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

Dr.-Ing. D. V. Koulocheris, Dr.-Ing. C. G. Vossou, C. G. Provatis, 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 to 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[2,3]. 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.

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%.

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 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.

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