吉林农业大学学报 2011, 33(4) 459-463 DOI: CNKI:22-1100/S.20110607.0923.0 ISSN: 1000-5684 CN: 22-1100/S

论文

基于ANSYS的耕整机机架有限元分析

肖成林[1], 周德义[1]|王永强[2], 杨翔宇[3], 李东来[4], 王子佳[1]

1.吉林大学生物与农业工程学院,长春130022; 2.吉林大学机械科学与工程学院,长春130022; 3.吉林大学汽车工程学院,长春130022|4.吉林省农业机械试验鉴定站,长春130062

摘要:

机架是耕整机上的重要结构,其品质好坏直接影响着耕整机的使用寿命。应用CATIA实现了耕整机机架模型的参数化建模,分析了耕整机工作时的受力情况,用有限元分析软件ANSYS完成对耕整机机架的有限元模型建立和强度、刚度分析,得到了机架的应力分布情况,对应力大的结构提出了改进措施,改进后效果较好。

关键词: 耕整机机架 强度; 刚度 CATIA ANSYS

Finite Element Analysis of Tillage Machine′s Frame Base on ANSYS

XIAO Cheng-lin[1],ZHOU De-yi[1],WANG Yong-qiang[2],YANG Xiang-yu[3],LI Dong-lai[4],WANG Zi-jia[1]

1. College of Biological and Agricultural Engineering, Jilin University, Changchun 130022,China; 2. College of Mechanical Science and Engineering, Jilin University, Changchun 130022,China; 3. College of Automotive Engineering, Jilin University, Changchun 130022,China; 4. Jilin Province Agricultural Machinery Test Appraisal Center, Changchun 130062, China

Abstract:

Frame of tillage machine is an important structure, the quality of which has direct impact on the life of tillage machine. Using the software of CATIA,the frame′s parametric model of 3D entity is built. The analysis of typical condition is finished when tillage machine is working. The finite element model isbuilt, the analysis of strength and stiffness is carried out by ANSYS, and thestress distribution is obtained. The improvement measures to the large stress structure are proposed. Some good results are obtained. This paper provides references for further improvement of frame and obtains a higher value in engineeringapplications

Keywords: tillage machine frame strength; stiffness CATIA ANSYS

收稿日期 2011-02-08 **修回日期** **网络版发布日期**

DOI: CNKI:22-1100/S.20110607.0923.0

基金项目:

科学技术部科技人员服务企业项目（2009GJB10036）

通讯作者:

作者简介:

作者Email:

参考文献:

本刊中的类似文章

文章评论

反馈人		邮箱地址	

反馈标题		验证码	8859