DESIGN AND NUMERICAL MODELING OF IMPACT ATTENUATOR

Student : Phu, Ma Quoc
Supervisor : Ing. Zdeněk Poruba, Ph.D.
Date of issue : 08.12.2017
Date of submission: 21.05.2018
Bachelor Thesis Assignment

Student: Phu Ma Quoc
Study Programme: B2341 Engineering
Study Branch: 3901R003 Applied Mechanics
Title: Design and Numerical Modeling of Impact Attenuator
Návrh a numerická simulace tlučiště nárazu

Description:
1) Study and describe rules and requirements on the Formula Student car design with the emphasis on the impact attenuator.
2) Propose and describe several designs of impact attenuators.
3) Perform crash simulation of chosen impact attenuators using finite element method.
4) Evaluate results calculated and recommend further works for deployment of impact attenuator into Formula Student car.

References:

Extent and terms of a thesis are specified in directions for its elaboration that are opened to the public on the websites of the faculty.

Supervisor: Ing. Zdeněk Poruba, Ph.D.
Date of issue: 08.12.2017
Date of submission: 21.05.2018

Ing. Martin Fasek, Ph.D.
Head of Department

doc. Ing. Ivo Blažej, Ph.D.
Dean
ABSTRACT

The main objective of this thesis is to develop an Impact Attenuator for the car racing Formula Student competition by designing and performing Crash test simulation numerically. The thesis studies into the rules and requirements of Formula Student for the car design with the emphasis on the Impact Attenuator. A number of designs made of Aluminum Honeycomb material are proposed and tested using Finite Element Method. The data from the simulations is then evaluated and recommendations for further works are stated.

Keywords: Impact Attenuator, Formula Student, Crash test simulation, Aluminum Honeycomb
# TABLE OF CONTENTS

BACHELOR THESIS ASSIGNMENT ........................................................................................................ 2
ABSTRACT ............................................................................................................................................... 3
TABLE OF CONTENTS ........................................................................................................................... 4
TABLE OF FIGURES .................................................................................................................................. 6
TABLE OF TABLES .................................................................................................................................... 7
1. Introduction .............................................................................................................................................. 8
2. List of Terms and Symbols ........................................................................................................................ 9
   2.1. Terms ................................................................................................................................................. 9
   2.2. Symbols .............................................................................................................................................. 9
3. Literature Review .................................................................................................................................... 10
   3.1. Rules ................................................................................................................................................. 10
      3.1.1. Designing requirements ............................................................................................................ 10
      3.1.2. Testing requirements ................................................................................................................ 11
   3.2. Designed components ..................................................................................................................... 12
      3.2.1. Anti-Intrusion Plate ................................................................................................................... 12
      3.2.2. Front Bulkhead .......................................................................................................................... 13
      3.2.3. Full assembly ............................................................................................................................. 14
   3.3. Testing methods............................................................................................................................... 16
      3.3.1. Quasi-static test ........................................................................................................................ 16
      3.3.2. Drop test ................................................................................................................................... 16
      3.3.3. Dynamic crash test .................................................................................................................... 16
   3.4. Aluminum Honeycomb .................................................................................................................... 17
      3.4.1. Terminologies............................................................................................................................ 17
      3.4.2. Material ..................................................................................................................................... 17
      3.4.3. Profiles and Properties .............................................................................................................. 18
      3.4.4. Failure mode ............................................................................................................................. 20
   3.5. Crash simulation .............................................................................................................................. 23
      3.5.1. Definition .................................................................................................................................. 23
      3.5.2. Pros and Cons ............................................................................................................................ 25
      3.5.3. FEM method in Crash simulation .............................................................................................. 25
4. Methods and Results .............................................................................................................................. 26
   4.1. Analytical solution ........................................................................................................................... 26
   4.2. Materials .......................................................................................................................................... 27
TABLE OF FIGURES

Figure 1: General Formula Student car structure with Impact Attenuator highlighted [1] ............ 8
Figure 2: Real Crash test setup [4] .......................................................................................... 11
Figure 3: Real Crash test result [4] .......................................................................................... 11
Figure 4: Anti-Intrusion Plate drawing ...................................................................................... 12
Figure 5: Extension for Bolting drawing .................................................................................... 13
Figure 6: Assembly of Front Bulkhead with Extensions drawing ............................................. 13
Figure 7: Full Assembly of Anti-Intrusion Plate with Front Bulkhead ....................................... 14
Figure 8: Standard Impact Attenuator 3D model ...................................................................... 14
Figure 9: Proposed Aluminum honeycomb Impact Attenuator 3D model ................................. 15
Figure 10: Optimized Aluminum honeycomb Impact Attenuator 3D model ............................ 15
Figure 11: Hexagon Honeycomb terminology [7] .................................................................... 17
Figure 12: Typical Load-Deflection Curve of Honeycomb [10] ................................................ 19
Figure 13: Aluminum Honeycomb Crush Curve [10] ................................................................ 20
Figure 14: Out-of-plane Elastic Buckling of a Honeycomb cell [11] ......................................... 21
Figure 15: Sandwich structure with Honeycomb core under Uniform load [12] ....................... 21
Figure 16: Honeycomb core failure in Sandwich panel [13] ..................................................... 22
Figure 17: Real test of failure mode of Thin-walled Structure [14] ............................................ 22
Figure 18: Plastic Collapse of the Honeycomb cell [11] ............................................................. 23
Figure 19: Car crash simulation and real Crash test advertisement [15] ................................. 24
Figure 20: Small number of Honeycomb cells geometry .......................................................... 31
Figure 21: End base surfaces for attachment ........................................................................... 31
Figure 22: Constraints .............................................................................................................. 32
Figure 23: Small number of Honeycomb cells failure mode .................................................... 32
Figure 24: Bigger number of cells geometry .......................................................................... 33
Figure 25: Bigger number of cells failure mode ....................................................................... 34
Figure 26: Version 1 of Full Assembly geometry .................................................................... 35
Figure 27: Constraints ............................................................................................................. 35
Figure 28: Version 1 Honeycomb pack failure mode ................................................................. 36
Figure 29: Version 1 AIP failure mode in Top view ................................................................. 36
Figure 30: Version 2 geometry ............................................................................................... 37
Figure 31: Version 2 Honeycomb pack failure mode ............................................................... 37
Figure 32: Version 2 AIP failure mode in Top view ................................................................. 38
Figure 33: Version 3 geometry ............................................................................................... 39
Figure 34: Version 3 Honeycomb pack failure mode ............................................................... 39
Figure 35: Version 3 AIP failure mode in Top view ................................................................. 40
Figure 36: Version 4 geometry ............................................................................................... 41
Figure 37: Version 4 Honeycomb pack failure mode ............................................................... 41
Figure 38: Version 4 AIP failure mode in Top view ................................................................. 42
Figure 39: Version 5 geometry ............................................................................................... 43
Figure 40: Version 5 Honeycomb pack failure mode ............................................................... 43
Figure 41: Version 5 AIP failure mode in Top view ................................................................. 44
Figure 42: Version 6 geometry ............................................................................................... 45
Figure 43: Version 6 Honeycomb pack failure mode ............................................................... 45
Figure 44: Version 6 AIP failure mode in Top view ................................................................. 46
Figure 45: Summary of Simulation Data sheet ..................................................................... 46
Figure 46: Version 2 and Real Crash test with one (1) diagonal tube comparison [17] ........... 47
Figure 47: Version 6 and Real Crash test (cut and pre-crushed) comparison [18] .................. 48
Figure 48: New Front Bulkhead drawing ............................................................................... 51
Figure 49: New IA drawing .................................................................................................... 52
TABLE OF TABLES

Table 1: Properties of Aluminum Alloy 5052 [8] ............................................................................... 17
Table 2: Used profiles of Hexagon Honeycomb [9] ............................................................................... 18
Table 3: Comparison of Specific strengths between solid Aluminum and Aluminum Honeycomb ....................................................................................................................................................... 19
Table 4: Aluminum Alloy 5052 properties for Simulation .................................................................. 27
Table 5: Structural Steel properties for Simulation ................................................................................. 27
Table 6: Structural Steel Unreal properties for Simulation ................................................................. 28
Table 7: General setup of Components ................................................................................................. 28
Table 8: Shell and Size of elements of Components ............................................................................ 29
1. Introduction

Formula Student (FS) is a multi-national car racing competition established for engineering and business university students. The competition format is to provide an ideal environment for students to perform designing, testing, improving their capabilities to deliver a complex product in the framework of a motorsport competition. The final product representing the students’ home university is delivered to compete with other universities from different nations. [2]

The competition consists of two parts which are graded: the vehicle design (in technology and business wise) and the actual race. The team who scores the most is the winner. The vehicle is built of two types of components: own-designed and bought ones. The team earns points from own-designed components but not bought one.

The VSB team was using the commercially available Standard IA made by Foam. Success in designing its own IA following a number of requirements mentioned in the FS rules will gain the team extra points.

Figure 1: General Formula Student car structure with Impact Attenuator highlighted [1]
2. List of Terms and Symbols

2.1. Terms

**Front Bulkhead**: The planar structure which makes up the forward plane of the Frame. Its function is to protect the driver’s feet. [3]

**Impact Attenuator (IA)**: A deformable, energy-absorbing structure located forward of the Front Bulkhead. [3]

**Anti-Intrusion Plate (AIP)**: The plate in between the Front Bulkhead and the Impact Attenuator. It helps to protect the driver’s feet. [3]

**Impact Attenuator Assembly**: The assembly includes Front Bulkhead, Anti-Intrusion Plate, Impact Attenuator and some tubes representing the front part of the Frame.

**Aluminum Honeycomb**: A structure of hexagon cells made with thin layers of aluminum imitating the nature honeycomb shape. It is orthotropic material, widely used for minimal strength-to-weight and energy absorbing ability.

**Out-of-plane**: The plane is normal to the hexagon cell surfaces of the Honeycomb. The Honeycomb for application of energy absorption is mostly compressed in this plane.

2.2. Symbols

\( F \): force (N)

\( m \): mass (kg)

\( t_{\text{impact}} \): time of impact, approximation

\( v_{\text{impact}} \): velocity at impact (m/s^2)

\( a_{\text{average}} \): average acceleration (m/s^2)

\( K_\varepsilon \): kinetic energy (J)
3. Literature Review

3.1. Rules

3.1.1. Designing requirements

Front Bulkhead [3]:

- Made of round tube 25.0 (25.4) mm x 1.75 (1.6) mm or square 25.0 mm x 25.0 mm x 1.20 mm from mild or alloy, steel (minimum 0.1% carbon)
- If the Anti-Intrusion Plate extends beyond the Impact Attenuator more than 25 mm on any side, there must be a diagonal or X-brace made from round tube 25.0 mm x minimum 1.2 mm added

Impact Attenuator (IA) [3]:

- Must be mounted directly to the Front Bulkhead or the Anti-Intrusion Plate
- Is separated from the frame
- Not able to penetrate the Front Bulkhead
- Designed with a closed front section
- Dimensioning to at least 100 mm high x 200 mm wide x 200 mm forward the Front Bulkhead
- The proposed material to use for IA from the team consultant is Aluminum Honeycomb to replace the last year version, made of Foam.

Anti-Intrusion Plate (AIP) [3]:

- Made of 1.5 mm solid steel or 4.0 mm solid aluminum plate
- The same size as the outside dimensions of the Front Bulkhead
- If welded to the Front Bulkhead, it must extend at least to the centerline of the Front Bulkhead tubing in all directions
- If bolted to the Front Bulkhead, it must be attached by a minimum of eight 8mm 8.8 bolts distancing at least 50 mm from each other, with positive locking
3.1.2. Testing requirements

There are certain testing requirements mentioned in the rules [3]:

- During testing, the IA must be attached to the AIP using the real case attaching method
- The Impact Attenuator Assembly must be attached to a representative of a section of the frame. There must be at least 50 mm distance from the AIP to the fix surface

![Image 2](image2.jpg)

**Figure 2: Real Crash test setup [4]**

- No part of the AIP would deflect permanently more than 25 mm from its position before the test

![Image 3](image3.jpg)

**Figure 3: Real Crash test result [4]**
- Test data has to prove that the Impact Attenuator Assembly, when mounted on the front of a vehicle with a total mass of at least 300 kg impacting a flat rigid barrier, with the velocity of impact to be 7.0 meter/second, decelerates the vehicle at the rate of not exceeding 20 g’s in average and 40 g’s in peak. The energy absorbed in this case must meet or exceed 7350 Joules.

3.2. Designed components

The below components are from the current Vehicle (Vector 4) which is used in this year’s race (Formula Student 2018):

3.2.1. Anti-Intrusion Plate

The AIP was decided to be 1.5mm solid steel plate with bolting joining method:

Figure 4: Anti-Intrusion Plate drawing
3.2.2. Front Bulkhead

Figure 5: Extension for Bolting drawing

Figure 6: Assembly of Front Bulkhead with Extensions drawing
3.2.3 Full assembly

Figure 7: Full Assembly of Anti-Intrusion Plate with Front Bulkhead

Standard IA:

Figure 8: Standard Impact Attenuator 3D model
Proposed Honeycomb IA:

Figure 9: Proposed Aluminum honeycomb Impact Attenuator 3D model

Figure 10: Optimized Aluminum honeycomb Impact Attenuator 3D model
3.3. Testing methods

There are many methods to test the stiff-worthiness of the IA. Below are the most used tests:

3.3.1. Quasi-static test

The IA Assembly is fixed vertically under a “presser”- usually a hydraulic cylinder, on a rigid and horizontal surface. This cylinder can exert different levels of constant loadings and displacements in time on the structure. The loads and displacements are recorded. Through using the peak load and the average load divided by the mass of the vehicle, the analyzer can calculate the decelerating properties (peak and average decelerations) of the structure. The structure is pressed more uniformly and completely under this way of testing. [5]

Pros and cons: Low cost of preparation. The structure is crushed completely. Thus, there is not the final shape of it under real impacting situation.

3.3.2. Drop test

The IA assembly is set up vertically in line with a mass which will be dropped onto the structure. The mass and drop height will be calculated to have the potential energy as much as the kinetic energy of a 300 kg vehicle travels at 7m/s would have. How the structure is deformed and the decelerations of the part of the vehicle will be recorded with the help of a high speed camera and sensors. [6]

Pros and cons: Low cost of preparation. The dropped mass, after the testing, will remain on the structure. This exerts more energy onto the structure which causes more deformation.

3.3.3. Dynamic crash test

The IA assembly is set up horizontally in line with a “crusher”. The “crusher” is a flat rigid surface which will be accelerated to have the same kinetic energy as a 300 kg vehicle travels at 7 m/s would exert. This “crusher” will hit the IA assembly. The deformation of the structure and the decelerations of the “crusher” will be recorded with the help of a high speed camera and sensors. [4]

Pros and cons: High cost of preparation. The test yields the nearest result to the reality as an elastic collision.
3.4. Aluminum Honeycomb

3.4.1. Terminologies

Figure 11: Hexagon Honeycomb terminology [7]

The Honeycomb structure is made of layers of Aluminum sheets with T: cell thickness. As can be observed from the figure, the in between thicknesses are 2 times the incline walls. Those terminologies are important for choosing the profiles in the manufacturer’s catalogue for the application.

3.4.2. Material

The material of the Honeycomb is chosen to be Aluminum Alloy 5052 with following properties that are important for further setting in ANSYS:

<table>
<thead>
<tr>
<th>PHYSICAL PROPERTIES</th>
<th>UNIT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>2.68</td>
</tr>
<tr>
<td>Mechanical Properties</td>
<td></td>
</tr>
<tr>
<td>Tensile Yield Strength</td>
<td>193</td>
</tr>
<tr>
<td>Young Modulus</td>
<td>70.3</td>
</tr>
<tr>
<td>Poisson Ratio</td>
<td>0.33</td>
</tr>
</tbody>
</table>

Table 1: Properties of Aluminum Alloy 5052 [8]
### 3.4.3. Profiles and Properties

The following profiles with Mechanical Properties are chosen from the manufacturer’s catalogue:

<table>
<thead>
<tr>
<th>Name</th>
<th>Cell size inch</th>
<th>Nominal foil gauge inch</th>
<th>Nominal density pcf</th>
<th>Strength psi</th>
<th>Modulus ksi</th>
</tr>
</thead>
<tbody>
<tr>
<td>PAMG-XR1-3.0-⅜-20-N-5052</td>
<td>⅜ (9.525 mm)</td>
<td>.002 (0.0508 mm)</td>
<td>3.0 (0.0480 g/cc)</td>
<td>260 (1.8 MPa)</td>
<td>70 (0.5 MPa)</td>
</tr>
<tr>
<td>PAMG-XR1-5.4-⅜-40-N-5052</td>
<td>⅜ (9.525 mm)</td>
<td>.004 (0.1016 mm)</td>
<td>5.4 (0.0865 g/cc)</td>
<td>720 (5.0 MPa)</td>
<td>200 (1.4MPa)</td>
</tr>
</tbody>
</table>

Table 2: Used profiles of Hexagon Honeycomb [9]

The name in order: Aerospace grade aluminum- XR1 corrosion coating- Nominal Density- Nominal Foil gauge x10000- Not perforated (N)- Alloy of the foil. [9]

The Aluminum Honeycomb is orthotropic, its strengths are different in W and L direction (see terminologies). It is widely used in Aerospace, Automobile, Marine and Military due to its high strength/stiffness-to-weight ratio and excellent energy absorbing ability. Besides, Honeycomb has excellent and constant crush strength, structural integrity, high fatigue resistance, corrosion resistance and flammability resistance,... [10]
There are highlighted properties to be used in this application, which will be explained in details:

**STRENGTH TO WEIGHT RATIO**

<table>
<thead>
<tr>
<th>Material</th>
<th>Tensile strength (MPa)</th>
<th>Density (g/cc)</th>
<th>Specific strength (kN.m/kg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminum Alloy 5052 (solid)</td>
<td>193</td>
<td>2.68</td>
<td>72</td>
</tr>
<tr>
<td>PAMG-XR1-3.0-¾-20-N-5052</td>
<td>260</td>
<td>0.0480</td>
<td>5417</td>
</tr>
<tr>
<td>PAMG-XR1-5.4-¾-40-N-5052</td>
<td>720</td>
<td>0.0865</td>
<td>8324</td>
</tr>
</tbody>
</table>

Table 3: Comparison of Specific strengths between solid Aluminum and Aluminum Honeycomb

Aluminum Alloy 5052 after being constructed into Honeycomb structure has significantly high Specific Strength.

**Crush strength**

For energy absorbing application, the Honeycomb structures are usually loaded in the direction normal to the hexagon surfaces of the cells, namely Out-of-plane loaded. In case of compressive loading, after the Honeycomb structure reaches its ultimate strength, the structure will deform plastically and crush uniformly.

![Figure 12: Typical Load-Deflection Curve of Honeycomb [10]](image)

As can be seen from the figure, the structure will crush at a constant stress level. This gives the structure the excellent energy absorbability, in which the degree of structural deformation can even be pre-determined.
When being used in this manner, the Honeycomb is pre-crushed in order to skip the peak load region:

![Figure 13: Aluminum Honeycomb Crush Curve][10]

3.4.4. Failure mode

The behaviour of the Honeycomb used as IA is studied from previous researches and Crash test videos available on Internet to justify the reliability of the simulations.

Under the Out-of-plane compressive load, after reaching its yield strength, the structure experiences first the elastic buckling on the thin-walled cells within the free spaces of the cells. This will change the load path yielding the uniaxial load. The uniaxial load will cause the plastic collapse of the structure (over ultimate strength) through “dimpling” (the cell walls are “hinged” among the free space of the cells, which will be explained below). This is the most important stage as the kinetic energy exerts from the moving vehicle will be converted (ideally-completely) to the deformation energy of the Honeycomb pack.
Elastic Buckling

Figure 14: Out-of-plane Elastic Buckling of a Honeycomb cell [11]

There are diamond patterns where the cell walls are going to buckle.

Figure 15: Sandwich structure with Honeycomb core under Uniform load [12]

The behaviour of the Honeycomb as a core of a Sandwich structure (mostly used in aerospace or automotive industry for its stiffness) is also studied. The Sandwich structures consists of a Honeycomb core with two plates jointed on the honeycomb’s two end planes. In the figure, the Sandwich beam is placed under a uniform load on one plate and supported in the opposite plate to study the buckling behaviour of the core. This can be a good prediction of how the Honeycomb IA will deform as it is similar to the Crash test (Honeycomb pack is placed between the AIP and (hit) the flat Rigid wall).
There are two types of stresses acting onto the Honeycomb cores in this case: compressive stress and shear stress. “Dimpling” happens when the core of the Sandwich panel is made of cell-structure like Honeycomb cells and the plates are joined at the cells’ edges. The cell walls can deform within the free space between others under evenly distributed compression.

The picture below shows the “Dimpling” effect of a sample made of thin-walled metal plates:
Plastic collapse

Plastic collapse stage is when the cell walls are “hinged”. This happens after the “dimpling” effect which reduces the Honeycomb pack’s height significantly bringing up the specifically high energy absorbability of this material.

![Figure 18: Plastic Collapse of the Honeycomb cell](image)

3.5. Crash simulation

3.5.1. Definition

**Crash simulation** is the virtual representation of the destructive crash test of a vehicle with the help of a computer. It is used to determine the crashworthiness of the vehicle and the occupants’ safety in case of crashing. The Crash simulations are done during the stage of Finite Element Analysis work before the prototypes are manufactured for real testing. Important results are the deformation (of the vehicle, the occupants’ interaction with the vehicle’s components,...), the decelerations (e.g. head
deceleration,...). Those must satisfy a number of required values stated in the Car safety rules (relating to NHTSA). [15]

Crashworthiness of a vehicle structure is the ability to absorb and transfer the energy from crash into plastic deformation of the chassis in the manner of making sure that the occupants receive as less damage as possible. [20]

Figure 19: Car crash simulation and real Crash test advertisement [15]
3.5.2. Pros and Cons

The crash simulation allows the analyzers to:

- Solve the highly nonlinear problems which can never be done without the help of a computer
- Virtually present the destructive crash test in practice with relatively high accuracy before coming to the real test
- Reduce significantly the cost of manufacturing the testing samples
- Provide relatively trustworthy results for optimizing the design concepts (usually the Chassis) for better crash tests

However, there are some drawbacks, the method demands:

- High calculation time
- High efficiency of computational processing units (softwares and hardwares)

3.5.3. FEM method in Crash simulation

In a crash simulation, the calculation of the vehicle structure is carried out with the help of spatial discretizations. The method divides up the continuous movement of the whole structure in real time into relatively small changes in position over small and discrete time steps. This also divides the body into a finite number of quadrilateral or triangular elements which have calculatable size. Each node and element do effect of forces and moments caused by its linear movement to their neighboring nodes and elements. Behavior of each of them will be calculated and recorded to sum up the final result of the simulation. [16]
4. Methods and Results

4.1. Analytical solution

According to the rule:

a) The car of 300 kg travelling at 7 m/s hits a rigid wall. Design the IA so that it can fully stop the car and absorb at least 7350 Joules of kinetic energy exerted from the car in this event. The decelerations of the car are not more than 20 g’s average and 40 g’s peak.

- **Kinetic energy in this event:**
  \[ K_e = \frac{1}{2} m v_{impact}^2 = \frac{1}{2} \times 300 \text{kg} \times (7 \text{m/s})^2 = 7350 \text{J} \]

- **Time of impact, approximation:**
  \[ t_{impact} = \frac{v_{impact}}{a_{average}} = \frac{7 \text{m/s}}{20 \times 9.81 \text{m/s}^2} = 0.0357 \text{s} \]

b) No part of the AIP after impact deflects more than 25mm in any of their original position.

There are dependently complex behaviours of the IA, the AIP and the Front Bulkhead, even the collapse of the IA. Those cannot be solve by hand but to predict with FEM on computer with accessibly high accuracy.
4.2. Materials

There are three (03) separate materials- two (02) realistics and one (01) unrealistic:

<table>
<thead>
<tr>
<th>PROPERTY</th>
<th>VALUE</th>
<th>UNIT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>2680</td>
<td>kg.m^-3</td>
</tr>
<tr>
<td><strong>Isotropic Elasticity</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Young’s Modulus</td>
<td>70300</td>
<td>MPa</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.33</td>
<td></td>
</tr>
<tr>
<td><strong>Bilinear Isotropic Hardening</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Yield Strength</td>
<td>193 MPa</td>
<td>MPa</td>
</tr>
<tr>
<td>Tangent Modulus</td>
<td>1 MPa</td>
<td></td>
</tr>
</tbody>
</table>

Table 4: Aluminum Alloy 5052 properties for Simulation

<table>
<thead>
<tr>
<th>PROPERTY</th>
<th>VALUE</th>
<th>UNIT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>7850</td>
<td>kg.m^-3</td>
</tr>
<tr>
<td><strong>Isotropic Elasticity</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Young’s Modulus</td>
<td>2E+05</td>
<td>MPa</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.3</td>
<td></td>
</tr>
<tr>
<td><strong>Bilinear Isotropic Hardening</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Yield Strength</td>
<td>200</td>
<td>MPa</td>
</tr>
<tr>
<td>Tangent Modulus</td>
<td>1000</td>
<td>MPa</td>
</tr>
</tbody>
</table>

Table 5: Structural Steel properties for Simulation
### STRUCTURAL STEEL UNREAL

<table>
<thead>
<tr>
<th>PROPERTY</th>
<th>VALUE</th>
<th>UNIT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>2.14E+05</td>
<td>kg.m^-3</td>
</tr>
</tbody>
</table>

#### Isotropic Elasticity

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s Modulus</td>
<td>2E+05</td>
<td>MPa</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.3</td>
<td></td>
</tr>
</tbody>
</table>

#### Bilinear Isotropic Hardening

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield Strength</td>
<td>200</td>
<td>MPa</td>
</tr>
<tr>
<td>Tangent Modulus</td>
<td>1000</td>
<td>MPa</td>
</tr>
</tbody>
</table>

Table 6: Structural Steel Unreal properties for Simulation

The density was set so that the Vehicle will have the mass of 300 kg with relatively small thickness.

<table>
<thead>
<tr>
<th>COMPONENTS</th>
<th>MATERIAL</th>
<th>THICKNESS (mm)</th>
<th>TYPE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Honeycomb pack</td>
<td>Aluminum Alloy 5052</td>
<td>9.525x0.05x0.1/9.525x0.025x0.05 (base= 2x inclined)</td>
<td>Deformable</td>
</tr>
<tr>
<td>Plate</td>
<td>Structural Steel</td>
<td>1.5</td>
<td>Deformable</td>
</tr>
<tr>
<td>Section of Frame</td>
<td>Structural Steel</td>
<td>2/3/4</td>
<td>Deformable</td>
</tr>
<tr>
<td>Wall</td>
<td>Structural Steel Unreal</td>
<td>1 (1 mm away from Honeycomb pack)</td>
<td>Rigid</td>
</tr>
<tr>
<td>Vehicle</td>
<td>Structural Steel Unreal</td>
<td>10</td>
<td>Rigid</td>
</tr>
</tbody>
</table>

Table 7: General setup of Components
4.3. Meshing

The function Auto Meshing with Shell element type [21] was used. Then, the function Sizing was used for important components: the Section of Frame and the Honeycomb pack:

<table>
<thead>
<tr>
<th>Components</th>
<th>Type of Element</th>
<th>Size (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plate, Wall, Vehicle</td>
<td>Shell</td>
<td>10 (Default)</td>
</tr>
<tr>
<td>Honeycomb pack</td>
<td>Shell</td>
<td>7</td>
</tr>
<tr>
<td>Section of Frame</td>
<td>Shell</td>
<td>6</td>
</tr>
</tbody>
</table>

Table 8: Shell and Size of elements of Components

There are different local elements size comparing to global ones due to the complexity of the geometry (small edges, circular tubes, connection between tubes...).

4.4. Contacts

- The separate Honeycomb walls were connected by a Group. The Connection function Face/Face, Face/Edge, Edge/Edge between them were turned on (Yes) with Tolerance of 0.5 mm
- Bonded Face/Face connection between the surfaces in the base of the Honeycomb pack and the surface of the AIP
- Bonded Face/Face connection with Trim Contact turned off (Maximum offset 3mm) between the Front Bulkhead tubes and the AIP
- Bonded Edge/Face connection of the four connecting tubes to the Vehicle plane

4.5. Constraints

- The Rigid wall was placed 1 mm apart, parallel to and forward from the rearest surface of the Honeycomb pack
- The Rigid wall was constrained by Fix Support function
- All the other Components (Honeycomb pack, AIP, Frame and Vehicle plane) were constrained by moving in Oz direction with velocity of +7000 mm/s
It is essential to have all the other Components except the Rigid wall move at the same velocity. Because in Explicit dynamics simulation, a single element with a considerably high kinetic energy can yield extreme effect on the structure as it moves with considerably high velocity impacting in very short time. For example, some of the Honeycomb cell walls within the IA structure which are not fully constrained with their neighboring walls will change the result significantly.

Moreover, the time set up of the simulation in ANSYS plays an relevant role in the overall time of calculation. As the time of impact calculated analytically to be approximately 0.0357s, the expected minimum time for the 300 kg body to travel a small distance till hitting the rigid wall plus the time it takes to fully stop would be not less than 0.0357s. However, in the very first trials, failure of the Impact Attenuator Assembly can be witnessed early enough. Thus, the author started with only small set up time, from 0.01 s in time accompanied by continous improvement of it and the geometry.

4.6. Other setups

- ANSYS Workbench for Researcher, Version 18.2 with 4 processors was used to calculate the problem
- Automatic Mass Scaling was turned on (Yes) from Version 3, for the advantages of calculating time and avoiding the Error of too large Energy [22]

4.7. Geometries, Boundary conditions and Simulation results

The most important results in Crash test simulation are the Deformation shape of the structure, the Deceleration rate and the Final velocity of the vehicle.

The Simulation results pictures below were put in True Scale for virtually presentation of the deformation shapes. All the figure in the deformation scale to the left hand of the result screens are not correct due to the collapsed elements’ (which experience stress over the yield strength limit, denoted as red points) flying out of the structure and performed significant deformation. Data of Decelerations and Final velocities are listed then.
4.7.1. Failure mode simulation

Small number of cells

A portion of Honeycomb cells was created to figure out how the honeycomb cells would deflect under Out-of-plane compressive load:

Figure 20: Small number of Honeycomb cells geometry

The portion was 15 mm high, cell size of 9 mm x 0.1 mm constructed in ANSYS by 0 thickness surfaces which will be added later. Vehicle of 1.5 kg travels at 7m/s was constructed by a solid block.

Figure 21: End base surfaces for attachment

Base surfaces were constructed for attaching the Honeycomb pack with the front-most plane of the vehicle.
A rigid wall is placed at a small distance (1mm) away from the highest plane of the Honeycomb pack with Fixed support function. The full assembly of the car moves in Oz axis with speed at +7000 mm/s. Run time is 1.00E-03 s.

Shape of failure was presented by checking the Total Deformation result file. “Dimpling” happened within the middle length of the cell walls. In contrary to the fixed end edges of the vehicle, which experienced slightly bending, the free end edges were bent significantly and “hinged” into the free space of the cells in relation with the neighboring cells. This could be explained by the fact the free end of the cell walls were not joined to a rigid surface like in the Sandwich panel.
Bigger number of cells

A portion of honeycomb pack (more complex shape) was then constructed and simulated:

Figure 24: Bigger number of cells geometry

The base surfaces were distributed in five (5) important locations for attachment of the Honeycomb pack to the AIP. That was done because the ANSYS software has problem with creating one surface covering the whole base edges. Moreover, the Solid block car was replaced with a Plate of 0 mm thickness for reduction of calculation time. The Plate were added with thickness and mass later via the Shell elements and Unrealistic material with high density.
The whole structure was simulated impacting the rigid wall at the same velocity with simulation time of \(3.0\times10^{-2}\) s.

The fixed edges with five (5) surfaces experienced no significant deflection meaning that method of attachment using them could be used for further simulation. The cells around the rectangular Portion showed the General bending while the free end of the in between cells were “hinged” to collapse.

Figure 25: Bigger number of cells failure mode

The fixed edges with five (5) surfaces experienced no significant deflection meaning that method of attachment using them could be used for further simulation. The cells around the rectangular Portion showed the General bending while the free end of the in between cells were “hinged” to collapse.
4.7.2. Version 1

The proposed shape with fixed dimensions was created with the evolvement of the front representative of the frame. This was meant to see the relation of the frame’s tubes, the AIP in between and the Honeycomb pack (hidden for better view of the structure). The distance between the AIP and the Vehicle plane is a little more than 75 mm. The tubes used were round 25 mm x 2 mm thickness.

Figure 26: Version 1 of Full Assembly geometry

Figure 27: Constraints
The Honeycomb pack barely deformed. There were red dots flying out of the pack because this is how the ANSYS deletes the affect of those elements which experience overstress. The red scale of 3000.7 mm max was indicator of the distance travelled by those elements which had high initial kinetic energy.

The AIP deformed significantly. As can be seen from the figure, its middle zone touched the rigid surface of the vehicle, which is more than 75 mm away. The cause was realized later by the improper contact between the tubes and the AIP. The author’s intention was the AIP to be be welded to the tubes and should not be torn out from the tubes edges that easily.
4.7.3. Version 2

There is one diagonal tube which was added to the Front Bulkhead:

Figure 30: Version 2 geometry

The diagonal tube was to make sure that the AIP would work as an Anti-Intrusion as it should.

Figure 31: Version 2 Honeycomb pack failure mode

The Honeycomb pack deformed more in the bottom. The diagonal tube did make sure that there was stress to the Honeycomb pack.
Figure 32: Version 2 AIP failure mode in Top view

The AIP still deformed significantly because in this version, the improper contact between the AIP and the tubes was not solved.

However, this brought up a good direction for later simulations. As can be observed from the simulating video, the load when impacting was firstly transferred through the cell walls to effect the AIP and the Front Bulkhead. After the AIP and the diagonal tube touched the rigid wall of the vehicle (the Front Bulkhead cannot deform more) that the Honeycomb pack started to deform as the vehicle moved forward. This was expected to yield complete crash of the Honeycomb pack as an energy absorbent until the vehicle stopped.

There were three factors to keep in mind: the stiffness of the Honeycomb pack, the stiffness of the Front Bulkhead including the additional tubes, and the connection between the AIP and the Front Bulkhead.
4.7.4. Version 3

There are two diagonal tubes, one was newly added to make the structure stronger, the Honeycomb pack was modified to have the shoulder pattern:

![Version 3 geometry](image)

**Figure 33: Version 3 geometry**

As mentioned in the rule and from previous observation, one (1) more diagonal tube was added to make the Front Bulkhead stronger. The shoulder pattern was created to change the load path as and attempt to make the Honeycomb pack less stiff as a reverse to improve the structural stiffness. [19]

![Version 3 Honeycomb pack failure mode](image)

**Figure 34: Version 3 Honeycomb pack failure mode**

The Honeycomb pack showed interesting shape of deformation as there was collapse at the shoulder edges.
Figure 35: Version 3 AIP failure mode in Top view

The collapse of the Honeycomb pack can be easier observed from the top view. The collapse started as the shoulder edges and the shoulders were folded inward. The AIP stayed in touch with the tubes. The simulation time was almost three (3) times the previous version with lesser deformation of the tubes.
4.7.5. Version 4

More modification of the honeycomb pack:

The double shoulder pattern was created and the two (2) diagonal tubes were kept. The Honeycomb cells were halved the thickness in order the reduce its stiffness.

Figure 36: Version 4 geometry

The Honeycomb pack showed a significant deformation as it was self folded at its shoulder.

Figure 37: Version 4 Honeycomb pack failure mode
The figures showed better view of the deformation of the Honeycomb pack. The plate was not in proper contact with the tubes.

What can be observed from this simulation is that the two (2) longest side tubes of the rectangular Front Bulkhead were bent significantly as they experienced biggest bending moment in the middle. This brought up the idea of changing the direction of the diagonal tubes to reduce the overall deformation.
4.7.6. Version 5

Direction of the X-brace was changed:

Figure 39: Version 5 geometry

The diagonal tubes orientation was changed so that each of them is parallel with the rectangular side tubes. Tubes thicknesses were increased from 2 mm to 3 mm.

Figure 40: Version 5 Honeycomb pack failure mode

The Honeycomb pack showed not so much different to the previous simulation. However, the idea of supporting tubes to the longest rectangular tubes worked.
Figure 41: Version 5 AIP failure mode in Top view

The AIP deformed less significantly comparing to the previous version. As can be seen from the figure, the intersection of the X-brace kept the plate from touching the Vehicle plane.
4.7.7. Version 6

More dimensional change to the honeycomb pack, tube thicknesses increased:

Figure 42: Version 6 geometry

More shoulder pattern was added. The tube thicknesses were increased to 4 mm to utilize the previously mentioned concept of design.

Figure 43: Version 6 Honeycomb pack failure mode

The Honeycomb pack performed a successful crash. The Honeycomb walls were crushed evenly and uniformly throughout its longitudinal edges.
The AIP deformed the furthest as around 40mm witnessing a significant improvement comparing to the previous simulation.

Due to the time allowed for the Bachelor work, this was the last version, the most successful and informative one as it suggested directions for further developments of the IA made by Aluminum Honeycomb.

### 4.8. Numerical solution

<table>
<thead>
<tr>
<th></th>
<th>Time [s]</th>
<th>Final velocity [mm/s]</th>
<th>Stopped? [Y/N]</th>
<th>Peak acc. [mm/s²]</th>
<th>Average acc. [mm/s²]</th>
<th>Cell size</th>
<th>Tube [mm]</th>
<th>Result (Fail/Not fail)</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>Version 1</td>
<td>1.25E-02</td>
<td>3838.0 N</td>
<td>Y</td>
<td>29680</td>
<td>-7025.5</td>
<td>9°.1</td>
<td>25*2</td>
<td>Fail</td>
<td>No X-brace</td>
</tr>
<tr>
<td>Version 2</td>
<td>1.25E-02</td>
<td>5458.6 N</td>
<td>Y</td>
<td>21590</td>
<td>-169932</td>
<td>9°.1</td>
<td>25*2</td>
<td>Fail</td>
<td>1 diagonal X-brace</td>
</tr>
<tr>
<td>Version 3</td>
<td>4.00E-02</td>
<td>-493.1 Y [@ 1.2E-02]</td>
<td></td>
<td>29970</td>
<td>-2.14E+05</td>
<td>5.525<em>0.1</em>0.2</td>
<td>25*2</td>
<td>Fail</td>
<td>2 diagonal X-brace</td>
</tr>
<tr>
<td>Version 4</td>
<td>4.00E-02</td>
<td>240.6 N</td>
<td></td>
<td>162030</td>
<td>-1.16E+05</td>
<td>5.525<em>0.05</em>0.1</td>
<td>25*2</td>
<td>Fail</td>
<td>2 diagonal X-brace, new cells</td>
</tr>
<tr>
<td>Version 5</td>
<td>4.00E-02</td>
<td>1656.7 N</td>
<td></td>
<td>19480</td>
<td>-1.15E+05</td>
<td>5.525<em>0.05</em>0.1</td>
<td>25*3</td>
<td>Fail</td>
<td>2 perpendicular + brace, new tubes</td>
</tr>
<tr>
<td>Version 6</td>
<td>5.58E-02</td>
<td>377.9 N</td>
<td></td>
<td>15820</td>
<td>-1.23E+05</td>
<td>5.525<em>0.05</em>0.1</td>
<td>25*4</td>
<td>Nfail</td>
<td>2 perpendicular + brace, new tubes</td>
</tr>
</tbody>
</table>

**Figure 45: Summary of Simulation Data sheet**

The Version 6 fulfilled the three designing criteria. However, the AIP having deflected more than 25 mm is what should be considered for future development.
5. Discussion

5.1. Failure mode in Simulation and Real Crash test

Comparison between the Simulations and the Real Crash tests carried by other Formula Student teams:

Figure 46: Version 2 and Real Crash test with one (1) diagonal tube comparison [17]
Figure 47: Version 6 and Real Crash test (cut and pre-crushed) comparison [18]
Figure 46 shows the deformation shape of the simulated Version 1 with one (1) diagonal tube added to the Front Bulkhead in comparison with the same Front Bulkhead setup in Real Crash test. The longest rectangular tubes of the Front Bulkhead are bent in the middle where they experience the biggest bending moment. In the Real Crash test, after the Honeycomb pack reaches the rigid plane of the vehicle, the free end of the Honeycomb pack starts to crush uniformly. The Version 1 is expected to behave the same if the simulation time is longer and the AIP is properly connected to the Front Bulkhead tubes.

Figure 47 shows the Deformation shape of the simulated Version 6 with the same Front Bulkhead setup, two (2) X-brace tubes which are parallel to the rectangular side tubes. The Honeycomb pack in the Simulation is pre-crushed to win the peak load (relating to Figure 13). In the simulation, Version 6 is cut with three (3) shoulder patterns instead. The results are the even and uniform crush of the Honeycomb pack which are ideal for the energy absorbing application.

5.2. Structural stiffness criteria

There are a certain factors affecting the stiff-worthiness of the structure with adjustable parameters which can be helpful for future developments:

- Stiffness of the Aluminum Honeycomb pack: dimensions (general sizing, shaping (cutting)), special treatment (pre-rushing to win the peak load), profiles (according to the Bare compressive stress information, the smaller the wall thickness, the “softer” the Honeycomb; perforated or not)

- Stiffness of the Front Bulkhead: dimensions (general sizing, outer shape), tube thicknesses

- Material of the AIP: 1.5 mm solid Steel or 4 mm Aluminium

- Connecting method of the AIP to the Front Bulkhead: bolting, welding
5.3. Pros and cons of the thesis work

Pros:

- Provide virtually highly reliable presentations of real crash tests
- Provide reliable suggestions for real testings and further developments

Cons:

- Can not estimate the time of simulation (simulations took 2-4 days and the last simulation took 1 week)
- Only basic calculations of the problems can be done
- There has not been any real physical testing to support the simulation
6. Conclusion

In conclusion, the rules and the requirements for the design of the Impact Attenuator were studied thoroughly. The material Aluminum Honeycomb with its specific properties and failure mode were studied. There were six (6) versions of possible versions of Impact Attenuator were proposed and put into simulations. The deformation shape and the numerical data were collected yielding good suggestions for further developments of the Impact Attenuator.

The final work uses the X-brace in the direction parallel to the other rectangular tubes of the Front Bulkhead with tubing of 25 mm x 4mm. The AIP is welded to the Front Bulkhead. In addition, the modified Aluminum Honeycomb pack is used. The final drawings of the Front Bulkhead and the IA are below:

![Figure 48: New Front Bulkhead drawing](image-url)
Figure 49: New IA drawing
7. References


[22] ANSYS®Academic Teaching Advanced, Release 18.2, help system, ANSYS, Inc.

**Read more:**

- Christopher Wright, PE. *Introduction to Structural Impact*. Christopher Wright, PE. 2012.
- J.E. Akin. *Impact Load Factors for Static Analysis*.
ACKNOWLEDGEMENT

Foremost, I would like to thank my supervisor Ing. Zdeněk Poruba, Ph.D., for the patience and the proper guide I received. His room was always cozy and welcoming to me. He was there to support me with his valuable advices and encouragements steering me to the right direction when I was in hard time. Words alone cannot express my thanks for him helping me to grow professionally and personally. I do appreciate and would keep the memory of working with him as a treasure of my student life.


Thank you David Krzikalla, Jakub Měšiček, Václav Šelong, Tomáš Pasterňák, Kubík Šmiraus, Kristýna Kutiová, Filip Lapuník, and others from the Formula Student of Technical University (2017-2018) for the support and experiences that I received being a team member. Without Formula Student, there would not be this thesis topic.

I would also like to thank Mr. Jukka Nisonen- my Degree Manager, Mr. Timo Eloranta, Mr. Seppo Toivanen and all other teachers in Saimia University of Applied Sciences, Lappeenranta, Finland for the supports, the knowledge that I received from the Bachelor Degree in Finland.

Finally, I must express my gratitude towards my parents, my younger brother and my girlfriend for providing me the right support and encouragement throughout my years of study. Thank my close friend. This accomplishment would not be possible without them, those persons I love from Czech Republic, Finland, and Vietnam.