Parameterized optimization analysis with finite elements, using ANSYS[®], of a reference point under a torque action

F G Marian^{1,2} and G Grebenişan¹

¹University of Oradea, Faculty of Managerial and Technological Engineering, Romania, str. Armatei Române nr. 5, 3700 Oradea, Bihor, România

marianflaviu77@yahoo.com

Abstract. To determine the elastic/plastic deformation value of the part, we used Ansys R19.0 software(CAE), to simulate a torque moment applied to the functional area part, which provided the information set about the part behavior. The mold of the part has 4 cavities, and the part material is a polyamide. Having a requirement imposed by the customer on the drawing, we considered this finite element analysis it is necessary before start manufacturing the molding and starting injection the parts to demonstrate that the customer requirement is achievable. The results obtained from the analysis with finite elements in ANSYS R19 showed that the constructional part could not fulfill the design requirement, namely to withstand a torque of 4500Nmm the part being fixed in the hole with a diameter of 2.5 mm. After analyzing the results, the client will have to modify the current drawing based on the results obtained by us, namely to reduce the torque value from 4500Nmm to 4000.3 Nmm.

1. Introduction

Injection of plastics has been a significant development, especially among the automotive industry. PLASTOR, one of the largest manufacturers of plastics and molds from Romania, has met along the years valuable clients with more prominent and drastic requirements for plastic injection from the automotive industry.

In this article will use the ANSYS software to analyze and optimized using as a variable value the torque moment, which is applied on the part in a specified area and compare the results, and then draw conclusions. Ansys software is used to design products and semiconductors, as well as to create simulations to test product durability, temperature distribution, fluid movements, and electromagnetic properties. Ansys develops and markets finite element analysis software used to simulate engineering problems (CAE-Computer-Aided Engineering), [1]. The software creates simulated computer models of structures, electronics or machine components to simulate strength, hardness, elasticity, temperature distribution, electromagnetism, fluid flow, and other attributes. Ansys is used to determine how a product will work with different specifications without building test products or performing impact tests. For example, Ansys software can simulate how a bridge will manage after years of trafficking, how to best process salmon at a factory to reduce the amount of waste, or how to design a slide that uses less material without sacrificing safety, [2], [3], [4].

Most Ansys simulations are performed using the Ansys Workbench software, one of the company's main products. Typically, Ansys users break larger structures into small components, each being individually modeled and tested. A user can start by defining the dimensions of an object and then adding weight, pressure, temperature, and other physical properties. Finally, Ansys software simulates and analyzes motion, fatigue, fractures, fluid flow, temperature distribution, electromagnetic efficiency, and other effects over time[5], [6].

2. Information about the part



Figure 1. 3D model of the part

Length	Value (mm)	Volume (mm ³)	Material
 Х	41.05		
Y	22.00	2064.4	PC
Z	11.00		(polycarbonate)

3. Testing part requests

Besides dimensional control, the client has a particular requirement. This requirement is that the part has to withstand a torque T=minimum 4500 Nmm applied in the hole with diameter 2.5 mm +0.1 mm. The part must be blocked against rotation in functional area to sketch according to the drawing below



Figure 2. Part test requirement by the project



Figure 3. Part test requirement imposed and defined in ANSYS R19, [7]

4. Finite Element Analysis

Based on the values obtained, result that 4500 Nmm torque moment value it is to raised and the part can be damaged during his functionality.





Figure 4. Total deformation (mm)

Figure 5. Maximum Equivalent stress (von Mises) (MPa)

The maximum value of the total deformation (plastic deformation), defined as displacement, is maximum 11.541 mm. The maximum value of the equivalent stress is 186.5 MPa.

To see the maximum torque moment value that can be applied on the part without to affect his functionality, an optimization procedure will be needed.



Figure 6. Optimization Tradeoff and Spider sensitivity (Response Points)

5. Optimization using Design of Experiments, and results [7], [8]

Optimizing a project using the Design of Experiments technique involves performing a finite element analysis before allowing the setting of some convenient limits for the chosen input parameters, ie the achievement of objectives imposed by design, technological or design requirements operation. Do not aim here, minimize a cost function, or maximize efficiency, of any class. After the analysis of finite elements can be determined, through the algorithms characteristic of Design of Experiments techniques, the data set consisting of design points, generating new projects, similar to the one initially analyzed, but with results configured for the new input parameter values. Based on these design points, the behavior of the output parameters is intended, which in fact determines the behavior of the project itself. These design points, in relation to the results obtained (output parameter values), generate the behavioral diagrams of the project, called Response Surfaces. Response Surfaces provide the operator with interpolation results with the degree of influence indicated in the Local Sensitivity Diagrams. The new torque moment value is 4000.3 Nmm, which is the maximum value applied on the part without to affect it. In this case, the client must change the draw and the value of the torque moment from 4500 Nmm in 4000.3 Nmm.



After optimization, the maximum value of the total deformation is 0.0192 mm. The maximum value after optimization of equivalent stress is 550.47 MPa.

6. Conclusion

The optimization was performed using several well-established, complementary methods: parameter correlation, linear optimization, and Response Surface's method after the project was analyzed by the finite element method. Each of these phases of project implementation is determined by the results obtained. Thus, by correlating the parameters, the degree of influence of the input parameters on the output parameters was determined to decide which of them decisively influences the functioning and the degree of safety of the operation of the project. The degree of safety of the project is given by the degree of certainty regarding the quality of the product, as well as the degree of satisfaction of the client, about the behavior of the finished product, which is also the high quality of the product. This phase is viably linked to the analysis of the finished part of the required piece at the torque imposed by the customer. Thus, the lifetime or safety factor of the project determines the quality of the obtained piece and the economic efficiency of the production. Direct optimization takes into account parameter correlation but does not use results as the basis for optimization itself. Optimization using Response Surfaces is an extremely laborious method but provides the most concise results with optimization checkpoints performed on one or all three candidate points found in the optimization process. The candidate points are a "statistical summary" of the best design points for which the algorithm provides the user with the optimization results but also the deviations from a statistical average deviation and the degree of compliance or convergence of the solution, the deviations being displayed in percentage. Only after the client changed the draw and the torque value, the producer will proceed the project and

begin the mold manufacturing and then injecting the parts.

Using ANSYS R19 before starting a project, everything that is on the drawing it is checked and analyzed reducing time, energy and material resources are efficient costs, the credibility and your professionalism to the client give you next opportunities for projects and the delivery term is respected.

This study we show as that sometimes even the client can make a mistake that's why the automotive parts manufacturers need to be updated with everything means CAM, CAD, CAE. Following the analysis, the client sent the design with the modifications requested by us. The project will continue by designing the mold (CAD) and then manufacturing it (CAM).





a)-Upper mold

b)-Lov Figure 9. The part mold

After the mold is received by the production department, the first injection trials will be made using the parameters agreed by the customer and then, after the CTC control give the feedback to the production department, the final parts will be injected and ready to be shipped to the customer.



Figure 10. The injection parts (4 cavities)

Acknowledgments

Thanks to advisors for all support: Prof. dr. Eng Ion Sereș - General Manager, Eng. Dan Buzdugan – Quality Manager (PLASTOR SA Oradea)

References

- [1] https://en.wikipedia.org/wiki/Ansys
- [2] Jianping L, Hongwei Z, Xiaoli H, Lin Z, 2016 Pengliang H, Cong L, Ning L, Yuexi Za, and Changyi L 2016, Ins. And Exp. Tech. Volume 59, Issue 5 P762–767
- [3] Grebenişan G., Bogdan S. 2017 Parameterized Finite Element Analysis of a Superplastic Forming Process, Using ANSYS, MATEC Web of Conferences, 126, 2017, <u>https://doi.org/10.1051/matecconf/201712603001</u>
- [4] Negrău D C, *Finite Element Analysis, and Solution Optimization for a bending device,* 2018, University of Oradea.
- [5] Huei-H L, 2017 Finite Element Simulations with ANSYS Workbench 17- Theory, Applications, Case Studies, SDC Publications
- [6] Xiaolin C, Yijun L 2018 Finite Element Modeling and Simulation with ANSYS Workbench, Second Edition CRC Press DOI: 10.1201/9781351045872
- [7] ANSYS 19, Workbench User's Guide <u>http://www.ansys.com</u> (accessed in April 2019)
- [8] Negrau D. C., Grebenisan G., Indre C. 2019 Experimental approach and finite element analysis of the behavior of a steel bending machine, IManE&E 2019