基于有限元分析对轿车自动化生产线运载辅助夹具进行优化设计

摘　　要

轿车运载辅助夹具是汽车工业中、外形结构繁琐、稳定性能要求高的重要零部件。从轿车运载辅助夹具结构材料的稳定性和生产成本两方面出发，本文采用有限元分析方法对其进行优化设计，具体内容如下：

（1）本文以 CATIA 和 ANSYS Workbench 为工具，对其进行参数化设计，建立了轿车运载辅助夹具三维模型，并进行了运载辅助夹具底部框架的主要组件设计，构成底部框架主体结构。

（2）对初步设计的运载辅助夹具底部框架整体结构在不同工况条件下进行力学分析，找到危险工况和薄弱环节，应力出现在右端的支撑组件上，最大值为 46.634MPa 小于 156MPa，满足使用要求，可以看出运载辅助夹具有很大优化空间。

（3）通过 ANSYS 软件对初始设计的底部框架进行了形状优化、尺寸优化、拓扑优化，重新建模进行应力分析，最大的应力值为 117.66MPa，与优化前比较增大 71.026MPa，没有达到底部框架材料的屈服极限 235MPa，满足使用要求。计算出来的固有频率数值在 1.7Hz，相应的振动有很小的差别，为发生振动的现象提供了参考数据。

夹具整体力学分析解决了局部零件应力大容易变形破损等问题，优化设计的研究表明：优化后的底部框架结构的弯曲刚度、最大应力有增加，但仍能保证运载辅助夹具的正常使用。并且底部框架的重量比优化前减轻了 15.44%，实现了运载辅助夹具底部框架优化目的。

关键词：轿车运载辅助夹具；有限元分析；优化设计；底部框架

Based on the finite element analysis method, the optimal design of the carrier auxiliary fixture for the automatic production line of cars was carried out

ABSTRACT

The vehicle carrier assist fixture is an important part in the automobile industry, which has complicated appearance and high stability requirement. From the two aspects of the stability and production cost of vehicle carrying auxiliary fixture structure materials, this paper adopts the finite element analysis method to optimize the design, the specific content is as follows:

(1) In this paper, CATIA and ANSYS Workbench were used as tools to carry out parametric design, a THREE-DIMENSIONAL model of the vehicle carrier assist fixture was established, and the main components of the bottom frame of the carrier assist fixture were designed to constitute the main structure of the bottom frame.

(2) Carry out mechanical analysis on the overall structure of the bottom frame of the preliminarily designed carrier auxiliary fixture under different working conditions, and find out the dangerous working conditions and weak links. which meets the use requirements.

(3) Stress analysis was carried out by re-modeling. The maximum stress value was 117.66mpa, which was 71.026mpa larger than that before optimization. The calculated natural frequency value is 1.7Hz, and the corresponding vibration has little difference, which provides reference data for the occurrence of vibration.

The overall mechanical analysis of the fixture has solved the problem of large stress and easy deformation and damage of local parts, and the research on optimal design shows that the bending stiffness and maximum stress of the bottom frame structure have been increased, but the normal use of the carrier auxiliary fixture can still be guaranteed. In addition, the weight ratio of the bottom frame was reduced by 15.44% before optimization, which realized the optimization purpose of the bottom frame of the carrier auxiliary fixture.

Key words: car carrier assist fixture; Finite element analysis; Optimize the design; At the bottom of the frame.