**Abstract**

This work consists of studying the structure of a mechanical system by numerical analysis in order to optimize its topology, and studying the mechanical behavior of the optimal structure under mechanical stresses. The accomplishment of this work is based on using SolidWorks to design the mechanical system, using the calculation code by finite element ABAQUS for the numerical analyses and simulations, and using the optimization tool TOSCA to optimize and redesign the mechanical system. We have created a numerical analysis to study the structure under mechanical solicitations, and a topology optimization task in order to get an optimal design starting from an initial one. The results show that the optimal design is acceptable and it can be manufactured by molding. We can create new designs with increased sustainability and reduced weight using a topology optimization