ISSN No:-2456-2165

Analysing the Side Impact Safety of Vehicle Frame Structure using Optimal Control Technique

Khaled Mohammed Ahmed alkandari[1], Ahmed Alhumaidah[2]
Specialist Trainer (B) in Public Authority for Applied Education and Training, Temple University, Philadelphia, USA.

Corresponding Author:- Khaled Mohammed Ahmed alkandari[1]

Abstract:- To prove the security of the side effect of the structure of the vehicle frame, and to minimise the overall number of the accidents that are met by the members that leads to some injured. A model has been built by the finite element that studies the side effect of the vehicle has been established with the help of an ANSYS model. However, this model has been built based on the basic principle of anti-side collision optimization, as well as the principle of the simulation. The constraint and the boundary conditions have been set based on the GB20071-2006 requirements. The analysis has been conducted by the software of the finite element analysis. Thus, the deformation speed as well the invasion amount of the B-pillar within the main parts of the side wall should be normal. The security of the original vehicle can be verified by several things such as: increasing the sheet metal thickness, optimizing the anti-collision beam position, adopting several cross-section shape for the anti-collision beam, adding some structural strengthening parts, and using the ultra-high strength steel. The outcomes showed that several approaches can enhance and optimise the side effect behaviour of the vehicle. However, the optimal way is by increasing the reinforced parts without affecting the original parts function. The method that is used for optimisation purpose, the conclusion, as well the measures that used to enhance the performance of the side impact provide a specific reference regarding the design of the vehicle side impact security behaviour.

Keywords:- *Vehicle Impact Side, Finite Element Analysis, Moveable Deformable Barrier, ANSYS Model, Structure of the Vehicle Frame***.**

I. INTRODUCTION

Cars are becoming a significant part of people's everyday lives due to the rapid economy developments. Nevertheless, as car usage has increased quickly, so an increase has been observed in the frequency of car traffic incidents, that not only result in significant property damages but also pose a severe public safety risk [1].

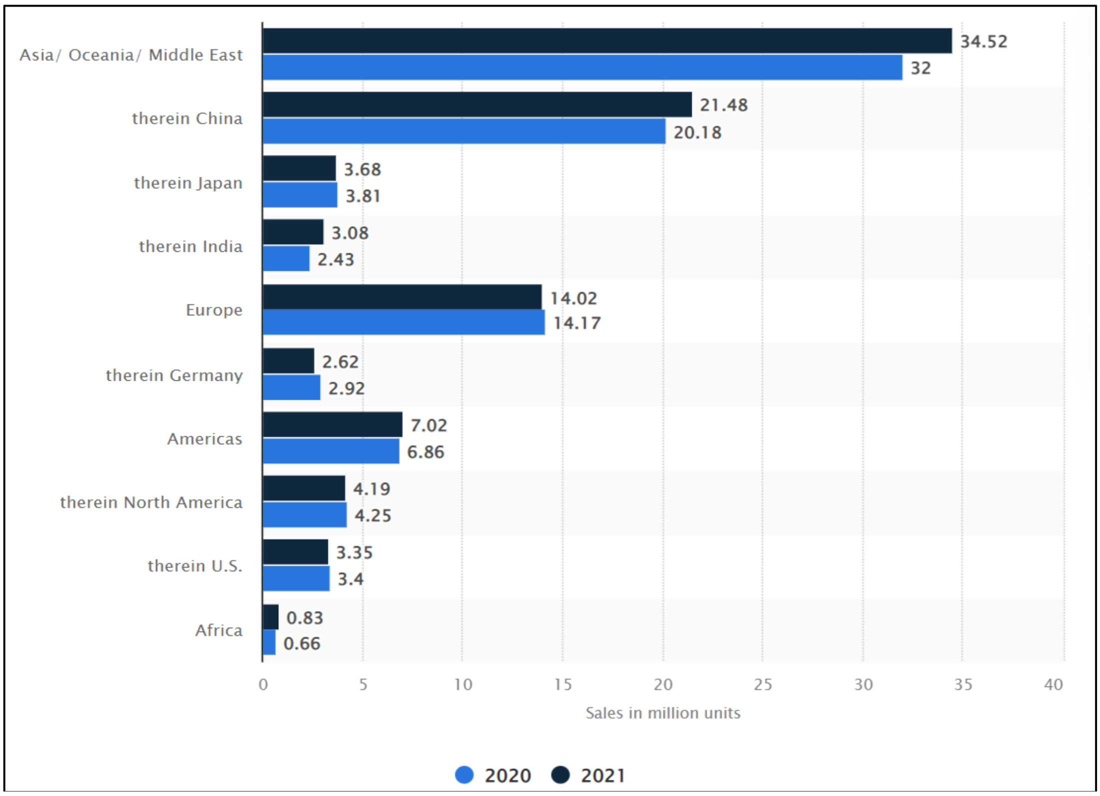

Fig 1 A Globally Car Sales Statistic [2]

ISSN No:-2456-2165

The vehicles accidents are the main reason behind the development of the passenger's vehicles to minimize the injuries and the death rates of the passengers as much as possible. One of the development processes concerns with the deformation structure design to enhance the safety performance for the vehicle body regarding mitigating the vehicles accidents. However, the increasing in the passenger's car popularity in the global market, there is increasing in the environmental issues which caused by the emissions which generated from the fuel vehicles [3].

Both side and frontal impacts occur most frequently in all types of car accidents. Side accidents lead to large door's excessive intrusion volume, as well as deformation speed, which might result in catastrophic injuries to the car users. Since side accidents represent 30% of the entire accidents, they are considered as the most common accident type. In car accidents, neck and head injuries make up 58% of all injuries, followed by trunk injuries that represents 32% of all injuries, and abdomen injuries represent 21% of all injuries [4].

Accordingly, the impact of side regarding safety became an important field to be investigated and studied, which attracted the attention of automakers and academic institutes globally. In general, there are two key indexes that help evaluate the automobile resistance against side impact, which are B-pillar and door. The speed and displacement variations of different components effected by external force are primarily studied for B-pillar and door since these two attributes are connected directly to the severity of passenger injuries [5].

The impact force must be distributed equally throughout the body to successfully limit side-impact injuries to passengers. This must be done while maintaining the lowest possible car speed and displacement. The car body performance is subject to reciprocal limitations due to its large size and high complexity [6].

The interplay of several functions and multiple disciplines must be carefully taken into account throughout the Research-and-Development (R&D) of a vehicle body. Consequently, when researching vehicle side impact safety, it is important to take into account the effects of NVH, lightweight, dynamic performance, as well as other disciplines [7].

Multidisciplinary optimization technique was developed to address the issue that it is challenging to produce an effective design of the vehicle performance using a single discipline and single goal optimization technique. The Multidisciplinary-Design-Optimization (MDO) technique accomplishes interdisciplinary convergence and optimization, as well as achieves the system's optimal design while taking the impact of performance into full consideration [8].

Accordingly, this study utilises various optimization approaches to improve the side structure's crash protection in side impacts from various angles, and it confirms the efficacy of the studied optimization techniques in principle. A consistent testing set is employed to create the test, as well as the regression analysis substitute model is created using the testing data in order to address the issue of reciprocal restrictions between body lightweight, as well as the safety of body side impact. Lastly, sequential-quadratic programming approach as well as the adaptive weighting technique are employed in this study. Optimization outcomes that fulfil design goals are attained. Computer simulation software is used to confirm the security results.

According to [9] after the collision between the vehicle and trolley there will be a deformation for the side part of the vehicle. The increasing in the deformation resulted in further increasing in the deformation resistant, and the acceleration of the vehicle gradually initiated due to the deformation resistance deformation. When the collision trolley and the vehicle have the same speed, the collision trolley and the vehicle are assumed to be the same speed. The collision trolley mass assumed to be M_1 whereas the vehicle mass assumed to be M_2. In addition, the collision trolley initial speed assumed to be V_0 whereas the vehicle initial speed assumed to be V_2, where the collision trolley speed after the collision assumed to be V_1 whereas the final vehicle combined speed assumed to be V_C. According to the energy conservation law the following equation can be written as follows:

$$M_1 V_\circ = M_1 V_1 + M_2 V_2 = (M_1 M_2) V_C \qquad \text{Eq.1}$$

The curve of the velocity-time curve is represented in Figure 2.

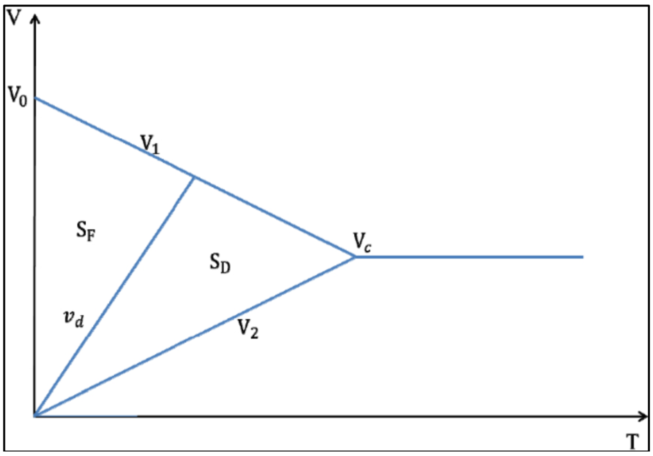

Fig 2 Velocity-Time Curve

In Figure 2, the enclosed area within the time-velocity curve represents the displacement . If the velocity of the internal panel is assumed to be V_d, then the barrier collision trolley deformation is denoted as S_F, and the body deformation is denoted by S_D. However, in the side impact the side-buffer space for the passenger is not large enough, in addition to a limitation in the deformation space. A small deformation amount in the B-pillar, floor-beam, side-door-anti-collision-beam, and the roof-cross-rail have the ability to absorb the impact's kinetic-energy. Consequently, to guarantee the car body deformation, the deformation of the barriers should be increased [9].

II. METHODS AND MATERIALS

A. *Analysis Steps*

The FEA involves several steps. Firstly, the appropriate model for analysis should be selected. The analysis model should involve the required steps to enhance the product based on the results. The involved steps are development for the product, test the selected model, testing for the remodelling and evaluating process [10].

ANSYS software is used for several purposes. In specific, ANSYS software is used for achieving a comprehensive simulation process in several disciplines of vibration, structural, heat transfer, physics, fluid dynamics, and electromagnetic. As a results, ANSYS software helps in 3D simulating the work and test conditions, helps in simulating and modelling in virtual environment before the fabrication of products. In addition, it helps in determining the weakness point to improve it before achieving the manufacturing process, it helps in assessing the lifetime, and predicting the possibility of occurring any problem in future. Another advantage of ANSYS software is the ability of integration with other engineering software's, such as CAD and FEA in the ANSYS module [11].

B. *Finite Element Analysis*

There is a high difficulty and complexity in understanding the process of the human body's dynamic reaction in the case of an actual accident process. This is due to the accident's short time, as well as the large resulting vehicle deformation. Accordingly, to ensure the side impact safety of the vehicle, automobile producers and research companies must simulate the actual accident process.

Therefore, there were several stages of accident simulation process development. The early stages of this process started by collecting the required data from actual vehicle experiments, where a dummy or corpse is used to simulate the human body's reaction in the case of an accident.

In recent years, there has been a massive development in the computer and software fields, besides the mathematical model's improvement, which led to the development of several computer simulation software to be utilised in the simulation processes of vehicle accident safety, and the related feasibility and accuracy analysis [12].

These simulation process have several advantages, including achieving high accuracy results, improving the vehicle crash safety, achieving a notable reduction in cost of development, as well as increasing the market competitiveness. All these advantages highly encourage expanding the automobile passive safety research [13].

Accordingly, Finite-Element-Analysis (FEA) is the one of the common engineering analysis methods that help reduce the physical prototypes, as well as experiments number and improve components during the design stage in order to achieve better products, faster while saving on expenses.

➢ *FEM is one of the numerical method which used to provide a general numerical by dividing the engineering problems. There are several advantages for using the FEM, including [14]:*

- It can be applied for all material types and complicated geometry.
- It can be used in several disciplines such as biomechanical, automotive industries, aerospace, to test the strength, stiffness, and collision in virtual before conducting the prototype, or fabrication process, which minimizing the cost significantly.
- The FEM helps in accelerating the designing process, hence the manufacturing stages.
- FEM can provide a successful design schemes.
- A general mathematical solution can be used for different problems.
- The computer simulation techniques are used to replace the representation of the actual vehicle accidents with a computer simulation relying on the collision models.
- To achieve the accuracy of computation, and reduce the calculation scale at the expense of the efficiency of calculation precision, an appropriate FEM with mesh generation becomes an essential key of the FEA. The calculation efficiency in the FEM depends on the element types, FEM simplicity, and the contact algorithms, the accuracy of the mesh generation, and the setting of material parameters.

C. *Meshing*

This is an important step that aims to determine the finite element's most suitable dimensions. The importance of this step is that selecting wrong dimensions leads achieve improper results. The main concept of the meshing process is to divide the elements of a large size into several small portions that will be solved individually. As a result, in order to totally dissolve the surface elements, finite elements will be existing on the surface' sides. After that, the width of the finite element will be chosen to be entered the bottom. ANSYS uses several mesh assessment criteria to assess the meshing appropriateness level after determining the overall and local mesh settings. However, these standards should be used to assess meshing. Also, based on Skewness-Criterion, the meshing suitability level was assessed in the simulation process.

D. *Skewness Criteria*

Skewness criteria is a common mesh evaluation principle is the most widely used. The following equation is used to obtain the criterion.

$$max\left[\frac{\theta_{max}-\theta_e}{180-\theta_e},\frac{\theta_{max}-\theta_e}{\theta_e}\right] \qquad \text{Eq.2}$$

Furthermore, the Equilateral-Volume-Deviation can be obtained from the following formula, which can be used with tetrahedron and triangles Skewness only:

$$\frac{Optimal\ cell\ size - Cell\ size}{Optimal\ cell\ size} \qquad \text{Eq.3}$$

Furthermore, the scale of the Skewness mesh spectrum is provided in Table 1.

Table 1 The Scale of the Skewness Mesh Spectrum

Excellent	Very good	Good	Acceptable	Bad	Unacceptable
0-0.25	0.25-0.50	0.50-0.80	0.80-0.94	0.95-0.97	0.98-1.00

E. Material Selection

The automobile framework is consisting of several elements which were obtained depending on using several materials, such as aluminium, steel and other precision material. The most common material used in automobile's framework manufacturing are the aluminium and stainless steel. Since the forces in these two materials are approximately equals, this means that the behaviour of these materials will be the same under the effect of the same load. When the steel was compared with the aluminium, the steel showed more toughness and heavier weight than aluminium. In the recent years, it is notable that there was a significant reduction in the fuel cleanliness and economy in the automotive industry sector.

Accordingly, the steel sheets with ultra-high strength properties have a great potential in the automotive industry. It is predictable that the depending on 1st and 3rd ultra-high generation steel for manufacturing the automotive vehicles will spread extensively in short time. As a result, the combination between these two types of steel sheets is inventible in the electrical-resistance spot-welding. Therefore, the selection of the best material for manufacturing the vehicles framework is compulsory, in this regards the focus is going on the Dual-Phase (DP), and composite steel. In this research, the selected steel type is DP1000. The DP distinguished by its high formability, high strength, availability in local and global market, good ability for welding, and cost effectiveness.

The utilised steel in this work is DP 1000 steel. DF Steels is widely utilised because of their several advantages, some of these advantages are summarised in Figure 3.

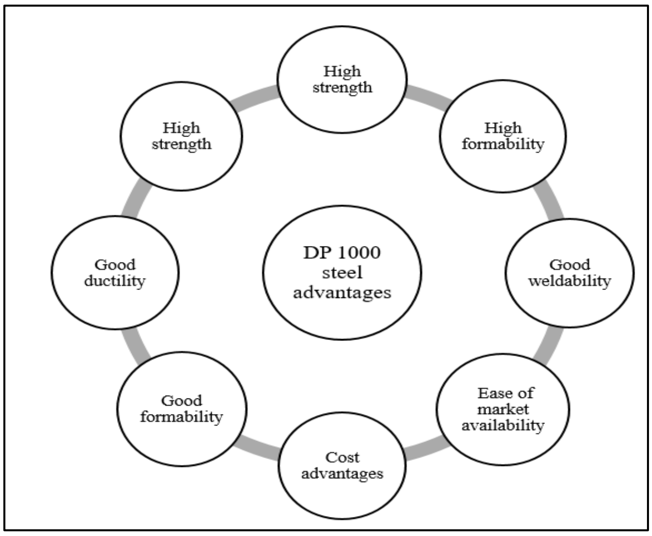

Fig 3 DP 1000 Steel Advantages

There are two main phases in the structure of DP steel, which are ferrite which contributes to providing structural ductility, as well as martensite which gives it exceptional strength. The automobile sector already makes substantial use of the DP1000, DP800, as well as DP600 steels. Furthermore, the mechanical characteristics of the utilised DP1000 steel are provided in Table 2.

Table 2 DP1000 Steel Properties

Mechanical Properties	Min	Max	Unit
Young's modulus	200	221	GPa
Specific Stiffness	25.5	28.1	MN.m/kg
Yield Strength	600	750	MPa
Tensile Strength	980	1,1e3	MPa
Specific Strength	76.4	95.6	kN.m/kg
Elongation	10	17	%strain
Compressive Strength	600	750	MPa
Flexural modulus	200	221	GPa
Flexural Strength	600	750	MPa
Shear Modulus	76.9	84.8	GPa
Bulk Modulus	167	184	GPa
Poisson's ratio	0,286	0,315	
Shape Factor	37		
Hardness-Vickerss	290	323	HV
Elastic stored energy	863	1,33e3	kJ/m^3
Fatigue strength at 10^7 cycle	333	368	MPa
Stress range	308	398	MPa

III. VERIFICATION AND FEM STRUCTURE OF THE MOVEABLE DEFORMABLE BARRIER (MDB)

Frontal collision happens when a vehicle is impacted with a rigid wall at the front side. On the other hand, the side impact accident occurs when a stationary vehicle is impacted with a moving obstacle. Consequently, this study prepared a movable deformable obstacle model in order to simulate the side impact.

A. The FEM Establishment of MDB

The establishment of FEM essentially involves preparing a mesh for the MDB, and defining the components properties and materials, as well as the assembly relationship among the vehicle components. After preparing the FEM, the constraints and load will be applied. In addition to that, the impact velocity applied at the tested vehicle is 50 km/h. Figure 4 shows the detailed dimensions of the simulated system.

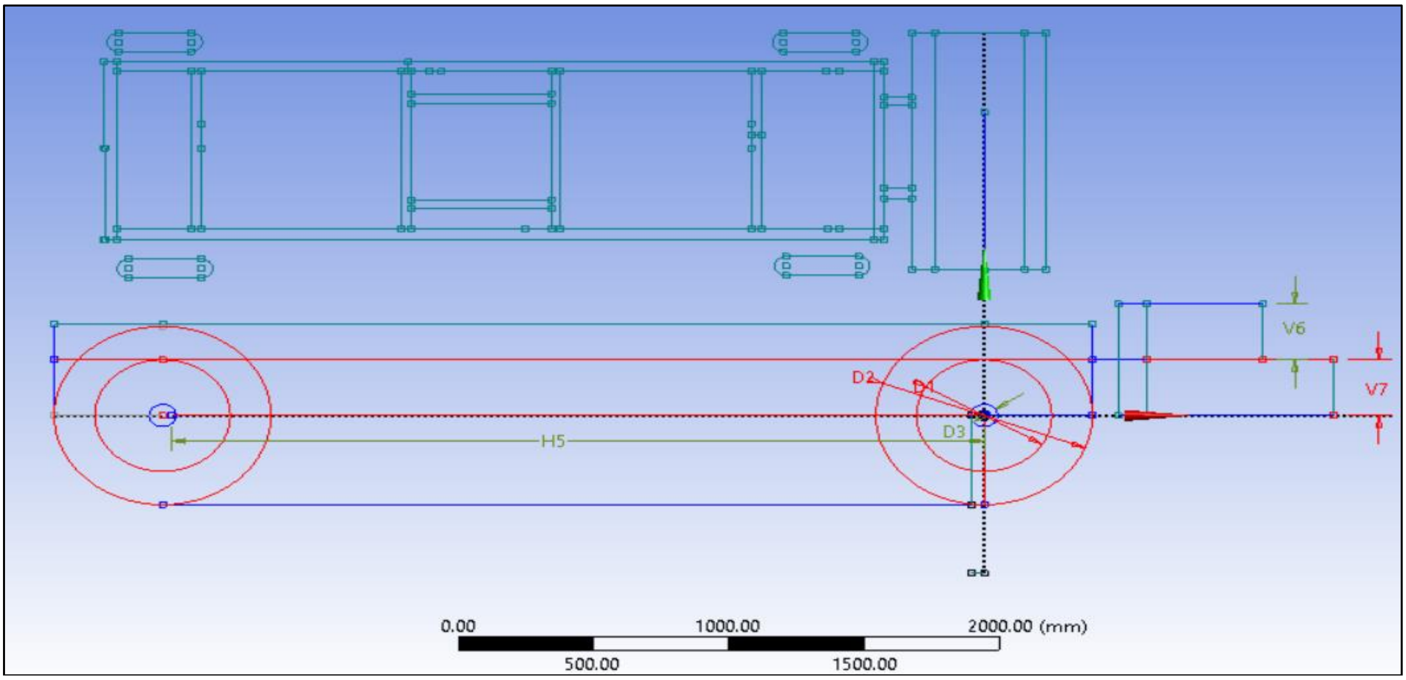

Fig 4 The Dimensions of DB (in mm)

The dimensions provided in Figure 4 are considered important requirements of the FEM of the MDB. Furthermore, the model front end dimensions are clarified in Figure 5.

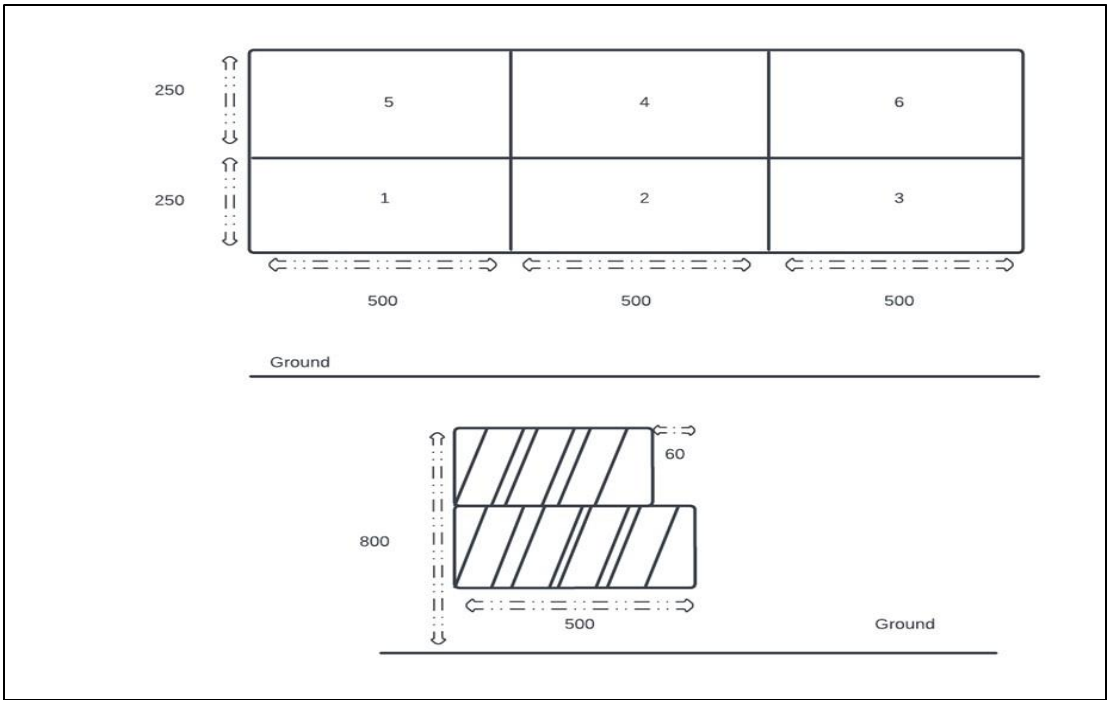

Fig 5 The Model Front end Dimensions

Additional essential details include the MDB's mass, which equals 950±20 kg, as well as the mass centroid, which was ±10 mm behind the x-axis symmetry plane. The FEM of the MDB includes, the models of the moving frame and energy absorbing block, as well as the central glue panel in addition to the rear and front connecting panels.

Furthermore, shell element will be used in order to simulate the moving frame since it will not be deformed during the collision. Whereas, solid element will be used in order to simulate the energy absorbing block, which is located in the frame's front. Also, beam element will be used for simulating the glues that are located at the middle of the MDB. Furthermore, the analysed trolley model depends on the frame, in addition to other related components, including key mass, wheels, and rear connecting plates. Also, tie contacts are used to connect the rear plate with the energy absorbing block. Accordingly, the resulted FEM is provided in Figure 6.

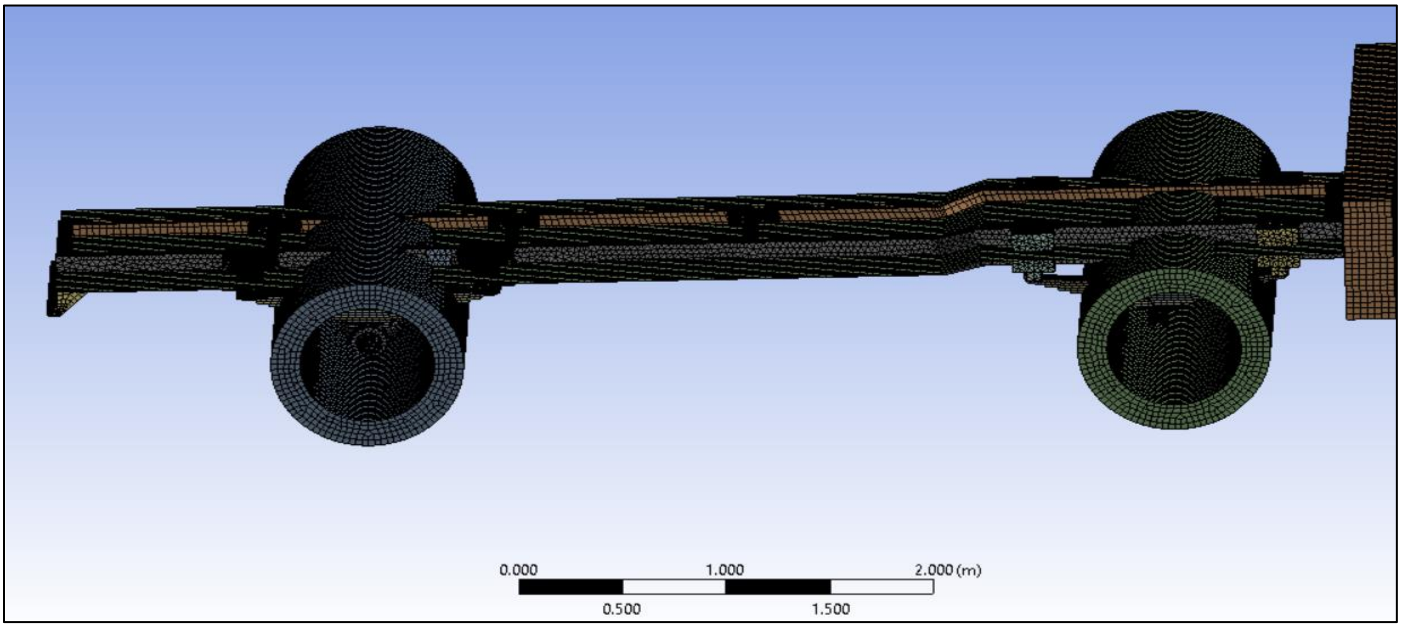

Fig 6 The Established FEM of the MDB.

In addition to that, the most important parameters utilised in the FEM are summarised in Table 3.

Table 3 The Most Important Parameters Utilised in the FEM

Mass of MDB (kg)	952.16
Location of MDB centroid (mm)	X = 0.45, y = -1859, 77 z = 511.43
Geometric parameters of mobile vehicle (mm)	Wheelbase 3000, wheelbase 1500, ground clearance 300
Initial speed of MDB (m/s)	9.722
Total number of units	239765
Number of deformable elements	18039
Number of rigid body elements	181375
Time step (ms)	0.76

B. The Verification MDB's FEM

Since the energy-absorbing block is primarily used in the rigidity simulation, its performance must be dynamically verified. The MDB verification process involves examining the entire force displacement characteristics of the energy-absorption block force displacement. Furthermore, force displacement properties must fall inside the defined band range by the lower and upper bounds, as well as the absorbed energy by the entire energy-absorbing block must fall within a particular range. Figure 7 shows an explanation of the verification test of the mobile.

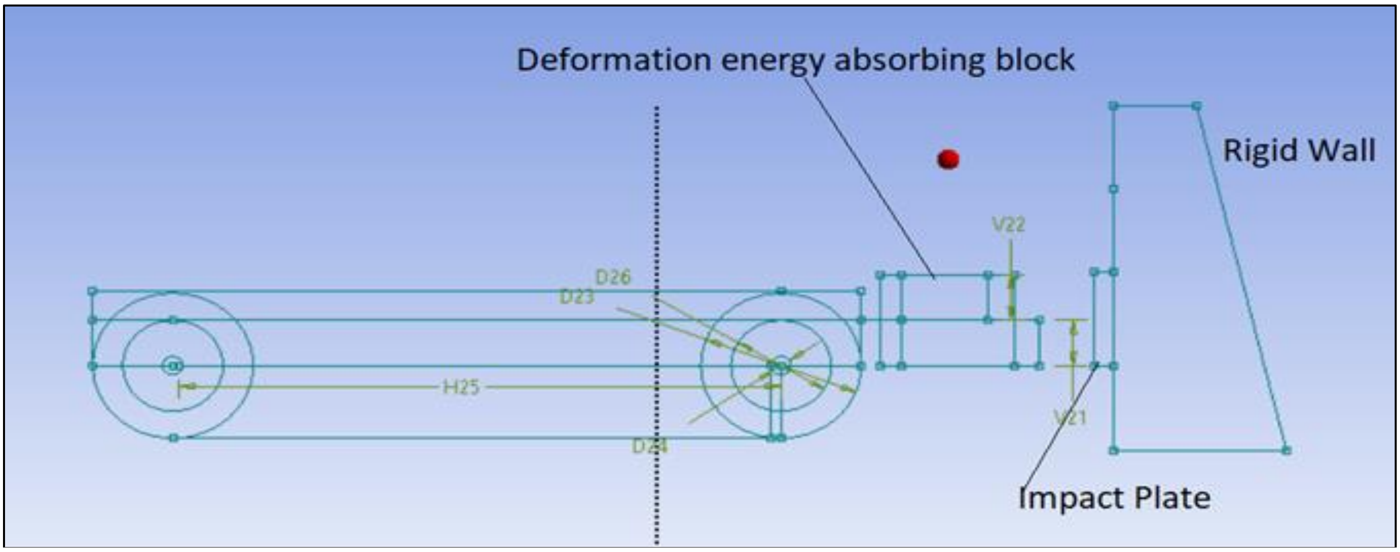

Fig 7 The Verification Test System

In order to investigate the condition of the energy-absorption block, which is located at the moving barrier's front end, as well as the force displacement curves, the rigid wall will be impacted with a speed equals 35 km/h in the vertical dimension. The utilised rigid wall in the test is made from reinforced concrete, the wall width is 3 m, and height is 1.5 m, while the weight is 70 t. Furthermore, the contact plane of collision is perpendicular with the movement axis. Also, load sensors will be used in order to detect the entire load of the MDB at the time of impact. Furthermore, a force computing was also used in order to estimate the impact force value throughout the impact action. During the simulation of the MDB, the connection between the ground, barrier, as well as rigid wall depend on the inner space. The barrier's velocity and displacement curves in the collision process are extracted based on the location of the mass centroid, allowing the MDB motion parameters in the full-frontal impact to be calculated.

The model outputs force-time history curves of (6) energy-absorption blocks that are translated to force fluctuation with displacement curves to assess the stiffness properties of the energy-absorption block deformation.

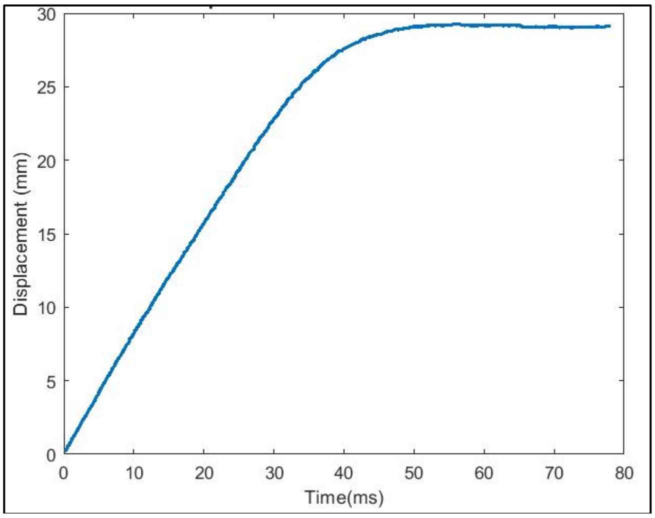

Fig 8 The MDB Centroid Curve

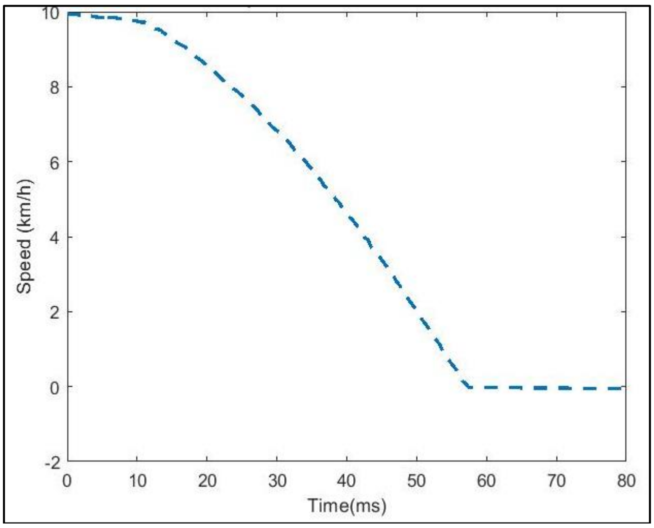

Fig 9 The MDB Time vs. Speed Curve

Figure 8 provides the MDB's displacement diagram showing the relationship between the displacement and time, while Figure 9 provides the resulted MDB's change diagram showing the relationship between the speed and time. Based on Figure 9, the movable energy-absorbing block's maximum deformation was 341 mm. Also, based on results showed in Figure 10, the moving barrier required time to reduce the speed from 35 to 0 km/h was 56.5 ms. After that, renounce occurs, and the speed become - 1.2 km/h.

Furthermore, Figure 10, Figure 11, Figure 12, and Figure 13 provide the relationship between the power and displacement for the six studied blocks. The dashed lines represent the maximum and minimum force-displacement curves' limit, as well as the area surrounding by it represents the impactor stiffness range that meets regulatory criteria.

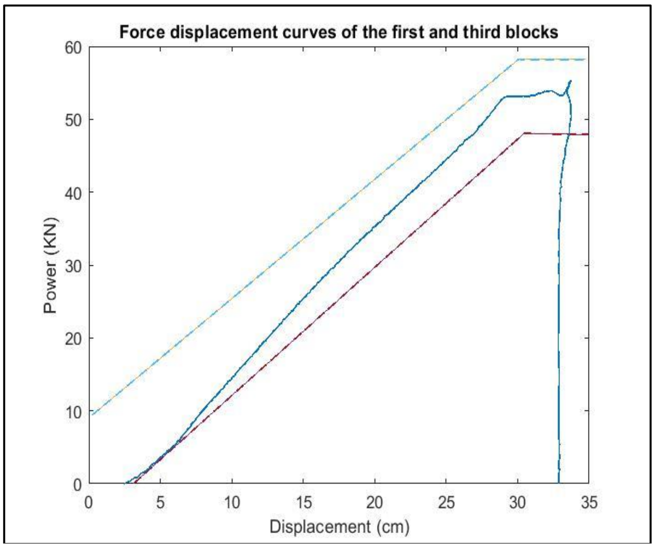

Fig 10 Power vs. Displacement Curve (Block 1 & Block 3)

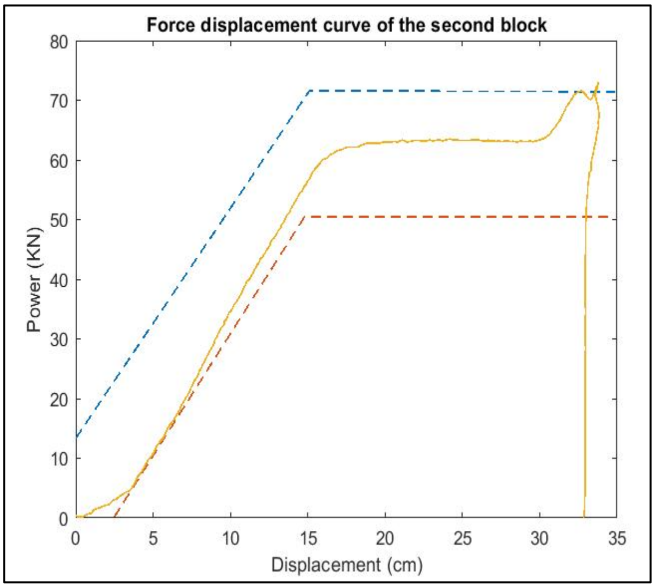

Fig 11 Power vs. Displacement curve (block 2).

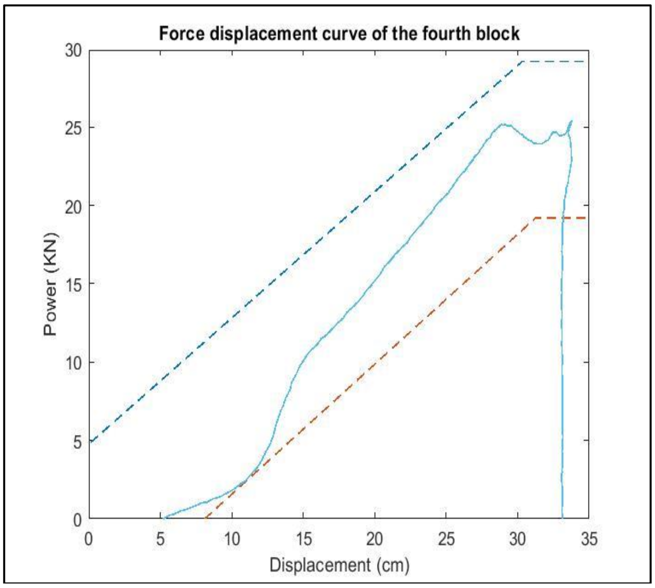

Fig 12 Power vs. Displacement curve (block 4).

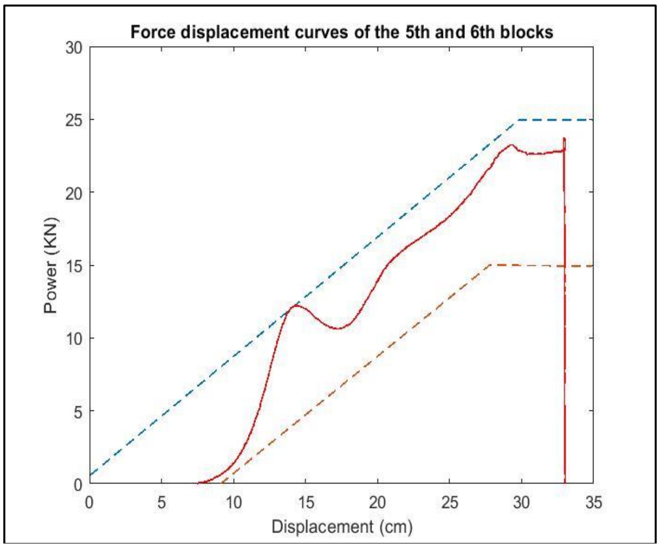

Fig 13 Power vs. Displacement curve (block 5 & block 6).

Results showed that, the obtained stiffness parameters of the tested deformation blocks were in the acceptable range. Furthermore, block 1 and block 3 showed the same stiffness characteristics. Similarly block 5 and block 6 also showed the same stiffness characteristics.

Additionally, the right and left sides of the barrier model are symmetrical. Additionally, Table 4 demonstrates that all energy-absorption blocks satisfies regulatory criteria. As resulted from the validation test, it was determined that the FEM of the MDB satisfies the test specifications and is able to be utilized as an impactor within the side-impact simulating.

Table 4 The Absorbed Energy by Every Absorption Energy Structure

Project	The first piece	The second piece	The third piece	The fourth piece	The fifth piece	The sixth piece	Population
Simulation	9.55	14.26	9.55	4.27	3.93	3.93	45.49
Regulatory Requirements	9.5±2	15±2	9.5±2	4±1	3.5±1	3.5±1	45±3
Result	Qualified	Qualified	Qualified	Qualified	Qualified	Qualified	Qualified

IV. THE FEM ESTABLISHMENT AND VERIFICATION

A. The Vehicle FEM Establishment

In this study, the vehicle simulation model is constructed using an ANSYS software. A small mesh is utilized in the primary deformation areas of the side impact, while the bigger mesh is employed in the supplementary parts, to regulate the model size under the presumption of guaranteeing the model's correctness and saving computation time. Moreover, the internal panel is made from piecewise linear-plastic material in the case that the driver's side front door is combined with the internal trim panel. Furthermore, all of the body panel, BIW, as well as seat structure of the FEM of the entire vehicle are reproduced using shell-element mesh. The motion relations have a significant role in the definition of the door hinge, suspension system, as well as tire.

Furthermore, the components connection relationship depends on the manufacturing process of the body, as well as the welding location is controlled. Also, the engine can be simplified as a solid body, as well as the sealing strip, harness, and pipeline components that have minimal influence on the side impact are substituted or eliminated by the mass unit.

B. The Vehicle FEM validation

To assess the validity and correctness of simulation findings, quantitative and qualitative assessment approaches are commonly utilized [15]. Qualitative assessment approach is primarily used to examine and evaluate the shape of deformation at the impact region in simulation and test, the impact properties of the major components, as well as the vehicle motion and MDF after the collision occurrence. The quantitative assessment compares the acceleration, deformation, as well as the impact load of various vehicle and MDB sections.

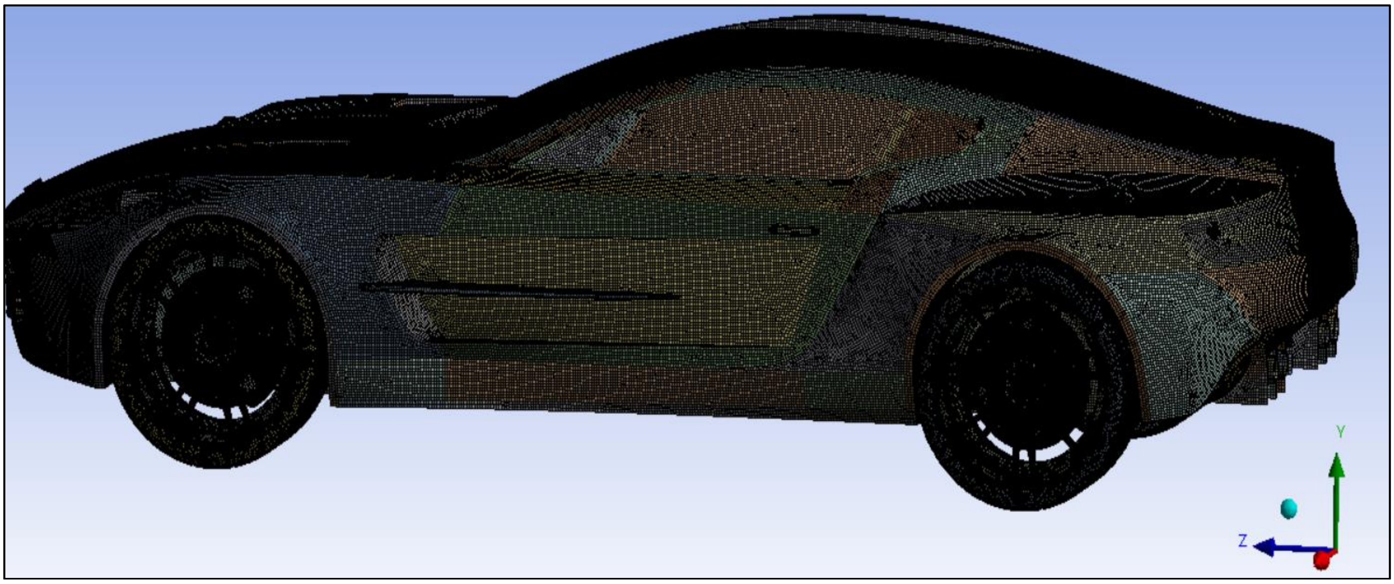

Fig 14 The FEM of the studied vehicle.

The simulation model of the side impact is conducted in this study as illustrated in Figure 15.

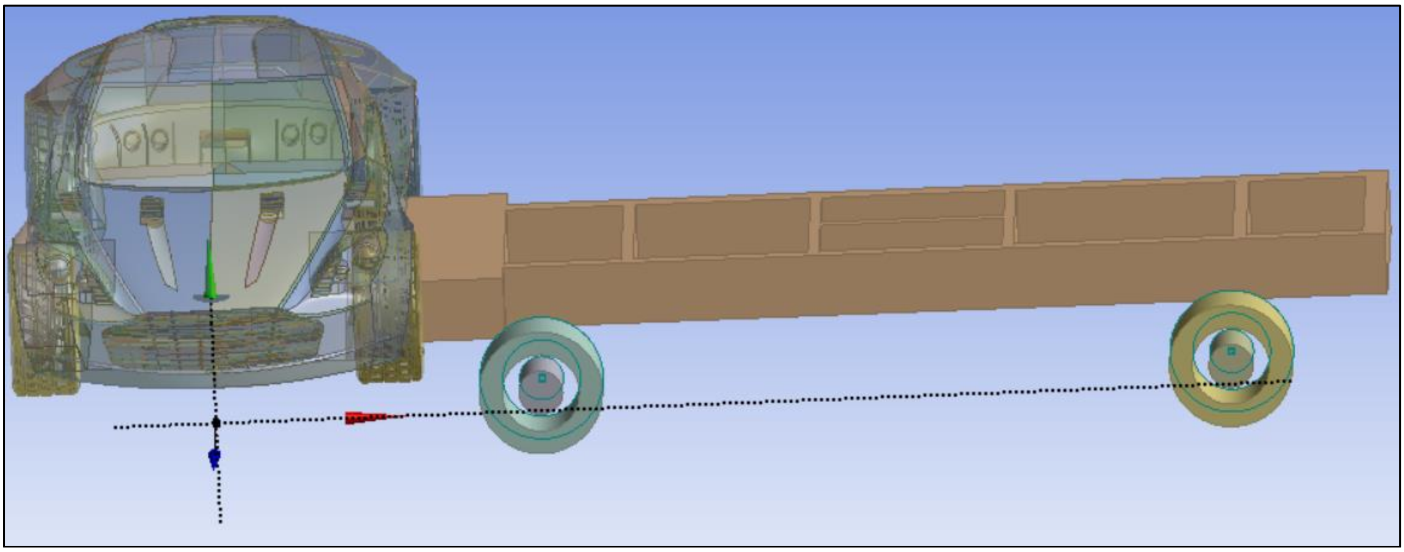

Fig 15 The Simulation Model Of The Side Impact.

At a velocity equals 50 km/h, the MDB strikes the halted vehicle vertically. Its longitudinal line goes via the (R-point) at the front seat. Also, the time of simulation was 140 milliseconds. Verify if the characteristics of the simulation model are appropriate based on energy changes. Also, the required time for simulation was 140 Ms. Reasonable side impact parameters should be ensured based on the energy change. Figure 16 shows the system's energy curve.

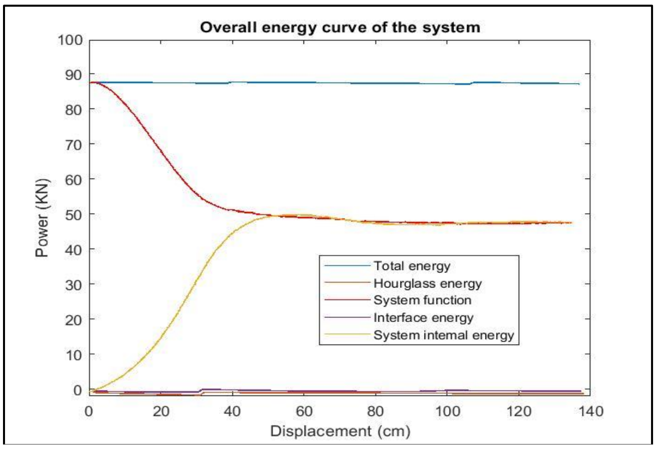

Fig 16 The system's energy curve.

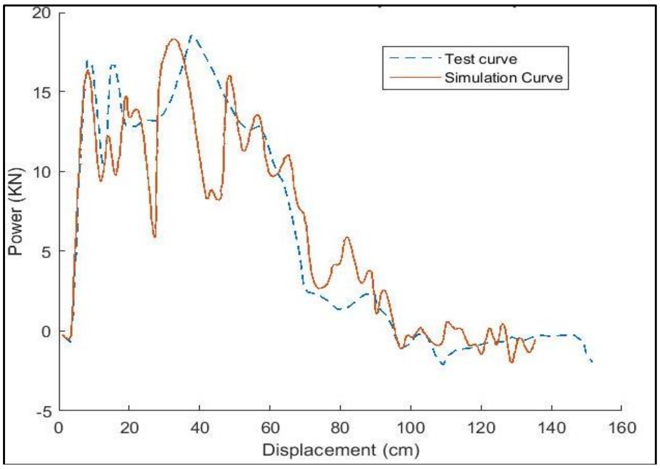

Fig 17 The acceleration vs. displacement at B-pillar lower end.

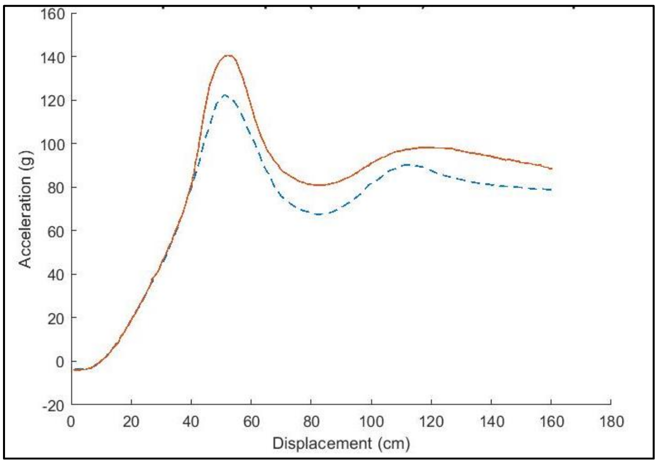

Fig 18 The acceleration vs. displacement at B-pillar lower end after and before the optimization process.

Based on the results provided in Figure 16, reasonable results regarding the system energy composition were obtained. Also, it was noted that the overall total energy was preserved, both slip interface and hourglass energy have relatively small that do not surpass (5%) of the entire energy. As a result, it is possible to argue that the connections of the model, FE mesh in addition to the setup of different solution cards' standards were suitable throughout modelling process.

Moreover, based on Figure 17, there is a constant acceleration change trend, as well as a notable agreement between the occurrence and the peak value. An error might be occurred by the exclusion of specific body attachments, material properties, or the solder joint models that vary from the real scenario, although the total error is lower than 5%, particularly during the initial and second peaks. It was reached that this model may be utilized in place of the actual vehicle in the subsequent side impact modelling study.

V. SAFETY IMPROVEMENT DESIGN FOR THE SIDE IMPACT OF THE VEHICLES

The real vehicle testing method is not the best choice for optimizing the safety design performance for the side impact, the primary method is depend on analysis technique, such as CAE technique, and multidisciplinary theories for optimizing the design of the body. The main concept behind the multidisciplinary optimization theories is to discover the effective multidisciplinary collaboration strategy, the dependence on the simultaneous designing of subsystems, and finally reach the optimal solution for body systems.

A. The Main Design Objectives

The main objectives of achieving the safety design for vehicles body should be determined before any multidisciplinary optimization process. During the testing method for the competitive analysis of the products, the competitive vehicles which has a high safety against the side impact was considered to achieve the design objectives.

Table 5 represent the side impact safety performance, and the design objectives for a certain model. The optimization of the constraints and objectives depends on selecting the impact acceleration of the B-pillar and the internal plate intrusion of the B-pillar. However, the verifications of the feasibility conditions for the optimum design scheme depends on using the performance objectives.

ISSN No:-2456-2165

Table 5 The Performance of the Side Impact and the Target Design for the Reference Vehicle

Key position	Reference car	Target value
Front door inner panel (chest) invasion (mm)	147	≤ 140
B-pillar inner plate (chest) invasion (mm)	156	≤ 140
Invasion velocity of lower part of B-pillar (m·s-1)	7.4	≤ 7
B-pillar (head) acceleration (g)	142	≤ 135

B. The Optimization Collaboration between Multidisciplinary Depending on Adaptable Weighting

The main spirit of the problem which related to the optimization of a multidisciplinary collaboration is the finding of a vector set which made-up of the design variables within the feasible region. Hence, the alternative values of the objective function for the deferent performance for the body system which can be improved as possible.

In past cases, the studies which were conducted for discussing the problems of the multidisciplinary depends on the single objective optimization process. Still there is a difficulty in determination the total optimum system solutions depending on a single objectives optimization technique due to the existence of conflicts between the objectives. A multidisciplinary collaboration optimization technique depending on adaptive weighting which is suggested to consider the optimum solution problem of multi-objective collaboration optimization. The suggested mathematical model is defined as follows;

$$min \ \ldots \ F(t) = 0 < \qquad \text{Eq.4}$$

$$\min \ F(t) = \sum_{i=1}^{n} w_i F_i(t) \quad 0 \le w_i \le 1 \cup \sum_{i=1}^{n} w_i$$

$$s.t. \quad g_i(t) < 0 \quad i = 1,2,\ldots,p$$

$$g_i(t) = 0 \quad j = 1,2,\ldots,q$$

- $F(t)$: Represents the total weighted objective function
- $F_i(t)$: Represents the objective function for the performance
- $g_i(t)$: Represents the inequality constraint function
- $g_j(t)$: Represents the equality constraint function
- w_i : Represent the weighting coefficient
- n, p, q : Represents the number of corresponding functions
- t: Represents the vector composed of design variables.

During the design optimization, the (wi) was chosen using the adaptive adjustment, as well as the F(t) was constructed.

After that the multi-objective was reduced to become a single objective, interdisciplinary collaborative optimization was performed under the design restrictions. After finishing the optimization process, the resulted $F_i(t)$ value was verified to check if it satisfy the requirements. In the case that the $F_i(t)$ satisfy the requirement, the optimization outcome will be used as the required outputs. If not, the (wi) will be hanged based on the design specifications, and the optimization design should be repeated.

Lastly, the most effective multi-objective optimization design, which satisfy the design requirements will be adapted, and the (wi) will be called as the adaptive (wi). Relying on the optimization process' objectives and outcomes, the adaptive weighting procedure can adaptively raise or reduce the response function's weight ratio. Lastly, it can achieve the best solution automatically, considerably reducing the required time for optimization. Both side-impact safety, as well as the lightweight performance are enhanced simultaneously in the optimization design. In the case of stabilizing the body mass conditions, the B-pillar acceleration and intrusion will have the minimum values, in order to increase the side impact's safety, the following formula can be used:

$$\min \ F(t) = w_1 d_b + w_2 a_b \qquad Eq.5$$

$$s.t. \quad m < m_0$$

$$d_b \le 120\, mm \qquad a_b \le 120\, mm$$

$$0.8\, mm \le t \le 2.5\, mm$$

Where, w1 & w2 represents the adaptive weighting-coefficients, as well as the (m_0) represents the mass of the vehicle before the optimization, which was 1674.3 kg.

Depending on both optimization objectives as well as the design variables, the Sequential-Quadratic-Programming (SQP) approaches was utilised in order to establish the multi-objective optimization utilizing MATLAB. Hessian matrix was calculated utilising the quasi Newton approach. Several iterations were conducted, and the outcome are provided in Table optimization results are shown in Table 6.

Table 6 The Optimization Outcomes.

Name	w_1	w_2	F	d_b /mm	a_b/g	m/kg	$t1$	$t2$	$t3$	$t4$	$t5$	$t6$
Initial value	0.6	0.4	147.93	146.40	150.22	1674.3	1.6	2.0	2.8	1.8	1.8	1.60000
Optimization value	0.6	0.4	120.57	113.48	131.20	1673.9	2.0	1.5	1.6	2.0	2.0	0.8873

By considering the real manufacturing and production, the most suitable values for the design parameter are provided in Table 7.

Table 7 The Design Parameters Values

The design parameters	**t1 (mm)**	**t2 (mm)**	**t3 (mm)**	**t4 (mm)**	**t5 (mm)**	**t6 (mm)**
Value	2.0	1.5	1.6	2.0	2.0	0.8

In accordance with the resulted optimization design variables values, the optimized model is constructed to be used in the simulation test after evaluating the optimization findings to determine whether they have any practical importance. The outcomes of the simulation were compared to those without optimization, and are provided in Table 8.

Table 1 The Performance Index before and after Improvement

Name	Before improvement	After improvement
Front door (chest) intrusion (mm)	146.98	122.46
B-pillar (chest) invasion (mm)	142.00	122.05
B-pillar (head) acceleration (g)	150.20	132.8
B-pillar (bottom) invasion velocity (m·s-1)	10.70	6.3
Vehicle mass (kg)	1674.30	1673.8

Based on the results shown in Table 8, the optimized structure performance surpasses the design objectives, as well as the safety of the side impact showed a notable improvement without affecting the mass of the vehicle.

Furthermore, a comparison between the acceleration, invasion, as well as instruction velocity with displacement before and after optimization process are provided in Figure 19, Figure 20, and Figure 21.

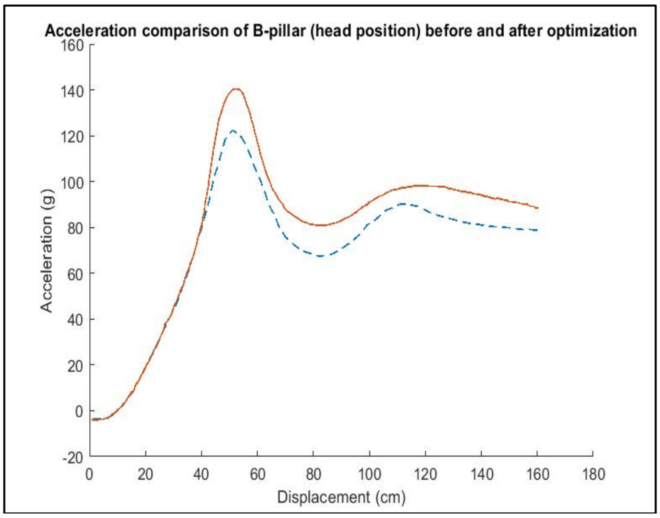

Fig 19 The acceleration vs. displacement after and before the optimization process.

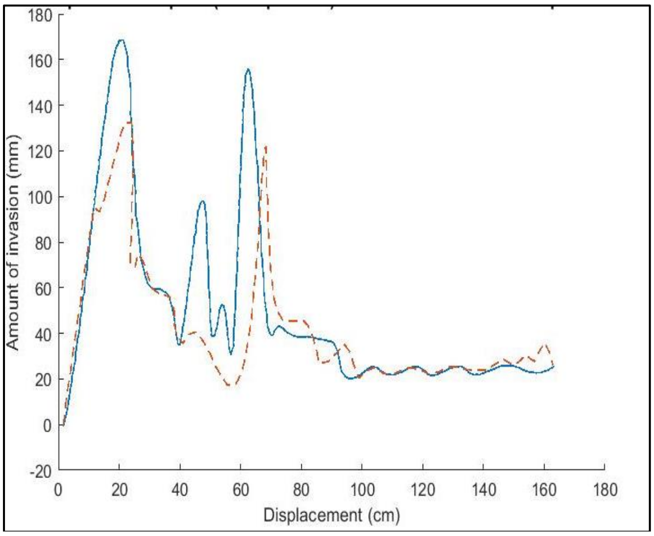

Fig 20 The Invasion Amount vs. Displacement after and before the Optimization Process

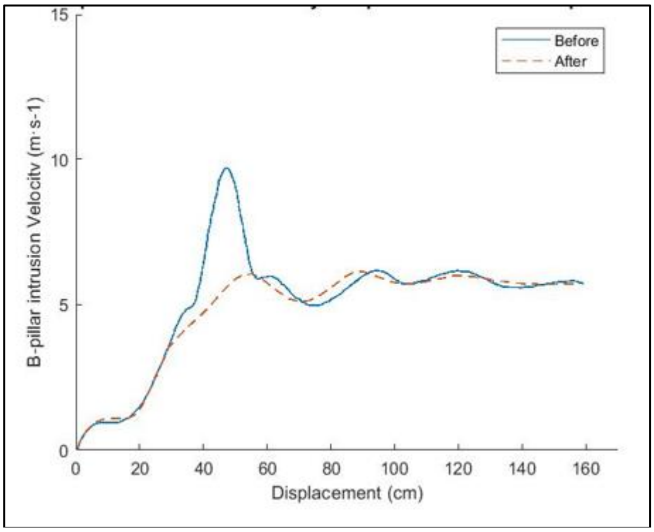

Fig 21 The Intrusion Velocity vs. displacement after and before the Optimization

Results showed that the instruction velocity, invasion, as well as acceleration were reduced after the optimization process by 5.5 (m.s-1), 130 mm, and 20 g respectively. Through the aforementioned study, it was observed that the established optimized structure offers much more side-impact safety when it is compared with the original vehicle, demonstrating the viability of the suggested optimization strategy in this work. This behaviour is caused depending on the design strategy described in this study, which improves the anti-collision beam's position by employing ultra-high strength steel, thicker plates, more structural stiffeners, and anti-collision beams with various section forms. Through simulation testing, the actual vehicle's safety was confirmed.

VI. CONCLUSION

Reasonable and acceptable simulation results depend on the valid simulation model, and since there is a possibility of finding flaws in the simulation analysis, errors are more likely to appear. Therefore, the model is defined as reliable after checking and ensuring that the optimization design and analysis are effective. Typically, the moving vehicle and deformable barriers are verified based on the side impact standards and regulations done through this study. The verification outcomes demonstrate that the FEA model is effective and reliable.

The simulation model of the vehicle side impact was performed and verified based on ANSYS simulation software. Furthermore, the verified simulation model has the ability to be employed for additional investigation and research. The vehicle body has a more extensive pillow space layout for rear-end collision or front impact, while the side just delivers a slight collision pillow space. Consequently, improving the side impact resistance of the vehicle and an anti-collision technology is complicated. For the purpose of enhancing the side impact, the critical issue is to transmit the formed collision energy to additional protective columns, beams, roofs, floors, as well as other vehicle car parts. These parts spread and absorb the exerted force, which confines the possible injury degree to the minimum level.

Various modification strategies are offered to enhance the vehicle crashworthiness via research about the vehicle impact characteristics, particularly at the side impact. Based on that, the scheme's effectiveness was analysed and assessed. The simulation outcomes indicate that the intrusion velocity and volume, as well as the acceleration of the optimized body, are decreased by 5.5 m.s-1, 130 mm, and 20 g, respectively, which is remarkably lower than the original vehicle model. The optimization outcomes' feasibility is confirmed via the assessment of the optimization strategy, and the side impact safety is highly enhanced.

REFERENCES

[1]. S. Kim, S. Lee, E. Ko, S. Kim, S. Lee, E. Ko, K. Jang and J. Yeo, “Changes in car and bus usage amid the COVID-19 pandemic: Relationship with land use and land price,” vol. 103168, p. 96, 2021.

[2]. M. Carlier, “Passenger cars - sales by region 2021,” statista, 6 May 2021. [Online]. Available: https://www.statista.com/statistics/257653/passenger-car-sales-by-region/. [Accessed 11 November 2022].

[3]. X. Liu, M. Liang and Q. Luo, “Innovative Electric Vehicle Body Design Based on Insurance Institute for Highway Safety Side Impact Conditions.,” Automotive Innovation, vol. 2, no. 3, p. 201–211, 2019.

[4]. S. F. Abdulqadir and F. Tarlochan, “Composite Hat Structure Design for Vehicle Safety: Potential Application to B-Pillar and Door Intrusion Beam,” Materials, vol. 15, no. 13, p. 108, 2022.

[5]. M. Wu and X. Zhang, “Optimal Control Method for Side Impact Safety of Vehicle Frame Structure,” Jordan Journal of Mechanical & Industrial Engineering, vol. 16, no. 1, 2022.

[6]. B. Fu, R. L. Giossi, R. Persson, S. Stichel, S. Bruni and R. Goodall, “Active suspension in railway vehicles: a literature survey ,” Railway Engineering Science , vol. 28, no. 1, pp. 3-35, 2020.

[7]. D. Elliott, W. Keen and L. Miao, “Recent advances in connected and automated vehicles,” Journal of traffic and transportation engineering, vol. 6, no. 2, pp. 109-131, 2019.

[8]. A. Papageorgiou, M. Tarkian, K. Amadori and J. Ölvander, “Multidisciplinary design optimization of aerial vehicles: A review of recent advancements,” International Journal of Aerospace Engineering, 2018.

[9]. C. Chou, K. Aekbote and J. Le, “A review of side impact component test methodologies,” Int. J. Veh. Saf., vol. 1/2, no. 2, p. 141–184, 2007.

[10]. A. Muhammad and I. H. Shanono, “Simulation of a Car crash using ANSYS,” International Conference on Electronics, Computer and Computation (ICECCO) , pp. 1-5, 2019.

[11]. B. W. Lenggana, A. R. Prabowo, U. Ubaidillah, F. Imaduddin, E. Surojo, H. Nubli and R. Adiputra, "Effects of mechanical vibration on designed steel-based plate geometries: behavioral estimation subjected to applied material classes using finitelement method," Curved and Layered Structures, vol. 8, no. 1, pp. 225-240, 2021.

[12]. D. A. Reed and J. Dongarra, "Exascale computing and big data," Communications of the ACM, vol. 58, no. 7, pp. 56-68, 2015.

[13]. M. N. Perc and D. Topolšek, "Using the scanners and drone for comparison of point cloud accuracy at traffic accident analysis ," Accident Analysis & Prevention, vol. 135, p. 105391, 2020.

[14]. J. M. Rodríguez, J. M. Carbonell and P. Jonsen, "Numerical methods for the modelling of chip formation," Archives of Computational Methods in Engineering, vol. 27, no. 2, pp. 387-412, 2020.

[15]. L. Busetto, W. Wick and C. Gumbinger, "How to use and assess qualitative research methods," Neurological Research and practice, vol. 2, no. 1, pp. 1-10, 2020. Zilina, Slovak Republic.

[16]. Tomasikova, M., Brumercik, F., & Nieoczym, A., Vehicle simulation model creation. LOGI, Scientific Journal on Transport and Logistics, 6(1), 2015, 130–136.

[17]. Wallmark, O., & Nybacka M., Design and implementation of an experimental research and concept demonstration vehicle. In Vehicle Power and Propulsion Conference (VPPC), IEEE, 2014.

[18]. Zhou, J., Shen, X., & Liu, D., Modeling and simulation for electric vehicle powertrain controls. In Transportation Electrification Asia-Pacific (ITEC Asia-Pacific), IEEE Conference and Expo. Beijing, 2014.