Development of Tata Super Ace Finite Element Model

Muhamad Fareez Laili[1,*], Khairil Anwar Abu Kassim[1], Mohd Hafiz Johari[1], Yahaya Ahmad[1], Wira Jazair Yahya[2]

[1] ASEAN NCAP, Malaysian Institute of Road Safety Research (MIROS), Taman Kajang Sentral, 43000 Kajang Selangor, Malaysia
[2] Malaysia – Japan International Institute of Technology (MJIIT), Universiti Teknologi Malaysia

ABSTRACT

Finite Element Method (FEM) widely used to solve engineering problems and mathematical models. In this paper, Tata Super Ace was chosen as a model of vehicle to generate finite element model on its body structure called chassis. It also presents a detailed finite element model of a Super Ace Tata model called meshing. In this study, mesh is made based on Finite Element Model. Meshing is carried out through Computer Aided Engineering (CAE) Software called Altair Hyperworks. As the result of these efforts, the finite element vehicle model was developed with the following characteristics: number of parts 3544, number of nodes 3 million, and number of elements 3 million. The average element size used is 8 mm with a minimum size of 4 mm. Thus, the model can successfully be used in computational frontal impacts analysis.

Keywords:
Finite Element, discretization, Tata Super
Ace, Computer Aided Engineering

1. Introduction

Small truck is one of the transportation vehicles that are being used to deliver customer things. Nowadays, online shopping is an important thing for people to buy their needs without need to drive or ride all the way to the shop (foods, accessories, etc) for example during pandemic COVID-19. Thus, there will be a lot of delivery order transportation will be needed to send their customer's item to their location.

Trucking is a well-known important element for freight movement and economic development. Studied by Zheng et al. [13], truck crashes contribute to a large number of injuries and fatalities due to additional risks, such as a larger vehicle size, heavier weight, and possible hazardous material release. In 2014, the total number of fatalities in truck crashes was 3903 (Federal Motor Carrier Safety Administration, 2014). Compared with the total number of fatalities in strictly passenger car crashes, 28,559, truck crashes do not seem as alarming. However, truck crashes are overall more likely to result in more severe outcomes such as a fatality. In 2014, there were 14 fatalities in large truck crashes per 100 million vehicle miles traveled by large trucks, while only 10.5 fatalities in passenger vehicle crashes per 100 million vehicle miles traveled by passenger vehicles. Additionally, there were

---

Corresponding author.
E-mail address: muhamadfareez@miros.gov.my

29.4 injury crashes involving large trucks per 100 million vehicle miles traveled by large trucks, compared with 58.5 for passenger vehicles (Federal Motor Carrier Safety Administration, 2014).

The need to improve commercial trucking company safety performance has been a major social concern. In order to reduce the impact of crash to the vehicles, the vehicle must be able to absorb the energy created during the impact. Although modern vehicles are designed with composite materials, that can absorb most of the energy created during impact, yet the risk of injury to the passenger or driver is still high [1]. Safety is one of the design concerns of the automotive industry. The National Highway Traffic Safety Administration (NHTSA) is supporting the PNGV to ensure that the new generation vehicles meet existing and anticipated safety standards. As part of this effort, NHTSA has a program to develop a set of finite element models for various vehicles that represent the full range of vehicle types currently on the road from a subcompact car up to a sport utility vehicle and full-size truck. This set of vehicle models will be used to establish the crash safety of future lightweight vehicles developed under the PNGV program [9].

The crash test is also an important step in validating the design of a new vehicle. However, the high cost of experimental research (virtual crash test) limits the number of crash tests and, as a result, sufficient data could not be collected. Alternatively, in addition to experimental research, computational modelling and simulation have been commonly used to analyses car collisions. As an efficient computational instrument, the Finite Element Method (FEM) plays a crucial role in the simulation of a crash test [7]. Vehicle collision by using computer simulation has been developed and improved significantly over the past few years. With the advances in computer technology and non-linear finite element method, simulation of crash test of vehicles is possible to perform. This paper presents a FEM on a Tata Super Ace vehicle and uses it for crash analysis in the future. The details of this method are briefly explained and discussed. Suggestion for further improvement in developing Finite Element Tata Super Ace are also included. Altair Hypermesh is used for developed a Finite Element Modelling.

2. Objectives

The objectives of the current study are as follow
- Literature survey on Finite Element, Tata Super Ace vehicle and method on meshing
- Generating finite element model for Tata Super Ace model vehicle

3. Literature Review
3.1 Vehicle Categories

Vehicles are categorized by a letter indicating their intended use. It is important that you know the category of your vehicle in order to fully understand your tax and test specifications. In this study, Tata Super Ace is categorized as N1 vehicles.
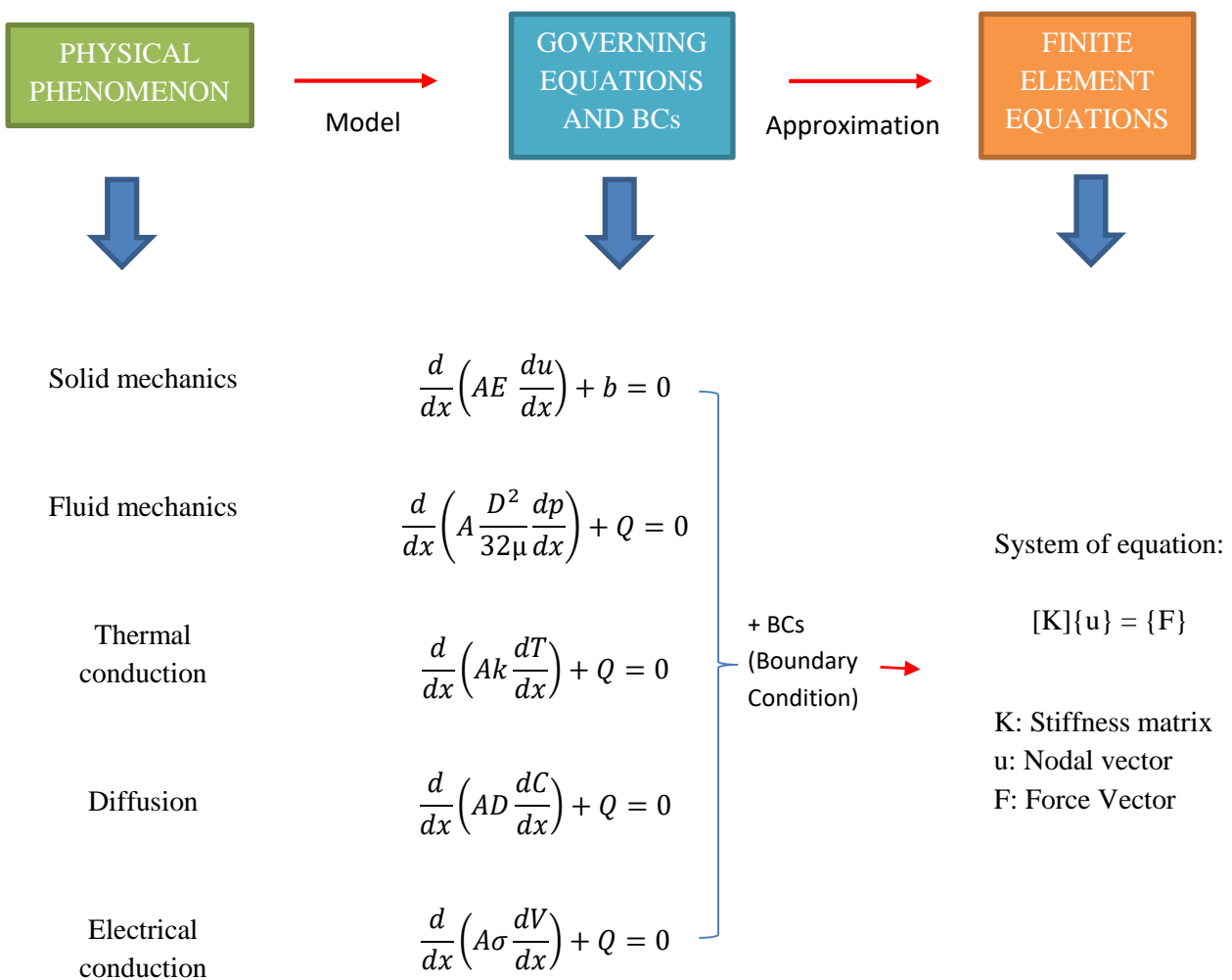
Vehicles from the category namely N1 vehicles engineered for the transport of goods, are designed to operate in a highly competitive, non-emotional market segment where sound quality is of no importance. Other than with passenger cars, this type of vehicle is acquired by purchase departments under economical aspects while the later driver has no influence on the choice [8]. Table 1 shows the commercial vehicle for 'N' category.

Table 1
Commercial vehicle for 'N' category

N1	Vehicles built and constructed for the transport of goods and having a maximum weight not exceeding 3,500 kg. Light vans and trucks, for example.
N2	Vehicles built and manufactured for the transport of goods with a maximum weight of 3,500 kg and not exceeding 12,000 kg, e.g. larger vans and trucks.
N3	Vehicles designed and constructed for the carriage of goods and having a maximum mass exceeding 12,000 kg e.g. trucks

3.2 Finite Element Method (FEM)

Finite Element Method (FEM) is a numerical technique that is used to solve a broad range of complex physical phenomena, especially those exhibiting geometrical and material non-linearities such as those that are often encountered in the physical and engineering sciences (Dean n.d.).

PHYSICAL PHENOMENON →(Model)→ GOVERNING EQUATIONS AND BCs →(Approximation)→ FINITE ELEMENT EQUATIONS

Solid mechanics

$$\frac{d}{dx}\left(AE\,\frac{du}{dx}\right) + b = 0$$

Fluid mechanics

$$\frac{d}{dx}\left(A\frac{D^2}{32\mu}\frac{dp}{dx}\right) + Q = 0$$

Thermal conduction

$$\frac{d}{dx}\left(Ak\frac{dT}{dx}\right) + Q = 0$$

Diffusion

$$\frac{d}{dx}\left(AD\frac{dC}{dx}\right) + Q = 0$$

Electrical conduction

$$\frac{d}{dx}\left(A\sigma\frac{dV}{dx}\right) + Q = 0$$

+ BCs (Boundary Condition) →

System of equation:

$$[K]\{u\} = \{F\}$$

K: Stiffness matrix
u: Nodal vector
F: Force Vector

3.3 Computer Aided Engineering (CAE) Software
3.3.1 Hypermesh

Research by Tamime [11] mentioned that he Hypermesh program is a product of Altair Hyperworks. It is a high performance and broad mesh generation finite element pre-processing programs. It is commonly used and compatible with commercially available CAD and CAE systems. In this project, the finite element model of each specimen is generated using the Hypermesh program. This program is needful in generating a finite element model of an already existing CAD model of a specimen needed for finite element analysis.

Hypermesh has evolved over the past 2 decades into the leading pre-processor for FEA high fidelity modeling, and its ability to quickly generate mesh for complex geometry has made Hypermesh popular amongst FEA researchers. This program supports a broad range of CAD and solver interfaces.

Hypermesh gains its relevance in the following:

    a. Automatic Shell Mesh generation
    b. Model Morphing
    c. Automatic Solid mesh generation
    d. Manual mesh generation
    e. Geometry Dimensioning
    f. CAD Interoperability and compatibility
    g. Batch Meshing for fast automatic high quality finite element mesh generation for assembly

Hypermesh is an advanced easy to use tool capable of editing CAE models and also capable of generating meshes in different element sizes and geometry as shown below.

    a. Tetra meshing
    b. CFD meshing
    c. High fidelity meshing
    d. Solid map hexa meshing
    e. Surface meshing

Hypermesh gains its relevance in converting solid 3D models into Finite Element Meshes by dividing the solid part into small parts. The quality of the mesh determines the degree of accuracy of the results of finite element analysis in simulations. The quality of mesh is determined by some criteria such as the aspect ratio of the meshes, the warpage and the geometry clean up.

4. Tata Super Ace
4.1 Model description

The geometrical model of Tata Super Ace (Figure 1) had been purchased from a legal website. This website known as 'Hum3D' provides many 3D models. Since 2005 Hum3D has been helping 3D artists from more than 80 countries to save time on 3D modelling and render realistic 3D models that are used in movies, video games, AR-apps and visualizations. All models are verified by lead 3D artists, who check their compliance with professional requirements and up-to-date 3D modeling standards.

By purchasing the vehicle model, it could reduce the development time and error related to detail structure as the geometry was verified. This website was chosen because it has the biggest car collection in the world with more than 9000 vehicles and 700 plus brands.

Fig. 1. Tata Super Ace

Fig. 2. Tata Super Ace 3D model (Hum3D)

Based on the figures, it can be compared that real Tata Super Ace model shows almost the same structure to the 3D Tata Super Ace model. The design of the tires, sport rims, location of the battery, the Tata Super Ace logo in front of the front parts shows almost the same design. It is just some of the additional parts from the 3D model that may not affected in simulation can be ignored.

4.2 Background

Tata Super Ace, launched by Indian multinational automotive company Tata Motors Limited, is a 1 tonne mini diesel truck. It has a loading deck length of 2630 mm, a top speed of 125 km / h (78 mph) and a turning radius of 5.1 m. It is based on the idea of the highly popular and acclaimed Tata Ace, India's first sub-one tonne four-wheel mini truck, which also received the BBC Top Gear's 2006 Best Commercial Vehicle Design Award. Figure 3 shows the summary of specification for Tata Super Ace vehicle from Tata Motors Company.

Tata Super Ace		
Engine		475 IDI TCIC 1405 cc
Max. Power		70 Hp @ 4500 rpm
Max. Torque		135 Nm @ 2500 rpm
Clutch		Single plate, dry friction, diaphragm type
Gearbox		GBS 65-5/5.07
Type		5-speed manual transmission
Steering		Hydraulically assisted power steering
Brakes	Front	Disc brakes
	Rear	Drum brakes
Suspension	Front	Independent strut type with anti-roll bar
	Rear	Semi elliptical leaf spring
Shock Absorber		Hydraulic double-acting telescopic type
Tyres		175 R14 LT 8PR radial tubeless
Fuel Tank		38 litres
Max. Speed		125 km/h
Max. Gradeability		39%
Key Dimensions & Weights		
Wheel base		2380 mm
Vehicle dimensions (l*b*h)		4340 mm x 1565 mm x 1864 mm
Cargo Box Dimensions(l*b*h)		2630 mm x 1460 mm x 300 mm
GVW		2350 kg
Kerb Weight		1260 kg

Fig. 3. Tata Super Ace
Source: (Choose n.d.)

5. Methodology

5.1 Importing model

The file format of the Tata Super Ace model was in Step file (.stp). In hypermesh, it is able to detect various file format automatically by importing the file geometry (Figure 4). It is consumed a lot of time if the file contains large number of geometries.

5.2 Geometry clean-up

Usually, CAD data is provided in .igs/.iges/.step format. Geometry clean-up is one of the parts in meshing activity. CAE engineers should at least have the basic knowledge of CAD. Before starting the meshing process, the geometry should be carefully checked for:

- Free edges
- Scar lines
- Duplicate surfaces

- Small fillets
- Small holes
- Beads
- Intersection of parts (assembly of components)

In order to reduce error when meshed, this factor need to be taken into consideration as it can avoid failure when doing a simulation [6].
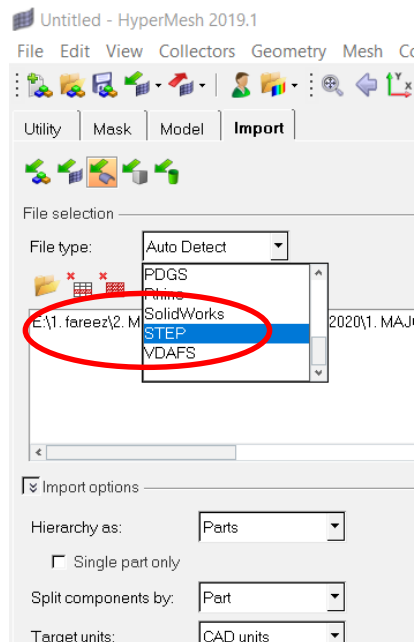

Fig. 4. Hypermesh importing geometry

5.3 Partition of Tata Super Ace components

Tata Super Ace model that was purchased need to be separate to their component because of some of the model come with 1 set of components only (Figure 5).
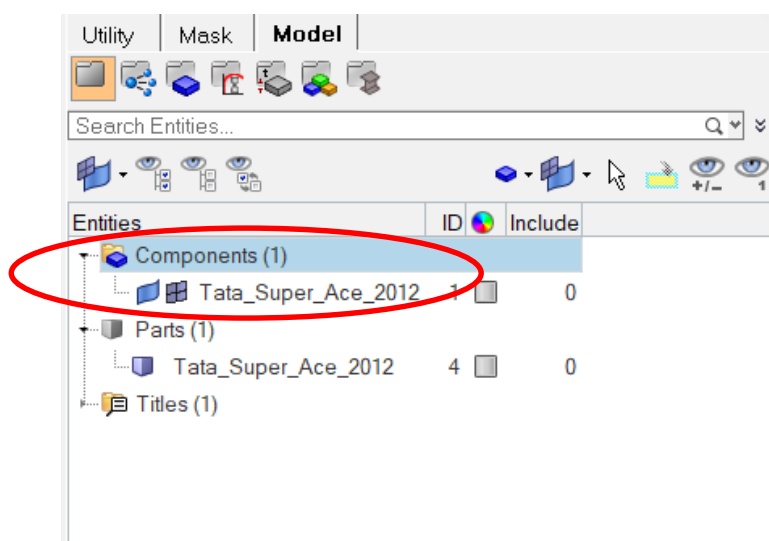

Fig. 5. Model panel in Hypermesh

Thus, the model should be separate in order to make the model easier to hide unnecessary part, assign materials and others. Also, meshing can be done according to their size of elements based on the more important parts that need to be mesh. As we know, more critical parts must be mesh with a finer element size to get the accurate result. While the less crucial parts, can be mesh with a bigger element size. This will reduce the overall time of processing for the model.

5.4 Midsurface

Usually, Computer Aided Design (CAD) provided that is not compatible for Computer Aided Engineering (CAE) operation like meshing. Some of the geometric-topological features are irrelevant and suppression of them does not harm the accuracy of the analysis-result to a large extent, but in turn provides significant leverage in terms of processing time-space. Thus, CAD models are often simplified-abstract-idealized to meet the needs of CAE analysis. One of the simplification techniques is called Midsurface, which is very suitable for thin-walled parts used in the field of plastics and sheet metal. Thin parts of the model are idealised to the surface along with thickness data later to be used for the simulation of shell components. Midsurface have been research for past few decades. Also, nowadays CAD and CAE can be able to do midsurface. Apart from CAE, midsurface has found applications in Visualization, Animation and Feature Recognition as well.

Fig. 6. Midsurface panel in Hypermesh

Midsurface may also be used in thin portions of the usual/mixed-dimensional/thick-thin sections, but there is a need to figure out the treatment of interfaces/joints/couplings considered to be beyond the scope of current research. Midsurface is supposed to have proper connectivity (no gaps) and to follow the shape of the base portion. As the face-pair wise mid-surface is generated, it needs to be stitched together to form a continuous shape. In midsurfacing techniques, there are two broad categories namely 'Medial Axis Transform (MAT)' and 'Midsurface Abstraction (MA)' [10].

5.5 Meshing

The basic idea of FEA is to make calculations at only limited (Finite) number of points and then interpolate the results for the entire domain (surface or volume). Any continuous object has infinite degrees of freedom and it just not possible to solve the problem in this format. Finite Element Method reduces the degree of freedom from infinite to finite with the help of discretization or meshing (nodes and elements) (Figure 8).

No. of Points = ∞
DOF per Point = 6

Total equations = ∞

No. of Nodes = 8
DOF per Node = 6
Total equations = 48
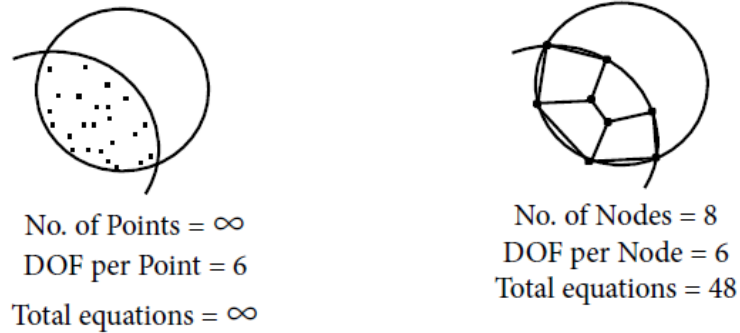
Fig. 7. Schematic diagram of infinite and finite surface

Mesh or grid generation represents an important consideration in attaining the numerical solutions to the governing partial differential equations of the CFD problem. We would like to stress that the term grid or mesh is generally used interchangeably with identical meaning [3].

5.5.1 Meshing technique

a) Batch meshing / Mesh adviser

Most of the software nowadays provide special programme that can automatically do geometry clean up and meshing with little or no interaction from the user. The user has to specify all the parameters such as diameter, fillet radius, average, minimum element length, quality parameters, etc. and the software will run to produce the best possible mesh by fulfilling all or most of the specified instructions. However, this programs is still in initial condition and for many applications the output is still not showing the best results (Do et al. 2014).
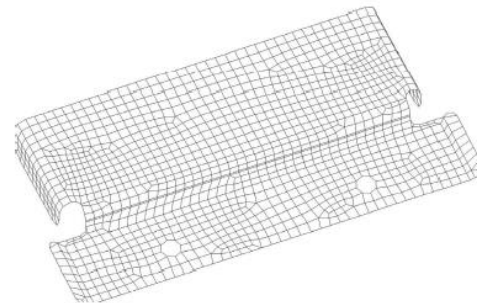
Nodes = 1400, Elements = 1309

Automatic meshing

Not acceptable
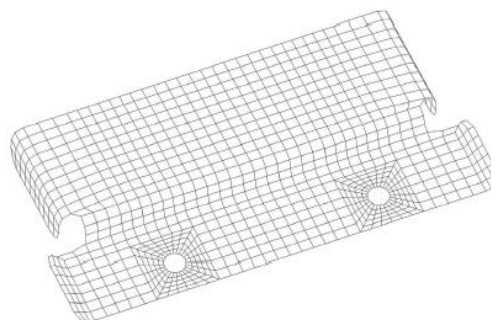
Nodes = 1073, Elements = 982

Mapped meshing

Fig. 8. Automatic mesh vs mapped/interactive / manual mesh [6]

	Automatic / Batch	Mapped (or Interactive)	Manual (Special commands: Spline, Ruled, Drag / extrude, Spin / rotate etc.)
Time required for meshing	i	≈ (intermediate i.e. more than auto but less than manual)	h
Geometry required	✓	✓	X
No. of nodes and elements generated	h	≈	i
User friendliness	h	≈	i
User's control over the mesh	i	≈	h
Structural mesh (flow lines)	i	≈	h
Experience or skill required	i	≈	h
Patience	i	≈	h (specially for brick / hex)

Fig. 9. Usage comparison between automatic/batch, mapped, manual meshing [6]

b)   Critical areas

Critical areas are locations where high stress locations will occur. Dense meshing and structured mesh (no trias / pentas) are recommended in these regions. Areas away from critical area are general areas. Geometry simplification and coarse mesh in general areas are recommended (to reduce the total DOF's and solution time). High stress regions are critical and could be remeshed with a smaller element length in the second run [6].
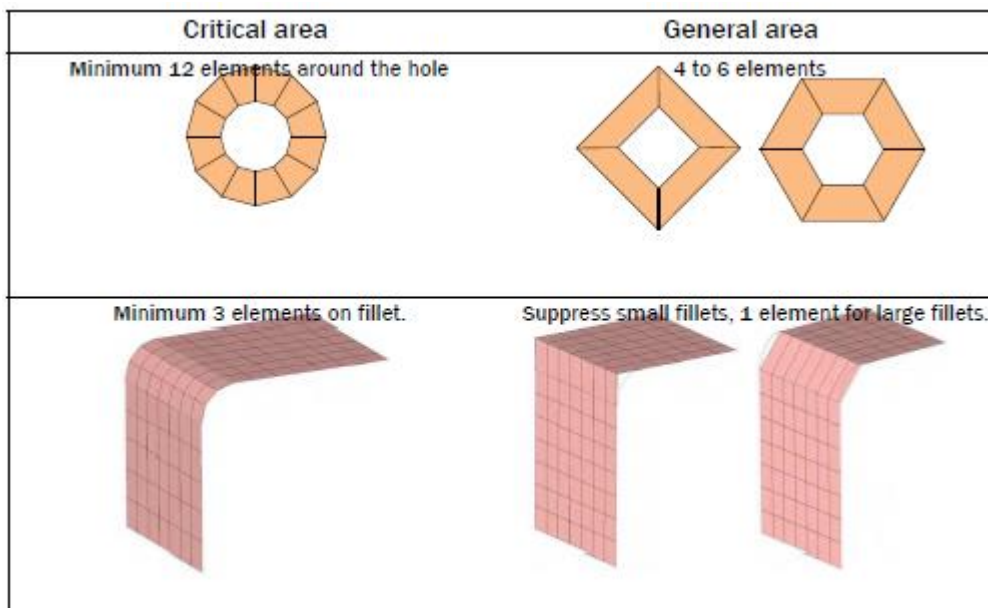
Fig. 10. Rules for modelling holes and fillets [6]
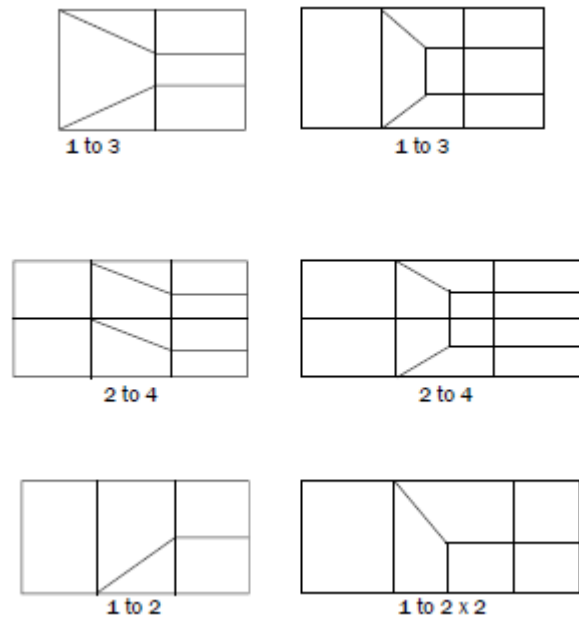
**Fig. 11.** Mesh transition techniques and flow lines [6]

c)   Degrees of freedom

DOFs are important because they dictate the ability of the elements to model a given problem and also dictate whether or not elements are compatible with each other.
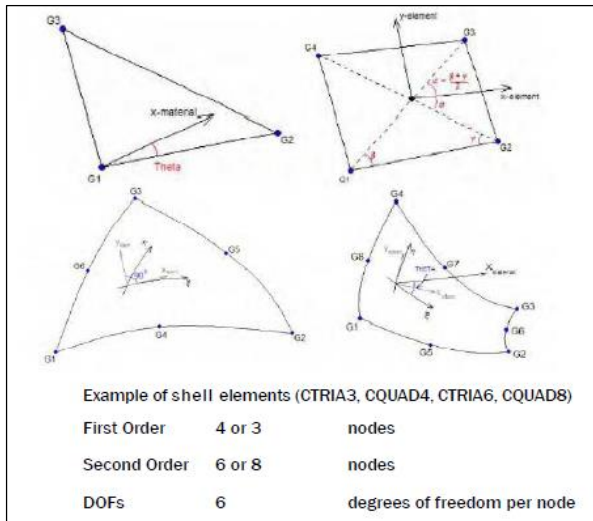
The following diagrams (Figure 12,13,14 and 15) depict single element models, each with several load cases applied in conjunction with a minimum set of boundary conditions. Adding more that the required boundary conditions can be used to learn even more about element behavior [6].
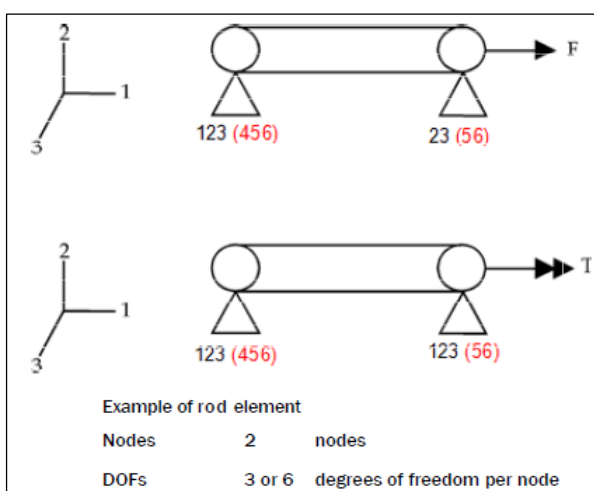
Fig. 12. Rod element
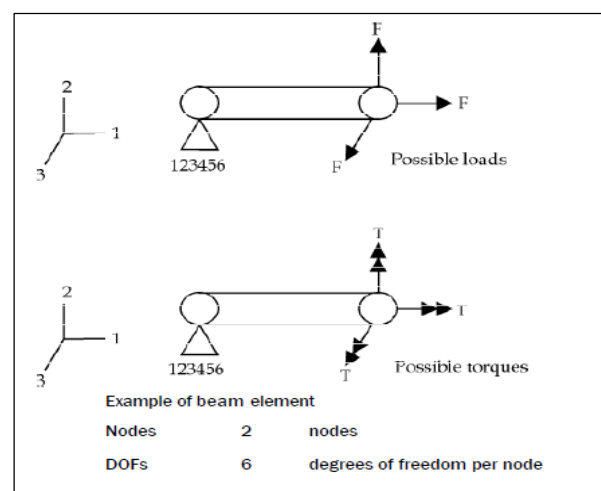
Fig. 13. Beam element

Fig. 14. Shell element
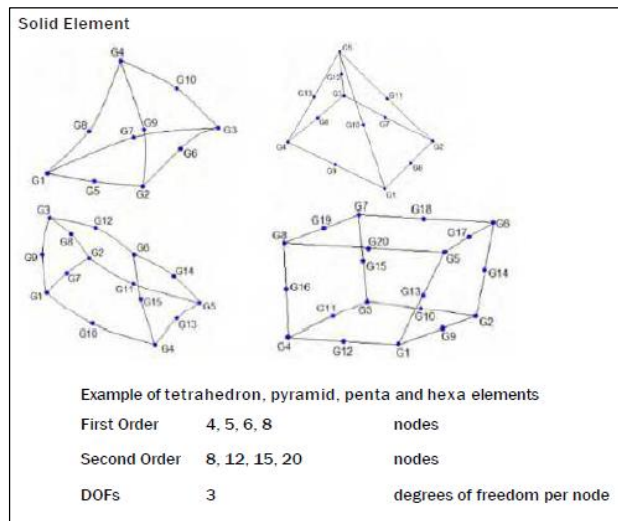
Fig. 15. Solid element

6. Result
6.1 Finite Element Model
6.1.1 Modelling

The complexity of this work is to separate the parts and meshing parts of the model. Since the model used is a purchased model on website, the model just needs to do a modification to make sure the simulation will run smoothly without an error.

The production of N1 vehicles Tata Super Ace model from India was used as the basis for the model. CAD-models of the full vehicle were purchased on a website. Each part was separate and meshed to create a representation of geometry models for finite element modeling. The parts were broken down into elements in such a way that critical features were consistent with the implications of element size on simulation processing times.

The resulting FE vehicle model has 800 thousand elements. This detailed FE model was constructed not included to the full functional capabilities of a suspension such as driveline and steering subsystems. The representation of the model in comparison with the actual vehicle is shown in Figures 6 and 7. The set of elements representing the vehicle was translated into an FE model by defining each as a shell, a beam, or a solid element in accordance with the requirements for using Altair Hypermesh Software. As the result of these efforts, the finite element vehicle model was designed with the following characteristics: number of parts 3544, number of nodes 3 million, and number of elements 3 million. The average element size used is 8 mm with a minimum size of 4 mm.
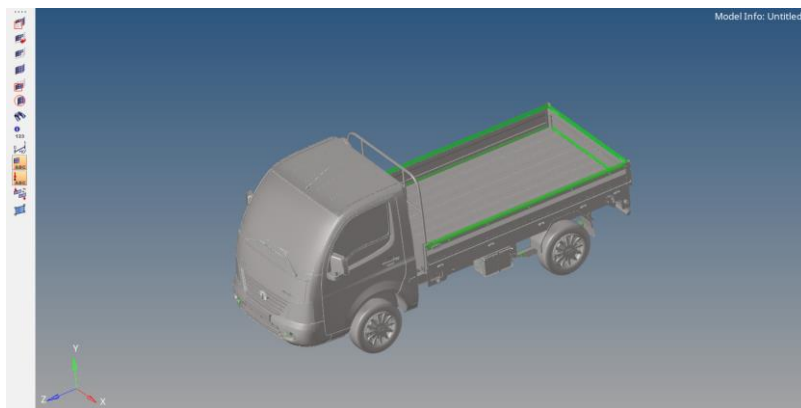
6.2 Tata Super Ace model
6.2.1 Introduction

In this work a Finite Element Model (FEM) of a N1 vehicles is involved. The model is based on a Tata Super Ace and was developed through CAE Software called Hypermesh. Figure 16 shows the model that have been imported from catia format (.cat) into hypermesh software. The design of the car frame in 3D modelling was purchased at the trusted website where the car was designed as a life size model to accurately examine the effects of a car crash.

5.2.2 Meshing

a)   Manual mesh (only include significant effect to the structure during crash)
As we know that manual mesh consumes a lot of time and focus into the surface of the model, the results may be more accurate as compared to automesh features. This manual mesh onto the Tata Super Ace model was done to the most important parts which is the front cab


Fig. 16. Isoparametric view
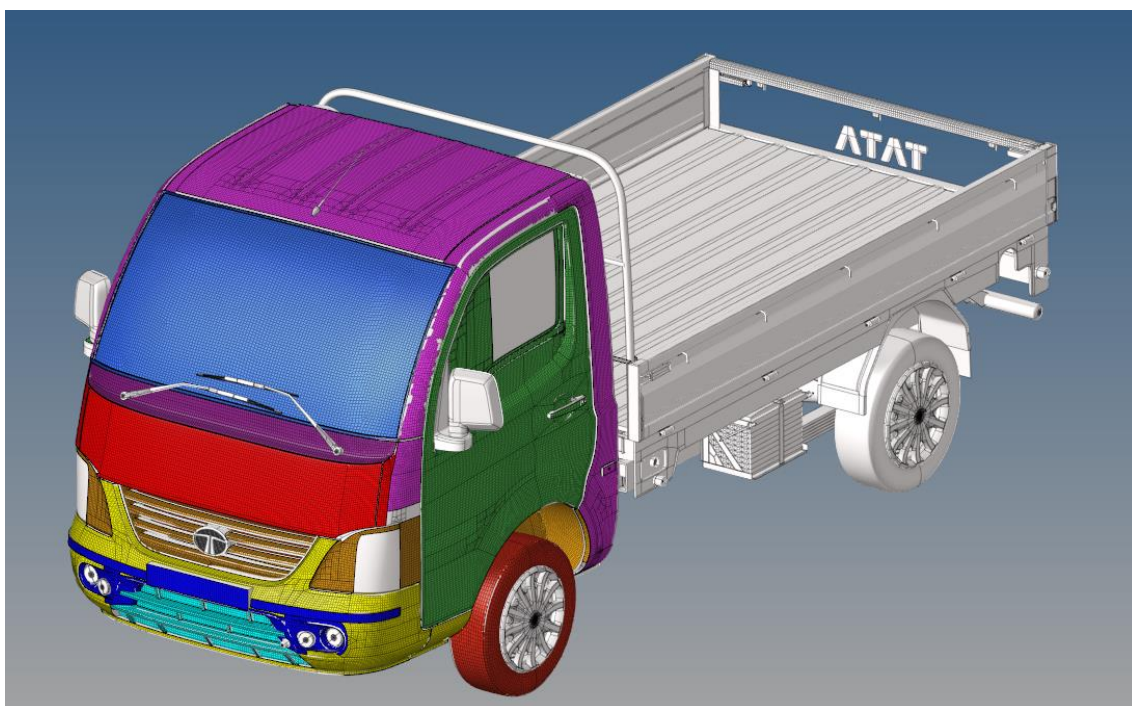
Fig. 17. the overall meshing of Tata Super Ace model

Based on the figure, the front cab undergoes meshing with quad elements. The front cab of the truck is an important part in frontal full width collision test because it will receive a high impact of energy during the crash test. Thus, it will be mesh with a proper and refine elements because it will need higher accuracy of results. It will also help to validate the simulation with the real crash test.

The Front cab are meshed with different colors in order to differentiate each part of the front cab. The front cab parts will undergo simplification in terms of unnecessary parts that are not giving much effect to the crash such as wipers, mirrors, logs and vehicle registration number.
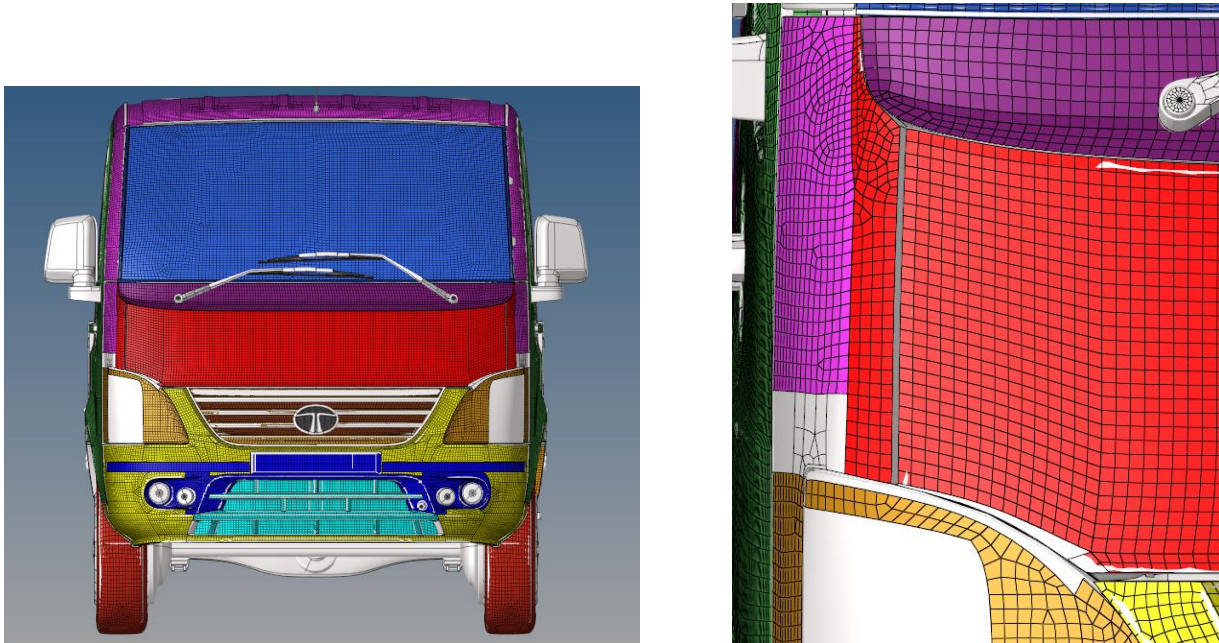
Fig. 18. Detail part of front panel

Figure 18 shows that the details part of the front cab that is front panel (red color) of the truck. It is shown that the front panel of the truck has a good meshing element and is arranged with the same type of elements. One of the problems when doing a meshing on surface is that the meshing element will have an error based on many factors. This part requires a very fine element of meshing just to make sure the simulation will get the most accurate result.

Fig. 19. Truck bed side

Figure 19 shows the truck bed side parts (white color) meshing elements. It is shown that the bed side consists of a bigger element as compared to the front cab of the truck. This is because the bed side of the truck undergoes automesh command that means the meshing element are made automatically. The characterization of the element is that the element is quad element shape with 10mm size of elements.

7. Discussion

FEM is applied to the model vehicle to perform a simulation of contact impact nonlinear dynamic analysis. The applied FEM simulation encompasses a number of individual problems, which should be given appropriate attention (Abdel-Nasser 2013). These problems are:

  a. Selection of elements (shape and size). The important parts of the model were the front parts of the vehicles which is the surfaces that are going to be collide. This surface should be done with a finer element (refinement) to get the accurate results. The finer the meshing, the result will be more accurate.

  b. Totally about 800 thousand elements have been generated in the entire model where the smallest size of the element is about 10 mm in the front contact area (front cab). Coarse mesh may be applied for areas located far from collision region to reduce CPU time. In this study, the part is the truck bed side.

Why used Tata Super Ace model?
Asean NCAP had performed crash on 3 types of small truck vehicle category which is Tata Super Ace, Suzuki Carry and Chery Transcab. Tata Super Ace is one of the light cab commercials that has been crashed by ASEAN NCAP. This flat head type vehicle was introduced in Malaysia market in year 2018. The frontal design also known as full forward flat head type.

  1. ASEAN NCAP Rating

Tata Super Ace was chosen in this study because of it is only achieved "0" star rating by Asean NCAP. There was no Electronic Stability Control, Anti-lock Braking System and Seatbelt Reminder System offered in the model. As the Tata Super Ace is a single cab vehicle, ASEAN NCAP did not perform the COP assessment. Hence, its overall score of 19.15 points was solely contributed by the points from the assessment under the AOP category. Based on this score, the Tata Super Ace achieved Zero Star ASEAN NCAP rating (Figure 20).
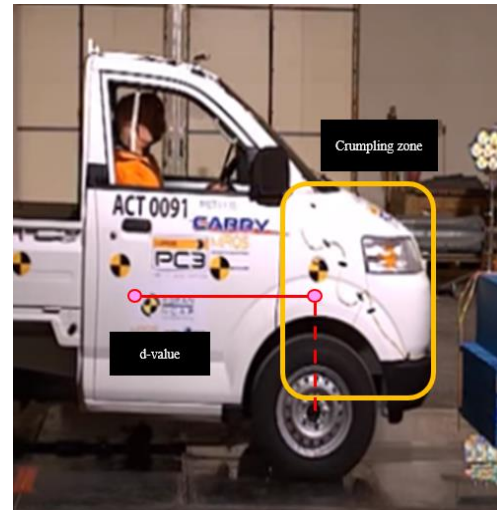

Fig. 20. ASEAN NCAP Rating [2]

Fig. 21. Tata Super Ace (before crash)

Fig. 22. Suzuki Carry

Figure 21 shows the condition of Tata Super Ace before the crash test. It can be compared to Suzuki Carry (Figure 22), Tata Super Ace has the smallest d-value between the front axle and the driver's seat. The d-value is in between 200 mm to 300 mm which is literally small. Then, it can obviously see that the driver's leg was placing in the crumpling zone (yellow area). Particularly that the structure design of Tata Super Ace is not safe as the dummy position was mostly in the survival space.

Fig. 23. Tata Super Ace (during crash)

Figure 23 was the action shot during the t = 0.01 ms of impact. The Figure indicate that the driver's survival space or the crumpling zone was not maintained well. It was approximately destroyed 86% of the residual space. The head of the dummy as well as directly hit the steering wheel as there were no airbag to protect the driver. Thus, the injury value for the head from this crash were somewhat high.

2. High impact of injury

Three types of small truck category had performed crash by ASEAN NCAP which is Tata Super Ace, Suzuki Carry and Chery Transcab. Among these 3-small truck, Tata Super Ace shows high injuries

criteria at three parts of driver body which is head, chest and leg. This can be proved by comparing the results of impact to the dummy after the crash had performed (Figure 24, 25, 26).
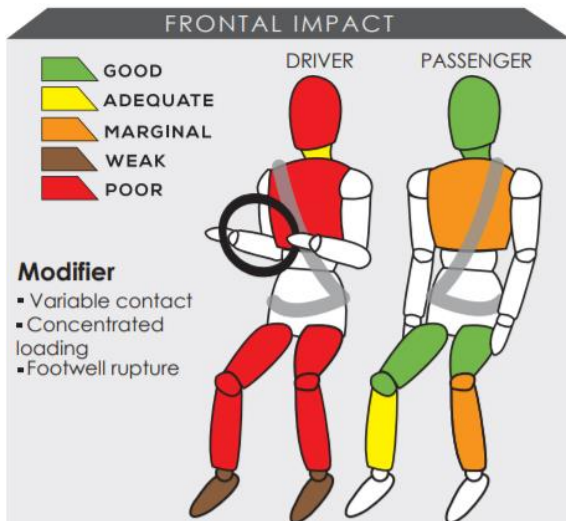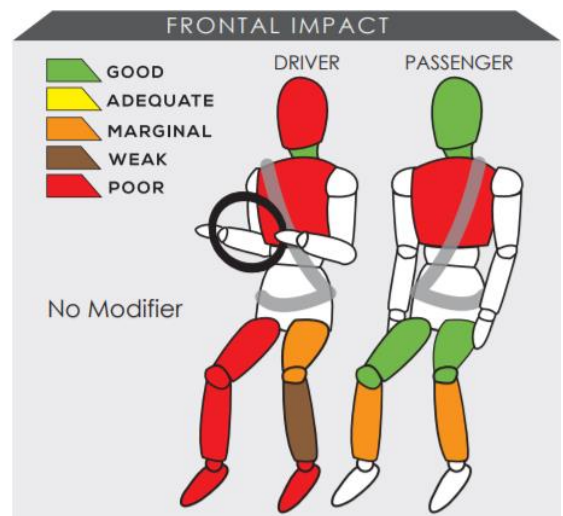

Fig. 24. Tata Super Ace
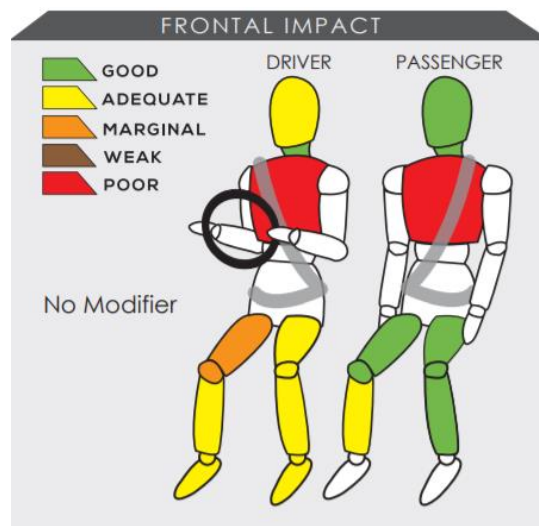

Fig. 25. Cherry Transcab

Fig. 26. Suzuki Carry

8. Conclusion

Tata Super Ace model was chosen because of its high frontal impact injury to the driver as compared to other two vehicles (Cherry Transcab and Suzuki carry). These three models of small trucks also chosen because of it already perform crash test by ASEAN NCAP. So, this study will be validated by using the existing data.

The model that is purchased on a website will undergo modification in terms of design, simplification of unnecessary parts, partition of each body parts, midsurface, meshing and element and quality checks. These modifications are needed to reduce the time of CPU to run and to make sure the simulation will run smoothly and get the most accurate result of the simulation.

As a conclusion, a finite element model has been built to the Tata Super Ace vehicle model to study structure deformation and injuries to the drivers due to impacts. Literature review on the finite element, Tata Super Ace model and method of have been studied and applied.

9. Future Works

The development of finite element model had been done on Tata Super Ace model. So, further study can be done on simplifying the parts of the model. This simplification may reduce the parts of the model that are going to be used for simulation part. It means that only parts that give significant impact to the crash test only will be include. The simplification also gives advantages that can reduce the time for analysis part and reduce error to run the simulation for the analysis.

Next, the parts that have been simplified, will be used for assigning properties such as material properties and boundary conditions. These properties and boundary conditions are important in order to get the best results when the crash simulation had performed.

Since one of the main results that are going to be validate are the injuries of the dummy after the crash had been perform, thus, dummy will be applied to the vehicle. The position of the dummy will be determined in order to get the correct position for the dummy to get the best result for the injuries of the dummy after the crash. Head, chest and leg are the parts of the body that gives significant impact to the dummy for the frontal crash of Tata Super Ace.

References

[1] Abdel-Nasser, Yehia A. "Frontal crash simulation of vehicles against lighting columns using FEM." *Alexandria Engineering Journal* 52, no. 3 (2013): 295-299.
[2] Aseancap. 2017.Dot Dot Holdings Sdn Bhd. https://aseancap.org/v2/ [15 November 2020].
[3] Tu, Jiyuan, Guan-Heng Yeoh, and Chaoqun Liu. "CFD Mesh Generation: A Practical Guideline." *Computational Fluid Dynamics. Elsevier* (2018): 125-154.
[4] Choose, F. Y. O. U. (n.d.). Direct Your Own Life. Direct your own life.
[5] Dean, J. (n.d.). Introduction to the Finite Element Method (FEM) Lecture 1 The Direct Stiffness Method and the Global StiffnessMatrix. Lecture Notes. Retrieved from https://www.ccg.msm.cam.ac.uk/images/FEMOR_Lecture_1.pdf
[6] Do, W., Carry, W. & Meshing, O. 2014. Proceedings of the 2014 International Conference on Artificial Intelligence, ICAI 2014 - WORLDCOMP 2014. Proceedings of the 2014 International Conference on Artificial Intelligence, ICAI 2014 - WORLDCOMP 2014 1–21.
[7] Hickey, Andrew, and Shaoping Xiao. "Finite Element Modeling and Simulation of Car Crash." *International Journal of Modern Studies in Mechanical Engineering (IJMSME)* 3, no. 1 (2017): 1-5.
[8] Informal, A. & Group, I. 2010. N1 Vehicles and ASEP : State of the discussion 04(February): 15–17.
[9] Kirkpatrick, Steven W., J. W. Simons, and T. H. Antoun. "Development and validation of high fidelity vehicle crash simulation models." *International journal of crashworthiness* 4, no. 4 (1999): 395-406.
[10] Sinica, A. P. 2013. 50(5): 2–5.

[11] OKORUGBO, PETER AZOR. "EFFECT OF IMPACT VELOCITY ON THE ENERGY ABSORPTION CHARACTERISTICS OF CRASH BOXES." PhD diss., NEAR EAST UNIVERSITY, 2019.
[12] Soliman, Davide. "Finite element modelling of vehicles and human body models for passive safety." PhD diss., Politecnico di Torino, 2020.
[13] Zheng, Zijian, Pan Lu, and Brenda Lantz. "Commercial truck crash injury severity analysis using gradient boosting data mining model." *Journal of safety research* 65 (2018): 115-124.