International Journal of Innovative Computing, Information and Control
Volume **13**, Number **2**, April **2017**
ICIC International ©2017 ISSN 1349-4198
pp. 659–670

DEVELOPMENT OF A NEW PART OF CASING CAP FOR THE PARKING BRAKE CABLE USING FINITE ELEMENT ANALYSIS

Siwawong Buthgate[1], Anakkapon Saenthon[1] and Somyot Kaitwanidvilai[2]

[1]College of Advanced Manufacturing Innovation
[2]Faculty of Engineering
King Mongkut's Institute of Technology Ladkrabang
Chalongkrung Road, Ladkrabang, Bangkok 10520, Thailand
drsomyotk@gmail.com

Received October 2016; revised January 2017

Abstract. *The brake system is an important safety system in every vehicle. This system is utilized for controlling of the vehicle movement. This paper proposes a new design and development of casing cap for the parking brake cable by applying the finite element technique to analyzing and evaluating the feasibility of the material and the design of the product. The study and analysis of influential factors affecting the parts of casing cap by JASO standard F903-75 control cable for automobiles are described. The results show that the plastic PA GF15 can be used to replace the conventional material that is steel casing cap, and will then be able to reduce the overall cost material by 2.5%. In addition, the results of the analysis illustrate that all features of the new product are complied with JASO F903-75, the control cable standard for vehicles.*

Keywords: Casing cap of the parking brake cable, Finite element method, JASO F903-75 control cable for automobiles, Plastic PA GF15

1. **Introduction.** Presently, the automotive industry is focusing on the trend of research and development in various research areas including the material engineering to improve both the productivity and cost. One of the most important parts in the vehicle is the brake system which is the safety system for all types of motor vehicles, e.g., cars and motorcycles. The parking brake is one of the important parts in the brake system; this part acts to make the car unmovable by pulling the lever on the hand brake up. When the users pull the lever of the hand brake, the exerted force will be transferred from the lever into the drum brake via the parking brake cable to the brake device reaching the enough friction which the wheel cannot move and the car is exactly stopped [1]. In [2], the lightweight design of the car using thermoplastic is an efficient method to reduce the CO_2 emission and fuel. The using of plastic presents an important role in the green concept for the design process by replacing the steel with the plastic material. The weight of the plastic car fender is 41% compared with the steel fender. Moreover, the production cost of plastic fender is cheaper than that of the steel [2]. In [3], the conventional metal tanks of the public buses were replaced by the plastic fuel tanks where the design and development of plastic fuel tanks was based on the static and dynamic analysis using finite element analysis. The samples were tested to meet the required properties of material with load test as the applied static and dynamic requirement. Results of static and dynamic analyses were performed using finite element method. The aim of the research work was to present the analytic methodology to prevent the damages [3]. In [4], Tube Hydro-Forming (THF) was known in the automotive and aerospace industry, where there is the more complex geometry of the tube and extrusions. The benefit of THF, for instance, is the using of the high strength material as the structure of automotive components which

results in lower weight and improved performance. The simulation model was analyzed by the FEM tool deform-3D [4]. In [5], the measurement of the temporal and true strain of polymer material was applied to the simulation of the plastic parts. The objective of the simulation was to simulate the behavior of the plastic strain rate which depended on the polymer. The results of the new approach based on time and relation methods were completely performed using FEA based optimization procedures which are similar to the inverse analysis. Thus, it only used the measured data in the mechanical testing to generate the true stress/strain-curves at different constant strain rates [5]. In [6], the polymer composites were shown that they had many efforts to explain the mechanical properties of the regions at the nanoscale. Thus, the increasing of resolution in nano-indentation and atomic force microscopy techniques was achieved; however, it is not always possible to identify thin inter-phases when the dimensions are similar to those in the indenter surface contact area. The results of demonstration by both experimentally and through finite element analysis were described, a method of quickly identifying indentations that are restricted by the reinforcement [6]. In [7], the taper roller bearings were applied to reducing the important gears. The influence was analyzed by finite element method in this work. Firstly, a finite element model of roller bearings was established by using the Reynolds equation and considering the surface roughness. Then, the stress fields in the roller and the raceway were calculated by adding load and solving, obtaining the maximum stress and strain of the bearings. Finally, the effects of the work property of the bearings were analyzed by the obtained maximum stress and strain. The results of this research work from the finite element analysis not only gave the maximum and a minimum shear stress in rolling bearings, but also showed the positions where the stress concentration is high [7]. In this paper, the static analysis of linear-elastic structures with uncertain parameters subjected to the deterministic loads was addressed. The uncertain structural properties were modeled as interval variables with assigned lower and upper bounds. The key idea of the novel method is to associate an extra unit interval to each uncertain parameter in order to keep physical properties linked to the finite elements in both the assembly and solution phases. This allows one to reduce overestimation and performs the standard assembly of the interval elementary matrices. In [8], a novel interval finite element method (IFEM) for the static analysis of linear structures with uncertain parameters has been presented. The key idea of the method is to model the uncertain parameters as interval variables handled by means of the improved interval analysis via extra unitary interval (IIA via EUI), recently introduced in the literature to reduce the overestimation affecting the classical interval analysis [8]. Coupling two different computational approaches, namely the finite element method (FEM) and meshless finite difference method (MFDM), in one domain was applied in solving thermomechanical initial-boundary value problem where the heat transport in the domain is non-stationary. In this method, the domain is divided into two subdomains for FEM and MFDM, respectively. Contrary to the other coupling techniques, the approach presented in [8] is defined in terms of mathematical problem formulation rather than at the approximation level. The scale parameter, depends neither on the type of the considered problem, nor on the applied approximation schemes. It depends on the discretization density only, and may be determined without any difficulties. It should be noted at this point that this scalar parameter is a common parameter for the mechanical and thermal parts and appears in the mixed problem formulation [9,10] including the movement of heat through the material shapes. As seen in many research works, the using of the finite element method (FEM) is a powerful tool for the analysis of material properties. The results of the hot air flow using FEM and conventional method were compared and applied to various fields [11,12]. Peri-dynamics is a new nonlocal theory that provides the ability to represent displacement discontinuities in a continuum body without explicitly modeling the crack surface. An explicit dynamics implementation of the bond-based peri-dynamics formulation is presented to simulate the dynamic fracture process in 3D elastic solid. The discontinuous Galerkin (DG) approach [13] was utilized to formulate the classical peri-dynamics governing equation. As the result, the spatial integration can be carried out through finite element approach to enforce the boundary conditions, constraints, contacts as well as to handle the non-uniform mesh in the engineering practices. The conclusion of the explicit dynamic formulas and numerical algorithms of bond-based peridynamics model is to predict the damage of 3D brittle material. In contrast to the mesh free version of peri-dynamics, the discontinuous Galerkin weak form of the peri-dynamics governing equations was considered [13]. Moreover, several papers suggested the parameter selection in FEM [14,15] and modified technique in FEM [16]. As seen in all documents reviewed, the FEM is a powerful technique to analyze the material properties.

The highlight of this research differed from the other articles is the development of a new parking brake cable which replaces the traditional steel with the plastic, which is lighter and lower cost. This part is an innovative part of the security of control cable according to the Japanese Standard, JASO standard F903-75 [1]. The development provides maximum efficiency with satisfactory specification; and the stress of the material must not exceed the yield point of the materials used in the design. The location and the stress can be calculated from theoretical mechanics; however, the mechanical calculation theory has several limitations in the calculation of the objects with complex shapes. In some cases, it is not possible to calculate by the analytical method. Thus, bringing the finite element analysis (FEA) to calculate the deformation behavior of the designed part is necessary.

This will reassure customers in the quality of the product and lower the cost in terms of lower weight, and lower cost of material and still retains the satisfactory mechanical properties. This solves the problem of production cost in material and leads to the future car components.

The parking brake cable is the important part in the brake system and this research work attempts to improve the casing cap for this cable. Figure 1 shows the diagram and photos of the main components in the parking brake cable, which are:

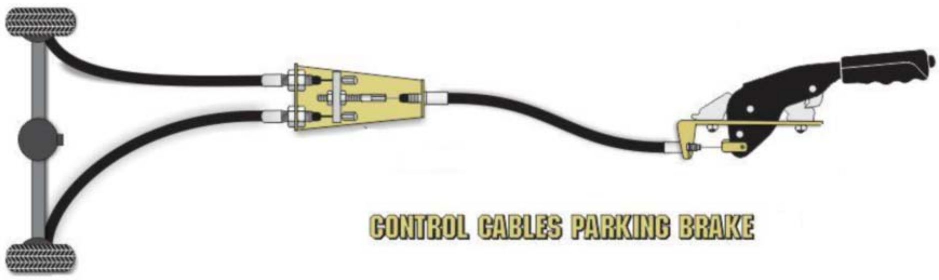
- The parking brake cable, the front end point is connected to the hand brake or lever and the other endpoint is connected to the left and right drum brakes.

- Fitting point is the point of connecting which composes of the casing cap assembling with a plate of the car body and the components are locked by the metal clip.

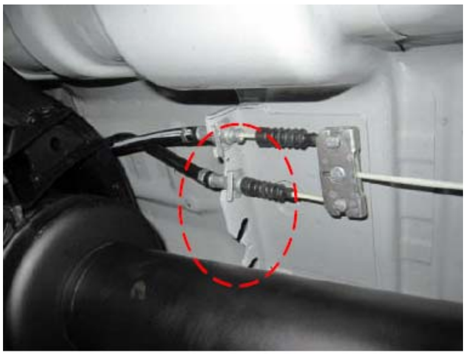
- Casing cap is a part of the fitting point which is the interesting part in our research study. This research focuses on the using of the plastic composite, glass fiber instead of steel in the casing cap.

As shown in Table 1 and Figure 2, there are 9 component parts of the parking brake cables.

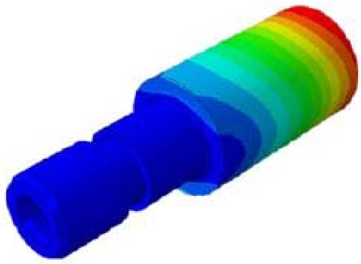
This paper proposes the study of the material analysis for the replacement of casing cap of the parking brake cable. The analysis of the proposed material is based on the finite element analysis and the results of mechanical properties of the new casing cap were investigated in comparison with the conventional material, steel casing cap. In addition, various types of material are studied to show the effectiveness of the proposed material for reducing the production cost and achieving the effectiveness of the proposed cable design. This paper is organized as follows. Section 2 presents the material and FEM analysis. The basic design of casing cap and its material are illustrated in Section 3. Simulation results and discussion are described in this section. Finally, Section 4 summarizes the research work and paper.



(a) The parking brake cable



(b) Assembly point



(c) Casing cap

FIGURE 1. The parking brake cable

TABLE 1. Details of the component on the parking brake cable

No	Components	Function
1	Inner cable	Loading capacity and passing it driven
2	Outer casing	Reflecting the strong support for protecting and maintaining the inner cable routing
3	Inner ends	Supporting the loading and connecting the related parts
4	Boot	Protecting dust and water
5	Casing caps	Reflecting the strong support and connecting related parts
6	Protector	Protecting the outer casing
7	Clamp	Keeping the routing of cables
8	Casing caps	Reflecting the strong support and connecting related parts
9	Inner ends	Supporting the loading and connecting related parts

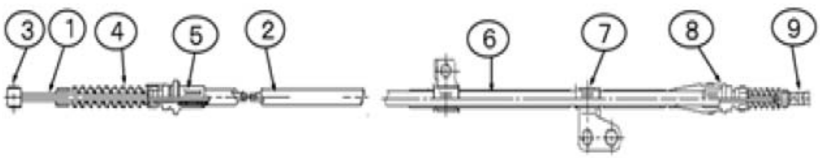


FIGURE 2. Parts on the parking brake cable

2. **Material and Finite Element Analysis.** At present, the casting cap is made of several materials; those are SS400 steel, ADC6 aluminum, and ZDC2 zinc which cause heavy weight of car body and high material cost. One of the most interesting materials

attempted to be studied in this research work is PA GF15 plastic which is light weight and low cost material.

2.1. **Material properties.** The casing cap is currently made of several different types of materials, which are shown in Table 2.

TABLE 2. Details of the material

Materials	Elastic modulus (GPA)	Poisson's ratio	Yield strength (MPa)
Steel – SS400	207	0.3	230
Aluminum – ADC6	71	0.33	159
Zinc – ZDC2	85.5	0.27	221

This paper focuses on the using of the following material instead of the original material mentioned above. Plastic: PA GF15, elastic 5.8 GPA, Poisson's ratio 0.42 and yield strength 200 MPa [17].

Generally, the important material properties, e.g., elastic limit, yield point, yield strength of the material can be specified by the stress-strain curve as shown in Figure 3. Stress is the force per unit area and is usually expressed in the unit of newton per square meter. Table 3 shows the examples of simple stresses and their definitions. When applying force to the material, the length or dimension of the material is normally changed. The ratio of the change in length of the material when applying force to the original length is called as strain. As seen in Figure 3, the characteristic curves of steel (1), plastic (2) and other material (3) are illustrated. Clearly, the characteristic curves of steel and plastic are different; however, the mechanical properties in terms of Poisson's ratio and yield strength are similar.

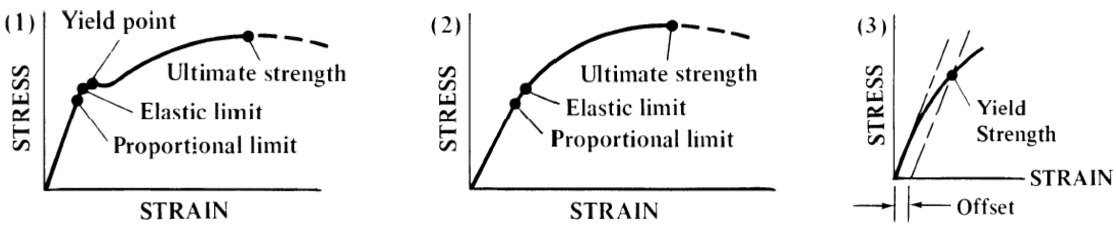
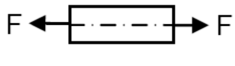
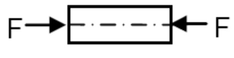
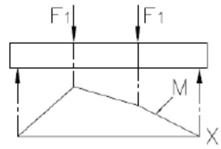
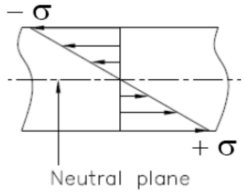


FIGURE 3. Stress-strain curves: (1) steel, (2) plastic, (3) other material

TABLE 3. The examples of stress and their definitions

Case	Type of Loading	Illustration	Stress Distribution	Stress Equations
1	Direct tension		Uniform	$\sigma = \frac{F}{A}$
2	Direct compression		Uniform	$\sigma = -\frac{F}{A}$
3	Bending	 Bending moment diagram		$\sigma = \pm M/Z$ $= \pm My/I$

2.2. **Finite element and boundary conditions.** To analyze the mechanical properties and the specification of the designed parts, especially in the complex shape, the finite element method based on the boundary and partial differential equations on the balance solids with the flexibility in three dimensions shown in (1) is utilized.

$$\begin{aligned} \frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + F_x = 0 \\ \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} + F_y = 0 \\ \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_z}{\partial z} + F_z = 0 \end{aligned} \tag{1}$$

σ_x, σ_y, σ_z are axial stress on the x, y, and z surfaces, respectively. τ_{xy}, τ_{xz}, τ_{yz} are the shear stress on the x, y, and z surfaces, respectively. F_x, F_y, F_z are the forces (body force) on the x, y, and z directions, respectively. The outer surface of the solid, as shown in Figure 3, contains various boundary conditions such as the set moves on the skin, subjected to the conditions of the stress on the surface (surface traction):

$$\overline{T} = T_x\hat{\imath} + T_y\hat{\jmath} + T_z\hat{k} \tag{2}$$

where T_x, T_y, T_z are the axial stress on the x, y, and z surfaces respectively. Generally, Equation (2) can be written in the general form of the stress subsidiaries as follows:

$$\begin{pmatrix} T_x \\ T_y \\ T_z \end{pmatrix} = \begin{pmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_y & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{pmatrix} \begin{pmatrix} n_x \\ n_y \\ n_z \end{pmatrix} \tag{3}$$

where n_x, n_y, n_z are the directional cosine vectors on the x, y, and z surfaces, respectively.

$$\hat{n} = n_x\hat{\imath} + n_y\hat{\jmath} + n_z\hat{k} \tag{4}$$

To illustrate the relationship on the boundary condition, Figure 4 shows the normal vectors and the forces exerted on the surfaces at the considering point.

In addition to these conditions, the extension of the conditions when the dimension of the object is changed, can be analyzed by considering the stress and strain. In the three-dimensional solid, the initial stress (prestrain) normally is applied first and then the relationship between stress and strain is typical as:

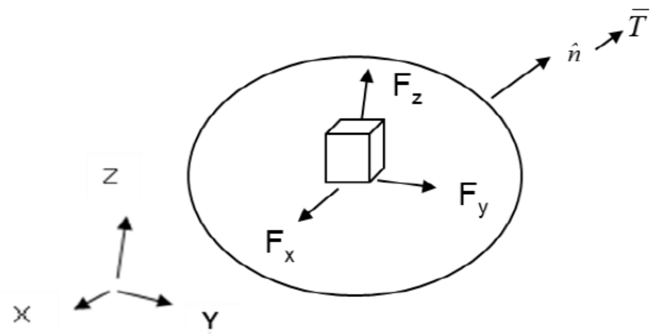


FIGURE 4. The balance on any position in the three-dimensional solid

so

$$\begin{aligned} \{\sigma\} &= [c]\{\varepsilon - \varepsilon_0\} \\ \{\sigma\}^T &= [\ \sigma_x \quad \sigma_y \quad \sigma_z \quad \tau_{xy} \quad \tau_{yz} \quad \tau_{xz}\ ] \\ \{\varepsilon\}^T &= [\ \varepsilon_x \quad \varepsilon_y \quad \varepsilon_z \quad \gamma_{xy} \quad \gamma_{yz} \quad \gamma_{xz}\ ] \end{aligned} \tag{5}$$

$$[C] = \frac{E}{(1+v)(1-2v)} \begin{pmatrix} 1-v & v & v & 0 & 0 & 0 \\ v & 1-v & v & 0 & 0 & 0 \\ v & v & 1-v & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{(1-2v)}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{(1-2v)}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{(1-2v)}{2} \end{pmatrix}$$

where v is the Poisson's ratio, and E is the Elastic modulus.

The most important tool to analyze the above mentioned boundary conditions is the finite element analysis in which the solid object needs to be firstly constructed. The complex shape object, in this case is the casing cap, which was designed and developed for the parking brake cable, is created by the CAD using CATIA software and is analyzed by the FEM software, Nastran. Firstly, the mesh on the designed CAD file is constructed by assigning the meshing value. The shape of casing cap model is similar to the cylindrical shape with several dimensions and diameters; the curve shape is designed to withstand the echo tension or force that has normally occurred in the brake system. The output of the analysis is the maximum pull load force exerted on the designed casing caps with different materials. The force must be within the value illustrated in the standard [1]. Figure 5 shows the example of mesh applied to a simple solid object. There are 4 materials needed to be analyzed in this research study [18].

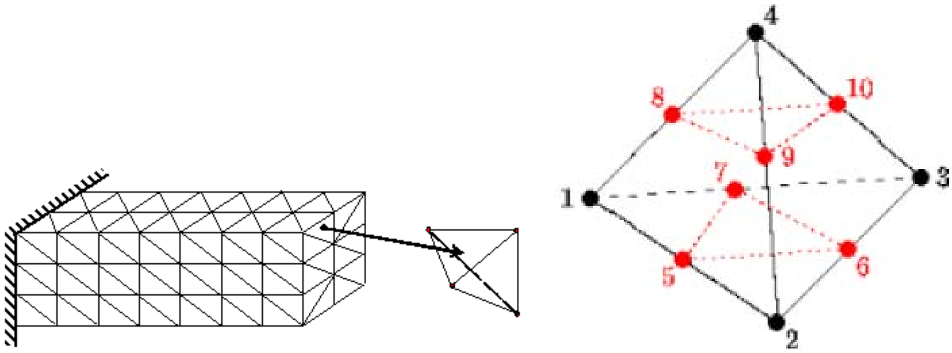


FIGURE 5. The mesh on the solid object and points for considering the boundary conditions with 10 node tetrahedron

3. **Casing Cap Design and Results.** In the design, the parameters of analysis in finite element analysis tool are shown in Tables 4 and 5 which are the properties and meshing conditions used in the finite element program: MSC Nastran software. All parameters of materials used in the simulation are from the material specification (material data sheet) and the conditions of FEM parameters are selected properly.

The materials adopted in this simulation study are shown in Table 6.

Objective: The specification of the designed parts, casing cap, must be complied with the maximum pull load force described in the standard JASO. As declared in the JASO F903-75, the standard of cable for automotive industry, the minimum value of the pull load for casing cap is 18 kgf (Refer to the Jason standard table 17 casing cap type D2).

By using the CAD files and assigned material properties, the analysis of the designed casing cap can be achieved. Figure 6 shows the FEM analysis on the 4 different casing caps.

As seen in Figure 7, the similar strain values, safety factor and maximum pull of the load can be analyzed. According to the specification in this industry, the safety factor

Table 4. Properties condition

No	Parameters	Meaning
1	Action: Create	Create the properties
2	Object: Isotropic	Select the 3-axis mechanical properties by Isotropic
3	Method: Manual	Select material method by Manual
4	Constitutive model: Linear elastic	Select analysis model by linear elastic
5	Elastic modulus	Input elastic modulus of material
6	Poisson ratio	Input Poisson ratio of material

Table 5. Meshing condition

No	Parameters	Meaning
1	Action: Create	Create meshing
2	Object: Mesh	Select mesh for analyzing force of area sample part
3	Type: Solid	Select type of sample part of solid
4	Element shape: Tet	Select shape of element by Tetrahedron
5	Mesher: Tetmesh	Select mesh type by Tetmesh
6	Topology: Tet 10	Select topology node 10 points by tetra 10
7	Value: 0.8	Input element size

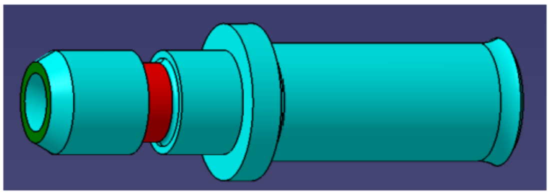
Table 6. Mechanical and material properties applied to the simulation

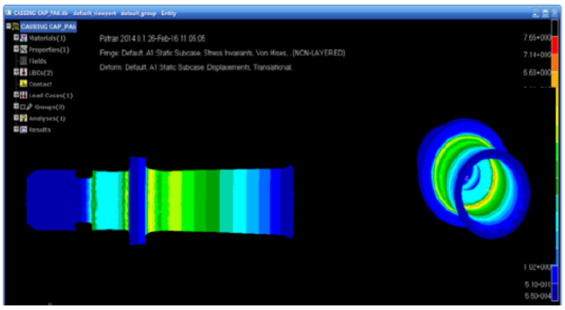
Material/Mechanical properties	PA GF15	SS400	ADC6	ZDC2
Elastic	5.8 GPA	207 GPA	71 GPA	85.5 GPA
Poisson's ratio	0.42	0.3	0.33	0.27
Yield strength	200 MPa	230 MPa	159 MPa	221 MPa

Table 7. Results of the stress on each material with different pull of load

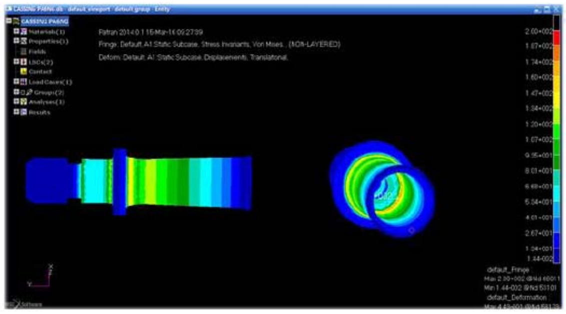
	Material	Pull of load				
		18 kgf	50 kgf	100 kgf	150 kgf	Maximum
Stress max (MPa)	PA GF15	7.65	21.20	42.50	63.70	200
	Steel	8.22	22.80	45.60	68.50	230
	Al	8.07	22.40	44.80	67.30	159
	Zn	8.36	23.20	46.40	69.70	221

is defined as the ratio of yield strength and the stress of the analyzed object. However, according to the JASO F903-75 standard, the minimum of the pull of the load for casing cap is 18 kgf min. As seen in the following results, the pull of load of each material has complied with the standard, and the high values of the safety factors are achieved. The FEA result shows that the maximum stress is 200 MPa and is not more than the value of the material yield point of PA6. Thus, according to the maximum stress, the gain traction of 471 kgf without incident can be determined. Next, the simulation result of the steel SS400 shows the maximum stress of 230 MPa which is not more than the value of the material steel yield. The maximum load at 505 kgf can be determined. The maximum stresses of Aluminum die-casting ADC6 and Zinc die-casting ZDC2 are 159 and 221 MPa. The maximum load of ADC6 and ZDC6 of 505 and 475 kgf can be determined. The following are the simulation results of all materials.

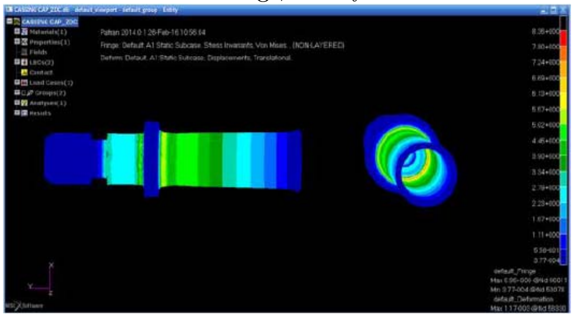




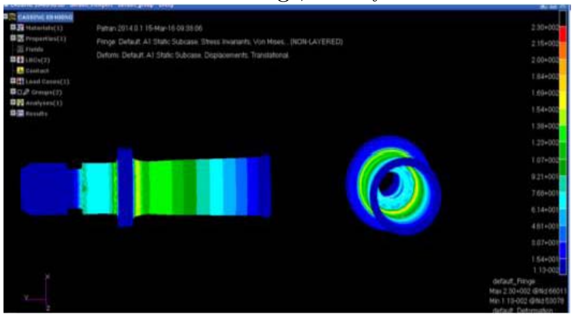
Material: PA GF15 %
Pull of load: 18 kgf, Safety factor: 26.14



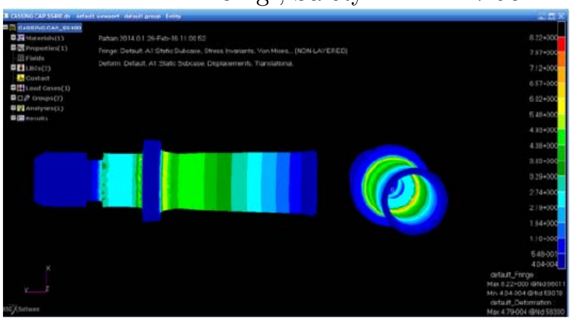
Material: PA GF15 %
Pull of load: 471 kgf, Safety factor: 1.0



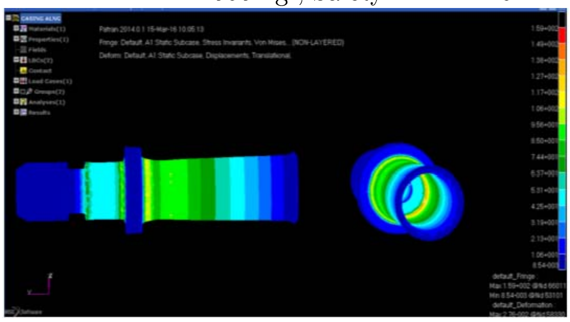
Material: steel SS400
Pull of load: 18 kgf, Safety factor: 27.98



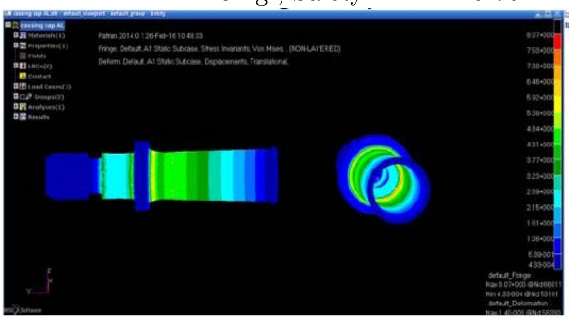
Material: steel SS400
Pull of load: 505 kgf, Safety factor: 1.0



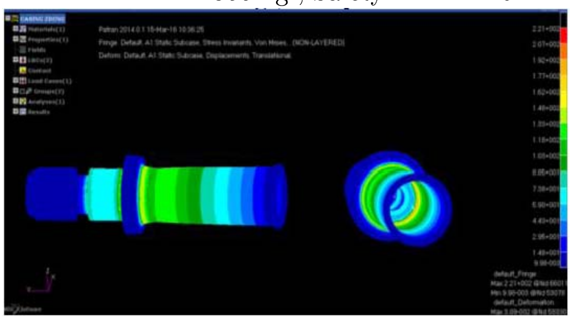
Material: Aluminium ADC6
Pull of load: 18 kgf, Safety factor: 19.70



Material: Aluminium ADC6
Pull of load: 355 kgf, Safety factor: 1.0



Material: Zinc ZDC2
Pull of load: 18 kgf, Safety factor: 26.43



Material: Zinc ZDC2
Pull of load: 475 kgf, Safety factor: 1.0

FIGURE 6. Finite element analysis on the 4 different casing caps

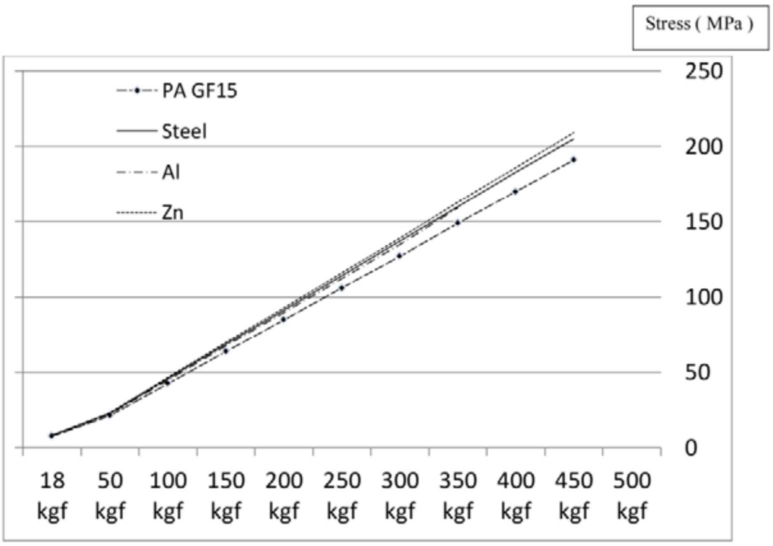


Figure 7. Estimated curves of stress of each casing cap

- Nylon6 Glass fiber 15

FEA analyze:

- Safety factor = yield strength 200 MPa/stress 7.65 MPa = 26.14
- Maximum pull of load 471 kgf

- Steel SS400

- Safety factor = yield strength 230 MPa/stress 8.22 MPa = 27.98
- Maximum pull of load 505 kgf

- Aluminum die-casting ADC6

- Safety factor = yield strength 159 MPa/stress 8.07 MPa = 19.70
- Maximum pull of load 355 kgf

- Zinc die-casting ZDC2

- Safety factor = yield strength 221 MPa/stress 8.36 MPa = 26.14
- Maximum pull of load 475 kgf

As the results indicated, the stress and maximum pull of load of each material analyzed from the finite element were found as the following.

Maximum pull of load of the
- Steel with the stress of 230 MPa load: 505 kgf;
- Zinc with the stress of 221 MPa load: 475 kgf;
- Plastic with the stress of 200 MPa load: 471 kgf;
- Aluminum with the stress of 159 MPa load: 355 kgf.

The important factors for the design engineer to decide the using of the plastic material instead of the conventional materials are the following.

a. The maximum pull of load,
b. Safety factor,
c. The yield stress of casing cap.

In the engineering viewpoints and the results from FEM, the steel is the best material for casing cap because of the maximum pull of the load and safety factor; however, by considering the weight and cost, the composite plastic is an attractive material for casing cap due to its lower weight and cost as compared to the conventional steel, while the maximum pull of load and safety factor are not much different. The weight of the plastic is approximately 21% of the weight of steel and the cost is about 40% of the cost of steel. Thus, this factor affects the decision of applying the composite plastic material for the casing cap.

4. **Conclusion.** From the results and analysis, this research work can be summarized below.

The results of the FEA model showed that the steel is the best material for casing cap to withstand the maximum pull of the load. However, the mechanical property of the plastic casing cap is not much different to that of the steel.

Considering the cost and weight, the composite plastic is an attractive material for the future casing cap. In addition, the eco-car for saving fuel is a new trend of the automotive industry, which requires the light weight of the automotive parts. The composite plastic helps to achieve this objective.

The results of the analysis in this research work suggest that based on the simulated values of tensile strength and maximum pull of load, all casing caps with 4 different materials comply with the standard of the control cable, JASO F903-75 standard.

Acknowledgements. This work was supported by the AMI (DSTAR), KMITL under the research grant no. DSTAR-RESERCH-01-58-02M. This work was also partially supported by Thai Steel Cable PCL for the case study of "casing cap", and the Sigma Solution Company for the Finite Element Analysis software.

REFERENCES

[1] *JASO F903-75, Japanese Automobile Standard Organization*, 1975.

[2] H. S. Park, X. P. Dang, A. Roderburg and B. Nau, Development of plastic front side panels for green cars, *CIRP Journal of Manufacturing Science and Technology*, vol.6, pp.44-52, 2012.

[3] H. B. Ustaoğlu, S. Ayhün, G. Simitçioğlu, S. Süsler, E. Akay, V. Z. Doğan, Z. Mecitoğlu, H. S. Türkmen and S. Atamer, Static and dynamic analysis of plastic fuel tanks used in buses, *International Conference on Material and Component Performance under Variable Amplitude Loading*, vol.3, pp.509-517, 2015.

[4] B. sreenivasulu and G. Prasanthi, FEA simulation analysis of tube hydroforming process using DEFORM-3D, *Global Congress on Manufacturing and Management*, vol.12, pp.1187-1197, 2014.

[5] V. A. Buryachenko and M. Brun, FEA in elasticity of random structure composites reinforced by heterogeneities of non canonical shape, *International Journal of Solids and Structures*, vol.48, pp.719-728, 2011.

[6] T. J. Young, L. E. Crocker, W. R. Broughton, S. L. Ogin and P. A. Smith, Observations on interphase characterization in polymer composites by nano-scale indentation using AFM and FEA, *Composites: Part A*, vol.50, pp.39-43, 2013.

[7] Y. Zhang, Q. Tan, K. Zhang and J. Li, Analysis of stress and strain of the rolling bearing by FEA method, *International Conference on Applied Physics and Industrial Engineering*, vol.24, pp.19-24, 2012.

[8] A. Sofi and E. Romeo, A novel interval finite element method based on the improved interval analysis, *Computer Methods in Applied Mechanics & Engineering*, vol.311, pp.671-697, 2016.

[9] J. Jaowiec and S. Milewski, Coupling finite element method with meshless finite difference method in thermomechanical problems, *Computers and Mathematics with Applications*, vol.72, pp.2259-2279, 2016.

[10] Y. Cao, Y. Chu, X. Zhang and X. Zhang, Immersed finite element methods for unbounded interface problems with periodic structures, *Journal of Computational and Applied Mathematics*, vol.307, pp.72-81, 2016.

[11] W. Sittiwong, Study of hot air flow behavior using easy FEM simulation and shadow graph techniques, *Journal of Thonburi university*, vol.8, pp.1-14, 2015.

[12] A. Chaimanutsakul, A study of stress in a horizontal pressure vessel and the saddle support using finite element method, *Journal of Thonburi university*, vol.8, pp.1-9, 2013.

[13] B. Ren, C. T. Wu and E. Askari, A 3D discontinuous Galerkin finite element method with the bond-based peridynamics model for dynamic brittle failure analysis, *International Journal of Impact Engineering*, vol.99, pp.14-25, 2016.
[14] L. Zhou and J. Ou, Study of parameters selection in finite element model updating based on parameter correction, *ICIC Express letters*, vol.4, no.5(B), pp.1831-1837, 2010.
[15] H. Ahmadian, G. M. L. Gladwell and F. Ismall, Parameter selection strategies in finite element model updating, *Journal of Vibration and Acoustics*, vol.119, pp.37-45, 1997.
[16] H.-C. Yang, A modified iteration technique in nonlinear finite element analysis with unloading, *ICIC Express letters, Part B: Applications*, vol.6, no.4, pp.1075-1082, 2015.
[17] N. Thamchot, *Engineering Material*, 2nd Edition, Se-education, Bangkok, 2006.
[18] P. Dechaumpai, *Finite Element Method in Engineering*, 2nd Edition, Chulalongkorn center, Bangkok, 2012.