A sensitivity analysis
for the design parameters
of a car side-view mirror

Ji Lu

The School of Mechanical Engineering,
The University of Adelaide,
South Australia 5005
AUSTRALIA

Thesis submitted for the degree of

Master of Engineering Science on 12th July 2005
Abstract

The reflected image in a motor vehicle side-view mirror can be distorted by vibration, to the extent that it may become impossible to clearly see a reflected object. The design parameters of a mirror (such as geometry, material and mass) will have a direct effect upon the level of vibration-induced distortion. The aim of this thesis was therefore to use finite element modelling to show how changes in the design parameters of the mirror affected by vibration-induced image distortion. A design engineer would then be able to predict how proposed design changes would affect vibration.

In order to develop a method of confidently predicting the effect of design parameter changes, a simple supported plate was initially analysed. The plate was modeled with Finite Element Analysis (FEA) software, ANSYS, where a modal analysis was performed to determine theoretical modal properties such as natural frequency and associated mode shapes. A harmonic response analysis was also performed to determine the Frequency Response Function (FRF) of the model. To verify these results a modal analysis of a physical plate (with identical geometry, material and constraint parameters) was performed and the results from this experiment were compared to those that were determined by the model. The experimental set-up and the parameters of the model were then both reviewed until there was good correlation between the modelled results and the results that were obtained from the experiment.

Once a modelling method that gave accurate predictions was established, the same skill was applied to modelling the mirror system in two stages of increasing complexity, to once again ensure confidence in the final overall results. Initially just the mirror bracket was modelled to determine the boundary conditions in the absence of any complexities that may arise with a complete assembly. Such complexities may have clouded any reasons for potential uncorrelated results between the model and the experiment. After achieving good correlation between the numerical results obtained from the model of the bracket, and the results of testing a physical bracket, a complete FE model of the mirror assembly was built. Since this was a combination of multiply parts, each with different material
properties, it was necessary concentrate on the simulation of the internal connections within the assembly. The results were also verified through experimental modal testing experiment.

By following this method of increasing model complexity and steadily increasing confidence in the model accuracy, it was possible to evaluate the effect of design parameter changes, such as increased dimensions or changes in the choice of materials, with a high level of confidence. After the model was changed to reflect a proposed design parameter change, it was possible to evaluate the sensitivity of the frequency response characteristics with respect to the excitation characteristics of the vehicle.
Table of contents

Abstract ................................................................. i
Statement of originality ........................................... xi
Acknowledgements .................................................. xii
Chapter 1 Introduction ........................................... 1
  1.1 Motivation and objectives of the research ................. 1
  1.2 The finite element method .................................. 3
  1.3 Modal analysis ................................................. 4
  1.4 Sensitivity analysis ........................................... 5
  1.5 Scope .......................................................... 6
Chapter 2 Literature Review .................................... 9
  2.1 Background .................................................... 9
  2.1.1 Side-view mirror and the vibration .................... 9
  2.1.2 Field tests on the side-view mirror ...................... 11
  2.1.3 Evaluation of field tests .................................. 13
  2.2 The finite element approach .................................. 15
  2.3 Finite element analysis on the rear-view mirror ............ 17
  2.4 A finite element analysis of a side-view mirror ................ 20
  2.5 Conclusion on the FEA mirror case studies ................. 24
Chapter 3 The Experimental Verification of a Simple Case Study ... 25
  3.1 Introduction .................................................. 25
  3.2 Properties of the plate ....................................... 26
  3.3 Finite element analysis of the plate ....................... 29
    3.3.1 Defining the geometry, material and physical properties .... 29
    3.3.2 Meshing .................................................. 29
    3.3.3 The boundary conditions ................................ 31
    3.3.4 The modal analysis solution ............................ 31
    3.3.5 Classical calculations .................................. 32
    3.3.6 Modal analysis results .................................. 32
    3.3.7 Harmonic response analysis results .................... 37
Table of Contents

3.4 Modal testing experiment..............................................................................40
3.4.1 The experimental configuration.............................................................40
3.4.2 Shaker excitation method........................................................................42
3.4.3 Results....................................................................................................42
3.5 Conclusion..................................................................................................44

Chapter 4 The Schenefacker Side-View Mirror.............................................45
4.1 Introduction..................................................................................................45
4.2 Description of the side-view mirror............................................................45

Chapter 5 Finite Element Analysis of the Mirror Bracket and Verification
......................................................................................................................50
5.1 Introduction..................................................................................................50
5.2 Description of the bracket..........................................................................50
5.3 Finite element analysis of the bracket.........................................................53
5.3.1 Modelling...............................................................................................53
5.3.2 Description of the FE model....................................................................54
5.3.3 Modal analysis results............................................................................57
5.3.4 Harmonic response analysis results.......................................................62
5.4 Experimental verification of the bracket FE model.....................................64
5.4.1 Experimental configuration.....................................................................64
5.4.2 Results....................................................................................................67
5.5 Conclusion..................................................................................................68

Chapter 6 FEA and Experimental Verification of the Side-View Mirror
assembly..........................................................................................................70
6.1 Introduction..................................................................................................70
6.2 Finite element analysis of the side-view mirror........................................70
6.2.1 FE model overview...............................................................................70
6.2.2 Defining the inter-connection.................................................................72
6.2.3 The connection between the bracket and the actuator holder..............72
6.2.4 The connection between the actuator and the glass holder....................75
6.2.5 Properties of the FE model.....................................................................77
6.3 Modal analysis results................................................................................79
6.4 Harmonic response analysis results............................................................83
6.5 Experimental verification of the complete mirror model...........................84

iv
# Table of Contents

6.5.1 Experimental configuration .................................................................................. 84  
6.5.2 Results .................................................................................................................. 88  
6.6 Conclusion ............................................................................................................... 89  

Chapter 7 Sensitivity Analysis of the Mirror Design Parameters ................. 91  
  7.1 Introduction ............................................................................................................. 91  
  7.2 Modifying the mirror plate geometry ...................................................................... 92  
  7.2.1 The Mirror Thickness ......................................................................................... 92  
  7.2.2 The Mirror Surface Area ..................................................................................... 93  
  7.3 Conclusion of the sensitivity analysis ...................................................................... 93  

Chapter 8 Conclusion and Recommendations ...................................................... 101  
  8.1 Conclusion ............................................................................................................. 101  
  8.2 Recommendations ................................................................................................. 102  

References: .................................................................................................................. 104  

Appendix-A ..................................................................................................................... 108  
  Basic theory of finite element analysis ....................................................................... 108  

Appendix-B ..................................................................................................................... 112  
  Theoretical calculation of natural frequency of the plate ......................................... 112  

Appendix-C ..................................................................................................................... 114  
  Structural damping in ANSYS ..................................................................................... 114