Case studies for automotive components using CAD and CAE techniques

M D Stroia[1], D E Moşteanu[2], I Virca[2], E Răduca[1], C Popescu[3] and C Haţiegan[1]

[1]"Eftimie Murgu" University of Reșița, Electrical Engineering Department, Piața Traian Vuia 1-4, Reşiţa, 320085, Romania
[2]"Nicolae Balcescu" Land Forces Academy of Sibiu, Sibiu, România
[3]"Constantin Brâncuşi" University of Târgu-Jiu, Târgu-Jiu, România

E-mail: m.stroia@uem.ro

Abstract. In the present paper we present model of 3D components, parts of power supply system found in automobiles. 3D designs of components are obtained using modern techniques of computer systems which implies the means of CAD software. Following components designing, an analysis of obtained models was made, considering static, mechanical and thermal stresses. These studies were made by means of numerical simulation, using CAE techniques, and have the purpose of optimizing geometrical shapes of designed components in the terms of their weight, in the same time maintaining their mechanical and electrical characteristics. From weight optimizing of automotive components ensues significant savings of material and energy consumption, this being a sensitive topic in automotive industry, whether we talk about classical, electrical or hybrid automobiles. Modelling and simulation was made using finite element analysis of SolidWorks2010 software and the obtained results could be used in the process of part production from automotive.

1. Case study framework

One of the most important component in automobile functionality is power supply system and wiring harness is an integrant part of the latter one. Battery connectors are constituent elements of wiring harness of an automobile and play a significant role in ensuring well-functioning of automobile. By means of these connectors the electrical energy transfer between accumulators and other components of electrical system is made. Therefore, it is of great importance for battery connectors to be designed properly and efficiently. Consequently, in order to be able to study their behaviour in terms of mechanical resistance and thermal stresses, we have designed a 3D model of classical battery connector [1].

1.1. 3D design of battery connector

Geometrical dimensions of straight connector and tightening screw assembly design shown in Figure 1 are the result of previous dimensioning calculus considering required restrictions with respect to mechanical and thermal resistance and electrical conductivity which designed connector must ensure.

The components used in engineering fields include section variation due to specific element like: thread holes, grooved wedge, thrust collar, etc. In these places the simulation studies show the local stress with an increased value compared to the stress value calculated by classical resistance formulas [2].

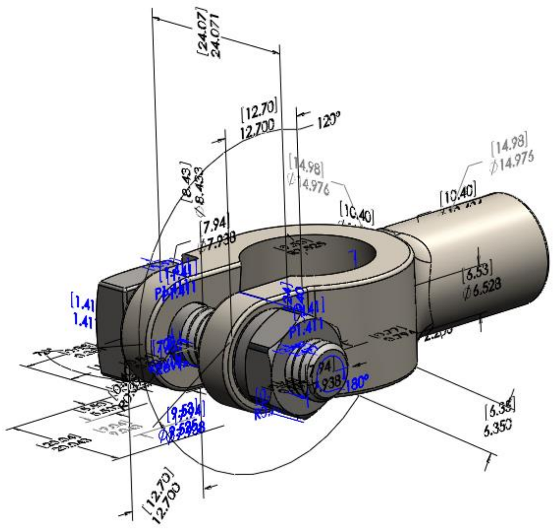


Figure 1. Geometric dimensions for straight connector and tightening screw assembly

1.2. Case study hypothesis

Formerly 3D designed connector by means of SolidWorks software, CAD module, will be subjected to static analysis, followed by thermal analysis, both made by simulations using finite element method. Used software contains a CAE module which allows performing such study, by simulating real conditions [2-4]. For the static case study, we chose a nickel material for straight connector and tightening screw assembly. Subsequently we define loads and constrains applied on various surfaces of design and set up meshing process. We perform simulations for static and thermal analysis and evaluate the scenarios for optimizing the considered design in terms of weight. In the last case study, we define as parameter the height of connector and follow to obtain a minimum material usage but still to meet the Von Misses limit condition. Loads will be in range 100÷1000 N, with 100 N increase. Temperature will be in range 25÷34 ˚C, with 1 ˚C increase.

2. Case study analysis

For present paper we have into account three case studies as follows: first case study assumes a static analysis, second case study implies a thermal analysis and third case study involves an analysis considering possibility of weight reduction of specified 3D connector model [5].

Loads and constraints were applied on surfaces of connector model as shown in Figure 2. We considered “Fixed” type constraints, marked with green arrows, on the inside contact surface of connector with battery terminal, and uniformly distributed load, marked with purple arrows, in the range of 100÷1000 N, with an increase of 100 N, on the contact surfaces of straight connector and tightening screw assembly [4], [5]. Constraints simulate by mechanical fixture the actual contact between connector and battery terminal.

Journal of Physics: Conference Series **1426** (2020) 012047 doi:10.1088/1742-6596/1426/1/012047

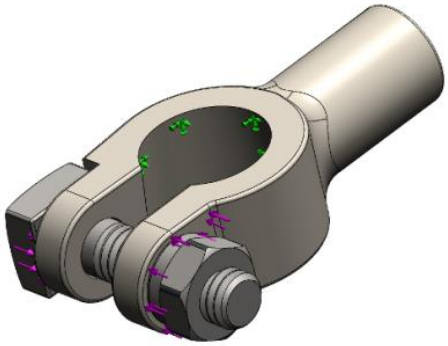


Figure 2. Appling constraints and loads for studied connector

For next step we applied finite element method for meshing connector model and obtained result from Figure 3. Solid meshing type was used and was generated a model having 214.882 elements and 354.214 nodes. Though a high number of elements and nodes increases simulation time, system provides more accurate results [6].

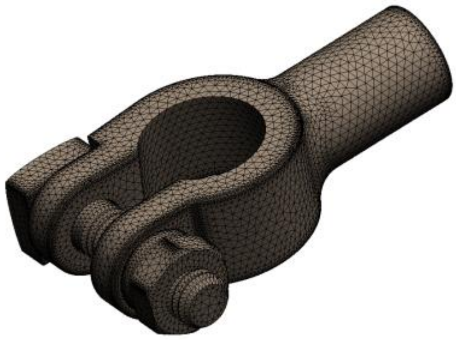


Figure 3. Meshing of studied connector design

2.1. Static analysis

Aims at determining tensile forces and tensile-stress intensity using von Mises criterion [6], by a progressive increase of loads, as well as establishing resultant displacement of finite elements of designed connector.

Von Mises criterion is based on von Mises-Hencky theory, also known as shearing energy theory or maximum distortion energy theory. Regarding tensile forces on the three axes, σ_1, σ_2, σ_3, von Mises stress can be formulated as:

$$\sigma_{vonMises} = \frac{1}{\sqrt{2}}\left[(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_1-\sigma_3)^2\right]^{\frac{1}{2}} \tag{1}$$

Theory asserts that a ductile material shows weakness in places where von Mises stress equals or even exceeds effort limit. Usually effort limit value is replaced by flow resistance value. However, the software allows adjusting this value [6-8].

Results for von Mises tensile stress obtained from simulation, for various load values, are shown in Table 1. Analysing determined values it can be concluded that maximum load that can be applied to connector model is around value of 600 N, where von Mises stress is close to flow limit of 59 MPa corresponding to Ni material. Von Mises stress shows a linear variation, proportional to load, according to Figure 4.

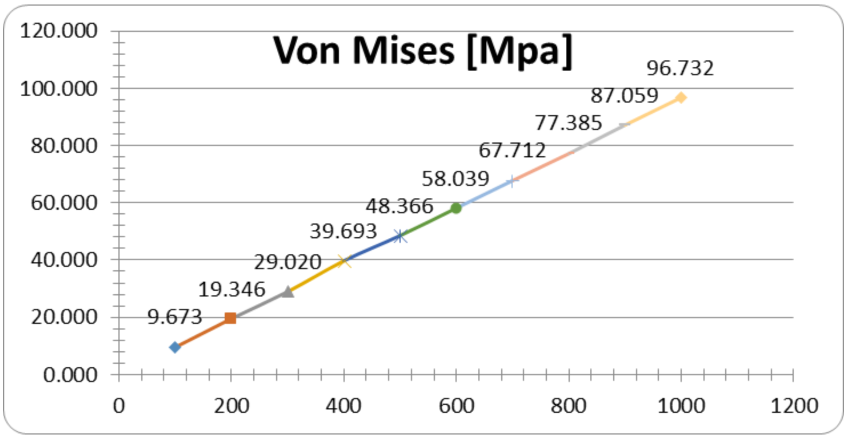


Figure 4. Variation trend of von Mises obtained values as response to various loads

Table 1. The values of Von Mises stress and displacement for applied forces

Load (N)	Stress Von Mises (MPa)	Displacements [mm]
100	9.673	0.000253
200	19.346	0.001101
300	29.020	0.001254
400	39.693	0.001124
500	48.366	0.001021
600	58.039	0.002320
700	67.712	0.002784
800	77.385	0.002412
900	87.059	0.002785
1000	96.732	0.003365

2.2. Thermal analysis

For thermal analysis we have chosen to gradually increase temperature by 1 ˚C in the range of 25÷34 ˚C, all the other prior imposed conditions, with respect to constraints and loads, remaining unchanged [9].

A synthesis of obtained results for von Mises stress can be found in Table 2, for load value of 500 N. Analysing determined values by simulation, from Table 2, we can state that temperatures below 32 ^{0}C don't have a significant influence upon stresses, what we can't say for situations simulated in temperature conditions above 32 ^{0}C.

For higher values than 32 ^{0}C, values for von Mises stresses exceed flow limit, σ_{limit} =59MPa, of material used for modeling battery connector of power supply system, nickel.

Journal of Physics: Conference Series **1426** (2020) 012047 doi:10.1088/1742-6596/1426/1/012047

Table 2. Von Mises stress for 500 N load and various temperatures

Temperature (˚C)	Stress Von Mises (MPa)
25	48.366
26	48.021
27	47.747
28	48.497
29	48.271
30	49.070
31	49.760
32	54.137
33	59.784
34	63.709

Affected areas, in terms of von Mises stress for a 27 ˚C temperature, are shown in Figure 5. It can be observed that maximum von Mises stress value is located in the area where the shape of the model changes from round to straight profile. In addition, Figure 6 presents the manner the connector model is affected by displacements, mostly at fixture areas, which appear when a load of 500 N is applied at temperature of 27 ˚C.

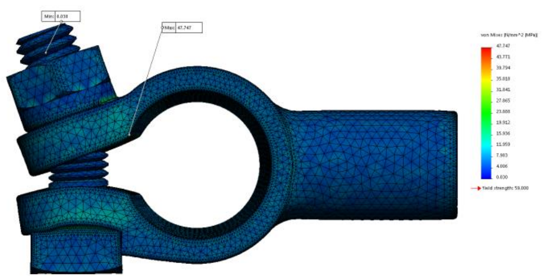


Figure 5. Von Mises stresses obtained for 500N load and 27˚C

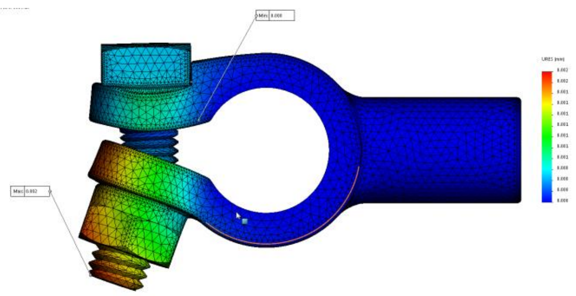


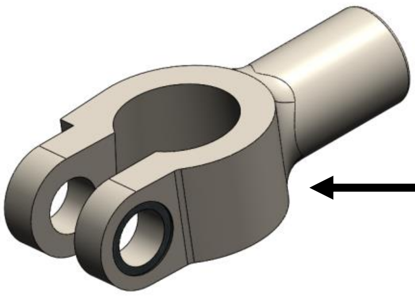
Figure 6. Displacements obtained for 500N load and 27˚C

2.3. Weight optimization study

In order to pursue optimization of battery connector from weight point of view we considered and defined as variable of 3D design the height of analysed connector, which has a significant impact on its geometrical shape [8], [9], as shown in Figures 7 and 8.

The design Study function allows us to generate multiple versions of part by incrementally changing certain parameters, such as fillet radius or extrusion height. From obtained results, we can pick optimal configuration, which may be drastically different from our initial concept but still meet the required factor of safety, load conditions or stress tolerance. Used software offers two methods for running a Design study: evaluation and optimization.

For this case study we used evaluation and optimisation design methods, where we specified discrete values for each variable and used sensors as constrains [2], [3]. The software runs study using various combinations of values and reports the output for each combination. This module lets us evaluate certain scenarios of the design and see results without performing optimization. After we define simulation sensors for constrains, simulation runs the associated studies to track sensor values for each scenario.



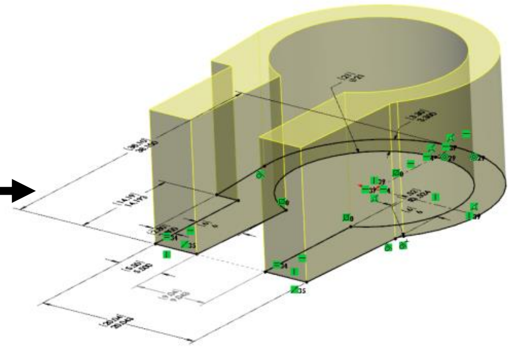


Figure 7. 3D model for studied connector **Figure 8.** Variable thickness of connector

In Figure 9 we present creation of Design Study for optimization mode where we specified values for each dimension, either as discrete value or as a range and used the *as* constraints and *as* goals. The actual value for chosen variable *height* is 15,24 mm, this variable is used in evaluation process by defining 8 scenarios. After running the study, we obtain as result optimum combination of values which meet our specified goal, for this case minimum of mass connector, and specified constraints, for this case Von Mises stress<59MPa (yield strength).

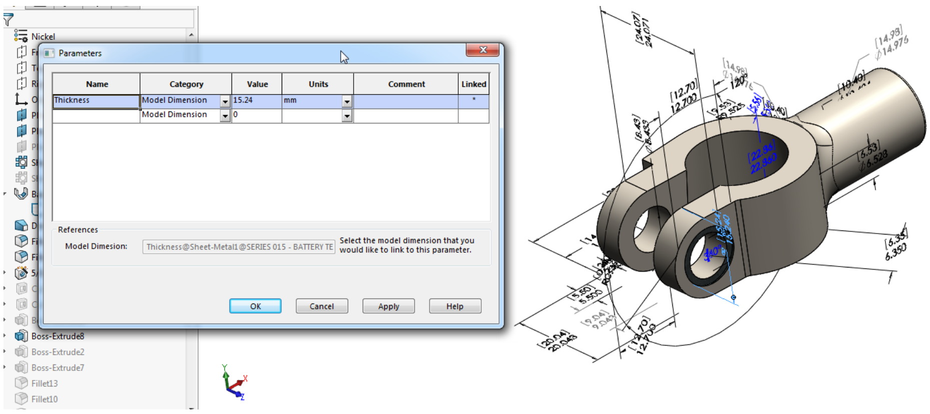


Figure 9. Creation of the design study for evaluation

In Figure 10 are sown result values obtained from design study evaluation, for Von Mises stress, considering a load of 500N, a connector thickness of 15,240 mm, and mass of connector, m=65,061g.

In this case study, for evaluation design we used thickness connector dimension acting as an input parameter with discrete values and stress, $\sigma_{Von\ Mises}$, as a constraint parameter, subjected to a monitoring operation. The goal is to minimize mass of connector.

Results obtained for weight optimization using optimization design study for discrete values of connector thickness in range 12mm÷19mm, an applied load F=500 N and initial mass of connector, m_i=65.061 g can be observed in Figure 11.

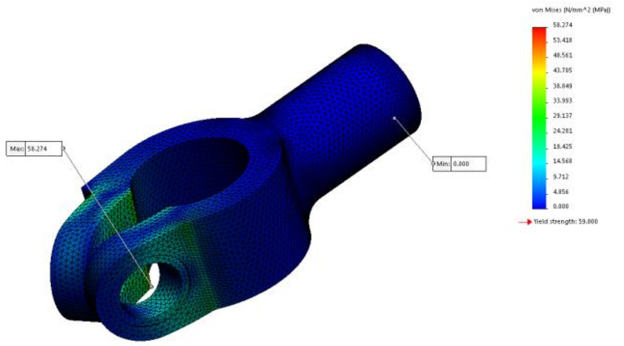


Figure 10. Von Mises Stress for F=500N, thickness=15,240 mm, m=65,061 g

After optimization of connector (see Figure 11), the mass decreases at m_o=53,35g for a Von Mises stress 55,034MPa < 59MPa, limit condition as constrain. The difference of masses, m_d=12 g, consists of a significant material saving at battery connector manufacturing process, considering the mechanical strength of the connector is maintained.

Variable View | Table View | Results View

10 of 10 scenarios ran successfully. Design Study Quality: High

		Current	Initial	Optimal (1)	Scenario 1	Scenario 2	Scenario 3	Scenario 4	Scenario 5	Scenario 6	Scenario 7	Scenario 8
Thickness		15.24mm	15.24mm	12mm	12mm	13mm	14mm	15mm	16mm	17mm	18mm	19mm
Stress1	< 59 N/mm^2	58.2678 N/mm^2	58.2678 N/mm^2	55.0344 N/mm^2	55.0344 N/mm^2	54.5934 N/mm^2	54.7658 N/mm^2	50.4885 N/mm^2	51.2119 N/mm^2	51.217 N/mm^2	50.2021 N/mm^2	48.6052 N/mm^2
Mass1	Minimize	65.061161 g	65.061161 g	53.352201 g	53.352201 g	56.931771 g	60.558352 g	64.264817 g	67.90016 g	71.654222 g	75.411418 g	79.150576 g

Figure 11. Results for optimization design study

3. Conclusions

Using CAD and CAE techniques we can reduce costs in terms of time, material and employee savings and actively contribute to the rapid optimization of mechanical parts, in present paper the electric power supply system of motor vehicles. Studies made in described cases were based on SolidWorks 2010 software suite, which allows fast designing, optimizing, static and thermal analyzing of mechanical parts by finite element simulation, thus leading to more reliable components.

References

[1] Tufoi M, Vela I, Marta C, Mituleţu C, Amariei D and Stroia M D 2010 *Design, Optimization and Realization of Mechanical Parts Using CAD, CAE and CAM Techniques*, Annals of DAAAM for 2010 & Proceedings of the 21st International DAAAM Symposium, Vienna, Austria, October 2010, pp 799-800

[2] Lombard M 2010 *SolidWorks 2010 Bible*, Wiley Publishing Inc., Indianapolis, Indiana, USA

[3] Nedelcu D 2011 *Digital prototyping and numerical simulaion with SolidWorks*, Universitatea "Eftimie Murgu", Reşiţa, pp 475-479

[4] ***Dassault Systems 2010 *SolidWorks 2010 Advanced Part Modeling*, 300 Baker Avenue, Concord, Massachusetts, 01742, USA.

[5] Haţiegan C, Gillich E V, Vasile O, Nedeloni M D, Jurcu M and Magheţi P 2015 Finite Element Analysis of thin plates clamped on the rim of different geometric forms. Part II: The Absolute and Relative Variation of Natural Frequencies, *Romanian Journal of Acoustics & Vibration* **12**(1) 81-86

[6] Mănescu T S, Praisach Z I, Pomoja F, Afronie E M and Stroia M D 2011 *Stresses and Displacement FEM Analysis on Biplane Disks of the Butterfly Valves*, The 4th WSEAS International Conference on Finite Differences/Elements/Volumes-Boundary Elements, Paris, Franta, pp 88-91

[7] Barboni L, Haţiegan C, Gillich G R, Hamat C O, Vasile O and Stroia M D 2018 Natural frequencies of thin rectangular plates clamped on contour using the Finite Element Method, *IOP Conf. Ser.: Mater. Sci. Eng.* **294** 012033

[8] Hațiegan C, Stroia M D, Popescu C, Răduca E, Racoceanu C 2018 Using finite element method to determin eigenfrequencies of two l- shaped platens assembly embedded on a surface. Part II, *Analele Universității "Constantin Brâncuși" Târgu Jiu* **2** 59-64

[9] Haţiegan C, Molnar M, Trocaru S, Pădureanu I, Jurcu M R and Ilie F 2016 *Modeling and Simulation of Thermal Analysis of a Teflon Coated Plate*, International Conference Knowledge-Based Organization, Vol. 22, Issue 3, pp 639-643