

# 考慮扭取硬化之循環塑性力學分析

呂學育, 廖國基, 蘇承維, 彭彥翔

中華科技大學航空機械系

## 摘要

本文考慮材料降伏面扭曲現象，以準確探討結構承受循環負荷作用下之彈塑性行為。有限元素商業化分析軟體具有泛用之彈塑性分析功能，惟均尚未能提供考慮降伏面扭曲的模擬功能。因此，本文基於 ABAQUS 所提供使用者副程式的功能，採用文獻上所推導的扭曲硬化模式，編寫模擬材料降伏面扭曲現象，以擴充 ABAQUS 的考慮循環負荷作用之彈塑性分析功能。分析案例採用 CS1026 材料，探討其彈性安定、棘輪、低週期破壞等行為，分析結果並與文獻上的實驗結果及數值分析結果比較，以驗證副程式編寫案例分析模擬之正確性。本文研究成果可提供不同類型的負荷及結構的相關彈塑性行為分析，未來也可延伸考慮不同材料降伏面扭曲現象的模擬。

**關鍵字：**降伏面扭曲、棘輪、有限元素彈塑性分析、單軸循環負荷

## ABSTRACT

In this thesis, we aim to investigate elastic-plastic analysis of structures subjected to cyclic loads considering yield surface distortion. Finite-element commercial codes provide elastic-plastic analysis for multi-purposes. However, they do not take into account yield surface distortion. In the thesis, we enhance the capability of ABAQUS in the simulation of yield surface distortion by editing the user subroutine. The advanced constitutive model considering the distortional hardening available in the literature is adopted. Case studies are performed by ABAQUS and compare with those experimental and numerical results of CS1026 published in the literature. In the future, the research results of the thesis can extend to consider various types of loadings, structures and materials.

**Keywords:** Yield surface distortion, Ratcheting, Finite-element elastic-plastic analysis, Cyclic uniaxial loading.