Designing and testing a stand used to simulate the dummy head impact with different surfaces using CAD software

A I Radu¹, D D Trusca¹, G R Toganel and B C Benea¹

¹Automotive and Transport Department, Faculty of Mechanical Engineering, University ‘TRANSILVANIA’ of Brasov, Romania

E-mail: alexandru.radu@unibv.ro

Abstract. The aim of the paper was to design, assemble and test a stand used to analyse various crash scenarios for a crash test dummy head. This stand can be used to test the head impact with different surfaces, with or without a helmet, on soft or hard surfaces at different impact angles. The idea was to design the stand in the CAD software SolidWorks because this particular software has the feature of dynamic simulation of assemblies known as Motion Analysis. The results show that designing the stand was a success and the simulations show a good behaviour of the assembly with direct results generated in term of acceleration and force values obtained from the drop test of the head on a surface. Also, the surface can be adjusted for different impact angles and with different contact surfaces properties to extend the usability of the assembly.

1. Introduction

Crash testing has been around since the development of the automotive industry [1]. Usually, crash tests are performed in the field or inside a building in order to improve the safety of vehicles [2]. Inside buildings, these tests are performed using stands specially designed to simulate real life conditions [3]. The design stand aims at testing the impact between the head and an impact plate at various angles (oblique) [4]. Usually in the real-world oblique impacts are present with motorcycle accidents. Oblique impacts induce a radial and tangential force to the head, leading to a linear and a rotational head acceleration [5, 6].

Today, we can design models using CAD software to create prototypes that can be tested in the virtual environment in order to assure a correct functionality before building it in real life [7]. SolidWorks has a module that allows for dynamic simulation of models based on input parameters and outputting the results. This model consists of another software called ADAMS and it is a multibody numerical simulation software that uses rigid or elastic bodies linked together with kinematic joints or elastic connections such as springs. The key parameter of this software is the solver, the mathematical model that takes the input parameters and calculates the output parameters based on constraints and material contact [8, 9, and 10].

Also, the software allows for the use of the finite element model that was used in order to study the material resistance. This method is used for structure analysis to determine the material strength and weak points. It uses a numerical method for solving partial differential equations in multiple space variables [11, 12, 13, and 14]. In our case, the critical part was the impact plate, we tested the FEM
analysis to assure that the impact plate can withstand multiple drop tests of the head for a good functionality of the design.

2. CAD model design

The designed stand was created using the SolidWorks CAD software due to the primary advantage of this particular software to test dynamically by the means of motion analysis. The model consists of multiple components, illustrated in the figure below.

![Figure 1. Stand model and components.](image)

The model assembly has 7-part bodies, presented as followed: **base** – it secures a good stability and provides an anchor point for the impact plate. **Beam** – is the part that mounts to the base and gives the stand its drop height. On the beam, the **adjustable stand** can be mounted on multiple levels, setting the drop height of the **dummy head** that couples to the adjustable stand by a **release pin**. The **impact plate** is mounted on an articulation on the base and it’s able to be adjusted and locked to a specific angle using the **angle adjusting beams**.

The stand dimensions are custom designed to assure multiple variations of the test parameters that the user desires. Therefore, the following dimensions of the stand are presented in the next figure.

The maximum height of the stand is about 1500 mm with the effective drop distance for the head of 1103 mm. The impact plate has a thickness of 35 mm to assure multiple tests can be completed without bending. The dummy head has a radius of 110 mm and it’s similar in size and weight to a human head. The distance between the head CG and the drop beam is 169 mm giving a good clearance for the drop. There are 11 holes on the drop beam with 100 mm apart one from another to assure multiple drop velocities for the head. The base is a square with a length of 750 mm. Also, the weight parameters for each component is presented in table 1.

| Table 1. Weight parameters of the components |
|---------------------------------------------|
| Weight parameters                           |
| Nr.  | Component | Weight [kg] |
|------|-----------|-------------|

The model assembly has 7-part bodies, presented as followed: **base** – it secures a good stability and provides an anchor point for the impact plate. **Beam** – is the part that mounts to the base and gives the stand its drop height. On the beam, the **adjustable stand** can be mounted on multiple levels, setting the drop height of the **dummy head** that couples to the adjustable stand by a **release pin**. The **impact plate** is mounted on an articulation on the base and it’s able to be adjusted and locked to a specific angle using the **angle adjusting beams**.

The stand dimensions are custom designed to assure multiple variations of the test parameters that the user desires. Therefore, the following dimensions of the stand are presented in the next figure.

The maximum height of the stand is about 1500 mm with the effective drop distance for the head of 1103 mm. The impact plate has a thickness of 35 mm to assure multiple tests can be completed without bending. The dummy head has a radius of 110 mm and it’s similar in size and weight to a human head. The distance between the head CG and the drop beam is 169 mm giving a good clearance for the drop. There are 11 holes on the drop beam with 100 mm apart one from another to assure multiple drop velocities for the head. The base is a square with a length of 750 mm. Also, the weight parameters for each component is presented in table 1.

| Table 1. Weight parameters of the components |
|---------------------------------------------|
| Weight parameters                           |
| Nr.  | Component | Weight [kg] |
|------|-----------|-------------|
The stand measures a weight of 92.6 kg in total, with the most important component, the dummy head having 4.2 kg. The weight parameters for the stand were generated by the CAD software based on the material attributed to each component. The material used was STAINLESS STEEL.

3. Results
For the dynamic analysis, in the software objects have 6 degrees of freedom, 3 translations and 3 rotations along the main axis (XYZ axis system).

In order to study the contact between the components, contact parameters between them were necessary. In order to achieve this, the software has built-in material contact to simplify the process. Since the material for the stand was stainless steel, the contact between all components was steel with steel. This is illustrated in the table below. Only for the head and the impact plate, the materials were chosen different due to the head having a softer material. So therefore, between the head and the impact plate a acrylic and steel parameters were set. The acrylic assures a greater elasticity and a softer stiffness than steel.

| Table 2. Components contact parameters |
|---------------------------------------|
| **Contact parameters**                |
| **First component** | **Second component** | **Material contact** | **Damping [N/(m/s)]** | **Stiffness [N/m]** |
| Dummy head                | Impact plate         | Acrylic - Steel      | 588                    | 1150000              |
| Base                      | Beam                 |                        |                        |                      |
| Beam                      | Adjustable stand     | Steel - Steel         | 49915                  | 10000000             |
| Adjustable stand          | Release pin          |                        |                        |                      |
| Release pin               | Dummy head           |                        |                        |                      |
| Impact plate              | Base                 |                        |                        |                      |

The main parameters that are set when choosing the contact materials are the stiffness and the damping between components. In case of steel to steel, the stiffness is high enough not to allow component penetration.

For the simulation to be completed, a couple of elements needed to be configured for the calculation. Along with the component contacts, the gravitational acceleration was set. The kinematics and the time phases of the dummy head are presented in the figure 2.
The simulation was analysed for a time frame of 100 msec. The impact between the head and the impact plate was around 480 ms after the release. After the impact, it takes about 200 ms for the head to rebound off the impact plate. These simulations show a kinematic behaviour of the assembly design and would provide good results further on. The next step was to study the impact of the head at different angles with the impact plate. We chose 4 angles for the analysis, at 0 degrees, meaning straight impact, at 10 degrees, 20 degrees, 30 degrees and 40 degrees. In figure 3 it is shown how the impact plate is configured for each of the proposed angles.

While the impact force is at maximum when the impact plate is at 0 degrees, when an angle is configured, the forces are split in 2 directions such as presented in figure 4.
When the head hits the impact plate, the force is split in 2 directions, one being the normal direction of the force G and the other along the X axis, and it’s noted with FX. The resultant force is the Fr and indicates the direction of the head after the rebound.

The stand offers multiple velocities to be tested by positioning the adjustable stand along the drop beam at any of the 11 positions. In figure 5, we can see all the velocities that can be obtained for each of the 11 positions.

As we can see, the increase of the velocity is mainly linear up to a maximum value of 4.65 m/s. Since we wanted to test the maximum results that the stand can obtain, we have done all the angle impact tests at the maximum velocity. The tested velocity was at 16.8 km/h and it’s presented in figure 6.
To understand the capability of the stand using the dynamic CAD simulation, 2 main parameters were chosen for study, the head deceleration obtained at the impact with the plate and the contact force between these 2 components. Multiple simulations were completed at the impact angles mentioned earlier. In figure 7, the head acceleration variation is presented for all the simulation configurations.

We can observe that the highest acceleration value was obtained with the impact straight on, while when the angle increases, the acceleration drops from the maximum of 650 m/s$^2$, down to 160 m/s$^2$, thus we see a value reduction of 25%. This is caused by the fact the force splits in 2 directions when at an angle is applied to the impact. Also, we can analyse the contact force values in figure 8.

![Figure 7. Head accelerations.](image1)

![Figure 8. Contact forces of the head with the impact plate.](image2)
This diagram shows similar reduction of the contact force values to the acceleration diagram. In this case the maximum impact force was 2800 N when the collision is straight on, but then the angle is applied, the force drops all the way down to 800 N granting a reduction of 28.5%. This confirms that the simulation has good results with a good dynamic and kinematic behaviour of the components.

In order to test the resistance of the impact plate, a stress simulation was also conducted. This simulation applies a fix static force to the impact plate mounted with the angle adjusting beams.

![Static stress simulation](image)

The simulation showed that the impact plate can accept static loads. The force applied was 2800 N, identical to the force obtained after the dynamic simulation. The analysis revealed that the maximum strain was 38078000 N/m², in the articulation area. The angle adjusting beams reported a strain of 19039000 N/m². This stress was a static continued test to evaluate the strength of the impact plate assembly. A drop test would cause less stress and less probability of failing.

4. Conclusions
The results of this study show that the assembly design was successfully simulated using the dynamic module and the input parameters. The collision between the head and the impact plate shows good results when configuring different impact angles and outputting and comparing result diagrams.

Limitations are also taken into account when simulating the assembly. The main limitation is material fatigue that is relevant when, in real life, the stand would be used multiple times in the same conditions and the impact plate material would suffer deformations. Also, the two angle adjusting beams will have similar problems such as fatigue when exposed to multiple tests. Solutions could be found such as replacement parts or making the parts thicker and stronger.

5. References
[1] Tolea, B., Trusca, D., Antonya, C. and Beles, H 2015 The influence of the frontal profile design of a vehicle upon the pedestrian safety at low velocity. 26th Annals of DAAAM and Proceedings.
[2] Shaw, G., Crandall, J. and Butcher, J. 2000 September. Biofidelity evaluation of the THOR advanced frontal crash test dummy. In IRCOBI Conference on the Biomechanics of Impact.
[3] Bliven, E., Rouhier, A., Tsai, S., Willinger, R., Bourdet, N., Deck, C., Madey, S.M. and Bottlang, M. 2019 Evaluation of a novel bicycle helmet concept in oblique impact testing. Accident Analysis & Prevention, 124, pp.58-65.
[4] Bourdet, N., Deck, C., Serre, T., Perrin, C., Llari, M. and Willinger, R. 2014 In-depth real-world bicycle accident reconstructions. International journal of crashworthiness, 19(3), pp.222-232.
[5] Willinger, R., Deck, C., Halldin, P. and Otte, D. 2014 November. Towards advanced bicycle helmet test methods. In International Cycling Safety Conference (pp. 18-19).
[6] Chang, K.H., 2019. Motion Simulation and Mechanism Design with SolidWorks Motion 2019. SDC publications.
[7] Kurowski, P. 2013 Engineering Analysis with SolidWorks Simulation 2013. SDC publications.
[8] Kurowski, P.M. 2015 Engineering analysis with solidworks simulation. Mission: SDC Publications.
[9] Ryan, R.R. 1990 ADAMS—Multibody system analysis software. In Multibody systems handbook (pp. 361-402). Springer, Berlin, Heidelberg.
[10] Bowling, M. and Veloso, M. 2000 An analysis of stochastic game theory for multiagent reinforcement learning (No. CMU-CS-00-165). Carnegie-Mellon Univ Pittsburgh Pa School of Computer Science.
[11] Calimanescu, I., FATIGUE COMPARATIVE STUDY OF A GEAR TRANSMISSION BY USING FINITE ELEMENTS ANALYSYS.
[12] Zienkiewicz, O.C., Taylor, R.L., Nithiarasu, P. and Zhu, J.Z. 1977 The finite element method (Vol. 3). London: McGraw-hill.
[13] Shih, R. 2014 Introduction to finite element analysis using solidworks simulation 2014. SDC publications.
[14] Logan, D.L. 2011 A first course in the finite element method. Cengage Learning.