



VERIFYING THE ENGINE COVER OF A PROTOTYPE USING THE FINITE ELEMENT METHOD (FEM)

Anisoara STANCIU, Monica IORDACHE*

University of Pitesti, Romania, monica.iordache@upit.ro

Article history:

Received: 20.01.2016; Accepted: 23.03.2016.

Abstract: *In this paper will be analyzed, by the finite element method, the resistance to deformation of a prototype car engine cover under its own weight, but also the elements that are assembled on it. It examines the stress and displacements resulting from simulation to more constructive solutions. First we will research the influence of sheet thickness on the maximum stresses and strains, and also will develop more solutions to support the vehicle engine cover. Initially, the sheet has a thickness of 0.7 mm, and 12 support pads. We will consider ways of reducing the sheet thickness, and the number of supports, but without significantly changing the internal stress values initially obtained. The research is aimed to obtain low internal stresses on the engine cover prototype studied, out of yield strength to support a force greater than their own weight without deforming as a possible minor impact.*

Keywords: engine cover, finite element method (FEM), stress, sheet, simulation

INTRODUCTION

The finite element method is based on the matrix method of structural analysis of movements. The finite element method attempts to find a solution to a problem approximate by admitting that the domain is divided into subdomains or finite elements with simple geometric shapes and function status. Unknown variables are defined around each item. The main objective of such an analysis is to get the best solutions to a number of conditions imposed. First the user will design a virtual system and will analyze it according to real functional requirements [3]. Finite element analysis model of a structure is, in fact, a checking account number. Thus, for a given geometry of the dimensionally defined, to obtain values of deformation, stress, reaction forces in the bearings or their frequency [1], [2].

Researchers are focusing to use a wide range of tools and techniques to ensure that the designs they are created it saves. Probably, sometimes accidents could happen while they work in the laboratories or factories. Industries need to know whether a product failed or how many percentage need to success it. That is because the design was inadequate. Nevertheless, they have to ensure that the product works well under a wide range of conditions and try to avoid from failure due to any cause. In this regard, finite element modeling (FEM) could help to avoid the failure and improve the success.

FEM is a very important tool for machining application analysis because it can produce very accurate and significant for results [5].

The automotive industry faces world-wide serious challenges: fierce market competition and strict governmental regulations on environment protection. The strategies of the automakers to meet these challenges is sometimes called the 3R Strategy: Reduction in time-to market, reduction in development costs to gain competitiveness, and reduction in the vehicle weight to improve fuel efficiency. The solutions to achieve this triple goal are essentially based on the implementation of CAD/CAE/CAM technologies in product development and process design. [8].

* Corresponding author. Email: monica.iordache@upit.ro

This paper aims to check a prototype car hood resistance to deformation under its own weight on / off, and to seek another solution more economical in terms of the item studied. Specifically, it will study whether the sheet thickness can be reduced without risk, and if the number of surfaces of contact between the hood and the vehicle can also be reduced.

In this paper, there will be a series of simulations by changing the values of the two above mentioned parameters. Initially, the engine cover has been designed with a thickness of 0.7 mm and 12 contact points. Starting from this assumption will change the two parameters sequentially. It is intended that following these changes, internal stress values to be close to those of the original solution to ensure a good resistance in various conditions of engine cover. For this, besides subjecting it to the force of gravity, it will be subjected to a force equal to 1000 N.

This analysis seeks to determine the optimum solution in terms of the contact points of the engine cover, and its sheet thickness, implicitly amount of force.

NUMERICAL PROCEDURE

Materials used

The material used for the engine cover is a steel alloy that has yield strength $R_{p02} = 260$ MPa. The mechanical properties of the material are presented on table 1.

Table 1. Mechanical properties of the material used

Young's modulus	$2 \cdot 10^{11} \text{ N/m}^2$
Poisson's ratio	0.266
Density	7860 kg/m^3
Thermal expansion coefficient	$1.17 \cdot 10^{-5} \text{ K}^{-1}$
Stress	$2.6 \cdot 10^8 \text{ N/m}^2$

Numerical procedure

CATIA V5. CATIA V5 software is used for the tests, exactly CATIA Generative Structural Analysis module interface which is accessed from Analysis & Simulation menu. CATIA provides the user with several types of FEA: Static Case, Case Frequency, Buckling Case, and Combined Case. The first step in addressing an analysis consists in choosing the type of analysis. Also, the finite element method has an important role in the achievement tests.

It should be noted that in succession CAD - FEA - CAM design there is an iterative process - computing - execution. This process is achieved successful operation of synthesis and analysis of the prototype and model for finite element calculation.

FINITE ELEMENTS ANALYSIS. The engine cover is meshed using parabolic tetrahedral elements. The tetrahedral element is a ten node iso-parametric solid element having 3 (translational) degrees of freedom. It has a quadratic displacement behavior and is well suited to model irregular meshes. The element also has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

To achieve the simulation, first apply the appropriate material piece, then set constraints 12 fixed supports, the contact surfaces of the hood, and two pivot links between the hood and hinges. It will also determine the point of application of force, the center of gravity of the piece. All sets are shown in Figure 1. The analysis seeks to determine the optimum solution in terms of the contact points of the engine cover, and sheet thickness of which is made, implicitly amount of force.

The basic model has a thickness of 0.7 mm, and 12 support points arranged symmetrically left - right. To lower the costs of fabrication for the engine cover, must decrease sheet thickness from 0.7 mm to 0.65 mm, and reduce the number of supports from 12 to 8 and even 6. However, these improvements are intended to not change the values very much, the maximum stress or maximum displacement.

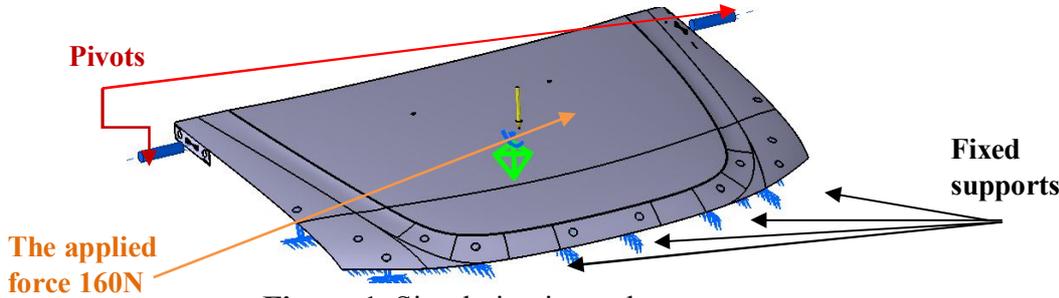


Figure 1. Simulation input data

Also, in this paper, 5 variants will be simulated. These cases are presented in table 2. The fifth case is the most unfavorable, and it will be achieved only if, in the simulations for cases 2, 3 and 4 will obtain low values close to those of the original solution (case 1).

Table 2. Simulations Planning

The analyzed case	Thickness [mm]	No. of supports	Applied force
Case 1	0.7	12	160
Case 2	0.65	12	140
Case 3	0.7	8	160
Case 4	0.7	6	160
Case 5	0.65	6	140

These changes are aimed to obtain low values, far from the limit values and to confer resistance, also to a larger force request. For example, in the case of a minor impact with another body, the engine cover should not deform. This will be check by applying a load of 100 kg weights corresponding to force of 1000 N. So, the 5 cases from the Table 2, will be checked by performing other five simulations, applying a force of 1000 N.

RESULTS AND DISCUSSIONS

Simulation for the case 1. Originally, the piece is made of steel with a thickness of 0.7 mm, it weighs 16 kg default, so a load of 160N. As restrictions, the piece is equipped with 12 fixed supports in front, and two pivot in the hinge zone. The results of the simulation in this case, are shown in figure 2.

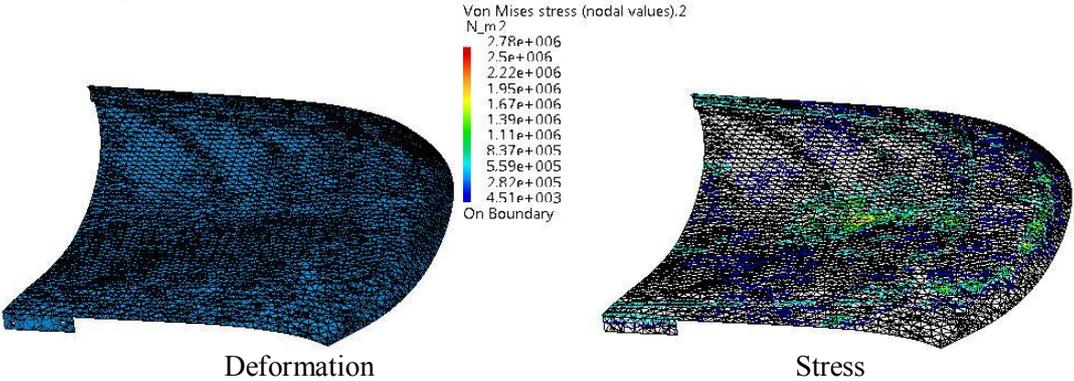


Figure 2. The results for Case 1

As shown in Figure 2, the maximum displacement is 0.0101 mm and maximum stress is $2.78 \times 10^6 \text{ N / m}^2$. Blue colors indicate low values and the colors yellow to red elevated. The yield strength of the used steel is $R_{p02} = 260 \text{ MPa} = 2.6 \times 10^8 \text{ N/m}^2$, so it can be concluded that the model studied will resist to the force of 160N without any risk.

Simulation for the case 2. In this case, it will only change the sheet thickness from 0.7 mm to 0.65 mm. This decreases the amount of force of gravity, from 160N to 140N. In this case, the results do not change very much as shown in Figure 3, the maximum stress is $2.8 \times 10^6 \text{ N / m}^2$ and the maximum displacement is 0.0107 mm.

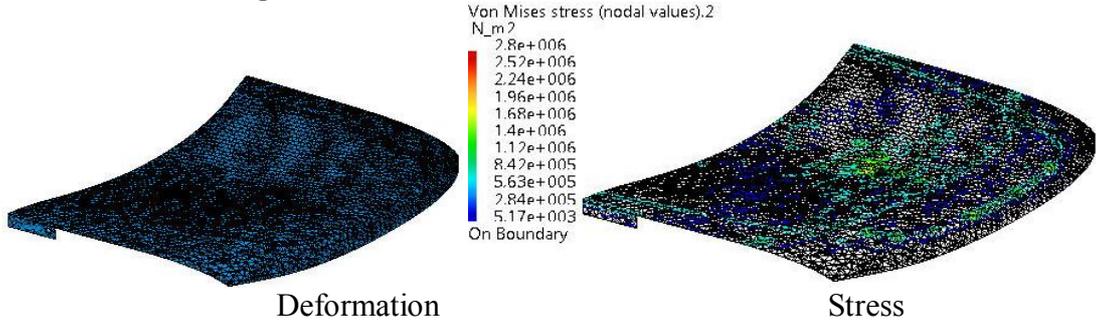


Figure 3. The results for case 2

Simulation for the case 3. Because by changing the thickness, the results are very small, we can reduce the number of the support pads, for checking if these 12 supports are really necessary. Thanks to these small results obtained in the precedents cases, the number of removed supports will be 4 and even 6. First, will be removed the 4 supports marked in figure 4 with red color.

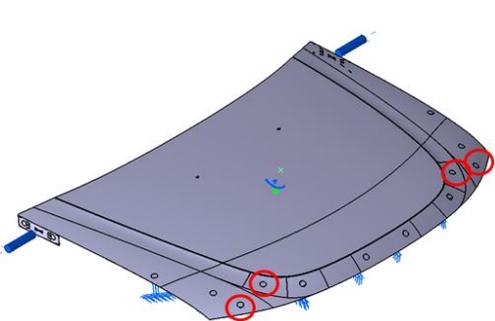


Figure 4. The supports removed in 3th case

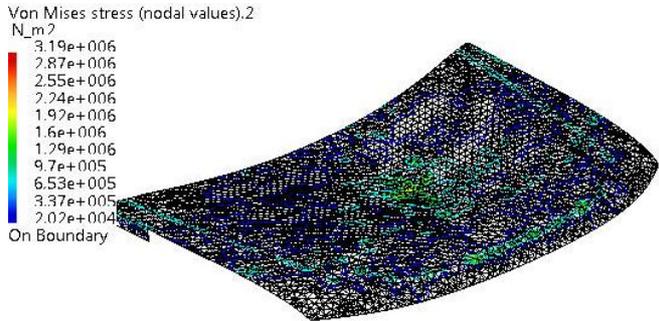


Figure 5. The resultat for Case 3

In this case, the maximum displacement and stress increase, but not significantly, according the figure 5. Also the maximum stress is $3.19 \times 10^6 \text{ N / m}^2$ and the maximum displacement is 0.011 mm.

Simulation for the case 4. On the strength of the small values obtain, the number of pads can be more reduced. Also, will be removed the 6 supports marked in figure 6 with red color.

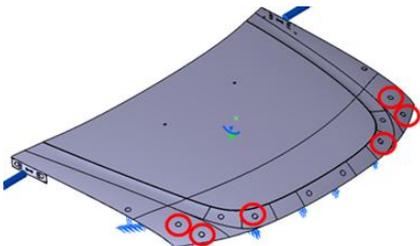


Figure 5. The supports removed in 4th case

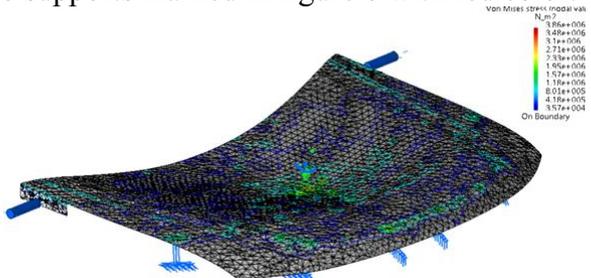


Figure 6. The resultat for Case 4

For these constraints, the result of the simulation is greater than other, but not close to limit. The maximum stress is $3,86 \times 10^6 \text{ N / m}^2$ and the maximum displacement is 0.018 mm.

Simulation for the case 5. Because in all these simulated cases, the values obtained are very small compared to limit values, we can try the worst-case simulation: thickness = 0.65 mm and number of support pads = 6. In this case, the maximum stress is $3.93 \times 10^6 \text{ N / m}^2$ and the maximum displacement is 0.02 mm.

All the results for the five cases, are summarized in the table 3 for better visualization and comparison.

Table 3. Results of the simulations

The analyzed case	Constraints	Maximum Stress [N/m ²] x 10 ⁶	Maximum Deformation [mm]
Case 1	t = 0.7 mm; No. Supports = 12; F = 160N	2,78	0.0101
Case 2	t = 0.65 mm; No. Supports = 12; F = 140N	2,80	0.0107
Case 3	t = 0.7 mm; No. Supports = 8; F = 160N	3,19	0,01
Case 4	t = 0.7 mm; No. Supports = 6; F = 160N	3,86	0.019
Case 5	t = 0.65 mm; No. Supports = 6; F = 140N	3,93	0,02

With the help of the table, it was created a graph which show the variation of the maximum stress according to the studied parameters (number of supports and sheet thickness), shown in figure 7.



Figure 7. The variation of the maximum stress

Simulation for a greater force. This simulation is aimed to check if by increasing the force from 140/160 N to 1000 N, all the studied cases are self to use. The purpose off the simulation for the 1000 N load, is to obtain small values, also highly lower than limit values. Because this simulation studies a possible deformation of the engine cover to a minor impact, the applied force will be distributed in front, as is shown in figure 8.

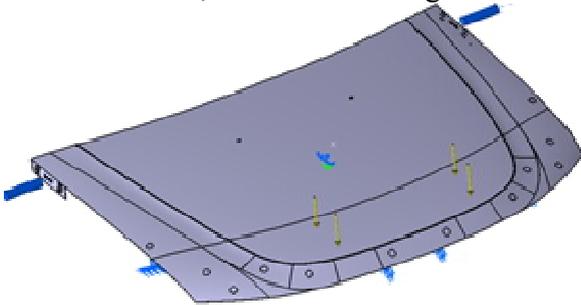


Figure 8. The distributed force

By simulation all the five cases with this distributed force of 1000 N, there were obtained also small values, shown in table 4.

Table 4. The results for 1000N force

The analyzed case	Max. Stress [N/m ²] x 10 ⁶	Max. Deformation [mm]
t = 0.7 mm; No. Supports = 12; F = 1000N	4.86	0.006
t = 0.65 mm; No. Supports = 12; F = 10000N	5.29	0.006
t = 0.7 mm; No. Supports = 8; F = 1000N	5.09	0.007
t = 0.7 mm; No. Supports = 6; F = 1000N	5.33	0.007
t = 0.65 mm; No. Supports = 6; F = 1000N	5.35	0.008

CONCLUSIONS

In this work is described the variation of stress into a prototype engine cover, according to two parameters: the sheet thickness and the number of supports. This study was achieved for searching the best solution regarding the cost off this piece.

From the bibliographic study and by analyzing the results, the following conclusions can be drawn:

- the Catia V5 is the most adequate to use for a static simulation with Finite Elements Analysis, because it needs very few input data (type of material, load and clamps);
- with increasing number of support points, the internal stresses and the maximum displacement decrease;
- by reducing the sheet thickness, the internal stresses and the maximum displacement increase, but not significantly;
- thickness affects not so much when the engine cover is required only at the closing / opening of its own weight thanks to the small force;
- in the case of requiring a larger force, the differences between the results and the limit values are greater, but even in this case are not very significant;
- by applying a force equal to 1000 N, even in the worst case (t = 0.65 mm and 6 supports), the results for internal stresses are very low, 5.35×10^6 N/m²;
- the prototype engine cover, can be made of sheet with thickness equal to 0.65 mm, and it can have only 6 supports, without any risk. This version is very safe to use.

REFERENCES

- [1] Ghionea Ionuț G., *Proiectare asistată în CATIA V5*. București: Editura Bren, 2007
- [2] Ghionea Ionuț G., *Aplicații în ingineria mecanică CATIA V5*. București: Editura Bren, 2008
- [3] Maksay Stefan, Bistriian Diana, *Introducere în metoda elementelor finite*. Iași: Editura Cermi, 2008
- [4] O.C. Zienkiewicz, R.L. Taylor, *The finite element method*, Butterworth-Heinemann, 2000
- [5] Geoffrey Boothroyd, Winston A. Knight, *Fundamentals of machining and machine tools*, Third Edition 2005
- [6] Carl T.F. Ross, *Advanced Applied Finite Element Methods*, Pages 185-251, 1998
- [7] Frank Rieg and Reinhard Hackenschmidt, *Finite Element Analyses for Engineers*, 2014
- [8] A. Makinouchi, C.Teodosiu and T. Nakagawa, *Advance in FEM Simulation and its Related Technologies in Sheet Metal Forming*, *CIRP Annals - Manufacturing Technology*, Volume 47, Issue 2, 1998, Pages 641-649