

Proton new saga  
torsion beam axle  
modal analysis  
engineering essay



**ASSIGN  
BUSTER**

# CHAPTER 1

## INTRODUCTION

### 1. 1 Overview

Since the invention of automobiles, the quantity of cars on the road around the world is increasing over the years. In Malaysia, 548 115 motor vehicles were registered in the year 2008 compared with only 97 262 registered in the year 1980 [1]. With cars and roads, men can travel from one place to another with ease. Apart from that, it is important to ensure the journey is safe and comfortable. This could be done by the car suspension system. It is playing a major role in keeping the car in control while weakening the unpleasant shock due to irregularities on the roads travelled.

Proton New Saga (also known as Proton BLM) was launched in January 2008. It is an affordable passenger car with a capacity of 5 passengers. Besides, it is equipped with MacPherson struts and a stabilizer bar in the front suspension system and torsion beam axle in the rear suspension system.

The Proton New Saga is well received as evidenced by bookings of more than 120 000 units since its launch [2].

### 1. 2 Problem Statement

The success in Proton New Saga does not stop Proton from willing to improve this car model. Studies on the rear suspension system are essential in order to further enhance the handling and ride comfort of this popular car [3].

Modal analysis is needed, especially, in order to study its vibration behavior. Besides determining the natural frequencies and mode shapes, the outcome

of a modal analysis can also be used to perform more detailed dynamic analyses.

The torsion beam axle found on the rear suspension system of Proton New Saga has an inverted “ V” cross-sectional shape. Other car models with similar rear suspension system such as Perodua Myvi and Toyota Vios have a “ U” cross-sectional shape torsion beam axle. However, the “ open” side of the “ U” cross-sectional shape torsion beam axle on Perodua Myvi is oriented toward the front of the car while the Toyota Vios is oriented toward the rear of the car. Consequently, it is of interest to study the effect of the cross-sectional shape of the torsion beam axle on the vibration behavior.

### **1. 3 Objectives**

The objectives of this thesis are:

To model the Proton New Saga torsion beam axle by using finite element software.

To perform modal analysis on the Proton New Saga torsion beam axle.

To compare the vibration behavior of different cross-sectional shape torsion beam axles.

## **CHAPTER 2**

### **LITERATURE REVIEW**

#### **2. 1 Vehicle Suspension System**

The suspension system of a vehicle is the structure which links the wheel to the vehicle body while allows relative motion between them [4]. It has two

important functions. The primary function is to isolate the vehicle structure whenever practicable from shock loading and vibration due to irregularities of the road surface travelled. Furthermore, the suspension system must at same time, maintains the stability, steering and general handling qualities of the vehicle. The first requirement can be met by using flexible elements and dampers. On the other hand, the second function can be achieved by controlling the relative motion between unsprung masses - wheel-and-axle assemblies - and the sprung mass by the use of mechanical linkage [5].

### **2. 1. 1 Types of Vehicle Suspension System**

The vehicle suspension systems can be generally categorised into dependent and independent types [6]. In a dependent suspension system, two wheels are physically linked by a rigid axle whereby mutual wheel influence exists between the wheels connected by the axle. Solid-axle, torsion-beam axle, De Dion axle are the examples of dependent suspension systems. On the other hand, in an independent suspension system, the vertical motion of one wheel does not affect directly the motion of another wheel. Some of the common independent suspension systems include double wishbone suspensions, MacPherson struts suspension, trailing arm suspension, and multilink suspension.

### **2. 1. 2 Torsion beam Axle Suspension System**

Torsion beam axle suspension system is also called twist-beam suspension. It is often used as the rear suspension in front-engine, front wheel drive (FF) type vehicles since it has less parts and simple configuration. The torsion beam axle suspension system consists of two trailing arms that are welded to a twistable cross-member (often torsion beam) and fixed to body through <https://assignbuster.com/proton-new-saga-torsion-beam-axle-modal-analysis-engineering-essay/>

trailing links with bushing. This twistable beam absorbs all vertical and lateral force moments [7]. It must be stiff enough to support lateral forces during cornering and, at the same time, be flexible enough to allow the right- and left-hand wheels to displace differently when driving over a bump. The relative balance between the stiffness and the flexibility of the constituent members has to be considered during the design stage in spite of the simple configuration [8].

Analyses have to be done on the torsion beam axle suspension system in order to evaluate its performance. It has to be analysed in terms of structural strength, durability, and kinematic and compliance characteristic. Modal analysis can be performed to study the vibration characteristics of the suspension system [3]. On the other hand, the elasto-kinematic analysis can be performed by using multibody method where the torsion beam has to be modelled as a flexible body [9].

Figure 2. 1. A typical torsion beam axle suspension system [8].

This suspension type has many advantages in terms of installation, suspension and kinematics.

From an installation point of view:

the assembly and dismantling are easy;

little space is needed;

springs and shock absorbers can be easily fitted;

few components to be handled .

From a suspension point of view:

there is a favourable wheel to spring damper ratio;

springing is hardly affected since there are only two bearing points;

low unsprung masses;

the cross-member can function as an anti-roll bar [7].

From a kinematic point of view:

toe-in and track width change on reciprocal and parallel springing is negligible;

the change of camber under lateral forces is low;

the load-dependent body roll understeering of the whole axle is low;

good radius-arm axis location whereby tail-lift during braking could be reduced [7].

However, there are also disadvantages for this type of suspension system.

The disadvantages are:

a tendency to lateral force oversteer as a result of control arm deformation;

torsion and shear stress exist in the cross-member;

there is high stress in the weld seams, therefore, the permissible rear axle load is limited;

the limited kinematic and elasto-kinematic opportunities for determining the wheel position;

the establishment of the position of the instantaneous centre by means of the axle kinematics and rigidity of the torsion-beam axle;

there is mutual effect on the wheels;

the difficult decoupling of the vibration and noise from the road surface;

the considerable need for stability of the bodywork in the region of those points on the front bearings where complex, superposed forces have to be transmitted [7].

Owing to the advantages of torsion beam axle suspension system, it is widely adopted in passenger car, especially in the rear suspension of hatchback. Some examples of passenger cars available in Malaysia which use torsion beam axle as rear suspension system are Proton New Saga, Proton Savvy, Proton Exora, Perodua Myvi, Perodua Viva, Toyota Altis, Toyota Vios, Honda City, Nissan Grand Livina, etc.

## **2. 2 Vibrations**

Vibration is any motion that repeats itself after an interval of time. Generally, a vibratory system includes a means for storing potential energy (spring or elasticity), a means for storing kinetic energy (mass or inertia), and a means by which energy is gradually lost (damper). In vibratory systems, potential

energy is transferred to kinetic energy and kinetic energy to potential energy alternately. If damping exists, some energy is dissipated in each cycle of vibration and the vibration will eventually die out [10].

In order to study the vibration of a dynamic system, it is important to know the number of degrees of freedom of the system before proceeding to analysis. The degree of freedom of a vibratory system is defined as the minimum number of independent coordinates required to determine completely the motion of all parts of the system at any instant of time [11].

Vibration can be classified into free or forced vibrations. Free vibration takes place when a system oscillates under the action of forces inherent in the system itself, and when external impressed forces are absent. The system under free vibration will vibrate at one or more of its natural frequencies. The natural frequencies are properties of a dynamical system established by its mass and stiffness distribution [12].

Forced vibration is the vibration that takes place under excitation of external forces. A system will be forced to vibrate at the excitation frequency if the excitation is oscillatory. If the frequency of excitation coincides with one of the system's natural frequencies, resonance occurs. When resonance occurs, dangerously large oscillations may result and cause failure to the system. Therefore, the calculation of the natural frequencies is of vital importance in the study of vibrations [12].

A vibration is known as undamped vibration if no energy is lost or dissipated in friction or other resistance during oscillation. On the other hand, if any

energy is lost in this way, it is called damped vibration. In many physical  
<https://assignbuster.com/proton-new-saga-torsion-beam-axle-modal-analysis-engineering-essay/>



systems, the amount of damping is so small that can be neglected for most engineering purposes. However, consideration of damping becomes extremely important in analyzing vibratory systems near resonance [10].

Vibrations can also be categorised as linear vibration or nonlinear vibration. If all the basic components of vibratory system (the springs, mass, and damper) behave linearly, the resulting vibration is known as linear vibration. However, if any of the basic components behave nonlinearly, the vibration is called nonlinear vibration. All vibratory systems tend to behave nonlinearly where the principle of superposition is not valid [10].

## **2.3 Finite Element Method**

The finite element method (FEM), sometimes called finite element analysis (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. A boundary value problem, also called field problem, is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. The field is the domain of interest and most often represents a physical structure. The field variables are the dependent variables of interest governed by the differential equation. They may include physical displacement, temperature, heat flux, fluid velocity and so on. On the other hand, the boundary conditions are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field [13].

## 2. 3. 1 How Does Finite Element Method Work?

Figure 2. 1 shows a volume of some materials with known physical properties. The volume represents the domain of a boundary value problem to be solved. For simplicity, it is assumed to be a two-dimensional case with a single field variable  $\ddot{\Gamma}(x, y)$  to be determined at every point  $P(x, y)$  such that a known governing equation is satisfied exactly at every such point. This implies an exact mathematical solution is obtained. The solution is a closed-form algebraic expression of the independent variables. However, the domain may be geometrically complex in practical problems. Thus, approximate solutions based on numerical techniques and digital computation are most often obtained in engineering analyses of complex problems [13].

Figure 2. 2. (a) A general two-dimensional domain of field variable  $\ddot{\Gamma}(x, y)$ . (b) A three-node finite element defined in the domain. (c) Additional elements showing a partial finite element mesh of the domain [13].

Figure 2. 2(b) shows a small triangular element that encloses a finite-sized subdomain of the area of interest. This element is not a differential element of size  $dx \ddot{\Gamma} dy$  makes this a finite element. The vertices of the triangular element are nodes. A node is a specific point in the finite element at which the value of the field variable is to be computed, explicitly. Nodes located on the boundaries of the finite element are called exterior nodes. They may be used to connect an element to adjacent finite element. Interior nodes, meanwhile, do not lie on the finite element boundaries and cannot be connected to any other finite element. The number of degrees of freedom

associated with a finite element is equal to the product of the number of  
<https://assignbuster.com/proton-new-saga-torsion-beam-axle-modal-analysis-engineering-essay/>

nodes and the number of values of the field variable (and possibly its derivatives) that must be computed at each node [13].

In finite element method, the values of the field variable computed at the nodes are used to approximate the values at nonnodal points by interpolation of the nodal values. This is the crux of finite element method. For the three-node triangle example, the nodes are all exterior. At any other point within the element, the field variable is described by the approximate relation

$$\dots\dots\dots(2. 1)$$

where  $\bar{I} \cdot 1$ ,  $\bar{I} \cdot 2$ , and  $\bar{I} \cdot 3$  are the values of the field variable at the nodes while  $N_1$ ,  $N_2$ , and  $N_3$  are the interpolation functions. The interpolation functions are predetermined, known functions of the independent variables. These functions describe the variation of the field variable within the finite element [13].

The finite element equations are formulated such that, at the nodal connections, the value of the field variable at any connection is identical for each element connected to the node. As a result, continuity of the field variable at the nodes and across interelement boundaries is ensured [13].

### **2. 3. 2 General Procedure for Finite Element Analysis**

There are certain common steps in formulating a finite element analysis of a physical problem, whether structural, heat transfer, fluid flow, or some other problem. These steps are usually embodied in commercial finite element software packages. The steps are described as follows [13].

## Preprocessing

The preprocessing steps is described generally as defining the model and includes

Define the geometric domain of the problem.

Define the element type to be used.

Define the material properties of the elements.

Define the geometric properties of the elements (length, area and the like).

Define the element connectivities (mesh the model).

Define the physical constraints (boundary conditions).

Define the loadings [13].

The preprocessing (model definition) step is extremely important. A perfectly computed finite element solution is of absolutely useless if it corresponds to the wrong problem [13].

## Solution

In this step, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary field variables. Then, the computed values are used by back substitution to perform computation on the additional, derived variables, such as reaction forces, element stresses, heat flow, etc [13].

However, it is very common for a finite element model to be represented by number of equations. Special solution techniques are used to reduce data storage requirements and computation time. For example, in static, linear problems, a wave front solver, based on Gauss elimination, is commonly used [13].

### Postprocessing

In postprocessing step, the analysis and evaluation of the solution results are performed. Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution. The operations that can be accomplished by the postprocessor software include

Sort element stresses in order of magnitude.

Check equilibrium.

Calculate factors of safety.

Plot deformed structural shape.

Animate dynamic model behavior.

Produce color-coded temperature plots [13].

Although solution data can be manipulated many ways in postprocessing, the most important objective is to apply sound engineering judgement in determining whether the solution results are physically reasonable [13].

### **2. 3. 3 Meshing**

Meshing is the process of representing a physical domain with finite elements. It results set of elements, known as the finite element mesh. It is generally impossible to include the entire physical domain in the element mesh if the domain includes curved boundaries. However, by using smaller and more numerous elements, more of the physical domain can be included and the curved boundaries are more closely approximated. As the interpolation functions satisfy certain mathematical requirements, a finite element solution for a particular problem converges to the exact solution of the problem. This can be done by increasing the number of elements while decreasing the physical dimensions of the elements [13].

Figure 2. 3. (a) Arbitrary curved-boundary domain modelled using square elements. Stippled areas are not included in the model. A total of 41 elements is shown. (b) Refined finite element mesh showing reduction of the area not included in the model. A total of 192 elements is shown. [13]

## **CHAPTER 3**

### **METHODOLOGY**

#### **3. 1 Introduction**

The main concern of this project is to compare the vibration behavior of different cross-sectional shape torsion beam axles. Modal analysis has to be performed to determine the natural frequencies and mode shapes of the torsion beam axles. Due to the large size of the specimen (torsion beam axle) and lack of appropriate equipments, experimental method cannot be used. However, with the available of finite element software package, such

as Abaqus Unified FEA and ANSYS Multiphysics, finite element analysis can be used to obtain approximate results.

### **3. 2 Materials and Equipments**

In this project, the Abaqus Unified FEA software was used due to its strong ability to solve nonlinear problems. The torsion beam axle is suitable to be modelled by using Abaqus Unified FEA since it behaves nonlinearly. Besides, the software can be used as preprocessor, solver and also the postprocessor. Certainly, modal analysis can also be performed by using it.

Besides, high performance computer was used to perform the finite element analysis as the analysis is a heavy task. The higher performance computer used will result faster analysis.

### **3. 3 Procedures**

First, the dimension of the torsion beam axle of a Proton New Saga was requested from Proton. The geometry was then modelled. Mesh generation was performed after geometry modelling. After that, modal analysis was performed to determine the natural frequencies and the corresponding mode shapes. Only the first ten natural frequencies and mode shapes will be taken. The geometry modelling, meshing and modal analysis were performed by using Abaqus Unified FEA. The same procedures were repeated for “ U’ cross-sectional shape torsion beam axle. Results for the two different cross-sectional shape torsion beam axles were then compared.