不同桥梁数值模型车桥耦合对比分析

毛旺涛，王涛，王琪，刘修平，程建平，邓路路

摘要：选取湖北省天门市境内一座跨径 30m 的简支 T 梁为试验对象，探讨车桥耦合振动的建模分析问题。首先，建立了三种不同有限元模型——梁格模型、壳-梁模型及实体模型。其次，结合现场试验，采用有限元模型修正方法，对以上三种有限元模型进行修正，同时基于有限元软件 ANSYS 中的参数化编程和瞬态分析功能，实现车桥耦合分析。最后，计算得到三种模型在移动荷载工况下的位移及应力时程，并与实测值对比。结果表明，实体模型的位移及应力时程与实测值吻合度最高，壳-梁模型次之，从而为车桥耦合数值分析提供了一种建模的思路。

关键词：车桥耦合分析；有限元模型；模型修正；时程；实体模型

引言

1 车桥耦合振动分析基本理论
1.1

DOI: 10.3724/SP.J.1224.2019.00099
图 1 三轴车动力分析模型

\[ M \ddot{Z}_v + C \dot{Z}_v + K Z_v = F_v \]
\[ M \ddot{Z}_b + C \dot{Z}_b + K Z_b = F_b \]

\[ F_v = F_{vi}(Z_v, \dot{Z}_v, \ddot{Z}_v, Z_b, \dot{Z}_b, \ddot{Z}_b, i) \]
\[ F_b = F_{bi}(Z_v, \dot{Z}_v, \ddot{Z}_v, Z_b, \dot{Z}_b, \ddot{Z}_b, i) \]

2 模型修正

2.1 模型修正
图 2 跨中横断面图（单位：mm）

表 1 不同桥梁有限元模型

|   |   |   |   |
|---|---|---|---|
| 1 |   | 280个Beam4单元 |   |
| 2 |   | 70个Beam4单元 |   |
| 3 |   | 196个Shell63单元 |   |
| 4 |   | 8685个Solid45单元 |   |

2.2
图 3 静力工况车辆加载位置（单位：m）

2.3

2.3.1

2.3.2

表 2 修正前后各个参数的变化

| 参数          | 修正前 | 修正后 | 变化量 |
|---------------|--------|--------|--------|
| 弹性模量/10^4 MPa | 3.45   | 4.13   | 19.7   |
| 密度/kg/m³    | 2549   | 2588.5 | 1.5    |
| 剪切模量/10^4 N·m² | 859.05 | 1023.84 | 19.2  |
| 压缩模量/10^4 MPa | 3.45   | 3.685  | 6.8    |
| 弹性模量/10^4 MPa | 3.45   | 3.88   | 12.5   |
| 密度/10^4 MPa | 2549   | 2503.2 | -1.8   |
表 3 修正前后计算频率与实测频率的比较（单位：Hz）

|       |       |       |       |       |       |       |       |       | MAC(%) |
|-------|-------|-------|-------|-------|-------|-------|-------|-------|--------|
| 1     | 4.78  | 4.91  | 4.71  | 2.72  | -1.46 | 100.00 |
| 1     | 7.14  | 7.55  | 7.24  | 5.74  | 1.40   | 98.98  |
| 1     | 16.71 | 17.88 | 16.39 | 4.79  | -1.92  | 99.43  |

3 不同桥梁有限元模型下车桥耦合数值模拟

3.1 ANSYS MATLAB

1 ANSYS

4 APDL

Δt≤ T/15

表 4 车辆技术参数

|       |       |       |       |       |       |       |       |       |       |
|-------|-------|-------|-------|-------|-------|-------|-------|-------|-------|
|       | kg   | kN/m² | kN/m   | kN/s/m | kN/m² | kN/m   | kN/s/m |       |
|       | 9100  | 1577  | 26.6   | 3146   | 22.4   | 33.4   | 33.4   |       |
|       | 15450 | 2362  | 20     | 2362   | 33.4   | 33.4   |        |       |
图 4 跨中梁底位移时程

图 5 跨中梁底应力时程

结语
Comparative Analysis of Vehicle-Bridge Coupling for Different Bridge Numerical Models

Mao Wangtao, Wang Tao, Wang Qi, Liu Xiuping, Cheng Jianping, Deng Lulu

(School of Highway, Chang’an University, Xi’an 710064, China)

Abstract: Commercial general finite element software has been increasingly used in vehicle-bridge coupling analysis in recent years. This paper uses a simple supported T-beam with a span of 30 m as an engineering example. Firstly, three different finite element models are established: grillage, shell-beam, and solid. Second, combined with field tests, these three finite element models are modified using the finite element model updating method. Third, based on the parametric programming and transient analysis functions of ANSYS finite element software, vehicle-bridge coupling analysis is realized. Finally, the displacement and stress time histories of the three models under moving loads are calculated and compared with the measured values. The results show that the displacement and stress histories of the solid model most successfully agree with the measured values, followed by the shell-beam model. Hence, this paper provides a method for the modeling and analysis of vehicle-bridge coupling.

Key Words: vehicle-bridge coupling analysis; finite element model; model updating; time history; solid model