



航空学报 » 2006, Vol. 27 » Issue (6) :1088-1091 DOI:

论文

最新目录 | 下期目录 | 过刊浏览 | 高级检索

<< Previous Articles | Next Articles >>

### 基于ANSYS的复合材料层合板单钉连接件参数化结构仿真

张爽<sup>1</sup>, 王栋<sup>1</sup>, 酈正能<sup>1</sup>, 寇长河<sup>1</sup>, 章怡宁<sup>2</sup>

1. 北京航空航天大学 飞机设计研究所, 北京 100083; 2. 沈阳飞机设计研究所, 辽宁 沈阳 110035

### Parameterized Simulation for Composite Laminates with Single Mechanically Fastened Joint Based on ANSYS

ZHANG Shuang<sup>1</sup>, WANG Dong<sup>1</sup>, LI Zheng-neng<sup>1</sup>, KOU Chang-he<sup>1</sup>, ZHANG Yi-ning<sup>2</sup>

1. Aircraft Design and Research Institute, Beijing University of Aeronautics and Astronautics, Beijing 100083, China; 2. Shenyang Aircraft Design and Research Institute, Shenyang 110035, China

摘要

参考文献

相关文章

Download: PDF (562KB) HTML 0KB Export: BibTeX or EndNote (RIS) Supporting Info

**摘要** 应用ANSYS参数化设计语言(APDL)开发复合材料层合板机械连接结构的ANSYS分析模块,实现有限元分析前处理过程的参数化建模和后处理过程的累积损伤可视化仿真。建立三维累积损伤有限元模型,采用ANSYS中柔体对柔体的面-面接触单元对螺栓表面和复合材料层合板孔壁间的接触行为进行模拟。进行的T300帘子布/QY8911双马来酰亚胺树脂层合板单钉单搭螺栓挤压强度试验验证此分析模块的有效性,对不同几何参数、不同铺层类型的复合材料层合板机械连接结构的承载性能进行可视化仿真模拟。

**关键词:** 结构仿真 复合材料层合板 机械连接 累积损伤 接触模型

**Abstract:** A parameterized simulation program is developed to predict the response of composite laminates with single bolt structure. The three-dimensional non-linear progressive damage model is developed on the platform of ANSYS using APDL. Parameterized modeling and progressive damage simulation could be achieved in this analysis. The analysis takes into account the contact at bolt-hole interface. The effects of stacking sequences and geometrical parameters on the bearing strength are determined experimentally. For all the analyses performed, the damage progression of single bolted joint is predicted.

**Keywords:** structure simulation composite laminates mechanically joints progressive damage contact model

Received 2005-05-26; published 2006-12-25

#### 引用本文:

张爽;王栋;酈正能;寇长河;章怡宁. 基于ANSYS的复合材料层合板单钉连接件参数化结构仿真[J]. 航空学报, 2006, 27(6): 1088-1091.

ZHANG Shuang; WANG Dong; LI Zheng-neng; KOU Chang-he; ZHANG Yi-ning. Parameterized Simulation for Composite Laminates with Single Mechanically Fastened Joint Based on ANSYS[J]. Acta Aeronautica et Astronautica Sinica, 2006, 27(6): 1088-1091.

#### Service

- ▶ 把本文推荐给朋友
- ▶ 加入我的书架
- ▶ 加入引用管理器
- ▶ Email Alert
- ▶ RSS

#### 作者相关文章

- ▶ 张爽
- ▶ 王栋
- ▶ 酈正能
- ▶ 寇长河
- ▶ 章怡宁