

應用 Abaqus 於按鍵手感最佳化之設計

張烜琥
富士康國際

摘要

隨著科技的發展，人們對於電子產品的依賴度也愈來愈高。有鑒於按鍵被廣泛的運用在消費性電子產品中，如手機、鍵盤、遙控器等等，如何設計出良好的按鍵結構，也就成了一項重要的課題。

本文旨在藉由Abaqus，以針對按鍵手感進行幾何結構之最佳化設計。文中以按鍵本體之厚度及幾何形狀、以及plunger之高度作為設計變因，對其進行參數化之調整。並以按鍵之手感為設計目標函數，以針對其進行按鍵最佳化之設計。

關鍵字：Abaqus、設計變因、按鍵、手感、最佳化

ABSTRACT

As the development of technology, people are tending to depend on electronic devices more often. According to the widely usage of keypad in consumer electronic devices such as mobile phone, keyboard, and remote control, it has become an important subject to design a good keypad structure.

The main goal of this thesis is to do an optimized design of a keypad by Abaqus solver. Firstly, it is needed to define the design variables such as the thickness, shape, and height of plunger of the keypad. Then by setting the tactile response of keypad as the constraint and objective, the optimized design can be achieved for the designer by running the optimization iteration.

Keywords: Abaqus, design variables, keypad, tactile, optimization

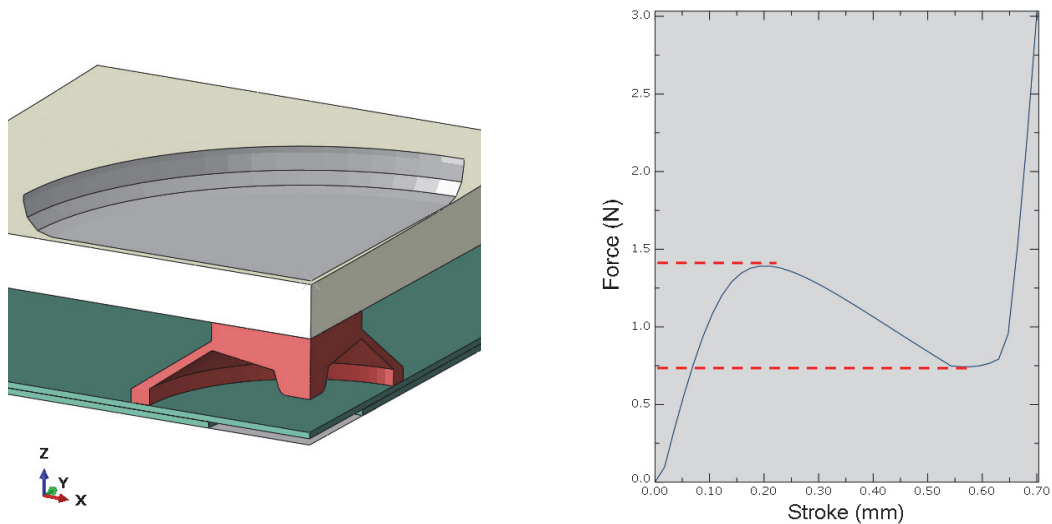


圖 1 按鍵手感之最佳化設計