->---6--->---7---
->---8
$  esort|  fmatrix|  niptets|  swlocl|  psfail|  T10JTOL|
      3
$
$
*CONTROL_SOLUTION
$-->---1--->---2--->---3--->---4--->---5--->---6--->---7---
->---8
$   soln|    nlq|   isnan|   lcint|
      2001
$
$
*CONTROL_TERMINATION
$-->---1--->---2--->---3--->---4--->---5--->---6--->---7---
->---8

```



```

$ endtim|   endcyc|   dtmin|   endneg|   endmas|   nosol|
  &TEND
$
$-----
-----
$
*CONTROL_TIMESTEP
$-->----1---->----2---->----3---->----4---->----5---->----6---->----7---
->----8
$ dtinit|   tssfacs|   isdo|   tslimt|   dt2ms|   lctm|   erode|
mslst|
          0.90
          &TSTEP
0
$
$=====$
=====$
$
$
$=====$
=====$
$
$
*DATABASE_BINARY_D3DRLF
$-->----1---->----2---->----3---->----4---->----5---->----6---->----7---
->----8
$   cycl|
      1
$
*DATABASE_GLSTAT
$-->----1---->----2---->----3---->----4---->----5---->----6---->----7---
->----8
$   dt|   binary|
      &THIST
$
*DATABASE_MATSUM
$-->----1---->----2---->----3---->----4---->----5---->----6---->----7---
->----8
$   dt|   binary|
      &THIST
$
*DATABASE_RCFORC
$-->----1---->----2---->----3---->----4---->----5---->----6---->----7---
->----8
$   dt|   binary|
      &THIST
$
*DATABASE_RWFORC
$-->----1---->----2---->----3---->----4---->----5---->----6---->----7---
->----8
$   dt|   binary|
      &THIST
$
*DATABASE_DEFORC
$-->----1---->----2---->----3---->----4---->----5---->----6---->----7---
->----8
$   dt|   binary|
      &THIST
$
*DATABASE_BINARY_D3PLOT

```



```

$   sfs|      sfm|      sst|      mst|      sfst|      sfmt|      fsf|
vsf|
      3          3
$   soft|    sofsc|    lcidab|    maxpar|    sbopt|    depth|    bsort|
frcfrq|
      2
$   penmax|  thkopt|    shlthk|    snlog|      isym|    i2d2d|    sldthk|
sldstf|

$   igap|    ignore|    dpfrac|    dtstif|                                flangl|
cid_rcf|
                                0.05      0.0

$
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_ID
$   cid|                                heading|
      202                                BALL_vs_COVER
$.>...>...1...>...2...>...3...>...4...>...5...>...6...>...7...
.>...8
$   ssid|      msid|      sstyp|      mstyp|      sboxid|      mboxid|      spr|
mpr|
      2          1110          3          3
$   fs|        fd|        dc|        vc|        vdc|      penchk|      bt|
dt|
      0.05      0.05
$   sfs|      sfm|      sst|      mst|      sfst|      sfmt|      fsf|
vsf|
      3          3
$   soft|    sofsc|    lcidab|    maxpar|    sbopt|    depth|    bsort|
frcfrq|
      2
$   penmax|  thkopt|    shlthk|    snlog|      isym|    i2d2d|    sldthk|
sldstf|

$   igap|    ignore|    dpfrac|    dtstif|                                flangl|
cid_rcf|
                                0.05      0.0

$
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_ID
$   cid|                                heading|
      202                                COVER_vs_RAIL
$.>...>...1...>...2...>...3...>...4...>...5...>...6...>...7...
.>...8
$   ssid|      msid|      sstyp|      mstyp|      sboxid|      mboxid|      spr|
mpr|
      2          1111          3          3
$   fs|        fd|        dc|        vc|        vdc|      penchk|      bt|
dt|
      0.4        0.4
$   sfs|      sfm|      sst|      mst|      sfst|      sfmt|      fsf|
vsf|
      3          3
$   soft|    sofsc|    lcidab|    maxpar|    sbopt|    depth|    bsort|
frcfrq|
      2
$   penmax|  thkopt|    shlthk|    snlog|      isym|    i2d2d|    sldthk|
sldstf|

$   igap|    ignore|    dpfrac|    dtstif|                                flangl|
cid_rcf|

```

```

                                0.05      0.0
$
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_ID
$   cid|                      heading|
    203                          duct_tape
$.>...1...>...2...>...3...>...4...>...5...>...6...>...7...
.>...8
$   ssid|      msid|      sstyp|      mstyp|      sboxid|      mboxid|      spr|
mpr|
    150         151         0         0
$   fs|        fd|        dc|        vc|        vdc|      penchk|      bt|
dt|
    1           1           2
$   sfs|      sfm|      sst|      mst|      sfst|      sfmt|      fsf|
vsf|
    3           3
$   soft|    sofsc|    lcidab|    maxpar|    sbopt|    depth|    bsort|
frcfrq|
    2           4.0        5.0
$   penmax|  thkopt|    shlthk|    snlog|    isym|    i2d2d|    sldthk|
sldstf|

$   igap|    ignore|    dpfrac|    dtstif|                                flangl|
cid_rcf|
                                0.05      0.0
$
*CONTACT_TIED_SURFACE_TO_SURFACE_ID
$   cid|                      heading|
    202                          COVER_vs_RING
$.>...1...>...2...>...3...>...4...>...5...>...6...>...7...
.>...8
$   ssid|      msid|      sstyp|      mstyp|      sboxid|      mboxid|      spr|
mpr|
    1           2           3           3
$   fs|        fd|        dc|        vc|        vdc|      penchk|      bt|
dt|
    0.2         0.2           2
$   sfs|      sfm|      sst|      mst|      sfst|      sfmt|      fsf|
vsf|
    3           3
$   soft|    sofsc|    lcidab|    maxpar|    sbopt|    depth|    bsort|
frcfrq|
    2           4.0        5.0
$   penmax|  thkopt|    shlthk|    snlog|    isym|    i2d2d|    sldthk|
sldstf|

$   igap|    ignore|    dpfrac|    dtstif|                                flangl|
cid_rcf|
                                0.05      0.0
$

*CONTACT_AUTOMATIC_SINGLE_SURFACE_ID
$   cid|                      heading|
    1100                          Global_Contact
$.>...1...>...2...>...3...>...4...>...5...>...6...>...7...
.>...8

```

```

$ ssid|      msid|      sstyp|      mstyp|      sboxid|      mboxid|      spr|
mpr|
  1100          0          2
$ fs|        fd|        dc|          vc|          vdc|      penchk|      bt|
dt|
  0.05        0.05
$ sfs|       sfm|        sst|        mst|        sfst|        sfmt|        fsf|
vsf|
  3            3
$ soft|      sofsc|      lcidab|      maxpar|      sbopt|      depth|      bsort|
frcfrq|
  2
$ penmax|    thkopt|      shlthk|      snlog|        isym|        i2d2d|      sldthk|
sldstf|
$ igap|      ignore|      dpfrac|      dtstif|                                flangl|
cid_rcf|
  0.05        0.0
$
*SET_PART_LIST
$-->---1--->---2--->---3--->---4--->---5--->---6--->---7---
->---8
$  sid|      da1|      da2|      da3|      da4|      solver|
  1100
$  pid1|     pid2|     pid3|     pid4|     pid5|     pid6|     pid7|
pid8|
  1          2          1110      1111
$
$=====$
$
$          20xx - Constrained:
$
$=====$
$
$=====$
$
$          40xx - Initial:
$
$=====$
$
$ Cylinder drop heighr 1500mm
$
*INITIAL_VELOCITY_GENERATION
$-->---1--->---2--->---3--->---4--->---5--->---6--->---7---
->---8
$  id|      styp|      omega|      vx|      vy|      vz|      ivatn|
icid|
  1110        2
$  xc|      yc|      zc|      nx|      ny|      nz|      phase|
irigid|
$
$
$=====$
$=====$
$
$          Part Definition
$

```

```

$=====
=====
$
$
*MAT_ADD_EROSION
$.>...1...>...2...>...3...>...4...>...5...>...6...>...7...
.>...8
$      MID      EXCL      MXPRES      MNEPS      EFFEPS      VOLEPS      NUMFIP
NCS
  01130001      12345
2
$.>...1...>...2...>...3...>...4...>...5...>...6...>...7...
.>...8
$      MNPRES      SIGP1      SIGVM      MXEPS      EPSSH      SIGTH      IMPULSE
FAILTM
  12345      140      12345      0.073      12345      12345      12345
$.>...1...>...2...>...3...>...4...>...5...>...6...>...7...
.>...8
$      IDAM      DMGTYP      LCSDG      ECRIT      DMGEXP      DCRIT      FADEXP
LCREGD
  1      1      10      0      Omitted
$.>...1...>...2...>...3...>...4...>...5...>...6...>...7...
.>...8
$      SIZFLG      REFSZ      NAHSV      LCSRS      SHRF      BIAXF

*DEFINE_CURVE
$      LCID      SIDR      SFA      SFO      OFFA      OFFO      DATTYP
  10      0      1.000      1.000      0.000      0.000
$
      A1 x      01 y
      Omitted
$
$
*INCLUDE
9999A_0111_01_bgr.k
$
$=====
=====
$
$
*END

```

```

$$Custom Drop weight test
$Thesis-dx2janat
*KEYWORD memory=250000000
$
$ -----
$ # !!! Einheiten: t, mm, s, N, MPa !!!
$ -----
$
$=====
=====
$                MATERIAL DEFINITIONS:
$=====
=====
$
$
$
*INCLUDE_PATH
../.../G0000/material/lsdyna/
$
*INCLUDE
$ - PA6-GF30-I feucht, 20C, Realfleisskurve
lsdyna_mat_0113_0001.k
$
*INCLUDE
$ - Steel, 20C, elastic
lsdyna_mat_5911_0062.k
$
$
$=====
=====
$                RIGID MATERIALs
$
$=====
=====
$
*MAT_RIGID
$ Steel_RIGID_sphere
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8
$   mid|      ro|      e|      pr|      n|      couple|
m|
   99001  5.5E-09  210000      0.3      0      0
0
$   cmo|      con1|      con2|
   0      0      0
$ card3 :

$
*MAT_RIGID
$ Steel_RIGID_rail
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8
$   mid|      ro|      e|      pr|      n|      couple|
m|

```

```

          99002  7.85E-09  210000  0.3  0  0
0
$      cmo|      con1|      con2|
          1          7          7
$ card3 :

$
$=====
=====
$
$                                PARTS DEFINITIONS
$
$=====
=====
$
$
$
*PART
$
$                                heading|
KETTENRADDECKEL
$...>...1...>...2...>...3...>...4...>...5...>...6...>.
...7...>...8
$      pid|      secid|      mid|      eosid|      hgid|      grav|
adpopt|      tmid|
          2          2  01130001
$
*SECTION_SOLID
$-->---1--->---2--->---3--->---4--->---5--->---6--->---
---7--->---8
$      secid|      elform|      aet|
          2          13          0
$
*PART
$
$                                heading|
RING
$...>...1...>...2...>...3...>...4...>...5...>...6...>.
...7...>...8
$      pid|      secid|      mid|      eosid|      hgid|      grav|
adpopt|      tmid|
          1          1  59110062
$
*SECTION_SOLID
$-->---1--->---2--->---3--->---4--->---5--->---6--->---
---7--->---8
$      secid|      elform|      aet|
          1          -2          0
$
*PART
$
$                                heading|
SPHERE
$...>...1...>...2...>...3...>...4...>...5...>...6...>.
...7...>...8
$      pid|      secid|      mid|      eosid|      hgid|      grav|
adpopt|      tmid|
          1110          1110          99001

```



```

$
*SECTION_SOLID
$--->----1----->----2----->----3----->----4----->----5----->----6----->
---7----->----8
$   secid|   elform|       aet|
      1110       13         0
$
*PART
$                               heading|
RAIL
$...>....1....>....2....>....3....>....4....>....5....>....6....>.
...7....>....8
$   pid|   secid|       mid|   eosid|   hgid|   grav|
adpopt|   tmid|
      1111       1111       99002
$
*SECTION_SOLID
$--->----1----->----2----->----3----->----4----->----5----->----6----->
---7----->----8
$   secid|   elform|       aet|
      1111       13         0
$
*INCLUDE
9999A_0111_01_elm.k
$
$
$
*END

```

Custom drop weight test – Analyse file

```

*KEYWORD
$
*TITLE
9999A_0110_kft0021_x01
$
*INCLUDE_PATH
../../../../k-modell/
$
*INCLUDE_PATH
../../../../G0000/material/lstyna/
$
$=====
=====
$
$

```