Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 174 (2017) 638 – 647

Procedia Engineering

www.elsevier.com/locate/procedia

13th Global Congress on Manufacturing and Management, GCMM 2016

Finite Element Analysis and Optimal Design for the Frame of SX360 Dump Trucks

Yuan Ren, Yongchang Yu, Binbin Zhao, Chuanhui Fan, He Li*

College of Mechanical and Electrical Engineering, Henan Agricultural University, Zhengzhou 450002, China.

Abstract

As bearing foundations of autos, the frame bears various forces and torques transmitted from engine, clutch, body, and cargo container. Analysis of natural vibration frequency and dynamic mode of the frame is an important process to ensure operating capabilities of autos. It is hard to yield accurate analysis results with traditional methods. However, the FEA software ABAQUS can provide accurate frame stress distributions, frame deflection laws at its two ends and the middle point, and transient frame dynamics. Most of previous finite element analysis only focused on frame mass and model optimization analysis, lacking of vibration and mode analysis. In this paper the large-scale software ABAQUS was used to analyze the natural vibration frequency and dynamic mode of the frame of SX360 dump trucks, providing necessary theoretic basis for optimal design of the frame.

Peer-review under responsibility of the organizing committee of the 13th Global Congress on Manufacturing and Management
Keywords: frame; finite element analysis; stress; transient dynamics; deflection

1. Introduction

As bearing foundations of autos, the frame supports all auto parts including the engine, body and cargo container, and bears various forces and torques transmitted from the engine, clutch, body and cargo container. In overall design of a vehicle, the frame strength and frame rigidity are very important, and insufficient rigidity will cause vibration and noise and reduce the passenger comfort, operational stability and reliability [1]. To provide necessary theoretic basis for structural improvement and optimal design of the frame, it is essential to understand the analysis of natural vibration frequency and dynamic mode. ABAQUS is powerful finite analysis software with abundant geometric elements library and various material models library. It can simulate many parameters of various engineering materials including capability and property, and address force and contact analysis of single parts and combined parts, and imitate analysis and calculation of many stresses and physical fields. Previous finite element analysis only

* Corresponding author. Tel.: +86-1551-5549-801.
E-mail address: chungbuk@163.com

Peer-review under responsibility of the organizing committee of the 13th Global Congress on Manufacturing and Management
doi:10.1016/j.proeng.2017.01.201

focused on frame mass and model optimization analysis. In this paper the large-scale software ABAQUS was used to build a parametric finite element model for the frame of SX360 dump trucks, in order to provide structural optimal design for the frame, ensure reasonable frame design and improve its overall operating performance [2-5].

2. Frame analysis model

2.1. Three-dimensional physical model of the frame

Main structural parameters of the frame: Q235 steel material, $7.85\times10^{-6}\,\mathrm{kg}/mm^{3}$ in density, 1500mm in length, 500mm in width, with 5# steel channel used as standard parts to support at 520mm and 980mm..As shown in Fig. 1.

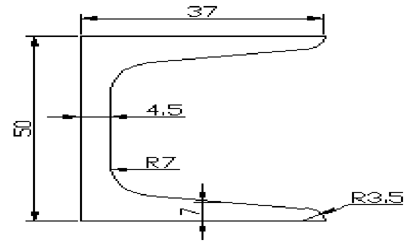

Fig. 1. Structure of steel channel.

At first in the functional module SKETCH in ABAQUS, a two-dimensional section was drawn for the frame and then in the functional module PART, it was stretched in different faces to form a three-dimensional physical model, only a half of which would be extracted for analysis and calculation based on symmetric frame structure and load. The simplified three-dimensional physical model of the frame is shown in Fig. 2.

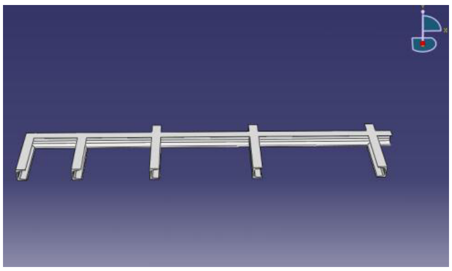

Fig. 2. Simplified three-dimensional physical mode of the frame.

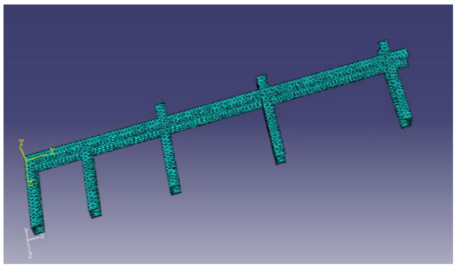

Fig. 3. Finite element model of the frame.

2.2. Finite element model of the frame

After incorporating with proper materials, section attributes and assembly parts, divide the built three-dimensional physical model into grids. To meet calculation accuracy and reduce calculation amount, use the three-dimensional physical unit C3D4 for sweeping grid division, and then with reduced integral calculation, divide the modelinto 19591 cells and 41440 nodes. Grid divisions are shown in Fig.3.

3. Analysis of calculation results

3.1. Static load analysis of the frame

The frame bears not only the engine, chassis and cargos, but also various forces and torques created during travel of the vehicle. Therefore, the frame strength not only concerns whether the whole vehicle operates normally, but also