ECT2014 9th International Conference on Engineering Computational Technology
Naples, 2-5 September 2014, Italy

DESIGN AND OPTIMIZATION OF A CONTROL LINK OF A VEHICLE SUSPENSION SYSTEM

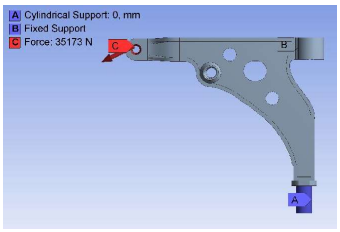
Dr.-Ing. D. V. Koulocheris, Dr.-Ing. C. G. Vossou, C. G. Provatidis, Th. Costopoulos
School of Mechanical Engineering, National Technical University of Athens
Athens, GR-15780, Greece
e-mail: dbkoulva@central.ntua.gr, web page: http://vlab.mech.ntua.gr

Abstract The purpose of a control link in the suspension system of a vehicle is to control the movement of the wheel without supporting the vehicle's weight. In the present paper, a commercial control link has been designed in Solidworks ® CAD software and its performance has been evaluated through static, modal and dynamic analysis in the Ansys Workbench ® CAE software. Additionally, optimization and design analysis tools, available in Ansys software, have been used in order investigate the robustness of the design and the material characteristics of this particular control link. More specifically the influence of its thickness along with different topologies and diameters of the holes that exist on its main body have been investigated in terms of achieving a more robust design. Finally, different material properties have been tested in the initial design in order to investigate the effect of different stiffness moduli to the static and the modal behavior of the component.
Keywords: *Control link, CAD, CAE, Finite Elements, Material Properties, Optimization*

Introduction

Three types of suspension links exist: (a) control, (b) support and (c) secondary (auxiliary) links. Their purpose is to control the movement of the wheel without supporting the vehicle's weight. Control arms are mainly subjected to horizontal forces that act at their joints[1]. In the present paper a commercial control link has been parametrically designed in Solidworks and its performance has been investigated through static and modal analysis in Ansys.

LC	F_x (N)	F_y (N)
1	-31460	-15730
2	-28460	-14230
3	-24200	-8200
4	-5900	-2020



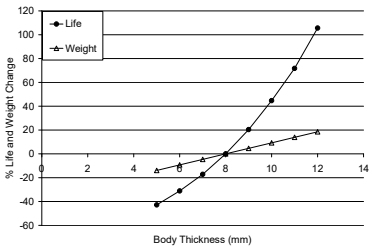
Materials & Methods

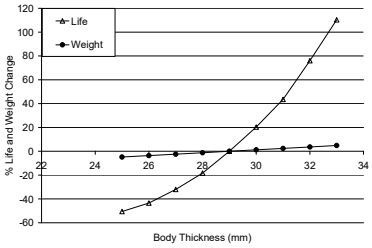
Modal and static analysis (4 different loading cases[2,3]) have been performed in order to investigate the effect of different *(a) material properties*, *(b) diameter* of the holes appearing on the main body of the control link [0.1, 45.0] mm, *(b) thickness* [5.00, 12.00] mm and (c) *height* of the rib surrounding the the main body [25.00, 33.00] mm to the **strength**, **stress factors** and **fatigue characteristics** of the control link. For the evaluation of the geometrical parameters the Design Assessment tool of Ansys Workbench has been used.

Results & Discussion

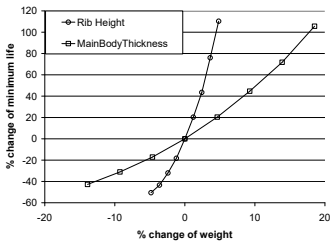
Using *materials* with lower density and elastic modulus leads reduces the eigenfrequency of the control link. The maximum total deformation increases as the values of the density and the elastic modulus of the manufacturing material decrease.

Minimum value for the *diameters* of the existing holes lead to maximum values of the minimum life as well as Safety factor The increase of the minimum life compared to the initial design is 0.7% while the increase of the safety factor is 0.25% with a simultaneous increase of weight of 1.5%.





Increase of the thickness of main body and the height of the rib leads to increase of minimum life.



Conclusions

- A 3D CAD model of a commercial control link has been built and analyzed in a CAD and a CAE software.
- Only Titanium seems to be an adequate alternative material, since the rest have an elongation limit lower than the resulting value of the analysis.
- The rib height influences the performance of the control link more than the thickness of its main body for the same weight change.
- The integration of the CAD and CAE software forms a very powerful tool for the development of new designs or the optimization of existing ones in the automotive industry.

References: *[1] HEIßING B., ERSOY M., "CHASSIS HANDBOOK: FUNDAMENTALS, DRIVING DYNAMICS, COMPONENTS, MECHATRONICS, PERSPECTIVES" 2011, SPRINGER VERLAG NY. / [2] NADOT Y., DENIER V., "FATIGUE FAILURE OF SUSPENSION ARM: EXPERIMENTAL ANALYSIS AND MULTIAXIAL CRITERION", 2004, ENGINEERING FAILURE ANALYSIS, 11, PP. 485–499. / [3] PINFOLD M., CALVERT G., "EXPERIMENTAL ANALYSIS OF A COMPOSITE AUTOMOTIVE SUSPENSION ARM", 1994, COMPOSITES, 25, PP. 59 – 63.*