

機構零件之最佳化設計 – 以手機彈片為例

張烜琥
富士康國際

摘要

本文旨在藉由Abaqus求解器及HyperStudy，以針對手機中常見之金屬彈片進行最佳化之設計。文中以彈片本體之厚度及幾何形狀、以及彈片之對手件 – 電池背蓋卡勾之幾何形狀作為設計變因，並對其進行參數化之調整，再經由HyperStudy中之DOE Study找出不同設計變因之敏感度，進而決定在所有設計變因中最具影響力之項目，以針對其進行彈片最佳化之設計。

關鍵字：Abaqus、HyperStudy、設計變因、手機、彈片、最佳化

ABSTRACT

The main goal of this thesis is to do an optimization study of a metal spring which is commonly used in a mobile phone by Abaqus solver and HyperStudy. Firstly, it is needed to define the design variables such as the material, shape, and thickness of the spring. Then find the sensitivities by using the DOE study module in HyperStudy and figure out the dominant design variables of all. Thus the optimization design can be achieved for the designer by using the Optimization study module.

Keywords: Abaqus, HyperStudy, design variables, mobile phone, spring, optimization

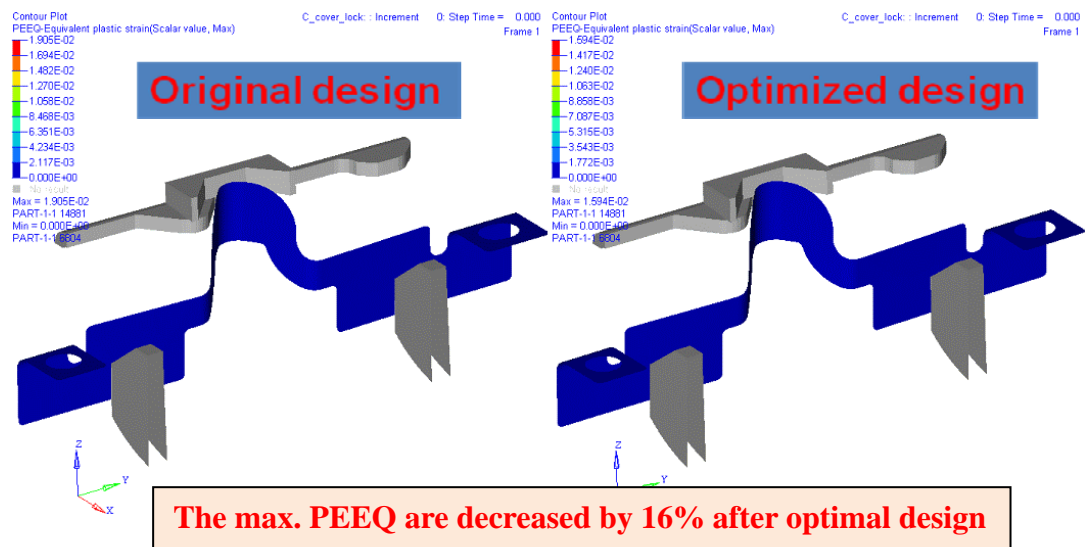


圖 1 手機彈片之最佳化設計