5-2016

2D Aeroacoustic Analysis of Flow in the Flame Trench

Meghan Pokorski

Follow this and additional works at: https://commons.erau.edu/edt

Part of the Aerodynamics and Fluid Mechanics Commons, and the Aeronautical Vehicles Commons

Scholarly Commons Citation
https://commons.erau.edu/edt/232

This Thesis - Open Access is brought to you for free and open access by Scholarly Commons. It has been accepted for inclusion in Dissertations and Theses by an authorized administrator of Scholarly Commons. For more information, please contact commons@erau.edu.
2D AEROACOUSTIC ANALYSIS OF FLOW IN THE FLAME TRENCH

A Thesis

Submitted to the Faculty

of

Embry-Riddle Aeronautical University

by

Meghan Pokorski

In Partial Fulfillment of the

Requirements for the Degree

of

Master of Science in Aerospace Engineering

May 2016

Embry-Riddle Aeronautical University

Daytona Beach, Florida
2D AEROACOUSTIC ANALYSIS OF FLOW IN THE FLAME TRENCH

by

Meghan Pokorski

A Thesis prepared under the direction of the candidate's committee chairman, Dr. Reda Mankbadi, Department of Aerospace Engineering, and has been approved by the members of the thesis committee. It was submitted to the School of Graduate Studies and Research and was accepted in partial fulfillment of the requirements for the degree of Master of Science in Aerospace Engineering.

THEESIS COMMITTEE

Chairman, Dr. Reda Mankbadi

Member, Dr. Anastasios Lyrintzis

Vladimir Golubev

Member, Dr. Valdimir Golubev

Member, Dr. Bruce Vu

Department Chair, Dr. Anastasios Lyrintzis

Dean of College of Engineering, Dr. Mohammadimi

Vice Chancellor, Dr. Christopher Grant

4/27/16

5/3/16

5/3/16
ACKNOWLEDGMENTS

I would like to thank Dr. Bruce Vu and David Chesnutt of NASA Kennedy Space Center for all of their help. I would also like to thank my family and Denean Kelson for their support. Lastly, I would like to thank my committee for their guidance.
# TABLE OF CONTENTS

TABLE OF CONTENTS........................................................................................................ iv
LIST OF TABLES.................................................................................................................. v
LIST OF FIGURES................................................................................................................ vi
SYMBOLS........................................................................................................................ ix
ABBREVIATIONS............................................................................................................. x
NOMENCLATURE................................................................................................................ xi
ABSTRACT.......................................................................................................................... xii

1. Introduction...................................................................................................................... 1
   1.1. Motivation................................................................................................................ 1
   1.2. Previous Impinging Jet Modeling ......................................................................... 1
   1.3. Objective ................................................................................................................ 4

2. Numerical Models............................................................................................................ 6
   2.1. Geometry................................................................................................................ 6
   2.2. Computational Mesh ............................................................................................ 7
       2.2.1. Grid Study...................................................................................................... 9
   2.3. Fluent Models......................................................................................................... 11
   2.4. Ffowcs Williams-Hawkings Methods .................................................................. 13

3. Benchmark Experiment .................................................................................................. 16

4. Results............................................................................................................................. 17
   4.1. Steady Base Flow.................................................................................................... 18
       4.1.1. Laminar Flow............................................................................................... 18
       4.1.2. Turbulent Flow......................................................................................... 21
   4.2. Unsteady Flow....................................................................................................... 24
       4.2.1. Flow Behavior........................................................................................... 29
       4.2.2. Flow Snapshots......................................................................................... 43
   4.3. Acoustic Field......................................................................................................... 55
       4.3.1. Frequency at Ffowcs Williams-Hawkings Surfaces.................................... 55
       4.3.2. Acoustic Locations.................................................................................... 73
       4.3.3. Acoustic Data Comparison....................................................................... 73

5. Conclusion....................................................................................................................... 80

6. Recommendations......................................................................................................... 81

REFERENCES...................................................................................................................... 82
LIST OF TABLES

Table 2-1 Geometry .................................................................................................................. 6
Table 2-2 Oscillation Resolution Verification ........................................................................... 13
Table 3-1 Ffowcs Williams-Hawkings Surface Probes ............................................................... 29
Table 3-2 Receiver Locations ................................................................................................... 73
LIST OF FIGURES

Figure 1-1 Ring and Wall Ffowcs-Williams Hawkings Surfaces (Mankbadi, et al., 2016) 2
Figure 1-2 Instantaneous Mach Distributions (Brehm, et al., 2013) ........................................ 4
Figure 2-1 Labeled Geometry ........................................................................................................ 6
Figure 2-2 Mesh in the Domain of Interest ..................................................................................... 7
Figure 2-3 Mesh on the Flame Deflector ......................................................................................... 8
Figure 2-4 Full Mesh ..................................................................................................................... 8
Figure 2-5 Coarse Grid (above) and Fine Grid (below) .............................................................. 9
Figure 2-6 Transition Coarse Grid (left) and Fine Grid (right) .................................................. 10
Figure 2-7 Velocity Magnitude (m/s) Coarse Grid (top) and Fine Grid (bottom) .............. 10
Figure 3-1 Experimental Microphone Locations (Vu & Harris, 2015) .................................... 18
Figure 3-2 Laminar Flow Velocity Magnitude (m/s) ................................................................ 19
Figure 3-3 Laminar Vorticity Magnitude (1/s) ........................................................................... 19
Figure 3-4 Laminar Centerline Velocity ..................................................................................... 20
Figure 3-5 Turbulent Flow Velocity Magnitude (m/s) ............................................................... 21
Figure 3-6 Turbulent Vorticity Magnitude (1/s) ....................................................................... 22
Figure 3-7 Ffowcs Williams-Hawkings Porous Surfaces (red) .............................................. 23
Figure 3-8 Turbulent Centerline Velocity ................................................................................... 23
Figure 3-9 Base Case Residuals for Iterations 0-1000, 110000-114550, and 877500-882250 .......................................................................................................................... 25
Figure 3-10 11 Hz Case Residuals for Iterations 0-1000, 105000-110000, and 875000-880000 .......................................................................................................................... 27
Figure 3-11 110 Hz Case Residuals for Iterations 0-1000, 105000-110000, and 875000-880000 ....................................................................................................................... 28
Figure 3-13 Ffowcs Williams-Hawkings Probe Locations .......................................................... 29
Figure 3-16 Nozzle Exit Total Pressure (Pa) ............................................................................... 31
Figure 3-17 Close View of Nozzle Exit Total Pressure (Pa) ...................................................... 32
Figure 3-18 Right Surface Total Pressure (Pa) ............................................................................ 34
Figure 3-19 Point 3 Right Surface Total Pressure (Pa) ............................................................... 35
Figure 3-20 Point 4 Right Surface Total Pressure (Pa) ............................................................... 36
Figure 3-21 Point 5 Right Surface Total Pressure (Pa) ............................................................... 37
Figure 3-22 Left Surface Total Pressure (Pa) ............................................................................. 39
Figure 3-23 Point 6 Left Surface Total Pressure (Pa) ........................................ 40
Figure 3-24 Point 7 Left Surface Total Pressure (Pa) ........................................ 41
Figure 3-25 Point 8 Left Surface Total Pressure (Pa) ........................................ 42
Figure 3-26 Base Case Density Contours (kg/m³) .............................................. 43
Figure 3-27 Base Case Static Pressure Contours (Pa) ........................................ 43
Figure 3-28 Base Case Velocity Magnitude (m/s) .............................................. 44
Figure 3-29 Base Case Mach Number Contours .............................................. 44
Figure 3-30 Base Case Vorticity Magnitude Contours (1/s) .............................. 45
Figure 3-31 Base Case Vorticity Contours (1/s) at Four Time Steps ............... 46
Figure 3-32 11 Hz Density Contours (kg/m³) .................................................... 47
Figure 3-33 Static Pressure Contours (Pa) ....................................................... 47
Figure 3-34 Velocity Magnitude Contours (m/s) .............................................. 48
Figure 3-35 Mach Number Contours ............................................................. 49
Figure 3-36 Vorticity Magnitude Contours (1/s) .............................................. 49
Figure 3-37 Dilatation Field .............................................................................. 50
Figure 3-38 110 Hz Density Contours (kg/m³) ................................................... 51
Figure 3-39 110 Hz Static Pressure Contours (Pa) ............................................ 51
Figure 3-40 110 Hz Velocity Magnitude Contours (m/s) ................................. 52
Figure 3-41 110 Hz Mach Number Contours .................................................. 53
Figure 3-42 110 Hz Vorticity Magnitude Contours (1/s) ................................. 53
Figure 3-43 110 Hz Dilatation Field Contours ................................................ 54
Figure 3-44 110 Hz Close View of Dilatation Field ........................................ 54
Figure 3-45 FFT for Static Pressure at Point 1 ................................................ 56
Figure 3-46 FFT for Total Pressure at Point 1 ................................................ 57
Figure 3-47 FFT for Static Pressure at Point 2 ................................................ 58
Figure 3-48 FFT for Total Pressure at Point 2 ................................................ 59
Figure 3-49 FFT for Static Pressure at Point 3 ................................................ 60
Figure 3-50 FFT for Total Pressure at Point 3 ................................................ 61
Figure 3-51 FFT for Static Pressure at Point 4 ................................................ 62
Figure 3-52 FFT for Total Pressure at Point 4 ................................................ 63
Figure 3-53 FFT for Static Pressure at Point 5 ................................................ 64
Figure 3-54 FFT for Total Pressure at Point 5 ................................................ 66
Figure 3-55 FFT for Static Pressure at Point 6 ................................................ 67
Figure 3-56 FFT for Total Pressure at Point 6 ................................................................. 68
Figure 3-57 FFT for Static Pressure at Point 7 ................................................................. 69
Figure 3-58 FFT for Total Pressure at Point 7 ................................................................. 70
Figure 3-59 FFT for Static Pressure at Point 8 ................................................................. 71
Figure 3-60 FFT for Total Pressure at Point 8 ................................................................. 72
Figure 3-61 SPL (dB) vs. 1/3 Octave Frequency (Hz) (Vu & Harris, 2015) ..................... 74
Figure 3-62 SPL (dB) vs 1/3 Octave Band Frequency (Hz) Receiver 1 ......................... 74
Figure 3-63 SPL (dB) vs 1/3 Octave Band Frequency (Hz) Receiver 2 ......................... 75
Figure 3-64 SPL (dB) vs 1/3 Octave Band Frequency (Hz) Receiver 3 ......................... 76
Figure 3-65 SPL (dB) vs 1/3 Octave Band Frequency (Hz) Receiver 4 ......................... 76
Figure 3-66 SPL (dB) vs 1/3 Octave Band Frequency (Hz) Receiver 6 ......................... 77
Figure 3-67 SPL (dB) vs 1/3 Octave Band Frequency (Hz) Receiver 7 ......................... 77
Figure 3-68 SPL (dB) vs 1/3 Octave Band Frequency (Hz) Receiver 8 ......................... 78
Figure 3-69 OASPL Polar Plot ......................................................................................... 79
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>dB</td>
<td>Decibels</td>
</tr>
<tr>
<td>Hz</td>
<td>Hertz</td>
</tr>
<tr>
<td>m</td>
<td>Meters</td>
</tr>
<tr>
<td>Pa</td>
<td>Pascals</td>
</tr>
<tr>
<td>s</td>
<td>Seconds</td>
</tr>
<tr>
<td>Abbreviation</td>
<td>Description</td>
</tr>
<tr>
<td>--------------</td>
<td>------------------------------------</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>FWH</td>
<td>Ffowcs Williams-Hawkings</td>
</tr>
<tr>
<td>SLS</td>
<td>Space Launch System</td>
</tr>
</tbody>
</table>
### NOMENCLATURE

- $a_0$: far-field speed of sound  
- $\delta(f)$: Dirac delta function  
- $f$: frequency  
- $H(f)$: Heavside function  
- $P$: total gage pressure  
- $p'$: sound pressure at the far field  
- $P_j$: total gage pressure, steady  
- $P_{ij}$: compressive stress tensor  
- $St$: Strouhal number  
- $t$: time  
- $T$: total temperature  
- $T_j$: total temperature, steady  
- $T_{ij}$: Lighthill stress tensor  
- $u_i$: fluid velocity component in the $x_i$ direction  
- $u_{jet}$: jet velocity, steady  
- $u_n$: fluid velocity component normal to the surface $f=0$  
- $v_i$: surface velocity component in the $x_i$ direction  
- $v_n$: surface velocity component normal to the surface  
- $\rho$: density at the surface  
- $\rho_0$: far-field density
ABSTRACT

We present here a methodology for using the commercial software ANSYS-FLUENT to predict the acoustic field associated with Space-Launch System (SLS). We consider a two-dimensional model of flame deflector, and flame trench. The ANSYS code is then used to simulate the internal flow. Both the steady state case is considered along with two other cases where the inflow has a harmonic component. A Ffowcs-Williams Hawking (FWH) surface is then constructed within the computational domain to use the computed flow fluctuations to obtain the acoustic field. The acoustic data was then compared to the experimental data.

When using the ANSYS code for this flow situation, the residuals did not converge but reached an asymptotic oscillatory state. The fluctuation in the residual is too large to consider the computed flow fluctuations to be the sound source. Therefore, the results should be interpreted with caution as a first-attempt preliminary results. However, the calculated sound field qualitatively resembles the experimental observations. This may support the need to address the conversion issue and the need to extend the approach to the realistic three-dimensional case.
1. Introduction

1.1. Motivation

Due to the variety of launch vehicles, both NASA developed and commercial, going to be launched in the future, experimental testing for each rocket size and configuration is unreasonable. Computational Fluid Dynamics (CFD) predictions of the launch environment are necessary to determine the safety of these future launch vehicles. Predictions of the temperature and pressure environments are in development (Brehm, et al., 2013) but predictions of the acoustic environment are needed as well. This paper will focus on the far field acoustics due to the ignition of the liquid rockets on the Space Launch System (SLS).

1.2. Previous Impinging Jet Modeling

There have been several CFD computations of the acoustic field resulting from impinging jets (Mankbadi, et al., 2016), (Plotkin, Sutherland, & Vu, 2009), (Sipatov, Usanin, & Chuhlantseva, 2010), and (Kurbatskii, et al., 2014). Due to the geometry of the flame deflector under the rocket nozzle, models of jets impinging upon inclined surfaces are especially of interest (Nonomura, Goto, & Fujii, 2010).

Mankbadi et. al. discuss the use of ANSYS Fluent for modeling and impinging jet, as well as the use of the Ffowcs-Williams Hawkings method to calculate the far field acoustics. The jet in this study is impinging against a flat plate, however the implementation of the Ffowcs-Williams Hawkings method is still of interest. Ffowcs-Williams Hawkings surfaces were placed at the wall where the jet impinged as well as a
permeable surface that was placed in the flow field. This is shown in Figure 1-1. A similar set up for the Ffowcs-Williams Hawkings surfaces was used in this thesis.

Figure 1-1 Ring and Wall Ffowcs-Williams Hawkings surfaces (Mankbadi, et al., 2016)

There has been less research conducted specifically on the acoustic field generated during rocket launch as opposed to the perpendicular impingement that occurs during vertical takeoff and landing of aircraft (Nonomura, Goto, & Fujii, 2010). The acoustic characteristics are not the same when the impingement is no longer perpendicular. Nonomura, Goto, and Fujii discuss how inclined plate impingement is more similar to Mach waves generated by large scale turbulence than the high frequency tone noises generated by a perpendicular plate. The feedback loop that is present with a perpendicular plate is also not present with an inclined plate. Their research included simulations where the jet was at several different temperatures and they concluded that the hotter jet produced higher sound pressure levels and a stronger spectrum in low frequencies than the cold jet at the same Mach number. It is important to note that this thesis was conducted using only low temperatures.
Nonomura, Goto, and Fujii also discuss the three sources of acoustic wave generation. These sources are: Mach waves generated in the shear layer of the jet before impingement, acoustic waves generated in the impingement region, and Mach waves generated downstream of the impingement. Prior empirical studies of acoustics during rocket launch had not considered the acoustic waves generated in the impingement region.

There has also been previous 2D modeling of the flame deflector and flame trench with the objective of modeling the temperature and pressure environments (Brehm, et al., 2013). The study by Brehm et. al. is part of ongoing research to model the 3D environment, so this preliminary report mainly focused on the steps that need to be taken in order to complete the 3D model. The 2D model does however provide a basis for comparison for this thesis.

Compressible Reynolds-Averaged Navier Stokes (RANS) equations were solved with Spalart-Allmaras turbulence models. LAVA and Overflow CFD codes were used on both Cartesian and unstructured grids. The instantaneous pressure plots, shown in Figure 1-2, were used for comparison purposes when determining if the results of this thesis were reasonable. This comparison will be discussed in Section 3.3.2.
The geometry of the rocket, launch pad, and flame trench in this study were provided by Dr. Bruce Vu for use in this thesis. This geometry will be discussed in more depth in Section 2.1. All of these previous studies were used during the process of modeling to ensure that the results during each step were reasonable.

1.3. **Objective**

The objective of this thesis was to create a 2D model that could recreate the overall trends of the far field experimental acoustic data collected by NASA (Vu & Harris, 2015). There is no expectation of an exact replica of the results because of
comparison of a 2D model to a 3D experimental setup. The flame trench and flame
deflector are the same geometry for a slice at any location in the z direction, however the
rocket is not. Due to these reasons, the exact values of the results are not as important as
the general trends of the results obtained.
2. Numerical Models

2.1. Geometry

The geometry of the 2D model contains a simplified lower portion of the SLS, the Launchpad, the main flame deflector, and the flame trench. Figure 2-1 shows this geometry labeled on a picture of the computational mesh that will be discussed in Section 2.2.

![Figure 2-1 Labeled Geometry](image)

Measurements for the geometry are shown in Table 2-1.

<table>
<thead>
<tr>
<th>Location</th>
<th>Symbol</th>
<th>Measurement (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rocket Nozzle Diameter</td>
<td>(D_n)</td>
<td>3.96</td>
</tr>
<tr>
<td>Height of Nozzle and Launchpad from bottom of Flame Trench</td>
<td>(h_n)</td>
<td>27.13</td>
</tr>
<tr>
<td>Height of Nozzle and Launchpad from bottom of Flame Trench</td>
<td>(h_f)</td>
<td>12.51</td>
</tr>
<tr>
<td>Width of Domain of Interest</td>
<td>(w_d)</td>
<td>50.79</td>
</tr>
<tr>
<td>Width of total Domain</td>
<td>(w_t)</td>
<td>210</td>
</tr>
</tbody>
</table>
2.2. **Computational Mesh**

The computational mesh was generated using Pointwise V12.2R2. The unstructured 2D mesh contains 554,850 cells. In the domain of interest, shown in Figure 2-2, the grid spacing is 0.1m.

![Figure 2-2 Mesh in the Domain of Interest](image)

The grid spacing is too small to see the mesh in Figure 2-2 so a closer view of this mesh along the flame deflector is shown in Figure 2-3.
For the remainder of the computational domain the grid spacing is gradually increased from 0.1 at the boundaries of the domain of interest to 0.3m on the far boundaries. This full grid is shown in Figure 2-4.
The grid was expanded past the domain of interest to prevent any outflow issues at the boundary from effecting the solution within the domain of interest. The solution in this region outside the domain of interest is neglected in the calculation of the acoustic field.

2.2.1. Grid Study

A comparison between the mesh shown above and an earlier mesh is shown to verify that the mesh being used will produce physically reasonable results. Both grids are shown in Figure 2-5.

Both grids have the same spacing in the domain of interest, however the coarse grid was expanded to 1m spacing at the far boarders compared to the 0.3m of the finer grid. A
closer view of the transition from the domain of interest to the rest of the domain is shown in Figure 2-6.

![Figure 2-6 Transition Coarse Grid (left) and Fine Grid (right)](image)

The grid spacing in the coarse grid expands extremely rapidly, but since the largest spacing on the fine grid is only 3 times larger than the spacing before the transition the expansion shown in Figure 2-6 appears to be more gradual. Other grids with more quantifiably gradual expansion were generated, however each of those grids had unknown errors when being imported into ANSYS Fluent. A different grid generator or grid generation method should be used in future models to ensure that the transition is not abrupt. Velocity magnitude contours for both grids are shown in Figure 2-7.

![Figure 2-7 Velocity Magnitude (m/s) Coarse Grid (top) and Fine Grid (bottom)](image)
The extremely rapid growth from small spacing to large has caused to flow to be reflected at the end of the domain of interest and no flow is entering the coarser grid. In the finer grid there does not appear to be any flow reflection at those same transition locations. The grid in the domain of interest was chosen to be fine enough to provide proper resolution for both high and low frequency oscillations. Table 2-2 Oscillation Resolution Verification shows the resolution in both time and space. Using a coarser grid in the domain of interest would not have allowed the high frequency oscillation to be fully resolved.

2.3. Fluent Models

ANSYS Fluent R16.0 was used to carry out the simulations. A pressure-based solver was used for both the steady and transient models. This was chosen due to the low Mach number away from the impingement surface. A density based solver would have required preconditioning. A SIMPLEC Solution method was used. The SIMPLEC algorithm allows for faster convergence than the SIMPLE algorithm when the convergence is limited due to pressure-velocity coupling (Van Doormaal & Raithby, 1984). Second order upwinding was used for the momentum, modified turbulent viscosity, and energy spatial discretizations. Second order discretization was used for pressure and least squares cell based was used for the gradient. Second order discretization was used for all of these parameters in order to reduce possible discretization error. For the transient model a first order implicit discretization was used for time. For time discretization, going to second order would increase computation time without improving accuracy noticeably.
The Spalart-Allmaras One Equation Model (Spalart & Allmaras, 1992) was used for the turbulence calculation. Spalart-Allmaras was chosen to allow for comparison to the 2D pressure and temperature study of the same geometry (Brehm, et al., 2013). The acoustic field was modeled using the Ffowcs Williams-Hawkings method. This method will be discussed in more detail in Section 2.4.

The launch pad, flame deflector, and flame trench were defined as wall boundaries. All outlets were defined as pressure outlets. ANSYS Fluent allows for backflow conditions to be set at pressure outlets in order to prevent flow reflection at the outlets. The outlet pressure was set as 101325 Pa. Since the outlets are so far from the domain of interest the main concern is to prevent flow reflection that could affect the solution. For the steady case the inlet was defined as a constant pressure inlet, \( P_j = 550375 \) Pa. This inlet pressure was chosen so that the inlet Mach number would be 2.5. This was chosen to prevent and unnecessary complications that could be caused by hypersonic flow at this early stage of modeling. A more realistic rocket launch Mach number would be over 10. For the transient cases a constant pressure case was run as well as two cases with sinusoidally oscillating pressure and temperature. The equation for the oscillating pressure is shown below.

\[
P = P_j + \frac{P_j}{5} \sin \left(2\pi \left(\frac{St \cdot u_{jet}}{d}\right) t\right)
\]

For the transient model the inlet temperature was sinusoidally oscillated using the equation shown below.

\[
T = T_j + \frac{T_j}{5} \sin \left(2\pi \left(\frac{St \cdot u_{jet}}{d}\right) t\right)
\]

Where \( St \) is the Strouhal Number (\( St = \frac{f_D}{u_{jet}} \)). For the two oscillating cases the frequencies were 11 Hz and 110 Hz.
To ensure that the grid spacing and time spacing were small enough such that the oscillations would be fully resolved the number of grid points per wave length and the number of iterations per period were calculated for both forced oscillation cases. The results of these calculations are shown in Table 2-2.

Table 2-2 Oscillation Resolution Verification

<table>
<thead>
<tr>
<th>Frequency (Hz)</th>
<th>Speed of Sound (m/s)</th>
<th>Wave Length (m)</th>
<th>Grid Spacing (m)</th>
<th>Points Per Wave</th>
<th>Period (s)</th>
<th>Timestep (s)</th>
<th>Steps Per Period</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>340</td>
<td>30.90909</td>
<td>0.1</td>
<td><strong>309.0909</strong></td>
<td>0.090909</td>
<td>0.00017</td>
<td><strong>534.7594</strong></td>
</tr>
<tr>
<td>110</td>
<td>340</td>
<td>3.090909</td>
<td>0.1</td>
<td><strong>30.90909</strong></td>
<td>0.009091</td>
<td>0.00017</td>
<td><strong>53.47594</strong></td>
</tr>
</tbody>
</table>

For both cases the oscillations should be well resolved in both time and space.

2.4. Ffowcs Williams-Hawkings Methods

The Ffowcs Williams-Hawkings Method is an integral technique used to calculate the acoustic field outside of the computational domain (Ffowcs Williams & Hawkings, 1969). The equation is derived from the continuity equation and the Navier stokes equations. There are three pressure terms; the monopole (thickness), the dipole (loading), and the quadrupole (volume). The quadrupole term contains all of the acoustic sources outside the control surface. The placement of the control surfaces allows the quadrupole term to be dropped, if it can be assumed that all of the acoustic sources are contained within the control surfaces. The original non-homogenous wave equation is shown below in
Equations 3, 4, and 5. In all of the following equations $a_0$ is the far-field speed of sound, $P$ is the total gauge pressure, $p'$ is the sound pressure in the far-field, pressure, $t$ is time, $u_i$ is the velocity component in the $x_i$ direction, $u_n$ is the fluid velocity component normal to the surface $f = 0$, $v_i$ is the surface velocity component in the $x_i$ direction and $\rho_0$ is the far-field density.

$$
\frac{1}{a_0^2} \frac{\partial^2 p'}{\partial t^2} - \nabla^2 p' = \frac{\partial^2}{\partial x_i \partial x_j} \left( T_{ij} H(f) \right) - \frac{\partial}{\partial x_i} \left[ \left( P_{ij} n_i + \rho u_i (u_n - v_n) \right) \delta(f) \right]
$$

where,

$$
T_{ij} = \rho u_i u_j + P_{ij} a_0^2 (\rho - \rho_0) \delta_{ij}
$$

$$
P_{ij} = p \delta_{ij} - \mu \left[ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right]
$$

Equation 4 is the Lighthill stress tensor. Equation 5 is the compressive stress tensor. $H(f)$ is the Heaviside function and $\delta(f)$ is the Dirac delta function.

For the non-moving porous surfaces, the equations for the monopole term, Equation 6, and the dipole term, Equation 6, are shown below (Lyrintzis, 2003).

$$
4\pi p_T' (\vec{x}, t) = \frac{1}{a_0} \frac{\partial}{\partial t} \int_S \frac{\rho_0 U_n}{r} \left[ \frac{1}{r} \right]_{ret} dS
$$

$$
4\pi p_L' (\vec{x}, t) = \frac{1}{a_0} \frac{\partial}{\partial t} \int_S \frac{L_r}{r^2} \left[ \frac{1}{r^2} \right]_{ret} dS + \frac{\partial}{\partial t} \int_S \frac{L_r}{r^2} \left[ \frac{1}{r^2} \right]_{ret} dS
$$

where

$$
U_n = \left( 1 - \frac{\rho}{\rho_0} \right) v_n + \frac{\rho u_n}{\rho_0}
$$

and

$$
L_r = P_{ij} \hat{n}_j + \rho u_r (u_n - v_n)
$$
In Equation 8 \( n \) is the dot product the unit vector in the surface normal direction and in Equation 9 \( r \) is the dot product of the unit vector in the radiation direction.

The Ffowcs Williams-Hawkings solver in Fluent works in two steps. First the flow solution is calculated at all of the source surfaces during the unsteady calculation. After the calculation is complete, the acoustic field at each of the selected receivers is calculated. Only the flow solution after the initial transient period is used in this calculation. For the three cases in this thesis, the period between 7.5s and 15s was used.

Since the model is 2D and the Ffowcs Williams-Hawkings solver in Fluent is 3D the sources must be extended into the third dimension. This extension into the third dimension is done by using the source correlation length. The data used in the third dimension is identical to the source data collected. Given an \( x \) and \( y \) location along the source, in this case the Ffowcs Williams-Hawkings surface, the flow solution is assumed to be the same for all \( z \) locations. For this model the source correlation length was set as 1m for all cases. This length was chosen because it is smaller than the diameter of the rocket nozzle. By keeping the source correlation length smaller than the diameter the solution can still be assumed to be realistic because the flow still exists in these locations. The source surfaces and receiver locations used in this model will be discussed in Sections 4.1.2 and 4.2.1, respectively.
3. Results

3.1. Benchmark Experiment

The experimental data used for comparison is from “Space Launch Systems (SLS) Mobile Launcher Rocket Exhaust Plume Induced Environment.” These tests were conducted at NASA Kennedy Space Center (Vu & Harris, 2015). The experiment involved the testing of the acoustics and vibration of the SLS at several conditions. The tests included liquid rockets only, solid rockets only, with water shielding and without water shielding. For this model the Hold Down, Dry (FA-HF-01) test was used for comparison. For this test the acoustic data was recorded for an average of 5-6.5 seconds after the RC-25 liquid rockets were ignited and before the solid rocket boosters were ignited. The exhaust velocity at sea level for the 4 liquid core engines was 14,377 ft/s, Mach 12.88. This is significantly higher than the velocities used in this thesis. The far-field data was collected at seven points distributed around the launch pad. These microphone locations are shown Figure 3-1.
No data was collected at location GA_FF05 so that location was also excluded from the CFD calculations.

3.2. Steady Base Flow

The laminar steady base flow was ran to ensure that the Fluent models used would produce reasonable results that could be expanded to the turbulent model and later the unsteady model. The steady base flows were run with a Reynolds number of \(2.57 \times 10^9\).

3.2.1. Laminar Flow

The laminar flow model ran for 2476 iterations until the residuals had become constant for several hundred iterations. The residuals, \(x\)-velocity, \(y\)-velocity, and
continuity) did not fully converge, but remained constant after initially decreasing. The velocity magnitude of the laminar flow model is shown in Figure 3-2.

![Figure 3-2 Laminar Flow Velocity Magnitude (m/s)](image)

The velocity contours show the jet separating from the flame deflector as well as some circulation further down the flame trench above \( x = -20 \) m. This circulation is better seen in the vorticity magnitude contour shown in Figure 3-3.

![Figure 3-3 Laminar Vorticity Magnitude (1/s)](image)
The large velocities on the upper surface of the flame trench at \( x = 0 \) to 5 m and \( x = -25 \) to -15 m are not physical and are a result of the laminar model. They do not occur in the turbulent models.

The large vorticity swirl over \( x = -20 \) m matches the expected behavior of the recirculation of the flow after the flame deflector. The vorticity swirl at the same locations as the large velocities on the upper surface of the flame trench are similarly non-physical and do not appear in the turbulent models.

Figure 3-4 shows the centerline velocity, at \( x = 0 \), from the jet nozzle to the flame deflector.

![Laminar Centerline Velocity](image)

**Figure 3-4 Laminar Centerline Velocity**

The velocity is shown as a fraction of the jet velocity and the distance is shown as a fraction of the vertical distance vs the diameter of the jet nozzle. The centerline velocity remains relatively constant until approximately 3-3.5 \( y/D \). As shown in Figure
2-1 the jet begins to deform around this area and there is circulation shown in Figure 2-3 that prevents the velocity from reaching a stagnation point on the deflector.

### 3.2.2. Turbulent Flow

The turbulent flow model ran for 7751 iterations until the residuals had become constant for several hundred iterations. Just like the laminar model, the residuals, (x- velocity, y-velocity, energy, and continuity) did not converge. For other test runs of several thousand more iterations, the residual remained constant.

The velocity magnitude of the turbulent flow model is shown in Figure 3-5.

![Figure 3-5 Turbulent Flow Velocity Magnitude (m/s)](image)

The velocity contours show the jet separating from the flame deflector as well as a velocity spike further down the flame trench above \( x = -30 \)m. The large velocities from
the laminar model are not seen on the upper surface of the flame trench at \( x = 0 \) to 5 m and \( x = -25 \) m to -15 m. There is a velocity spike on the back side of the flame deflector above \( x = 5 \) m.

The vorticity distribution for the turbulent model is shown in Figure 3-6.

Due to the concentration of the largest vorticity magnitudes between \( x = -35 \) m and 15 m the Ffowcs Williams-Hawkings surface can be placed at these edges of the domain of interest in the flame trench as well as at \( y = 0 \). The other surfaces used in calculating the acoustic field are the walls of the flame trench, the flame deflector, the launch pad, and the exterior of the rocket. Figure 3-7 shows the location of the porous Ffowcs Williams-Hawkings surfaces.
The black lines in Figure 3-7 are also sources used in the FFowcs Williams-Hawkings calculations.

Figure 3-8 shows the centerline velocity from the jet nozzle to the flame deflector.
The velocity is shown as a fraction of the jet velocity and the distance is shown as a fraction of the vertical distance vs the diameter of the jet nozzle. The centerline velocity begins to decrease just after the jet but remains near \(0.8u_j\) until around \(y/D = 3\). After that point the velocity begins to drop more rapidly. This can also be seen in Figure 3-5.

### 3.3. Unsteady Flow

The first unsteady case run was a constant base flow. The other two unsteady cases were run with perturbed temperature and pressure as discussed in Section 2.3. The base flow case had the same parameters as the steady turbulent flow. This model was run for a total of 15 seconds with a time step of \(1.7 \times 10^{-4}\)s. A total of 88235 time steps were run with 10 iterations per time step. Two periodic models were run, a low frequency sinusoidal oscillation and a high frequency sinusoidal oscillation. Both models were run with the same oscillation amplitude; 20 percent of the pressure input from the constant case. The high frequency, 110 Hz, case was run with a Strouhal number of 0.5 and the low frequency case was run with one tenth of the frequency, 11 Hz, which reduces the Strouhal number to 0.05. Residuals for the flow solution for the base case are shown in Figure 3-9. ANSYS Fluent does not allow the residuals to be saved or exported so the residuals at later time periods cannot be plotted with the initial residuals.
Figure 3-9 Base Case Residuals for Iterations 0-1000, 110000-114550, and 877500-882250

While the residuals show that the solution does not converge, the two later time periods do show the oscillatory behavior of the residuals has remained constant and that
there is no divergence. A fully converged solution would be preferable in terms of how well the results can be trusted, but since the solution does not blow up and remains constant over a long period of time, the results are assumed to be accurate enough for analysis in this thesis. Better convergence could be achieved by refining the grid, altering the nozzle shape to be more realistic, or by using different solution methods such as DES or LES. Another option would be to use a two equation turbulence model. The same trends can be observed in the residual plots for the 11 Hz case, shown in Figure 3-10, and the 110 Hz case, shown in Figure 3-11.
Figure 3-10 11 Hz Case Residuals for Iterations 0-1000, 105000-110000, and 875000-880000
Figure 3-11 110 Hz Case Residuals for Iterations 0-1000, 105000-110000, and 875000-880000
3.3.1. Flow Behavior

At each time step the pressure and velocity at several probe points was exported. These probe points include 8 points on the Ffowcs Williams-Hawkings surfaces. The pressure and velocity at each of these points is compared to the inlet conditions to see the behavior of the flow.

The locations of the eight probe locations on the Ffowcs Williams-Hawkings surfaces are shown in Figure 3-12.

![Figure 3-12 Ffowcs Williams-Hawkings Probe Locations](image)

The geometric locations are shown in Table 3-1.

<table>
<thead>
<tr>
<th>Probe</th>
<th>Location (m)</th>
<th>Probe</th>
<th>Location (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(-1.3,-0.1)</td>
<td>5</td>
<td>(17.27,-22)</td>
</tr>
<tr>
<td>2</td>
<td>(1.3,-0.1)</td>
<td>6</td>
<td>(-35.5,-10)</td>
</tr>
<tr>
<td>3</td>
<td>(17.27,-10)</td>
<td>7</td>
<td>(-35.5,-15)</td>
</tr>
<tr>
<td>4</td>
<td>(17.27,-15)</td>
<td>8</td>
<td>(-35.5,-22)</td>
</tr>
</tbody>
</table>
Due to the extremely large number of data points plotted many of the remaining figures with asymptotic behavior near the origin appear to start at above 0, they do not. In order to fit all of the data onto the same plot, the asymptotic behavior near the origin is not fully shown. The following plots show the total pressure for all three cases. The total pressure at points 1 and 2 from Figure 3-12 is shown in Figure 3-13.
Figure 3-13 Nozzle Exit Total Pressure (Pa)
A closer view to show the difference between points 1 and 2 and the inlet total pressure is shown in Figure 3-14. For all three cases there is an initial transient period at points 1 and 2. After 0.3s the flow is no longer transient at these points.

Figure 3-14 Close View of Nozzle Exit Total Pressure (Pa)
point 1, the light blue, cannot be seen because it is directly behind point 2, the orange. For the base case, points 1 and 2 are slightly lower than the input total pressure. In the 11 Hz case, the magnitude of the oscillations is slightly larger than the input and the phase has shifted by about $1/10^{th}$ of a period. Similar to the 11 Hz case, the magnitude at the nozzle exit is slightly higher than the input. However, the phase shift for this case is much larger. There is nearly a half period shift.

Figure 3-15 shows the total pressure at points 3, 4, and 5 compared to the inlet total pressure.
Figure 3-15 Right Surface Total Pressure (Pa)
A closer view of the total pressure at points 3, 4, and 5 from the right Ffowcs Williams-Hawkings Surface are shown in Figure 3-16, Figure 3-17, and Figure 3-18 respectively.

Figure 3-16 Point 3 Right Surface Total Pressure (Pa)
Figure 3-17 Point 4 Right Surface Total Pressure (Pa)
The pressure at all three points has a time average of around 0 Pa. In the 11 Hz case, the behavior after the flow has hit the flame deflector is no longer perfectly sinusoidal, however
it still has a period of about 0.1 seconds. For the 110 Hz case, none of the points have retained the original frequency. The behavior does become time independent after about 5 seconds. Loss of the original frequency could be because the RANS model cannot handle high frequencies well and so the high frequency was damped out after the flow hit the flame deflector. As discussed in Section 2.2, the grid and time spacing are both small enough that they should not be causing the dissipation of the frequency. All of the Ffowcs Williams-Hawkings surfaces are within the domain of interest so the grid resolution calculations were only done using that grid spacing, 0.1m.

The pressure oscillations at points 6, 7, and 8 on the left Ffowcs Williams-Hawkings surface is shown in Figure 3-19.
Figure 3-19 Left Surface Total Pressure (Pa)

A closer view of points 6, 7, and 8 are shown in Figure 3-20, Figure 3-21, and Figure 3-22 respectively.
The time average total pressure at point 6 is around -20000 Pa for all three cases.
The time average at point 7 is also below 0 for all three cases.
Figure 3-22 Point 8 Left Surface Total Pressure (Pa)

The behavior is no longer sinusoidal at any of the three points.
3.3.2. Flow Snapshots

Snapshots of different parameters were taken to observe the behavior of the flow in the base case. A snapshot of the density contours is shown in Figure 3-23.

The density is highest inside the rocket and there is also a region of high density at the base of the flame deflector.

Figure 3-24 shows the static pressure contours.
The pressure contours show that the static pressure is highest inside the rocket and there is a region of high pressure at the base of the flame deflector.

Figure 3-25 shows the velocity magnitude contours.

![Figure 3-25 Base Case Velocity Magnitude (m/s)]

The velocity contours show some separation from the surface of the flame deflector and the majority of the flow going to the left with small tufts going to the right.

Mach number contours are shown in Figure 3-26.

![Figure 3-26 Base Case Mach Number Contours]
The Mach number contours show the same behavior as the velocity magnitude contours. The flow behavior in this figure is similar to the behavior shown in Figure 1-2, especially the LAVA Unstructured and Cartesian-Unstructured solutions, shown in (b) and (c). The flow to the right of the flame deflector, and the separation on the surface of the flame deflector is similar in all four solutions shown in Figure 1-2.

Vorticity contours are shown in Figure 3-27.

![Figure 3-27 Base Case Vorticity Magnitude Contours (1/s)](image)

A closer view of the vorticity around the rocket and flame deflector at several time steps is shown in Figure 3-28.
The vorticity contours show the progression of the vorticity due to the tufts shown in Figure 3-25 to the right of the flame deflector. The vorticity on the flame deflector as well as the vorticity between the rocket and the flame deflector does not change.

Videos of the velocity magnitude, vorticity magnitude, and the wall shear stress along the bottom wall to the left of the flame deflector were created to observe the unsteadiness of the flow. These videos were used to determine that the unsteadiness was physical and not the result of an error with the solution.

Snapshots of different parameters were taken at different time steps were taken to observe the behavior of the flow in the 11 Hz case. These snapshots were taken at 14.62s, 14.65s, 14.67s, and 14.70s. These times were chosen because they represent 4 different times within one period and also it is near the end of the run time. Taking the snapshots so late ensures that the flow is no longer transient.

Figure 3-29 shows the density contours.
The density snapshots show that the density is highest inside the rocket body and also there is a high density region that moves down the flame deflector as the period progresses.

Figure 3-30 shows the static pressure contours.

The static pressure contours show the high and low pressure at the inlet at different points in the period. At times 14.62s and 14.67s the pressure in the rocket is high and at
14.65s and 14.70s the pressure is low. There is also a region of higher pressure midway between the rocket and the flame deflector at times 14.62s and 14.67s. This is due to the pulse of higher pressure. At times 14.65s and 14.70s there is a larger region of higher pressure on the flame deflector, this is because the pulse from the previous snapshot has reached the deflector.

Figure 3-31 shows the velocity magnitude contours.

![Figure 3-31 Velocity Magnitude Contours (m/s)](image)

The velocity magnitude contours also show the pulses of higher velocity and lower velocity. The high velocity pulse can be seen at times 14.62s and 14.70s.

Figure 3-32 shows the Mach number contours.
The Mach number contours are very similar to the velocity magnitude contours in terms of the behavior of the high and low Mach at the different times. At all four time steps the shocks can be seen in the flow exiting the rocket nozzle.

Figure 3-33 shows the vorticity magnitude contours.

While there is vorticity throughout the entire computational domain, the largest vorticity in at each of the four time steps can be seen near the rocket and the flame deflector.
This confirms the selection of the Ffowcs Williams-Hawkings surface locations since the largest magnitudes, shown in red and yellow, are not beyond the surfaces.

Figure 3-34 shows the dilatation field. The dilatation field is the divergence of velocity shown over a small range.

![Dilatation Field](image)

The dilatation field does not show any noise propagation. The noise propagation should be seen in right propagating outward from the noise sources. Due to the low frequency and high wavelength of this case the computational grid does not extend far enough to see the waves.

Snaps shots for the dilatation field are shown for the 110 Hz case. Figure 3-35 shows the density contours at 14.62s, 14.6226s, 14.625s and 14.6277s. These times are all within one period and were chosen to show the behavior at each point in the period.
Since the period of this case is much smaller there is not much change in the entire computational domain. However, in the rocket body the different density can be seen at times 14.62s and 14.6277s.

Figure 3-36 shows the static pressure contours.
The static pressure contours also show the pulse of higher pressure in the rocket body.

Figure 3-37 shows the velocity magnitude contours.

![Image of velocity magnitude contours]

Figure 3-37 110 Hz Velocity Magnitude Contours (m/s)

There is not much difference in the velocity magnitude contours over such a small time period, but some change can be observed in the plume below the rocket nozzle.

Figure 3-38 shows the Mach number contours.
Figure 3-38 110 Hz Mach Number Contours

Just like the velocity magnitude contours the main difference at the different time steps can be seen in the plume below the nozzle.

Figure 3-39 shows the vorticity magnitude contours.

Figure 3-39 110 Hz Vorticity Magnitude Contours (1/s)

Like the vorticity contours from the 11 Hz case, the largest vorticity magnitude is within the designated Ffowcs Williams-Hawkings surfaces.
Figure 3-40 shows the dilatation field contours.

Figure 3-40 110 Hz Dilatation Field Contours

The dilatation field for this case does show the noise propagation. Figure 3-41 shows the dilatation field closer to the rocket nozzle.

Figure 3-41 110 Hz Close View of Dilatation Field
The sound propagation going upwards appears to continue to the edge of the computational domain, but the rings below the launch pad break up around -10m and 10m. When observing a video of the dilatation field over more than a second, the sound waves still dissipated at this point. The waves dissipated well before the transition to the coarser grid. The grid spacing in this region should be small enough to fully resolve the waves, however a finer grid could be investigated along with different turbulence models or changing to a LES or DES solver. Reasonable acoustic results can still be expected because the sound waves do make contact with the launch pad which is also a Ffowcs Williams-Hawkings surface. The Ffowcs Williams-Hawkings surfaces that this frequency is present at are shown in red in Figure 3-41. The wave that propagates upwards also comes into contact with the launch pad as well as the porous Ffowcs Williams-Hawkings surface below the rocket nozzle.

3.4. **Acoustic Field**

3.4.1. **Frequency at Ffowcs Williams-Hawkings Surfaces**

A Fast Fourier Transform was performed on the static and total pressure data at each of the 8 locations on the Ffowcs Williams-Hawkings surfaces for all three unsteady cases. This was done to see if the forced input frequency was still dominant at all of the porous Ffowcs Williams-Hawkings surfaces. Figure 3-42 shows the FFT for the static pressure at point 1.
Figure 3-42 FFT for Static Pressure at Point 1
The base case does not show any dominant frequency. Both the 11 Hz case and the 110 Hz case show the dominant frequency at the input frequencies of 11 Hz and 110 Hz respectively. Figure 3-43 shows the total pressure FFT at point 1.
The total pressure shows the same trends as the static pressure. Figure 3-44 shows the FFT results for the static pressure at point 2.

Figure 3-44 FFT for Static Pressure at Point 2
The base case does not show any dominant frequency. Both the 11 Hz case and the 110 Hz case show the dominant frequency at the input frequencies of 11 Hz and 110 Hz respectively. Figure 3-45 shows the total pressure FFT at point 2.
The total pressure shows the same trends as the static pressure. Figure 3-46 shows the FFT results for the static pressure at point 3.

Figure 3-46 FFT for Static Pressure at Point 3
The base case does not show any dominant frequency. The 11 Hz case shows a dominant frequency at 11 Hz. The 110 Hz case does not show any dominant frequency. Figure 3-47 shows the total pressure FFT at point 3.
The total pressure shows the same trends as the static pressure. Figure 3-48 shows the FFT results for the static pressure at point 4.

Figure 3-48 FFT for Static Pressure at Point 4
The base case does not show any dominant frequency. The 11 Hz case shows a dominant frequency at 11 Hz. The 110 Hz case does not show any dominant frequency. Figure 3-49 shows the total pressure FFT at point 4.
The total pressure shows the same trends as the static pressure. Figure 3-50 shows the FFT results for the static pressure at point 5.

Figure 3-50 FFT for Static Pressure at Point 5
The base case does not show any dominant frequency. The 11 Hz case shows a
dominant frequency at 11 Hz. The 110 Hz case does not show any dominant frequency.

Figure 3-51 shows the total pressure FFT at point 5.
The total pressure shows the same trends as the static pressure. Figure 3-52 shows the FFT results for the static pressure at point 6.
The base case does not show any dominant frequency. The 11 Hz case shows a dominant frequency at 11 Hz. The 110 Hz case does not show any dominant frequency. Figure 3-53 shows the total pressure FFT at point 6.
Figure 3-53 FFT for Total Pressure at Point 6

The total pressure shows the same trends as the static pressure. Figure 3-54 shows the FFT results for the static pressure at point 7.
Figure 3-54 FFT for Static Pressure at Point 7
The base case does not show any dominant frequency. The 11 Hz case shows a dominant frequency at 11 Hz. The 110 Hz case does not show any dominant frequency. Figure 3-55 shows the total pressure FFT at point 7.

Figure 3-55 FFT for Total Pressure at Point 7
The total pressure shows the same trends as the static pressure. Figure 3-56 shows the FFT results for the static pressure at point 8.

Figure 3-56 FFT for Static Pressure at Point 8
The base case does not show any dominant frequency. The 11 Hz case shows a dominant frequency at 11 Hz. The 110 Hz case does not show any dominant frequency.

Figure 3-57 shows the total pressure FFT at point 8.
The total pressure shows the same trends as the static pressure.

3.4.2. Acoustic Locations

The receiver locations shown in Table 3-2 Receiver Locations were placed radially around the domain to match the microphone locations shown in Figure 3-1.

Table 3-2 Receiver Locations

<table>
<thead>
<tr>
<th>Receiver</th>
<th>X (m)</th>
<th>Y (m)</th>
<th>Z (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-14.0799</td>
<td>1.43256</td>
<td>-5.832</td>
</tr>
<tr>
<td>2</td>
<td>-10.165</td>
<td>1.43256</td>
<td>-10.165</td>
</tr>
<tr>
<td>3</td>
<td>-5.832</td>
<td>1.43256</td>
<td>-14.0799</td>
</tr>
<tr>
<td>4</td>
<td>0</td>
<td>1.43256</td>
<td>-15.24</td>
</tr>
<tr>
<td>6</td>
<td>10.165</td>
<td>1.43256</td>
<td>-10.165</td>
</tr>
<tr>
<td>7</td>
<td>14.0799</td>
<td>1.43256</td>
<td>-5.832</td>
</tr>
<tr>
<td>8</td>
<td>12.192</td>
<td>1.43256</td>
<td>0</td>
</tr>
</tbody>
</table>

As mentioned in Section 3.1, receiver 5 was skipped because no experimental data was collected at this location.

3.4.3. Acoustic Data Comparison

The experimental report contains plots of the Sound Pressure Level vs. the 1/3 Octave Frequency for each receiver as well as the Overall Sound Pressure Level for each receiver (Vu & Harris, 2015). This information is shown in Figure 3-58.
The SPL from each microphone was plotted along with the SPL calculated at the receiver locations in all three Fluent cases. The acoustic data for the Fluent cases was only collected from 7.5s to 15s to ensure that the solutions were time independent. The plots for receiver 1 are shown in Figure 3-59.
The blue, gray, and yellow lines show the Fluent data and the orange is the experimental. The fluent lines match the overall trend of the experimental data. Both the 11 Hz case and the 110 Hz case show a peak at the same frequencies that were dominant in the Ffowcs Williams-Hawkings FFT plots jet below the rocket nozzle. The 11 Hz case has a peak at 11 Hz and a smaller peak at 33 Hz. The 110 Hz case has a peak at 110 Hz and a smaller peak at 330 Hz. These same behaviors can be observed in the plots for receivers 2 through 8 which are shown in Figure 3-60 through Figure 3-65.

![Figure 3-60 SPL (dB) vs 1/3 Octave Band Frequency (Hz) Receiver 2](image_url)
Figure 3-61 SPL (dB) vs 1/3 Octave Band Frequency (Hz) Receiver 3

Figure 3-62 SPL (dB) vs 1/3 Octave Band Frequency (Hz) Receiver 4
Figure 3-63 SPL (dB) vs 1/3 Octave Band Frequency (Hz) Receiver 6

Figure 3-64 SPL (dB) vs 1/3 Octave Band Frequency (Hz) Receiver 7
A polar plot of the Overall Sound Pressure Level is shown in Figure 3-66. This plot shows the sound directivity of the experimental results vs the Fluent results. Both oscillatory fluent cases match the OASPL of the experimental case very closely for the first four receivers, shown from 337.5° to 247.5°. For the last three receivers the fluent model overestimated the OASPL by about 20 dB. For the Fluent base case, the OASPL at the first four receivers is underestimated by about 20 dB and for the last three receivers the base case is a closer match to the experimental model.
Figure 3-66 OASPL Polar Plot
4. Conclusion

The main objective of this thesis was to create a 2D model that could recreate the overall trends of the acoustic data collected by in the experimental model (Vu & Harris, 2015). In section 4.2 three cases, one with no forced oscillation, one with low frequency forced oscillation, and one with high frequency forced oscillation were discussed. Plots of the pressure at various locations in the flow field for the low frequency case showed that the forced oscillations remained present. However, due to the large wavelength caused by such a low frequency the dilatation field did not show any clear acoustic waves.

The high frequency case no longer showed the oscillations after the flow hit the flame deflector. The dilatation field did show clear acoustic waves that were damped out after the flow hit the flame deflector. This is likely the product of the RANS model not being able to resolve the high frequency.

As discussed in Section 3.4 the overall results of the Sound Pressure Level vs. 1/3 Octave Frequency have trends similar to the experimental results for all three cases. Both oscillatory cases produced sound directivity plots that closely resembled the experimental results. Due to the simplifications in the geometry of the Fluent model the predictions are not exact, but this model can be used as the basis for creating the 3D model which will replicate the geometry and flow conditions of the experimental model more exactly. Due to the lack of convergence seen in the residual plots, the methodology needs to be improved. In order to remove error caused by spurious noise several recommendations are made in Section 5.
5. Recommendations

To improve upon the accuracy of the results it is recommended to improve upon the transition between the coarse mesh and the fine mesh, as seen in Figure 2-6. While flow reflection did not appear to be a problem in any of the videos produced there is still a line of potentially spurious noise seen at the transition in Figure 3-41. Making the transition to the coarser grid more gradual would reduce any possibility of spurious noise at this location.

It is also recommended that a larger computational domain be used. Due to computation time constraints the computational domain was limited, especially above the launch pad. Using a larger computational domain would allow the sound waves from a lower frequency oscillation to be seen on the dilatation field. Expanding the computational domain will also allow for the inclusion of the noise due to the launch tower.

Another recommendation is to use LES or DES instead of RANS. The LES or DES model will eliminate the possibility of the oscillations being damped out after the flow hits the flame deflector. Because the LES and DES solvers would take significantly more computation time they were not feasible for the time constraints of this thesis.
REFERENCES


