Chassis strength is important aspect engineering essay

Chassis strength is important aspect that needs to be considered in the study of a road vehicle design. Chassis or frames are the main structure for road vehicle design. Since the strength of chassis can be effected on stability and safety of vehicle. Because most of load are fully distribute along the chassis. Its will be have some critical point due to section that have more load over the beam section especially driver weight.

This chapter introduce about the analysis of members under axial loading, beams, and frames. Structural members and machine components are generally subject to a push-pull, bending or twisting type of loading.

Beams play significant roles in many engineering application, including building, bridges, automobiles and airplane structures. Beams are commonly subjected to transverse loading, which is a type of loading that creates bending in the beam. The definition of beam is a cross-sectional dimension of structural member that are relatively smaller than its length. A beam subjected to a distributed load is shown in figure 1. 1.

formulation to generate finite element models. However, the axial loading is defined as a linear element for the structure. For example in this section is employed to introduce the basic ideas of one dimensional element and shape function. Steel columns are commonly used to support loads from various floors of structural. The loading from the floors causes vertical displacements of various points along the column.

Nowadays, with the high level of CAD to design, added with a computational technique in simulation and greater understanding in finite element analysis will help engineers to produce a more efficient in chassis building which is have lighten weight but have sufficient strength. This method also can improve of car from any factor such as twisting or actually have some deformation to the chassis.

Analysis of strength chassis of car is the best answer to solve this problem. A ¿½UiTM Shell Eco-Sprint¿½ car which will participate in Shell Eco Marathon Asia 2012 was selected in this study to investigate about their chassis strength and each critical point that have for every chassis section. This project will focus on the Finite Element Analysis (FEA) analysis as a method to analyze the strength characteristic of the car design with different of chassis types. After that, the obtained result will be compared each other to choose which are the best strength chassis.

The competition is split into two classes. The Prototype class focuses on maximum efficiency, while passenger comfort takes a back seat. The Urban Concept class encourages more practical designs. Cars enter one of seven categories to run on conventional petrol and diesel, biofuels, fuel made from natural gas (GTL), hydrogen, solar or electricity.

Over several days, teams make as many attempts as possible to travel the furthest on the equivalent of one litre of fuel. Cars drive a fixed number of laps around the circuit at a set speed. Organisers calculate their energy efficiency and name a winner in each class and for each energy source. The scopes of safety, teamwork, design, and technical innovation will be including in mark for this competition.

The competition inspires the engineers of the future to turn their vision of sustainable mobility into reality, if only for a few days. It also sparks passionate debate about what could one day be possible for cars on the road.

1. 3 Problem statement

When Formula 1 began in early 1950¿½s, most of Europe racing teams used basic space frame chassis(Figure 1. 2), that formed from the comprised of a series beams to be the complete shape of the car and consist of the engine, suspension, driver, and other vehicle sub-system. One of the main advantages of using the space frame design is its easy and logical construction process, of which can be performed by student with intermediate knowledge and experience using basic welding and metal working equipment.

For the UITM Shell Eco-Sprint car, we choose space frame chassis as our type chassis design. The chassis must have the best possible strength to minimize deformation to vehicle and consequently make the chassis more safety from any dangerous. And also prevent from any crack happen to the chassis.

In early stage, the chassis is designed with a lot of weakness in terms of strength of materials. This is because the load applied to the chassis is different from many directions. As example the driver load, tires holder, engine load. The entire factor will effect to the chassis strength and can make some deformation for any critical point.

This project ¿½UiTM Shell Eco-Sprint¿½ will be started with design and modeling the car chassis using CAD software (SOLIDWORKS 2010). This research will focuses on 3D analysis. After the design is completed, it will be go through with Finite Element Analysis (FEA) modelling and mesh optimization. The strain, stress and deformation of the chassis will be investigated and analyzed by using application in Solidwork software.

The strain, stress and deformation of the chassis also will be investigated and analyzed by using application in Abaqus software. Besides that, there have two drawing with different construction. The drawing is modelling by using Solidworks software. First, the beam of chassis constructed with the tube hollow without any modification to the beam surface. For the second, the beam of chassis was constructed with the same tube hollow but has some modification with horizontal holes along the beam structure as in figure 1. 3. This two sample analysis will analyzed by using ABAQUS software and compared which one less deformation or the hole can make much critical strength or can increase the deformation of the chassis. The parametric study for this project is thickness and holes diameter of hollow tube.

1. 3 Objective of research

To ensure the successfulness of this research, the objective must be achieved. The

objective of this research is as follow:

1) To reduce weight and making components more compact.

2) To improve the strength and rigidity characteristics.

3) To obtain the light weight chassis but have sufficient strength.

2. 0 Introduction

Finite Element Analysis (FEA) consists of a computer model of a design or material that is analyzed purposely to get a specific result. It is widely applied in a new product design, and existing product. A company is able to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction [1]. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition [1].

Nowadays, many of industry have analyzed by using 2D modelling and 3D modelling. While 2-D modelling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results [1]. 3-D modelling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively [1]. Inside each of these modelling schemes, the programmer can insert various functions to make the system perform linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation while non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture [1].

2. 1 How does Finite Element Analysis work.

Finite Element Analysis (FEA) is one of the most common examples of engineering analysis and one of most commonly used and powerful feature of the CAD software. It can be applied in structural and FEA usually used in problems where analytical solution not easily obtained. To carry out the analysis of the object by using FEA, the object is dividing into finite number of small elements of shapes like rectangular or triangular.

FEA used as a complex system of points called nodes which make a grid called a mesh [1]. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area [1]. Regions which will receive large amounts of stress usually have a higher node density than those which experience little or no stress. Points of interest may consist of fracture point of previously tested material, fillets, corners, complex detail, and high stress areas [1]. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties to the object, creating many elements [1]. The figure 2. 1 below show the sample of meshing step

2. 2 Advantages of Finite Element Analysis

The Finite Element Analysis is very important for every design. A new design may be modelled to determine its real world behaviour under a multiplicity of load category; hence it can be adjusted earlier to the design of drawings. There have many of advantages by using Finite Element Analysis.

Firstly, it is very important tools for stress and strain analysis because it provides accurate information. Once a detailed CAD model has been developed, FEA can analyze the design in detail, saving time and money by reducing the number of prototypes required [2].

Then, this method can help to modify an existing product which is experiencing a field problem, or is simply being improved, can be analyzed to speed an engineering change and reduce its cost [2]. That is the case of study where some analysis are run in the computer to see how the occurrence of a stress concentrator affects the behaviour of the elements. If the concentration is high, the element can be modified with ease and the subjected to analysis again, and depending on the result, a decision has to be taken to see if it needs more changes.

The combination of the software with other types of software is a very useful tool because the programme of the finite element analysis allows the designer to import models from other CAD software to another FEA software. FEA also can be performed on increasingly affordable computer workstations and personal computers, and professional assistance is available [2].

The Finite Element Analysis (FEA) has been widely implemented by automotive companies and is now used by design engineers as a design tool during the product development process. Some of modern FEA packages consist of detailed components such as thermal, electromagnetic, and structural working environments. In a structural simulation, FEA helps tremendously in producing stiffness and strength visualizations and also in minimizing weight, materials, and costs [3].

FEA allows detailed visualization of where structures bend or twist, and indicates the distribution of stresses and displacements [3]. FEA software provides a wide range of simulation options for controlling the complexity of both modelling and analysis of a system [3]. In the same way, the accuracy level and associated computational time requirements can be managed all together to most engineering applications. FEA allows entire designs to be constructed, refined, and optimized before the design is manufactured [3].

This powerful design tool has significantly improved both the standard of engineering designs and the methodology of the design process in many industrial applications [3]. The introduction of FEA has significantly decreased the time to take products from concept to the production line. It is primarily through improved initial prototype designs using FEA that testing and development have been accelerated. In summary, benefits of FEA include increased accuracy, enhanced design and better insight into critical design parameters, virtual prototyping, fewer hardware prototypes, a faster and less expensive design cycle, increased productivity, and increased revenue [3].

The definition of stress is a force exerted when one body or body part presses on, pulls on, pushes against, or tends to compress or twist another body or body part [4]. A normal stress, s as defined as:

Where dF is a differential normal force acting on a differential area dA. It can summarized that normal stress is s = P/A, where P is the resultant force on area A.

2. 4. 2 Stress Von Misses

The stress von misses known as yield criterion suggests that the yielding of materials begins when the second deviatoric stress invariant reaches a critical value [5]. Simple equations relate the tensile yield stress, shear yield stress and compressive yield stress to a material property. Von Misses stress is determined from the stress state as:

2. 4. 3 Strain

Strain is a measure of intensity of deformation, which is an importance variable in the development of formulas used in the design against deformation failures [6]. The change in structure shape can be described by the displacements of point on the structure. The strain, e as define as:

Where, ? L represents the deformation of the line and Lo the original value.

2. 4. 4 Tensile properties

Tensile properties illustrate the reaction from material to forces that applied in tension. A tensile test is a basic mechanical test where a carefully prepared specimen is loaded in a very controlled manner while measuring the applied load and the elongation of the specimen over some distance. Tensile tests are used to determine the modulus of elasticity, elastic limit, elongation, proportional limit, reduction area, tensile strength, yield point, yield strength and other tensile properties [7].

2. 4. 5 Elastic-plastic behavior

Elastic region is the region of the stress-strain curve in which the material returns to the undeformed state when applied forces are removed. The plastic region is the region in which the material deforms permanently. Yield point is the points separating the elastic from the plastic region [7]. The elastic behavior, plastic behavior and yield point can be described from stress ¿½ strain curve. The stress at yield point is called yield stress. The permanent strain when stresses are zero is called plastic strain. The stress strain curve describing an elastic ¿½ plastic behavior for a ductile material is show in Figure 2. 2.

2. 4. 6 Young¿½s modulus

Young¿½s modulus, E can be defined as the ratio of the uniaxial stress over the uniaxial strain in the range of stress in which Hooke¿½s Law holds [6]. It is used to measure the stiffness of an isotropic elastic material. It also called as the modulus of elasticity, elastic modulus or tensile modulus. It can be represents the gradient of the straight line in a stress-strain curve. Since the calculation of Young's modulus, E is equal to tensile stress dividing by tensile strain:

E = the Young's modulus (modulus of elasticity) (N/m¿½).

F = the force applied to the object, (N).

A0 = the original cross-sectional area through the force applied, (m¿½).

? L = the amount by which the length of the object changes, (m).

L0 = the original length of the object, (m).

2. 5 Material Selection

The alluminium alloy (6063 t5) was selected in chassis shell eco challenge. Because the characteristic of alluminium simply enough to produce high quality chassis. In addition, Aluminium is remarkable for the metal's low density and for its ability to resist corrosion due to the phenomenon of passivation. Structural components made from aluminium and its alloys are vital to the automotive industry and are important in other areas of transportation and structural materials. The most useful compounds of aluminium, at least on a weight basis, are the oxides and sulphates.

¿½Aluminum may be the majority of plentiful steel within the Earth¿½s brown crust area, and also the 3rd the majority of plentiful component, following air as well as silicon. This is the reason 8% through pounds from the Earth¿½s strong area. Because of simple avaibality, higher power in order to pounds percentage, simple machinability, long lasting, ductile as well as malleability aluminum may be the most favored non-ferrous steel within 2005 had been 31. 9 million tonnes¿½ [3].

2. 5. 1 Advantages of Aluminium

Aluminium is very light metal with a specific weight of 2. 7 gm/cm3, about a third that of steel. For example the use of aluminium in vehicles reduces dead- weight and energy consumption while increasing load capacity. Its strength can be adapted to the application required by modifying the composition of its alloys. The application of light weight, strong and long lasting aluminium alloy is shown in figure 2. 3 and 2. 4 [4].

actually creates the protecting oxide layer and is highly corrosion resistant. It is especially helpful for application where protection and conservation are necessary needed. The application of highly corrosion resistance aluminium alloy is shown in figure 1. 3 and 2. 5. [4].

Combining of aluminium and alloy will increased the strength and stiffness properties of aluminium compared to conventional metals and alloys. From the figure below can conclude that material aluminium have middle range of stiffness. In aspect of safety, the stress of chassis by using alluminium alloy in the high level safety. Alluminium alloy also can reduce the cost to build a racing car chassis. So far Alluminium alloy is the best material for the construction of vehicle chassis for Shell Eco-Marathon competition because many of benefits those have in combination of aluminium and alloys. The properties of alluminium alloy shown as in table 2. 6 below.

3. 0 Preparation for Finite Element Analysis (chassis)

Finite Element Analysis (FEA) is designed to help an engineer to understand the physical events that occur on the chassis or beam of the vehicle within designated objects. These events are related to the action and interaction of phenomena such as deformation, distorsion, crack and deflection due to the chassis. Because of the capabilities of the FEA software, FEA software now is very popular among the design engineer to help them in reduces cost because the actual simulation can be done before the design is modelled especially in automotive and aeronautic industry.

In this project, the purpose of FEA simulation was to simulate the chassis car vehicle model similarity to the real situation. In other side, FEA is used to study strength of the chassis vehicle, to obtain the deformation of the beam section and can get the critical point due to load that applied to the chassis. The strain, reaction and element force also can get by doing the FEA method.

The Finite Element Analysis was simulated by using software SOLIDWORKS 2010. In order to create the 3-D drawing for this simulation, SOLIDWORKS 2010 is used to draw the chassis due to complicated shaped and smooth surface. Then continue to make the simulation and analysis towards the completed drawing. To carry out the 3-D analysis, the work was divided into following procedure:

1. Define type of study.

2. Create material defination.

3. Assign material properties.

4. Assign fixed geometry.

5. Apply the external loads.

6. Create mesh for the subject.

7. Process Analysis result.

The sequences of simulation process are shown

3. 1 Create Model of UiTM Shell Eco-Sprint Chassis by Using CAD Software

The selection of the 3-D drawing for ¿½Uitm Shell Eco-Sprint¿½ car software is due to the capabilities and user friendly factor of the software. In the development of the 3-D drawing for ¿½Uitm Shell Eco-sprint¿½ car is done by using SOLIDWORKS 2010 software.

The figures 3. 2, 3. 3 and 3. 4 below show the isometric view, top view and side view of the ¿½Shell Eco-Sprint¿½ chassis.

3. 2 Analysis using SOLIDWORKS 2010 Software

3. 2. 1 Step in the Finite Element Analysis

There are seven important steps in the finite Element Analysis:

i. Define type of study.

This research focused on static study. Whenever a load to be put on the entire body, the body deforms and also the impact associated with lots is actually sent through the entire body. The actual exterior lots stimulate inner causes as well as responses in order to make your body right into a condition associated with balance [9].

The main subjects that calculated in static analysis are stress, strain, displacement and reaction forces from the loads that applied on chassis. The static stady is showing in figure 3. 5.

ii. Create material defination.

Before running the analysis, the material of chassis must be defined. This is very important step because there have much of material optional that can be choosing. Every material has different value of properties depend on analysis type. For example the static analysis stady is required a specific modulus elasticity value. Figure 3. 6 below show how to create material defination

iii. Assign material properties.

The material that selected in this analysis is Aluminium Alloys 6063-T5. There have the specific value for elestic modulus, poissons ratio, density and yield strength that will be use for the analysis. The material properties show in figure 3. 7 below.

iv. Assign fixed geometry.

After the material was selected, the chassis will be set up the fixed point. The fixed point usually located at tyre holder. There have four fixed points to be set up in this analysis. Two points located by the side of front chassis and balance located at the back chassis as in figure 3. 8.

v. Apply the external loads.

The main objective of the analysis is to observe the deformation of chassis. The deformation of chassis is caused by the force that applied toward the chassis. So, there have three external loads that involve in this analysis. The loads are illustrated as in figure 3. 9 and figure 3. 10.

Load Criteria Force(N)

vi. Create mesh for the subject.

Then, the program subdivides the model into small pieces of simple shapes connected at common points. This step called meshing process and the figure 3. 11 below illustrate how to create mesh type.

After all procedure has done, the analysis will run to get the result of analysis. The results can be visualized after the analysis was completed. Those results of analysis consist of displacement, stress, strain, and factor of safety.

3. 3 Analysis using ABAQUS 6. 10 Software

ABAQUS software can be a user-friendly non-linear specific aspect program code as it could manage the particular modeling treatments directly into web template modules. Each and every element includes related tools to perform a certain process. This analysis focused to determine the maximum load and deflection of the beam section due to the load applied. There have 8 modules that must be following in finite element analysis by using Abaqus software:

i. Parts module

ii. Property module

iii. Assembly module

iv. Step module

v. Interaction Module

vi. Load module

vii. Mesh module

viii. Load module

3. 3. 1 Parts module

For the simulation analysis process, the model of beam will be simulated by using ABAQUS software. In order to run the simulation, the model will be imported into Geometry for model preparation. The 3-D model of tube hollow is constructing by using Solidwork2010 and converts to IGES file to be import into Abaqus 6. 10. The figure 3. 12 below show the part that imported will be in shell. The scale of import part will multiply it length by 0. 001 because all dimensions in Abaqus are using in meter. The other parts that will involve in this analysis are two support span and the hammer that will contribute as force movement on the beam. Those parts were defined as discrete rigid geometry in 3-D modeling space in Abaqus workbench.

The material that has chosen on this chassis is aluminium alloy 6063 (T5). The material properties of aluminium alloy for general or mechanical (plastic and elastic) were filled up in this step as in figure 3. 13 below. Those properties that must be filled up are:

I. Density

II. Young¿½s modulus

III. Poisson¿½s ratio

IV. Yield stress

V. Plastic stress

be change to other value as in figure 3. 14 below. Then the assign section will apply into the beam to locate the material properties and the thickness value. In this module, the references point that locates at tools menu must be created to all parts except the beam.

Figure 3. 13: Material manager step.

For the section manager step, the beam parts will set as homogeneous type. This is very important step where the value of shell thickness will ditermine and can

3. 2. 3 Assembly module

All parts that created before will assemble together in this module. The instance parts will create and the position of parts will adjust using coordinates and translation command. There must have surface contact between parts and beam as shown in figure 3. 15 below.

3. 2. 4 Step module

This analysis focused on dynamic and explicit procedure. The time period will set at 0. 03s as in figure 3. 16 below. The incrementation and mass scalling will be in default setting.

There have history output request to be create for this analysis. First output will set up for the whole model and the output variable selection is energy as in figure 3. 17

Then, the history output request for hammer only was set as second output. The

rotations scope as in figure 3. 18 below. The translational will define hammer movement by make x, y and z axis as references.

output variables for the support part. Those outputs are reaction force and moment as in figure 3. 19 below. The direction of the reaction will refer in x, y and z axis.

3. 2. 5 Interaction module

General contact (explicit) was selected to analysis the contact behavior of the beam as in figure 3. 20. Then the contact property will create as a friction. The tangentional behavior and friction formulation with type penalty was selected to characterize the contacts between the beams and hammer during analysis. The friction coefficient was set at 0. 25 as in figure 3. 21 to avoid sliding of the structure.

3. 2. 6 Load module

The boundry condition is using to create a movement for every part. For the hammer, the displacement will set by 0. 07m only in y-direction. The negative sign show the movement will in opposite of axis direction depend on the assembly model. The smooth step was selected for amplitude of movement as in figure 3. 22. Besides, there have no movement for two support and was in fixed condition. So, the point for displacement and rotation remain zero for every direction and angle.

3. 2. 7 Mesh module

Meshing step is the program that subdivides the model into small pieces of simple shapes connected at common points. The sizes of mesh are depending on seed size. For this analysis, the seed size of beam was set to 0. 005m. Next, the element type is explicit and the structure technique was selected as technique of mesh control. Then, the step will go trough by select the instance region to be meshing as in figure 3. 23.

After procedure in Geometry and Meshing was complete, the model now ready to simulate in Setup software in order to solve the appropriate problem and visualization the result. The step involves in Setup software is to define the properties, the physical condition, and the visualization scene.

3. 4 Experimentals Preparation

This subtopic will introduce about the procedure preparation for tensile test and flexural test experiment. These two experiments are related with the analysis that have done by using CAD software. And the result of the experiment is used to make a comparison between the simulation results in Finite Element Analysis.

The tensile test is probably the simplest and most widely used test to characterize the mechanical properties of a material. The setup for the test as described in this tutorial and as performed in the laboratory is based upon standards established by the American Society for Testing and Material (ASTM). The main purpose of tensile test experiment is to get stress and strain of material. From there, the graph of stress against strain will be constructing and can obtain the stress-strain curve. The young¿½s modulus directly defined by calculating the slope of the stress-strain curve. Than the experimental value of the young¿½s modulus can be compare to the analysis value that use in finite element analysis.

3. 4. 1 Material testing (Tensile Test)

For the analysis beam, there will have a comparison between results of physical testing (tensile test) and FEA modelling result of the beam chassis. These steps are made in order to reduce any possible error due to analysis that have constructed. From the Abaqus software there has material input to define young¿½s Modulus. To prove the values of theses material properties as provided by the manufactures would hopefully reduce the error between modelled result and tested result for the chassis performance. So the young¿½s Modulus of alluminium alloy (6063 T5) can be calculated directly by doing tensile test experiment.

The properties that measured from this experiment are ultimate tensile strength, maximum elongation and reduction in area. Beside that, those properties also can calculate possion¿½s ratio and yield strength of the materials. The figure 3. 24 below show the schematic diagram of tensile test.

3. 4. 1. 1 Sample preparation

There have 5 samples that will contribute with tensile test experiment. The entire samples have same dimension and thickness. Because the only way to obtain the percise value is get the average result of the samples. All of the samples that will run trough this experiment are already in dog bone profile as shown in figure 3. 25.

3. 4. 1. 2 Method of Tensile Test Experiment.

Firstly, the speciemens file is created and all the parameters were set-up before start the experiment. The speed that set in this tensile test experiment is 5mm/min. So the speciemens start to install in lower grip and leave upper grip open. Then close the upper grip until the speciemens perfectly grip with the upper and lower clamps as in figure 3. 26. Then the experiment can be started by press ¿½start¿½ button until the test proceed through elastic range until yielding is clearly present on the scope.

The main ojective of flexural testing is to determine parameters such as bend strength, yield strength in bending and elastic modulus. Regarding with the project, the flexural experiment is use to investigate response of metal when subjected to bending. Bending as well as flexure measurement can be widespread throughout along with brittle resources as their multifunction behaviors are generally linear including concretes, stones, woodlands, pockets, cups along with ceramics. Other designs involving brittle resources including powdered ingredients metallurgy highly processed mining harvests along with resources tend to be screened underneath a

new transverse flexure. Bend over examination can be for that reason well suited for assessing energy involving brittle resources wherever model involving tensile examination response to a similar product can be tough on account of smashing involving specimens all-around specimen gripping.

3. 4. 2. 1 Sample prep