Steady Aeroelastic Response Prediction and Validation for Automobile Hoods

THESIS

Presented in Partial Fulfillment of the Requirements for the Degree Master of Science in the Graduate School of The Ohio State University

By

Justin Mark Pesich, B.S.

Graduate Program in Aeronautical and Astronautical Engineering

The Ohio State University

2017

Master's Examination Committee:

Jack J. McNamara, Advisor

Sandip Mazumder
Abstract

This thesis describes a strategy to predict steady aeroelastic response of an automobile hood at high speeds using coupled fluid dynamic and structural codes. The pursuit of improved fuel economy through weight reduction, reduced manufacturing costs, and improved crash safety can result in increased compliance in automobile structures. However, with compliance comes an increased susceptibility to aerodynamic and vibratory loads. The hood in particular withstands considerable aerodynamic force at highway speeds, creating the potential for significant aeroelastic response that may adversely impact customer satisfaction and perception of vehicle quality.

The goal of this thesis is to develop and couple high fidelity fluid and structural computational models to improve the understanding of fluid-structure interactions between automobile hoods and the surrounding internal and external flow. Computational analysis is carried out using coupled CFD-FEM solvers with detailed models of the automobile topology and structural components. The experimental work consists of wind tunnel tests using a full-scale production vehicle. Comparisons between the numerical and experimental results yield
important insights into required modeling fidelity, coupling, and challenges in validation for the aeroelastic response of automobile hoods.

Three separate vehicle configurations are considered. The first configuration resembles an initial design model or “styling” model which neglects the internal flow through the front fascia and has a simplified underbody and wheels. The second configuration is a “complete” model including all vehicle components. The last configuration is an adaptation of the complete model employing a simplified engine compartment and underbody. One motive for the last configuration is the complete model was found to have inadequate mesh cell quality to implement into the coupled simulation framework, but a model with reduced complexity is satisfactory. Furthermore, these configurations are used to study the importance of the internal flow. The degree of the mutual interaction between the fluid and structure is also considered. Investigations of computational uncertainty indicate low sensitivity of simulation results to small changes in fluid modeling. In addition, an examination of measurement compliance showed large margins of experimental uncertainty.
Dedication

To my family, Benjamin Grove, and Katie Wheeler.
Acknowledgments

I would like to thank my advisor, Dr. Jack McNamara for his incredible insight and passion for my research. Without his steep expectations for hard work and dedication to solve demanding research problems, I would not be the engineer and person I am today. He has given me an essential skill set that will allow me to become a very successful engineer. Further than academics, he has graciously reached out to his students to build friendships outside the classroom. I am grateful to have worked alongside a state-of-the-art researcher and mentor. I would also like to thank Dr. Sandip Mazumder for his insight to the research, and for serving on my defense committee.

I am extremely grateful to Honda R&D Americas, Inc. for funding this work. It was a pleasure working with a world-renowned automotive manufacturer that does great engineering. Specifically, I would like to thank Peter Kang for building and managing the structural model. Additionally, I would like to thank Austin Kimbrell, Allen Sheldon, Craig Kline and Annie Boh for their support and
management of this project. Last, I would like to thank the many engineers responsible for carrying out the experimental work required for this research.

This work would not have been possible without the contributions from the Lockheed Martin Wind Tunnel staff, the Ohio Supercomputing Center, and the Simulation and Innovation Modeling Center for computational resources. I would also like to acknowledge technical insights given by the Multi-Physics Interactions Research Group members, Dan Steen from CD-adapco, and Naethan Eagles from TotalSim. Last I would like to thank my family for their constant encouragement and support.
Vita

2014.................................................................B.S. Aeronautical and Astronautical Engineering, The Ohio State University

2014 – 2016..................................................Graduate Research Assistant, The Ohio State University

Fields of Study

Major Field: Aeronautical and Astronautical Engineering
# Table of Contents

Abstract .......................................................................................................................... ii

Dedication ....................................................................................................................... iv

Acknowledgments .......................................................................................................... v

Vita .................................................................................................................................. vii

Table of Contents .......................................................................................................... viii

List of Tables .................................................................................................................. x

List of Figures ................................................................................................................ xii

Chapter 1: Introduction and Objectives ........................................................................ 1

1.1 Introduction ............................................................................................................. 1

1.2 Literature Review .................................................................................................. 2

1.3 Objectives of this Thesis ......................................................................................... 3

1.4 Key Novel Contributions of this Thesis ................................................................ 5

Chapter 2: Experimental Setup ..................................................................................... 6

Chapter 3: Computational Models and Coupling Procedure .................................... 14
List of Tables

Table 1: Dimensions of Lockheed Martin Low Speed Wind Tunnel .................. 6
Table 2: RMSE of Experimental Displacement Measurements .......................... 11
Table 3: Inertial and Viscous Resistances of Radiator and Condenser ............... 16
Table 4: Cell Count for Each Vehicle Configuration......................................... 24
Table 5: Summary of the Grid Convergence Study ......................................... 25
Table 6: Hood Structural Components ............................................................. 28
Table 7: Structural Model Boundary Conditions .............................................. 29
Table 8: Structural Model Validation by Hood Sag Test ................................... 30
Table 9. Operating Conditions ........................................................................... 34
Table 10: $L_2$ and $L_\infty$ Error Norms of Hood Surface Pressure Data using FSI Prediction Pressures ......................................................................................... 44
Table 11: Coupled Displacement Results of V3 With and Without Lasers .......... 48
Table 12: Comparison of Uncoupled and Coupled Response ............................ 49
Table 13: Coupled Displacement Results of V1 and V3 ..................................... 52
Table 14: Coupled Displacement Results of V1 With and Without Internal Pressure Loading ......................................................................................................... 53
Table 15: Uncoupled Displacement Results of V1, V2, V3

64
List of Figures

Figure 1: Displacement Measurement Locations .................................................. 7
Figure 2: Laser Fixture on Driver’s Side Fender (Point 1) ........................................ 8
Figure 3: Laser Fixture on Trailing Edge Midline (Point 3) ..................................... 8
Figure 4: Surface Pressure Probe Locations ............................................................. 9
Figure 5: Pitot-Static Tube for Reference Pressure .................................................. 9
Figure 6: Data Acquisition Signal for Experimental Displacement Measurements
........................................................................................................................................ 10
Figure 7: Description of Measurement Apparatus Compliance ................................ 12
Figure 8: Measurement Apparatus Load-Displacement Data with Linear Curve Fit
........................................................................................................................................ 12
Figure 9: Apparatus Displacement due to Wind Load .............................................. 13
Figure 10: V1 Model Geometry .................................................................................. 15
Figure 11: V2 Model Geometry .................................................................................. 16
Figure 12: V3 Model Geometry .................................................................................. 17
Figure 13: Isometric View of Remaining Under-hood Boundaries in V3 .......... 17
Figure 14: Fluid Domain Boundary Conditions and Dimensions, Front View .... 19
Figure 15: Fluid Domain Boundary Conditions and Dimensions, Side View ..... 19
Figure 16: Pressure Boundary Condition Location.................................................. 19
Figure 17: Domains with Increasing Inlet/Outlet Distances from Vehicle.......... 20
Figure 18: Probes along Inlet of D1 (0 < z < 4.95 m)............................................. 21
Figure 19: Velocity Distribution of Each Domain (D1-D4)................................. 21
Figure 20: Top View of Volume Mesh on Floor of Fluid Domain ...................... 23
Figure 21: Side View of Volume Mesh on Planar Slice of Fluid Domain .......... 24
Figure 22: Under Hood Volume Mesh of Each Vehicle Configuration ............ 24
Figure 23: Discrete Pressure Probing Locations for Grid Convergence Study .... 25
Figure 24: Pressure Comparison along Probe Set 1 ............................................. 26
Figure 25: Pressure Comparison along Probe Set 2 ............................................. 26
Figure 26: Pressure Comparison along Probe Set 3 ............................................. 27
Figure 27: Hood Structural Assembly................................................................. 28
Figure 28: Structural Model Boundary Conditions............................................. 29
Figure 29: Partitioned FSI Simulation Workflow.................................................. 31
Figure 30: Explicit Coupling Scheme Diagram.................................................... 32
Figure 31: Steady-State Internal Pressure Distribution of V2............................ 35
Figure 32: Discrete Pressure Comparison along Strip 1, 100 kph...................... 37
Figure 33: Discrete Pressure Comparison along Strip 2, 100 kph...................... 37
Figure 34: Discrete Pressure Comparison along Strip 3, 100 kph...................... 38
Figure 35: Discrete Pressure Comparison along Strip 4, 100 kph...................... 38
Figure 36: Discrete Pressure Comparison along Strip 5, 100 kph ........................................ 39
Figure 37: Discrete Pressure Comparison along Strip 1, 160 kph ........................................ 39
Figure 38: Discrete Pressure Comparison along Strip 2, 160 kph ........................................ 40
Figure 39: Discrete Pressure Comparison along Strip 3, 160 kph ........................................ 40
Figure 40: Discrete Pressure Comparison along Strip 4, 160 kph ........................................ 41
Figure 41: Discrete Pressure Comparison along Strip 5, 160 kph ........................................ 41
Figure 42: Discrete Pressure Comparison along Strip 1, 200 kph ........................................ 42
Figure 43: Discrete Pressure Comparison along Strip 2, 200 kph ........................................ 42
Figure 44: Discrete Pressure Comparison along Strip 3, 200 kph ........................................ 43
Figure 45: Discrete Pressure Comparison along Strip 4, 200 kph ........................................ 43
Figure 46: Discrete Pressure Comparison along Strip 5, 200 kph ........................................ 44
Figure 47: Displacement Comparison of FSI Prediction to Experiment, 100 kph 45
Figure 48: Displacement Comparison of FSI Prediction to Experiment, 160 kph 45
Figure 49: Displacement Comparison of FSI Prediction to Experiment, 200 kph 46
Figure 50: CFD Modeling of Laser Instrumentation ................................................................. 47
Figure 51: Pressure Distribution of V3 With and Without Lasers .......................................... 47
Figure 52: Uncoupled vs. Coupled Structural Response ......................................................... 49
Figure 53: Steady Aeroelastic Pressure Distribution of V1 and V3 ........................................ 51
Figure 54: Cowl Top Geometry of V1 and V3 ..................................................................... 51
Figure 55: Steady Aeroelastic Structural Response of V1 and V3 .......................................... 52
Figure 56: Steady Aeroelastic Structural Response of V1 With and Without Internal Pressure Loading................................................................. 53

Figure 57: Wall $y^+$ Distribution of V1 ................................................................. 59

Figure 58: Wall $y^+$ Distribution of V2 ................................................................. 59

Figure 59: Wall $y^+$ Distribution of V3 ................................................................. 60

Figure 60: Discrete Pressure Comparison of V1, V2, V3 along Strip 1............. 61

Figure 61: Discrete Pressure Comparison of V1, V2, V3 along Strip 2............. 62

Figure 62: Discrete Pressure Comparison of V1, V2, V3 along Strip 3............. 62

Figure 63: Discrete Pressure Comparison of V1, V2, V3 along Strip 4............. 63

Figure 64: Discrete Pressure Comparison of V1, V2, V3 along Strip 5............. 63
Chapter 1: Introduction and Objectives

1.1 Introduction

Automobile hood design is complicated by many often competing factors, such as: pedestrian/crash safety, weight, durability, styling, aerodynamics, manufacturability, and cost. One important design feature is hood compliance, which must be appropriately balanced to meet the above objectives. However, the impact of hood compliance is not easily handled in the early stages of design due to: 1) the potential for aeroelastic interactions; 2) tight margins on allowable hood deflection, 3) the high cost of prototyping and experimentation, and 4) the fact that sub-discipline modeling errors tend to aggregate in coupled systems. The second and fourth issues indicate that a high level of model detail may be needed, while the third issue indicates that computational capabilities are critical. Thus, the development and assessment of aeroelastic prediction tools, and considerations for their validation, are important areas of study.
1.2 Literature Review

Previous studies published on the general problem of automobile aeroelasticity are relatively limited. One study focused on aeroelastic tailoring of an Indy car rear spoiler to reduce drag at high speeds and maximize downforce at low speeds [1]. The analysis was carried out by coupling the computational fluid dynamics (CFD) software ANSYS Fluent and finite element method (FEM) software MSC Nastran to solve the static structural response. The optimized spoiler obtained a 3% reduction in wing drag while maintaining the same downforce during cornering maneuvers. Consequently, the vehicle top speed was projected to improve by 1 kilometer per hour (kph). Another study investigated the increase in drag due to a deformed chin spoiler [2]. STAR-CCM+ was used to carry out the coupled analysis for both domains. The chin spoiler deflection was predicted to increase the drag coefficient by 0.004, corresponding to a 0.15 mile per gallon decrease at 80 kph. Gupta et al. [3] and Gaylard et al. [4] used an uncoupled approach to assess hood vibrations due to wake shedding of an upstream vehicle. Time-dependent pressure distributions on the trailing vehicle hood were predicted using the CFD software PowerFLOW. Subsequently, these pressure distributions were prescribed on an FEM model constructed with MSC Nastran. In some cases the wake shedding produced pressure fluctuations with frequency spectra near the free vibration modes of the hood structure, naturally leading to
relatively large vibratory response. In [5], the steady aeroelastic response of a Jaguar XK8 convertible car roof was predicted by coupling of the CFD software STAR-CD to a third-party FEM solver. The coupled procedure is said to provide displacement results within 20% of the uncoupled response. Ramsay et al. [6] predicted the static deflection of an automobile hood in an uncoupled manner using unspecified CFD and FEM solvers. The model did not include internal flow; however, pressure inlet/outlet boundary conditions were used on the front fascia openings to model the resistance provided by the engine compartment. Results indicated less than a 10% difference between the prediction and experimental results for displacement measured at two locations. Also, it was discovered that the externally mounted displacement measurement devices exhibited flow induced vibrations, leading to noisy data.

1.3 Objectives of this Thesis

This study is motivated by the need for a better understanding on aeroelastic interactions of automobiles, and specifically aeroelastic simulation of the hood. The goal is to assess the degree of aeroelastic coupling in a typical automotive hood, the importance of engine compartment flow, and also model validation. This is carried out through systematic development of coupled CFD-FEM for
simulation of the aeroelastic response of an automobile hood, and validated by comparisons with experimental data for hood surface pressures and deflections.

The specific objectives of this thesis are:

1. Develop high fidelity fluid and structural models that when coupled match the environment in which the experimental data was taken.

2. Compare the aeroelastic prediction to experimental measurements for validation.

3. Determine the sensitivity of the fluid-structure interaction response to internal flow through the engine compartment.

4. Determine the coupling requirements by comparing the uncoupled and coupled solutions.

The remainder of this thesis is arranged as follows: the experimental setup and uncertainty is detailed in Chapter 2; the computational configurations of the fluid and structural models, and the coupling procedure are described in Chapter 3; results and discussion are provided in Chapter 4; and the principal conclusions and suggested future work is given in Chapter 5.
1.4 Key Novel Contributions of this Thesis

The key novel contributions contained in this thesis are:

1. Determined on-vehicle measurement device attachment compliance and its impact on measurement uncertainty; and the influence the presence of the devices have on the local flow field.

2. Examined the sensitivity of the internal flow on the aeroelastic response of the hood. Specifically, the impact of the flow through the front fascia, and engine compartment pressure loading.

3. Determined a possible deflection threshold at which aeroelastic response is not significant for a typical automobile hood.
Chapter 2: Experimental Setup

Experimental results for this study were conducted in the Lockheed Martin Low Speed Wind Tunnel in Marietta, Georgia. The Low Speed Wind Tunnel is a single return, closed test section facility with a maximum wind speed of 320 kph. The dimensions of the test section are given in Table 1. The tunnel is equipped with *floor blowing*, a feature where a slot in the floor at the inlet of the test section inserts air at the equivalent dynamic pressure of the mean flow to eliminate the boundary layer. Floor blowing replicates an on-road aerodynamic environment and was used in the experiment.

<table>
<thead>
<tr>
<th>Dimension</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Height</td>
<td>4.95 m</td>
</tr>
<tr>
<td>Width</td>
<td>7.09 m</td>
</tr>
<tr>
<td>Length</td>
<td>13.1 m</td>
</tr>
</tbody>
</table>

The objective of the experiment was to record transverse hood displacement and surface pressure measurements of a full-scale vehicle at 100, 160, and 200 kph.
Displacement was recorded at three locations specified in Figure 1. Externally mounted lasers were used to measure the hood displacement, with an accuracy to within 5 microns. The lasers at Points 1 and 2 were placed in an airfoil-type enclosure to mitigate disturbances in the flow, and mounted on the fenders of the vehicle as shown in Figure 2. The laser at Point 3 was suctioned to the windshield and held by a rigid fixture as shown in Figure 3.

Figure 1: Displacement Measurement Locations
Surface pressure was measured on five strips of probes as shown in Figure 4. Strips 2–5 consisted of 20 probes, while strip 1 consisted of 18. In addition, a pitot-static tube was affixed to the right mirror to record a reference pressure in
the flow, as shown in Figure 5. The displacement and pressure data were taken separately so that the presence of the lasers would not affect the surface pressure measurements.

Figure 4: Surface Pressure Probe Locations

Figure 5: Pitot-Static Tube for Reference Pressure
A first order experimental uncertainty investigation was carried out to determine the reliability of the measurements, similar to the study conducted in Ref. [6]. Ref. [6] claimed the externally mounted triangular apparatus displayed flow-induced vibrations that led to noisy data. As a result, the airfoil shape was proposed to mitigate apparatus motion and was used in this study. The data acquisition signal over the sampling period for each measurement location is presented in Figure 6, showing small oscillations relative to the mean values. Complementary, the root-mean-square error (RMSE) indicates small fluctuation for each measurement and is listed in Table 2.

Figure 6: Data Acquisition Signal for Experimental Displacement Measurements
Table 2: RMSE of Experimental Displacement Measurements

<table>
<thead>
<tr>
<th>Location</th>
<th>Measurement (mm)</th>
<th>RMSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Point 1</td>
<td>0.68</td>
<td>0.050</td>
</tr>
<tr>
<td>Point 2</td>
<td>0.74</td>
<td>0.051</td>
</tr>
<tr>
<td>Point 3</td>
<td>-0.075</td>
<td>0.0054</td>
</tr>
</tbody>
</table>

However, although apparatus vibration was negligible, it was speculated that the steady wind load caused static deflection of the device. Compliance increases the experimental measurements of deflection at Points 1 and 2 as depicted in Figure 7. To quantify the static deflection, a test was conducted where the measurement apparatus was fixed into position, applied a known load, and the displacement was recorded. The load-displacement data and linear curve fit are shown in Figure 8. Estimation of the deflection of the lasers in the wind tunnel experiment was determined by computing the drag force on the laser geometry as modeled in the CFD domain. Following, the displacement corresponding to the drag load from the CFD solution was extracted using the linear curve fit in Figure 9. These values are set as the margin of uncertainty of all displacement measurements.
Figure 7: Description of Measurement Apparatus Compliance

Figure 8: Measurement Apparatus Load-Displacement Data with Linear Curve Fit
Figure 9: Apparatus Displacement due to Wind Load
Chapter 3: Computational Models and Coupling Procedure

Here, the fluid and structural models used in this study are described. For the fluid model, the various vehicle configurations, domain size, boundary conditions, flow modeling, and grid generation/convergence are presented. Following, a description of the structural model is given, including an explanation of the structural components and each component’s material, mesh size, element type, and boundary conditions. In addition, the structural model is validated against experimental data. Finally, the coupling procedure is described.

3.1 Fluid Model

3.1.1 Vehicle Configurations

Three separate vehicle configurations of the Acura 2015 TLX are considered for the present study. The first configuration resembles an initial design model or “styling” model. This configuration, shown in Figure 10, neglects the internal flow through the front fascia and has a simplified underbody and wheels. This vehicle geometry is denoted as V1.
The second configuration is a “complete” vehicle model, which includes all under-hood and underbody components shown in Figure 11. The radiator and condenser were modeled as porous media using inertial and viscous resistance values provided by the supplier, listed in Table 3. This vehicle geometry is denoted as V2.
Figure 11: V2 Model Geometry

<table>
<thead>
<tr>
<th></th>
<th>Inertial (kg/m^4)</th>
<th>Viscous (kg/m^3-s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radiator</td>
<td>91.62</td>
<td>901.86</td>
</tr>
<tr>
<td>Condenser</td>
<td>82.13</td>
<td>357.16</td>
</tr>
</tbody>
</table>

Due to the inability to obtain a coupled solution using the V2 model, a third configuration was considered using a simplified engine compartment and underbody as shown in Figure 12. As highlighted in Figure 13, the only underhood components retained are the radiator, condenser, front bumper support, chin spoiler and under-hood structure. Furthermore, the powertrain and exhaust
systems were completely removed from the underbody exposing the vehicle floor. This vehicle geometry is denoted as V3.

Figure 12: V3 Model Geometry

Figure 13: Isometric View of Remaining Under-hood Boundaries in V3
As indicated, aeroelastic computations were not achievable with V2. This is due to relatively poor grid quality around the complex internal topologies of the grid that yields negative volumes during grid morphing. Thus, only V1 and V3 are used for aeroelastic simulations. However, V2 provides the most accurate internal flow modeling and is used to provide a pressure boundary condition for the under-hood pressure.

3.1.2 Computational Domain and Boundary Conditions

For each vehicle configuration, a fluid domain is constructed using commercial CFD software STAR-CCM+. The boundary conditions and dimensions of the domain are shown in Figures 14 – 16. The inlet and outlet boundaries are specified as a velocity inlet and mass flow outlet, respectively. The mass flow outlet specifies the percentage of mass that flows through the boundary face, which for this setup is 100 percent. Using this boundary condition over the more traditional pressure outlet allows the user to specify the gauge pressure at a single \((x,y,z)\) location to anchor the pressure solution. This is set as the pressure measured by the probe positioned off the right mirror, shown in Figure 16. The top, bottom, and side walls are set to a slip wall boundary condition, while the vehicle surface boundaries are set to no-slip walls. The cross section of the fluid domains were set
to match the size of the Lockheed Martin Low Speed Wind Tunnel so that the blockage ratios are identical.

Figure 14: Fluid Domain Boundary Conditions and Dimensions, Front View

Figure 15: Fluid Domain Boundary Conditions and Dimensions, Side View

Figure 16: Pressure Boundary Condition Location
A challenge associated with constructing the size of the fluid domain is to determine the location of the inlet and outlet with respect to the vehicle. These boundaries should be positioned so that the presence of the boundary does not affect the flow solution. Therefore, a sensitivity study is carried out to determine the domain size. Four domains (D1, D2, D3, D4) with increasing inlet and outlet distances from the vehicle were created and shown in Figure 17.

![Diagram of domains with increasing inlet and outlet distances](image)

Figure 17: Domains with Increasing Inlet/Outlet Distances from Vehicle

To analyze the four domains, the velocity distributions along the set of probes indicated in Figure 18 are plotted in Figure 19. The freestream velocity for this study is 160 kph. The location of the probes is the same for each domain and
positioned on the inlet of D1. As the inlet and outlet distances increase, the inflow conditions approach a solution where the proximity has no impact on the flow solution. Since D3 and D4 give the same solution, the smaller of the two is used for the aerodynamic modeling so as to reduce computational cost.

Figure 18: Probes along Inlet of D1 (0 < z < 4.95 m)

Figure 19: Velocity Distribution of Each Domain (D1-D4)
3.1.3 Flow Modeling

The fluid domain is modeled by solving the incompressible Reynolds Averaged Navier-Stokes equations using STAR-CCM+. The Realizable K-Epsilon Two Layer model is selected as the turbulence model, a common approach when predicting external aerodynamics of ground vehicles [7-9]. The continuity, momentum and turbulence closure equations are discretized to second order accuracy. Regarding the turbulence model, the Realizable distinction allows a coefficient of the model to be expressed in terms of mean flow and turbulence properties rather than held constant, and is consistent with experimental observations in boundary layers [10]. In addition, the Two Layer distinction gives a more accurate prediction of the boundary layer when $y+$ values fall in the buffer layer ($1 < y+ < 30$), where boundary layer flow solutions typically exhibit inaccuracies [10]. This commonly occurs near stagnation regions in bluff body flows. A discussion of the wall $y+$ values for each vehicle configuration is located in Appendix A.

3.1.4 Grid Generation and Convergence

A grid of the outer domain is created for each vehicle configuration (V1, V2, V3) by surface wrapping to create a watertight geometry. This is then followed by surface and volume meshing. Each boundary on the vehicle has a surface size ranging from 2.5 – 10 millimeters (mm) depending on the geometric complexity
and location of the part. The volume mesh is composed of two types of cells: prism layers and trimmed cells. Prism layers are the first cells off the wall used to capture the boundary layer. The first cell height is calculated so that the wall $y+$ values fall within the log-law range ($30 < y+ < 300$). Three rectangular zones are created to locally refine the mesh around the vehicle. The cell sizes of the zones, shown in Figure 20, are 10, 20 and 40 mm. A nearfield top view of the floor and a planar slice ($y = 0$) in the streamwise direction are provided in Figures 20 and 21, respectively. Planar slices of the computational domain through the engine compartment for each vehicle configuration are shown in Figure 22. The under-hood region falls within the 10 mm refinement zone, and the grid size continues to decrease closer to the vehicle surface to capture the boundary layer. The total cell count for each configuration is presented in Table 4.

![Figure 20: Top View of Volume Mesh on Floor of Fluid Domain](image)

Figure 20: Top View of Volume Mesh on Floor of Fluid Domain
Mesh convergence is analyzed at a freestream velocity of 200 kph. V2 is used to confirm mesh convergence since it has the most complexity. The integrated lift force of the hood is used to determine convergence where percent error is measured against the finest grid. Results of the study are summarized in Table 5.
Additionally, discrete pressures on the hood are compared. The location of these pressures are indicated in Figure 23, and the comparison is given in Figures 24 – 26. The medium grid is considered converged with a percent error of 1.26% for the hood lift value and maximum percent error of 4.1% for the discrete pressure comparison. This grid is selected as the best balance between accuracy and modeling resources.

Table 5: Summary of the Grid Convergence Study

<table>
<thead>
<tr>
<th>Grid</th>
<th>Cell Count</th>
<th>Hood Lift (Newton)</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>60M</td>
<td>248.3</td>
<td>4.87</td>
</tr>
<tr>
<td>Medium</td>
<td>86M</td>
<td>257.7</td>
<td>1.26</td>
</tr>
<tr>
<td>Fine</td>
<td>105M</td>
<td>261</td>
<td></td>
</tr>
</tbody>
</table>

Figure 23: Discrete Pressure Probing Locations for Grid Convergence Study
Figure 24: Pressure Comparison along Probe Set 1

Figure 25: Pressure Comparison along Probe Set 2
3.2 Structural Model

The structural model, provided by Honda R&D Americas, Inc., is solved using the commercial FEM software Abaqus Standard. The model is an assembly of several structural components and accounts for geometric nonlinearity. Each component, material, mesh size, and element type is listed in Table 6. The mesh size for each component is 4 mm. All materials are linear and modeled using shell elements. The structural assembly is shown in Figure 27. The frame is the load-bearing component of the structure and is attached to each component. The skin is attached to the frame by a mastic material. The mastic interaction with the skin...
and the frame was solved as a contact problem using the penalty method. The latch and hinges are bolted to the frame. In addition, these components attach to the vehicle, where the boundary conditions for the latch and hinges are listed in Table 7, and shown in Figure 28. Both components are constrained in translation, but free to rotate about any axis.

![Diagram of Hood Structural Assembly](image)

**Figure 27: Hood Structural Assembly**

**Table 6: Hood Structural Components**

<table>
<thead>
<tr>
<th>Component</th>
<th>Material</th>
<th>Mesh Size</th>
<th>Element Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Skin</td>
<td>Aluminum</td>
<td>4 mm</td>
<td>Shell</td>
</tr>
<tr>
<td>Frame</td>
<td>Aluminum</td>
<td>4 mm</td>
<td>Shell</td>
</tr>
<tr>
<td>Hinge</td>
<td>Steel</td>
<td>4 mm</td>
<td>Shell</td>
</tr>
<tr>
<td>Latch</td>
<td>Steel</td>
<td>4 mm</td>
<td>Shell</td>
</tr>
</tbody>
</table>
Table 7: Structural Model Boundary Conditions

<table>
<thead>
<tr>
<th>Component</th>
<th>Constrained Degrees of Freedom</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hinge</td>
<td>u, v, w</td>
</tr>
<tr>
<td>Latch</td>
<td>u, v, w</td>
</tr>
</tbody>
</table>

The structural model is validated by a hood sag test. In the experiment, the hood is constrained at the hinges and one of the front corners near the headlight, while the structure deforms under gravitational loading. The same boundary conditions and loading are applied to the structural model, and the displacement is measured at the free corner. The accuracy is given in Table 8.
Table 8: Structural Model Validation by Hood Sag Test

<table>
<thead>
<tr>
<th></th>
<th>Displacement (mm)</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>20.0</td>
<td>10.0</td>
</tr>
<tr>
<td>Simulation</td>
<td>22.0</td>
<td></td>
</tr>
</tbody>
</table>

3.3 Coupling Procedure

STAR-CCM+ and Abaqus use a native co-simulation engine (CSE) to couple the domains. The CSE follows a partitioned approach where the fluid and structure response are computed using separate solvers and subsequently coupled through an exchange of boundary conditions at the interface of the domains. The fluid-structure interaction (FSI) workflow is shown in Figure 29. First, the fluid equations are solved to compute the static pressure. Next, the fluid load is mapped onto the FEM mesh by integrating over the surface face and interpolating to the surface grid points in a least squares sense. The structural equations are solved, and the displacement field is mapped to the CFD mesh by interpolating using the structural elemental shape functions. Last, the CFD mesh is morphed using multiquadric-biharmonic method to define the motion of the vertices and create a new hood shape. This process is iterated until the steady aeroelastic response is computed. The solution is considered sufficiently converged when the change in displacement at the measurement locations between successive time steps is less than 0.001 mm.
Despite the fact that the problem considered is steady-state in nature, the CSE is implemented by STAR-CCM+ in time-accurate mode. Thus the steady-state aeroelastic response is computed using a time step of 0.1 seconds. Convergence to the steady-state solution is accelerated by implementing critical Rayleigh damping. Subiterations were used to converge each fluid time step, where convergence is defined by when a change in hood lift force between successive time steps is less than 0.1 Newtons. An explicit coupling scheme, commonly used for weakly coupled problems, was implemented so the boundary conditions were exchanged once per time step. The scheme is staggered such that one solver leads and the other solver lags. The exchange of information over one time step for the explicit scheme is shown in Figure 30. For this study, the structure led and the
fluid lagged. Note that the steady-state flow solution of the rigid vehicle was used as the initial condition to the coupled simulation.

As noted earlier, one of the challenging aspects of the aeroelastic simulation is morphing the CFD grid to accommodate structural deformation. This is an issue for complex topologies with associated poor cell quality that are susceptible to the appearance of negative volumes during mesh deformation. This is also a challenge for two structures in close proximity to each other. For this work, the engine compartment mesh for configuration V2 experiences negative volumes during aeroelastic simulation. Furthermore, the hood skin and frame are close in proximity, and morphing these boundaries simultaneously leads to the appearance of negative volumes. The mesh resolution required to morph both
components is impractical for simulation. As a result, the coupling procedure
described herein is only applied to configurations V1 and V3 with the hood skin
as the lone boundary deformed.
Chapter 4: Results

4.1 Comparison Between Numerical and Experimental Results

Results are obtained for operating speeds of 100, 160, and 200 kph. Fluid properties are specified to be consistent with that of the experiment, shown in Table 9. Configuration V3 is used to compute the FSI baseline prediction. The steady-state internal pressure distribution of V2 is applied to the skin bottom, and frame top and bottom, shown in Figure 31. The skin bottom and frame top essentially have constant negative pressure distributions. The frame bottom is predominately negative excluding positive pressure regions near the cowl top and engine cooling aperture (flow stagnation zones).

<table>
<thead>
<tr>
<th>Velocity (kph)</th>
<th>Density (kg/m³)</th>
<th>Pressure (Pa)</th>
<th>Dynamic Viscosity (kg/m·s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>1.127</td>
<td>97986.5</td>
<td>1.762E-05</td>
</tr>
<tr>
<td>160</td>
<td>1.124</td>
<td>97886.9</td>
<td>1.761E-05</td>
</tr>
<tr>
<td>200</td>
<td>1.120</td>
<td>97797.7</td>
<td>1.760E-05</td>
</tr>
</tbody>
</table>
Comparisons of the hood surface pressure are shown in Figures 32 – 46. For each speed, the pressure data along strips 1 – 5 are shown for the experiment, FSI prediction, and rigid hood. The data for the 100, 160, and 200 kph cases are shown in Figures 32 – 36, Figures 37 – 41, and Figures 42 – 46, respectively. The FSI prediction and rigid CFD pressures are nearly identical, indicating that the pressure at the measured locations is not strongly sensitive to fluid-structural coupling. The simulation captures the overall trend of the data, but consistently overshoots the pressure on the trailing edge. $L_1$ (mean absolute error) and $L_\infty$ norms for each velocity are provided in Table 10 using the data from all probe locations. The agreement between predictions and experiment decreases with increasing wind speed. Overall, the results indicate reasonable agreement between the experiment and prediction.

The hood deflection results of Points 1, 2, and 3 are provided in Figures 47 – 49. As noted earlier, post-analysis of the experimental results indicated deflection
of the laser measurement devices at Points 1 and 2 due to aerodynamic force. Subsequently, bench testing of the laser with attachment was used to correlate the aerodynamic loading at different wind speeds with laser deflection. The wide uncertainty bars in the experimental results account for this deflection. This was not observed to be an issue at Point 3. Thus no error bars are included since the measurement uncertainty of the laser itself is smaller than the symbol used in Figures 47 – 49. In general, the predicted displacements are reasonably close to within experimental uncertainty of the measured displacements. Furthermore, the maximum displacements are $O(1\text{mm})$ or less, which is consistent with the pressure comparisons, suggests that this hood structure is not strongly susceptible to aeroelastic interactions.
Figure 32: Discrete Pressure Comparison along Strip 1, 100 kph

Figure 33: Discrete Pressure Comparison along Strip 2, 100 kph
Figure 34: Discrete Pressure Comparison along Strip 3, 100 kph

Figure 35: Discrete Pressure Comparison along Strip 4, 100 kph
Figure 36: Discrete Pressure Comparison along Strip 5, 100 kph

Figure 37: Discrete Pressure Comparison along Strip 1, 160 kph
Figure 38: Discrete Pressure Comparison along Strip 2, 160 kph

Figure 39: Discrete Pressure Comparison along Strip 3, 160 kph
Figure 40: Discrete Pressure Comparison along Strip 4, 160 kph

Figure 41: Discrete Pressure Comparison along Strip 5, 160 kph
Figure 42: Discrete Pressure Comparison along Strip 1, 200 kph

Figure 43: Discrete Pressure Comparison along Strip 2, 200 kph
Figure 44: Discrete Pressure Comparison along Strip 3, 200 kph

Figure 45: Discrete Pressure Comparison along Strip 4, 200 kph
Figure 46: Discrete Pressure Comparison along Strip 5, 200 kph

Table 10: $L_2$ and $L_\infty$ Error Norms of Hood Surface Pressure Data using FSI Prediction Pressures

<table>
<thead>
<tr>
<th>Velocity (kph)</th>
<th>$L_\infty$ (Pa)</th>
<th>$L_1$ (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>30.8</td>
<td>9.97</td>
</tr>
<tr>
<td>160</td>
<td>70.9</td>
<td>23.70</td>
</tr>
<tr>
<td>200</td>
<td>123.9</td>
<td>36.13</td>
</tr>
</tbody>
</table>
Figure 47: Displacement Comparison of FSI Prediction to Experiment, 100 kph

Figure 48: Displacement Comparison of FSI Prediction to Experiment, 160 kph
A potential cause of discrepancy between the predictions and experimental measurements is the alteration of the local surface pressure due to the presence of the laser devices. This effect is examined by adding the laser devices into the CFD domain, as shown in Figure 50, and repeating the coupled analysis using V3 at 160 kph. As indicated in Figure 51, there are significant local changes in surface pressure near the measurement locations. However, as indicated by the results listed in Table 11, these local pressure changes have a negligible effect on the predicted displacement at Points 1 and 2, and a modest improvement on the predicted displacement at Point 3. This is likely due to the relative stiffness of the considered hood. Thus, aeroelastic analysis of more flexible configurations may
exhibit stronger sensitivity to these local pressure variations, making this an important consideration in general.

Figure 50: CFD Modeling of Laser Instrumentation

Figure 51: Pressure Distribution of V3 With and Without Lasers
Table 11: Coupled Displacement Results of V3 With and Without Lasers

<table>
<thead>
<tr>
<th>Location</th>
<th>With Lasers</th>
<th>Baseline Prediction</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Point 1</td>
<td>0.46</td>
<td>0.47</td>
<td>2.2</td>
</tr>
<tr>
<td>Point 2</td>
<td>0.44</td>
<td>0.44</td>
<td>0.0</td>
</tr>
<tr>
<td>Point 3</td>
<td>-0.025</td>
<td>-0.012</td>
<td>52.0</td>
</tr>
</tbody>
</table>

The degree of fluid-structural coupling is assessed by comparing the uncoupled and coupled structural response at 160 kph. Here, V3 is used for the uncoupled analysis, and compared to the baseline prediction discussed above. The results are shown in Figure 52 and Table 12. As expected from the above results, the difference between the two predictions is relatively small. This is indicative of a relatively stiff hood construction for this vehicle. Furthermore, these results suggest weak aeroelastic coupling for hood deflections of 1 mm or less predicted using an uncoupled analysis.
Figure 52: Uncoupled vs. Coupled Structural Response

Table 12: Comparison of Uncoupled and Coupled Response

<table>
<thead>
<tr>
<th>Location</th>
<th>Uncoupled</th>
<th>Baseline</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Point 1</td>
<td>0.45</td>
<td>0.47</td>
<td>4.26</td>
</tr>
<tr>
<td>Point 2</td>
<td>0.43</td>
<td>0.44</td>
<td>2.27</td>
</tr>
<tr>
<td>Point 3</td>
<td>-0.0098</td>
<td>-0.012</td>
<td>18.3</td>
</tr>
</tbody>
</table>

4.2 Sensitivity to Internal Flow

Aeroelastic predictions using V1 are computed at 160 kph for both under-hood pressure included and neglected. This provides insight into the sensitivity of the aeroelastic response to flow through the front fascia and the resulting engine compartment pressure.

The impact of flow through the front fascia on the aeroelastic predictions is assessed comparing the V1 aeroelastic prediction, which has a closed front fascia,
to that of using V3. Similar to the V3 prediction, the internal engine compartment pressure computed using V2 is applied onto the FEM model. The effect of closing the front fascia on the exterior hood skin pressure is shown in Figure 53. The largest differences occur at the front and trailing edge regions of the hood. A larger suction on the front middle and fender regions is observed for V1. This is due to the closed apertures on the front fascia accelerating the flow over the hood. Conversely, the pressure on the trailing edge of V1 exceeds that of V3. This is due to flow through the cowl in V3 compared to the closed boundary in V1. These differences are highlighted in Figure 54. The impact on the structural response is provided in Figure 55 and Table 13. There is a small difference on Points 1 and 2, while the displacement for Point 3 of V1 is nearly five times that of V3. However, note that the displacement at Point 3 remains relatively small compared to the other locations.
Figure 53: Steady Aeroelastic Pressure Distribution of V1 and V3

Figure 54: Cowl Top Geometry of V1 and V3
Table 13: Coupled Displacement Results of V1 and V3

<table>
<thead>
<tr>
<th>Location</th>
<th>V1</th>
<th>V3</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Point 1</td>
<td>0.45</td>
<td>0.47</td>
<td>4.26</td>
</tr>
<tr>
<td>Point 2</td>
<td>0.43</td>
<td>0.44</td>
<td>2.27</td>
</tr>
<tr>
<td>Point 3</td>
<td>-0.054</td>
<td>-0.012</td>
<td>350</td>
</tr>
</tbody>
</table>

The impact of neglecting under-hood pressure on the aeroelastic predictions is considered in Figure 56 and listed in Table 14. The neglect of under-hood pressure tends to increase the displacement overall. This is due to the suction force induced by the negative engine compartment pressure, which tends to resist positive hood displacement.
Figure 56: Steady Aeroelastic Structural Response of V1 With and Without Internal Pressure Loading

Table 14: Coupled Displacement Results of V1 With and Without Internal Pressure Loading

<table>
<thead>
<tr>
<th>Location</th>
<th>With Internal Pressure</th>
<th>Without Internal Pressure</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Point 1</td>
<td>0.45</td>
<td>0.51</td>
<td>11.8</td>
</tr>
<tr>
<td>Point 2</td>
<td>0.43</td>
<td>0.52</td>
<td>17.3</td>
</tr>
<tr>
<td>Point 3</td>
<td>-0.054</td>
<td>-0.16</td>
<td>66.3</td>
</tr>
</tbody>
</table>
Aeroelastic simulation in automobile development and design is an important consideration as manufacturers vary component compliance to meet increasingly challenging, and sometimes conflicting objectives. Critical to this are the development of computational tools, as well as validation of these tools. This thesis examines this in the context of the aeroelastic response of a representative automobile hood using both a coupled CFD-FEM fluid-structure interaction framework and experimental measurement. Specifically, comparisons of the experimental and numerical results have been presented. In addition, the effect of the measurement apparatuses in the flow, and the experimental uncertainty due to compliance of these devices was assessed. Next, the degree of fluid-structure coupling was considered. Last, the sensitivity of the aeroelastic prediction to internal flow through the engine compartment was examined.

5.1 Principal Conclusions Obtained in this Study

The studies in this work provide several useful conclusions:
1. Poor cell quality around complex topologies, and morphing multiple structures in close proximity are susceptible to the appearance of negative volume cells during FSI simulation. This is currently a major issue for high fidelity representation of the vehicle during aeroelastic simulation.

2. Overall agreement between the experiment and aeroelastic predictions is reasonable, with agreement increasing with increasing wind speed.

3. On-vehicle laser measurement devices for displacement have a significant impact on local flow behavior, and are susceptible to aeroelastic interactions due to compliance of the attachment. This can have a significant impact on measurement uncertainty.

4. Results in this study indicate that hood deflections of 1 mm or less, as predicted from a preliminary uncoupled analysis, correspond to a hood structure with negligible aeroelastic interactions.

5. The internal flow and pressure loading have a modest impact in capturing the fluid-structural response, predominately on the trailing edge of the hood for the considered vehicle.

### 5.2 Recommendations for Future Research

The steady aeroelastic results presented in this study were computed on simplified geometries (V1 and V3) that exhibited acceptable cell quality to morph in the
coupling procedure. Therefore, an FSI solution has not been obtained on the complete geometry, V2. Investigations into the mesh morphing issues should be considered.

A comprehensive investigation of experimental uncertainty must be conducted. This should include extensive experimental data of measurement apparatus deflection at specific orientations and operating conditions common in on-road testing. Additionally, non-intrusive diagnostic tools should be pursued to mitigate experimental uncertainty.

The conclusions relating to the internal flow/pressure and degree of coupling are specific to this vehicle model, which did not exhibit strong aeroelastic interactions. This work should be carried out on a vehicle with more substantial hood compliance to further assess the importance of aeroelastic interactions, and the reliability of the predictive tool and gathering of validation data.

A higher fidelity structural model should be considered. Rubber seals contained in the structural model have been shown to add stiffness to the hood and may change the aeroelastic prediction.

Last, the unsteady aeroelastic response of the hood to upstream vehicle wake shedding should be considered. Several incoming flow conditions may be considered such as a turbulent wake generated by a preceding vehicle while varying separation and offset distances. The mutual interaction between the unsteady fluid and structural dynamics should be assessed to see what conditions,
if any, aeroelastic interactions alter the response from uncoupled application of wake induced pressure fluctuations on a trailing vehicle structural model.
Appendix A: Wall $y^+$

To ensure accurate modeling of the turbulent boundary layer, it is suggested that $y^+$ values fall in the viscous sublayer ($y^+ < 1$) or in the log-law region ($30 < y^+ < 300$). When $y^+$ values fall in the viscous sublayer, the exact solution of the boundary layer is solved. For the log-law region, the flow solution is calculated using a logarithmic function dependent on distance from the wall and is much less computationally expensive. For this study, the log-law approach was applied. Wall $y^+$ distributions for configurations V1, V2 and V3 are shown in Figures 57, 58, and 59, respectively. The Realizable Two Layer K-Epsilon turbulence model produces more accurate boundary layer flow solutions when $y^+$ values fall in the buffer region ($1 < y^+ < 30$), unavoidably found near flow stagnation zones.
Figure 57: Wall $y^+$ Distribution of V1

Figure 58: Wall $y^+$ Distribution of V2
Figure 59: Wall $y^+$ Distribution of V3
Appendix B: CFD and Uncoupled FSI Results of V2

The CFD pressure and uncoupled structural response results of configuration V2 are presented and compared to V1 and V3. The rigid CFD pressure is plotted in Figures 60 – 64 along strips 1 – 5. In addition, the uncoupled displacement results are given in Table 15 at Points 1, 2 and 3. The hood skin pressure of V2 and V3 agree very well, implying that the reduced complexity does not have a large effect on the exterior flow solution. In addition, the uncoupled structural response of V2 and V3 are similar.

Figure 60: Discrete Pressure Comparison of V1, V2, V3 along Strip 1
Figure 61: Discrete Pressure Comparison of V1, V2, V3 along Strip 2

Figure 62: Discrete Pressure Comparison of V1, V2, V3 along Strip 3
Figure 63: Discrete Pressure Comparison of V1, V2, V3 along Strip 4

Figure 64: Discrete Pressure Comparison of V1, V2, V3 along Strip 5
Table 15: Uncoupled Displacement Results of V1, V2, V3

<table>
<thead>
<tr>
<th>Location</th>
<th>V1</th>
<th>V2</th>
<th>V3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Point 1</td>
<td>0.44</td>
<td>0.50</td>
<td>0.45</td>
</tr>
<tr>
<td>Point 2</td>
<td>0.43</td>
<td>0.48</td>
<td>0.43</td>
</tr>
<tr>
<td>Point 3</td>
<td>-0.054</td>
<td>-0.0022</td>
<td>-0.0098</td>
</tr>
</tbody>
</table>
Bibliography


