Using Computational Fluid Dynamics To Accurately Model Agricultural Spray Nozzles

Zachary Chapman

South Dakota State University

Follow this and additional works at: https://openprairie.sdstate.edu/etd

Part of the Bioresource and Agricultural Engineering Commons, and the Mechanical Engineering Commons

Recommended Citation
https://openprairie.sdstate.edu/etd/3933

This Thesis - Open Access is brought to you for free and open access by Open PRAIRIE: Open Public Research Access Institutional Repository and Information Exchange. It has been accepted for inclusion in Electronic Theses and Dissertations by an authorized administrator of Open PRAIRIE: Open Public Research Access Institutional Repository and Information Exchange. For more information, please contact michael.biondo@sdstate.edu.
USING COMPUTATIONAL FLUID DYNAMICS TO ACCURATELY MODEL
AGRICULTURAL SPRAY NOZZLES

BY

ZACHARY CHAPMAN

A thesis submitted in partial fulfilment of the requirements for the
Master of Science
Major in Mechanical Engineering
South Dakota State University
2020
This thesis is approved as a creditable and independent investigation by a candidate for the master’s degree and is acceptable for meeting the thesis requirements for this degree. Acceptance of this does not imply that the conclusions reached by the candidate are necessarily the conclusions of the major department.

Zachary Chapman

Jeffrey Doom
Advisor Date

Kurt Basset
Department Head Date

Dean, Graduate School Date
I would like to dedicate this work to my parents who have provided the support necessary for me to continue my education.
ACKNOWLEDGMENTS

The first person I would like to thank is my advisor Dr. Doom. Dr. Doom was the professor who sparked my interest in computational fluid dynamics and also provided me with an interesting project to work on. Second, I would like to thank the students who worked on the Raven Sprayer Testbed who made it possible for me to gain experimental results to compare to my work. I would also like to thank the Mechanical Engineering Department at South Dakota State University for providing me with the resources necessary to conduct my research as well as providing me with an excellent education. Finally, I would like to thank my family who continued to provide support as I pursued my Masters degree.
## CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABBREVIATIONS</td>
<td>viii</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>ix</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>xii</td>
</tr>
<tr>
<td>ABSTRACT</td>
<td>xiii</td>
</tr>
<tr>
<td>1 INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>1.1 OVERVIEW</td>
<td>2</td>
</tr>
<tr>
<td>1.2 ORGANIZATION OF THE WORK</td>
<td>5</td>
</tr>
<tr>
<td>2 LITERATURE REVIEW</td>
<td>6</td>
</tr>
<tr>
<td>3 METHODOLOGY AND APPROACH</td>
<td>10</td>
</tr>
<tr>
<td>3.1 PROBLEM STATEMENT</td>
<td>10</td>
</tr>
<tr>
<td>3.2 CAD MODEL GENERATION</td>
<td>11</td>
</tr>
<tr>
<td>3.3 MESHING</td>
<td>11</td>
</tr>
<tr>
<td>3.4 PHYSICS</td>
<td>16</td>
</tr>
<tr>
<td>3.4.1 GOVERNING EQUATIONS</td>
<td>16</td>
</tr>
<tr>
<td>3.4.2 MULTIPHASE MODELING</td>
<td>17</td>
</tr>
<tr>
<td>3.4.3 PASSIVE SCALAR</td>
<td>19</td>
</tr>
<tr>
<td>3.4.4 COURANT-FRIEDRICHES-LEWY CONDITION</td>
<td>19</td>
</tr>
<tr>
<td>3.5 SOLVER SETTINGS</td>
<td>20</td>
</tr>
<tr>
<td>4 RESULTS AND DISCUSSION</td>
<td>24</td>
</tr>
<tr>
<td>4.1 MESH SIZE</td>
<td>24</td>
</tr>
<tr>
<td>4.2 MESH TYPE</td>
<td>26</td>
</tr>
</tbody>
</table>
ABBREVIATIONS

$\alpha_i$ Phase Volume Fraction

$f_b$ Body Forces

$j_i$ Diffusion Flux

$q$ Heat Flux

$v$ Continuum Velocity

$\Delta t$ Maximum Time Step

$\Delta x$ Dimension of the Cell

$\phi_i$ Passive Scalar Component

$\rho$ Density

$\sigma$ Stress Tensor

$\theta_s$ Spray Angle

$C$ Courant Number

$c$ Velocity

$E$ Total Energy per Unit Mass

$h$ Distance Below the Nozzle

$S_E$ Energy Source per Unit Volume

$S_{\phi_i}$ Source Term for Passive Scalar Component $i$

$V$ Volume of Cell
\[ V_i \quad \text{Volume of phase } i \]

\[ x_s \quad \text{Width of Spray Plume} \]
LIST OF FIGURES

1  Nozzle Nomenclature .................................................. 2
2  Raven Sprayer Testbed .................................................. 3
3  High Speed Video Shot .................................................. 4
4  Geometry of XR110-08 Spray Nozzle ............................... 11
5  Model Broken Into Regions ............................................. 12
6  Nozzle Broken Into Regions ........................................... 12
7  Mesh Size ................................................................. 13
8  Mesh Types ............................................................... 14
9  Mesh Used For Pulsed Simulation .................................... 15
10 Geometry Used For 20 in Simulation ............................... 16
11 Mesh Used For 20 in Simulation ...................................... 17
12 Geometry Used for Simulation of Liquid Jet Breakup .......... 17
13 Mesh Used for the Simulation of Liquid Jet Breakup .......... 18
14 Comparison of Volume Fraction of Water for Coarse and Fine Mesh .......................... 25
15 Comparison of Passive Scalar Between Coarse and Fine Mesh .................................. 27
16 Comparison of Velocity Between Coarse and Fine Mesh ........ 28
17 Comparison of Volume Fraction of Water Between Tetrahedral and Trimmed Cell Mesh .................................................................................. 30
18 Comparison of Passive Scalar Between Tetrahedral and Trimmed Cell Mesh .................. 31
19 Comparison of Velocity Between Tetrahedral and Trimmed Cell Mesh ......................... 32
20 Comparison of Volume Fraction of Water Between Polyhedral and Trimmed Cell Mesh .................................................................................. 33
21 Comparison of Passive Scalar Between Polyhedral and Trimmed Cell Mesh .................. 34
22 Comparison of Velocity Between Polyhedral and Trimmed Cell Mesh ........................ 35
<p>| 23 | Comparison of Volume Fraction of Water Between Polyhedral Broken into Multiple Regions and Trimmed Cell Mesh | 37 |
| 24 | Comparison of Passive Scalar Between Polyhedral Broken into Multiple Regions and Trimmed Cell Mesh | 38 |
| 25 | Comparison of Velocity Between Polyhedral Broken into Multiple Regions and Trimmed Cell Mesh | 39 |
| 26 | Comparison of Volume Fraction of Water Between Steady-State Simulation with Coarse Mesh and Unsteady Simulation | 40 |
| 27 | Comparison of Passive Scalar Between Steady-State Simulation with Coarse Mesh and Unsteady Simulation | 41 |
| 28 | Comparison of Velocity Between Steady-State Simulation with Coarse Mesh and Unsteady Simulation | 42 |
| 29 | Comparison of Volume Fraction of Water Between Steady-State Simulation with Fine Mesh and Unsteady Simulation | 43 |
| 30 | Comparison of Passive Scalar Between Steady-State Simulation with Fine Mesh and Unsteady Simulation | 44 |
| 31 | Comparison of Velocity Between Steady-State Simulation with Fine Mesh and Unsteady Simulation | 45 |
| 32 | Comparison of Volume Fraction of Water Between $k-\omega$ and $k-\epsilon$ Turbulence Modeling | 47 |
| 33 | Comparison of Passive Scalar Between $k-\omega$ and $k-\epsilon$ Turbulence Modeling | 48 |
| 34 | Comparison of Velocity Between $k-\omega$ and $k-\epsilon$ Turbulence Modeling | 49 |
| 35 | Pulsed Simulation Passive Scalar | 51 |
| 36 | Pulsed Simulation Velocity | 52 |
| 37 | Plot of Velocity Magnitude During Pulsing Simulation | 53 |
| 38 | Plot of Absolute Pressure During Pulsing Simulation | 53 |
| 39 | Pulsed Simulation Passive Scalar | 54 |</p>
<table>
<thead>
<tr>
<th>Page</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>40</td>
<td>Pulsed Simulation Velocity</td>
</tr>
<tr>
<td>41</td>
<td>Plot of Velocity Magnitude During Pulsing Simulation</td>
</tr>
<tr>
<td>42</td>
<td>Plot of Absolute Pressure During Pulsing Simulation</td>
</tr>
<tr>
<td>43</td>
<td>Mean Passive Scalar 20 Inches Below the Nozzle</td>
</tr>
<tr>
<td>44</td>
<td>Volumetric Flow Rate [GPM] 20 Inches Below the Nozzle Collected From Raven Testbed</td>
</tr>
<tr>
<td>45</td>
<td>Droplet Modeling Volume Fraction of Water</td>
</tr>
<tr>
<td>46</td>
<td>Droplet Modeling Passive Scalar</td>
</tr>
<tr>
<td>47</td>
<td>Droplet Modeling Velocity Magnitude</td>
</tr>
</tbody>
</table>
## LIST OF TABLES

1. Physics Models Selected For All Simulations .................................................. 20
2. Time and Turbulence Modeling For Mesh Effects, Pulse Width Modulation, and Spray Distribution simulations .................................................. 20
3. Turbulence and Time Modeling Selections for Turbulence and Time Modeling Simulations .................................................................................. 21
4. Automatically Selected Physics Models .............................................................. 21
5. Eulerian Phase Models ....................................................................................... 21
6. Phase Interaction Models .................................................................................... 21
7. Regions For Investigation of Mesh and Physics Models Simulations .............. 22
8. Regions For Pulse Simulations .......................................................................... 22
9. Regions For 20 in. Simulations .......................................................................... 22
10. Regions For Liquid Jet Breakup ....................................................................... 23
11. Summary of Spray Angles ............................................................................... 46
Computational fluid dynamics (CFD) is a tool used by engineers in many industries to study fluid flow. A relatively new industry to adopt the use of CFD is the agricultural industry. The present work seeks to understand whether CFD can be used to accurately model spray nozzles. A spray nozzle commonly used in agricultural spraying was simulated. First, the impact of factors such as mesh size, mesh type, and physics models have on the solution were investigated. Next, a method to pulse the spray was determined. This was required to compare simulation results with experimental data. A user-defined function was used to define a pulsed velocity inlet in order to pulse the spray. The domain was then extended to allow the examination of a slice 20 inches below the nozzle. The results were compared to experimental data collected from the Raven Sprayer Testbed. Results from these studies suggested that CFD could be used to model spray nozzles but the validity of the results is strongly related to the available computational resources. These simulations were carried out using Star-CCM+. Lastly, Large Eddy simulations were conducted to capture the liquid jet breakup within the spray plume. The results suggested that the liquid jet breakup could be modeled using CFD, but again sufficient computational resources are required. These simulations were performed in OpenFOAM.
1 INTRODUCTION

While sprayer nozzles may seem like a small part of the spraying system, the nozzle plays an important part in spray application. There are several different types of spray nozzles used for agricultural spraying and the use of an improper spray nozzle can result in problems such as over application, under application, or drift [10]. All of these problems are costly. Over application results in the waste of the chemical being applied while under application can result in the need for respraying. Drift is the deposition of spray particles to non-target locations. Drift can cause health and environmental risks as well as contribute to economic losses. The inhalation of chemicals can be harmful to humans and wildlife. Drift can also cause crops to become unsellable by injuring crops or contaminating crops that are not registered for the use of the particular chemical [7]. Several factors contribute to drift including:

- Wind velocity: higher wind velocity will carry the droplets easier.

- Distance between the nozzle tip and target area: allows wind velocity to contribute to drift more.

- Air temperature: can lead to evaporation.

- Droplet size: smaller droplets are carried easier.

The droplet size is one factor that can be controlled. Droplet size is affected by the geometry of the nozzle and the operating pressure. There have been many experimental studies on drift by several different industries. However, experiments are costly. There is also a desire to produce more efficient nozzles. This suggests that the study of sprayer nozzles using computational fluid dynamics may be beneficial.
1.1 OVERVIEW

Computational fluid dynamics (CFD) is a tool used by engineers in many industries such as the aerospace industry to study airflow over an airfoil or the bio-medical industry to study flow through a medical device. The need for the use of CFD comes as a result of the equations which govern fluid flow. These equations can not be solved analytically so we are forced to solved them using numerical methods i.e. CFD. A relatively new industry to adopt the use of CFD is the agricultural industry.

This work looks into the validity of using CFD to accurately model spray nozzles. To assess the validity of using CFD to model spray nozzles, a commonly used agricultural spray nozzle was studied. Figure 1 is the nomenclature for the nozzle which was studied.

![Nozzle Nomenclature](image.jpg)

Figure 1: Nozzle Nomenclature

First, factors such as mesh size, cell type, and physics models were studied. The commercially available package, Star-CCM+, was used to investigate these effects. Mesh size was the first factor that was studied. This study compared the effects of mesh size using a fine and a coarse mesh. Next, different cell types were studied by comparing the three cell types available in Star-CCM+. Finally, the effects of physics models such as time and turbulence modeling were investigated. These simulations were compared to a base case that used a trimmed cell mesh with approximately 1.2 million cells, unsteady time modeling, gravity enabled, and $k - \varepsilon$ turbulence modeling. The optimal mesh and
physics models were determined by comparing the spray angle of the simulation to the listed spray angle for the nozzle which was studied. These mesh and physics conditions were then used for further simulations.

The use of pulse-width modulation to pulse the spray, as opposed to spraying continuously, is commonly used to avoid the effects pressure drop during agricultural spraying. Once the effect of meshing and physics conditions were studied, a method to study whether the pulsing of the spray could be incorporated into a CFD simulation was determined. The method was to use a pulse function, entered as a user-defined function, to define the velocity of the nozzle inlet. This method was then used to determine whether CFD could accurately model the spray distribution of a nozzle. The fluid domain was extended to allow the spray distribution at a slice 20 inches below the nozzle to be examined. These results were compared to the Raven Sprayer Testbed. Figure 2 is a picture of the testbed.

![Figure 2: Raven Sprayer Testbed](image)

The Raven Sprayer Testbed was developed at South Dakota State University with
Raven Industries as a partner. The testbed can capture the spray angle using high-speed videos and the spray pattern distribution using the cup method and patternator. Figure 3 is a screen capture of the high-speed video capabilities from the testbed.

The possibility of simulating the liquid jet breakup during the spraying process was also investigated. These simulations looked into the possibility of using the volume fraction of water to model the water droplets during the simulation. Due to the requirement of the extremely fine mesh to capture the water droplets, the open-source solver OpenFOAM was used. Large Eddy simulations were used with the multiphase solver InterFOAM.
1.2 ORGANIZATION OF THE WORK

This work discusses research into the validity of using computational fluid dynamics to model spray nozzles. Following the introduction is a literature review that discusses past research into agricultural spraying, drift, and simulations. Next, the methodology and approach for the current work are discussed. Items include the problem statement, CAD model generation, meshing, a brief review of the physics, and an overview of the solver settings. Results and an interpretation of the results are given next. First, the results from the study of the effects of mesh size, cell type, and physics models are discussed. A brief summary of the results from the simulations is also given. The rest of the results are then given in the following order:

- pulsed nozzle simulation results
- a comparison of the spray distribution between the simulated and experimental results
- liquid jet breakup simulation results

Lastly, a discussion of the conclusions and the potential future work is given.
2 LITERATURE REVIEW

This section contains a review of literature relevant to agricultural spraying and simulations.

D. Dekeyser et al. [4] investigated the ability to accurately model three air-assisted sprayers using CFD. First, experiments were conducted to characterize the sprayers which were studied. The droplet size and velocity were measured using a Phase Doppler Particle Analyzer. A continuous scan was used to find the average values of the spray plume. A grid scan gave information on droplet characteristics in the horizontal plane under the nozzle. A CFD simulation was done for each of the sprayers being studied. The droplet size was modeled with the Rosin-Rammler distribution. CFD simulations were done to attempt to optimize the spout angle of the sprayer as well. Dekeyser found that the CFD model was successful in modeling airflow and spray patterns for axial, cross flow, and sprayers with individual spouts. The CFD model was also able to optimize the spout angle for a chosen application.

A. Endalew et al. [5] conducted a study to investigate the ability to accurately model three sprayers using CFD. Experiments were carried out in an experimental orchard. Ultrasonic anemometers were used to collect the jet velocity at the outlet of the sprayers. A CFD model of the sprayer moving through the orchard was done using a pulse function. The simulation had comparable results with the results of the experiments.

M. Sidamed et al. [9] took a different approach by creating a virtual nozzle to study spray patterns and droplet transport. A patternator was used to measure volumetric spray patterns and a Phase Doppler Particle Analyzer was used to measure droplet size and velocity. The size and initial velocity of several droplets were collected and recorded in a table. When creating the model, an assumption that the droplets originate radially from the liquid sheet breakup region, which was equal for all droplets, was made. Unlike a real nozzle where the drops are injected simultaneously, the virtual nozzle injects the
droplets individually. The droplet’s size and velocity are randomly selected from the collected data table. One injection event is completed once one whole sweep over the injection surface was completed. A flat plane was used to model the Phase Doppler Particle Analyzer for the target. A study was done comparing effective and convention drag to find which model gave favorable results. Sidamed concludes that CFD can be used to accurately simulate the dynamics of spray droplets in the spray cloud.

A. Miranda-Fuentes [8] conducted experiments to investigate the influence of liquid volume and airflow rate on the efficiency of sprayers. He suggests ”Accurate spray volume adapted to the characteristic canopy could reduce applied volume by 20% while maintaining quality.” Airflow and air velocity are related to drift. Volumetric rate and airflow rate studies were done. For the volumetric rate, volumetric rates of 1603, 619, 182 l/ha were used while holding the airflow rate at a constant rate of 12.03 $m^3/s$. The volumetric rates were chosen due to the commercial industry standard volumetric flow rate of 1600 l/ha. The airflow rates used for the airflow studies were 11.93, 8.9, and 6.15 $m^3/s$ with a constant volumetric rate set at 770 l/ha. Food dye was used as a trace to measure the deposits on the leaves. The findings of the studies showed that both the volumetric rate and airflow rate had strong influences on the spray pattern. Miranda-Fuentes concludes that increasing the volumetric rate increases deposit but decreases efficiency.

M. Fard et al. [6] developed a computer model to study splash-plate atomizers called BLSpray. Like sprayer nozzles, there is much interest in the droplet size of the spray leaving the splash-plate. BLSpray is used to predict liquid film characteristics such as film thickness, velocity, the pattern of the breakup, and droplet distribution. The fluid flow was assumed to be incompressible, Newtonian, and laminar with a Reynolds Number estimated around 2000. The computer model uses the Volume of Fluid model for the multiphase model. For verification purposes, the results were compared to experiments conducted to study droplet size, film thickness, and velocity distribution. The simulations corresponded well to the experiments.
K. Bade et al. [2] compared CFD results and experimental results at multiple injection angles. Experiments were done using a spray nozzle, wind tunnel, PDI system with traverse, and an LSI system. The tests were carried out in coflow and crossflow with a wind velocity of 15 m/s. The sprayer nozzle injected liquid water at ambient temperature with a nozzle pressure of 3.8 bar. Simulations were done using Fluent. The geometry used was the sprayer within the wind tunnel. The simulations were performed at steady state. Droplet size was measured using PDI measurements. A Rosin-Rammler distribution was used for the droplet size distribution. The spray angle of the CFD simulations was measured as $77^\circ$ which was within $1^\circ$ of the experimental results.

F. Xiao et al. [13] studied liquid jet primary breakup. A robust large eddy simulation algorithm was created to study the breakup. Xiao discusses the movement towards using unsteady LES/DNS simulations as opposed to experiments due to the additional details simulations can capture. Xiao discusses various numerical models used to simulate liquid jet atomization. In the developed algorithm, the coupled level set and Volume of Fluid method were used for the multi-phase modeling. The rescaling and recycling method was used for inflow conditions. The simulations conducted were compared to the experimental data used to create the test cases. Laminar and turbulent simulations were done. The laminar simulation showed the importance of correct inflow conditions. The turbulent simulations matched much better to the experimental results. All of the test cases showed good agreement with the experimental data.

T. Butts et al. [3] studied the effects of pulse width modulation duty cycle on droplet size. Experiments were conducted in a low-speed wind tunnel equipped with a SharpShooter PWM system. Several different nozzles were studied including both venturi and non-venturi nozzle types. The droplet size distribution was measured using a laser diffraction system. Regression analysis was conducted to allow for predictions of the change of droplet size dependent on the duty cycle. Butts et al. concluded that the droplet size was inconsistent when operated at lower duty cycles, PWM sprayers should be
operated at higher pressures due to the pressure loss across the solenoid valve, and non-venturi nozzles should be used with PWM sprayers as opposed to venturi nozzles.

From the literature review, we can conclude that CFD can be used to model agricultural spray nozzles. However, there has not been a study of the effects of mesh type, mesh size, or physics models. The present work first looks to explore the effects of these factors and eventually use these results for more advanced simulations.
3 METHODOLOGY AND APPROACH

This section contains a discussion of the method and approach taken. Items discussed include:

- Problem Statement
- CAD Model Generation
- Meshing
- Physics
- Solver Settings

3.1 PROBLEM STATEMENT

The problem discussed in the present research is stated as follows:

*How can computational fluid dynamics be used to accurately model agricultural spray nozzles?*

To determine how CFD can be used to accurately model spray nozzles, several factors were investigated. First, simulations were conducted to determine the effects of mesh and physics models. The best performing results were obtained by comparing the spray angle to the listed spray angle of the nozzle studied. Equation 1 is the equation that was used to calculate the spray angle.

\[ \theta_s = 2 \tan \left( \frac{x_s}{H} \right) \]  

(1)

Studies were also done to determine how well CFD simulations match up with experimental data. Simulation results were compared to experimental data collected from the Raven Sprayer Testbed. The possibility of using CFD to model the liquid jet breakup during the spraying process was also investigated.
3.2 CAD MODEL GENERATION

The geometry of the spray nozzle was obtained from a CT scan of the nozzle. The size of the CT scan was too large to open as a part in Solidworks, so the file size was reduced by decreasing the size of the surface mesh from the scan. Two open-source programs, FreeCad and MeshLab, were used to decrease the size of the surface mesh. Holes in the mesh were also filled in to create a better representation of the model. Another open-source program, Meshmixer, was used to fill in the holes.

Once the file was small enough to be opened in Solidworks, the CT scan was imported into Solidworks and opened as a solid part. Measurements were then taken from the CT scan to re-create the geometry of the nozzle. Figure 4 is the geometry of the nozzle. The geometry of the fluid was then created as a representation of the air using a subtract feature in Solidworks. The file was saved as a Parasolid file to be imported into Star CCM+.

![Figure 4: Geometry of XR110-08 Spray Nozzle](image)

3.3 MESHING

Once the geometry was imported into Star CCM+, the geometry of the fluid was broken into different regions. Additionally, the nozzle was broken into multiple regions for one of the polyhedral meshes and the tetrahedral mesh. This allowed for more control
of the cell size of the mesh around the nozzle. Figure 5 shows the model geometry broken into regions and Figure 6 shows the geometry of the nozzle broken into different regions.

Several different meshes were developed to investigate the effects of mesh size and cell type. To study the effects of mesh size, a coarse and a fine mesh were used. The coarse mesh contained approximately 600,000 cells while the fine mesh contained approximately 1.2 million cells. Figure 7 shows the coarse and fine mesh.

Three different cell types were also compared. These were the trimmed cell, polyhedral cell, and tetrahedral cell. The trimmed cell mesh model is primarily dominated by a hexahedral cell but allows some tetrahedral cells. The polyhedral cell mesh model uses polyhedral cells and the tetrahedral cell mesh model uses tetrahedral cells. Four
Figure 7: Mesh Size

(a) Coarse Mesh

(b) Fine Mesh
different meshes were compared to study the effects of mesh type. Figure 8 shows the four meshes used to study the effects of mesh type.

A volumetric control was used for the trimmed cell meshes. The volumetric control allows for the use of smaller cells in areas of more importance which was beneficial for the trimmed cell mesh. The size of the cells allowed with the volumetric control for the trimmed cell mesh was much smaller. This did not hold true for the polyhedral and the tetrahedral meshes so a volumetric control was not used. The difference between the two polyhedral meshes shown in Figure 8 is the mesh shown in 8d has the nozzle broken into multiple regions. This allowed for more control of the cell size around the nozzle. By breaking the nozzle into multiple regions, a lower base size was able to be used for the mesh while keeping a similar cell count. This was also done for the tetrahedral mesh.

The same geometry was used for the simulations investigating the use of pulse width modulation, as the simulations investigating mesh and physics model effects. However, a new mesh was made. The mesh was a trimmed cell mesh with approximately
2 million cells. The mesh also used a volumetric control to decrease the cell size in areas of more importance. Figure 9 is the mesh that was used for the pulsed simulations.

![Figure 9: Mesh Used For Pulsed Simulation](image)

When studying the spray pattern, a new geometry was created. The domain was extended to allow the examination of a slice 20 inches below the nozzle. To allow for a better comparison to experimental results, the shape of the domain was switched to a rectangular domain. Again, the mesh was a trimmed cell mesh that used a volumetric control. The mesh consisted of approximately 12.5 million cells. Figures 10 and 11 are the geometry and the mesh that was used for the simulations examining spray pattern.

Similar to the simulations examining the spray distribution, the geometry was also changed for the simulations investigating the liquid jet breakup of the spraying process. However, a short and narrow rectangular domain was used as opposed to a wider and longer domain. The representation of the air around the nozzle was also removed. This was done to help decrease the size of the mesh. The mesh was created in Star-CCM+. The mesh used the trimmed cell mesh model with a volumetric control. The mesh contained approximately 11 million cells. Figures 12 and 13 are the geometry and mesh used during these simulations.
3.4 PHYSICS

3.4.1 GOVERNING EQUATIONS

The equations which govern fluid flow are the Navier-Stokes Equations. These equations are the conservation of mass, conservation of momentum, and conservation of energy. Equations 2, 3, and 4 are the equations for the conservation of mass, momentum, and energy respectively.

\[ \frac{\partial \rho}{\partial t} + \nabla (\rho \mathbf{v}) = 0 \] (2)

\[ \frac{\partial (\rho \mathbf{v})}{\partial t} + \nabla (\rho \mathbf{v} \otimes \mathbf{v}) = \nabla \sigma + \mathbf{f}_b \] (3)

\[ \frac{\partial (\rho E)}{\partial t} + \nabla (\rho E \mathbf{v}) = \mathbf{f}_b \mathbf{v} + \nabla (\mathbf{v} \sigma) - \nabla \mathbf{q} + S_E \] (4)

For more information on the numerical methods used by the solvers, please refer to [1].
3.4.2 MULTIPHASE MODELING

The spraying process is considered to be a multiphase problem meaning multiple phases, in this case, air and water, are present. This requires multiphase modeling to be included in the simulations. There are several different multiphase modeling techniques. One of the oldest and most popular methods is the Volume of Fluid method. The Volume
of Fluid is an interface-capturing method that attempts to predict the distribution and movement of the interface between the different phases. The Volume of Fluid method determines the distribution of a fluid by determining the volume fraction of the phase in the cell. Equation 5 is the equation for a phase, i, in a particular cell and must satisfy Equation 6.

\[ \alpha_i = \frac{V_i}{V} \] (5)

\[ \sum_{i=1}^{N} \alpha_i = 1 \] (6)

The phases present in a particular cell can be determined from the volume fraction of each phase. If the volume fraction for phase i is zero, then the particular cell does not contain phase i. If the volume fraction for phase i is one, then the particular cell is full of phase i. If the volume fraction is between 0 and 1, the interface between the phases is present within the cell. For further information on the numerical methods for multiphase models, please refer to [1] or [11].
3.4.3 PASSIVE SCALAR

The Volume of Fluid method keeps track of the volume fraction of each phase present in a simulation. This would suggest that the volume fraction could be used to visualize the spray plume. However, this is very computationally expensive because the cell size needs to be sufficiently small. An alternative, which is less computationally expensive than viewing the spray plume with the volume fraction, is using a passive scalar. A passive scalar is similar to using a dye. The passive scalar is "passive" because the addition of a passive scalar does not affect the physical properties of the simulation. The transport equation for a passive scalar component is given by Equation 7.

\[
\frac{\partial}{\partial t} \int_V \rho \phi_i dV + \oint_A \rho \phi_i \mathbf{v} \cdot d\mathbf{a} = \oint_A \mathbf{j}_i \cdot d\mathbf{a} + \int_V S_{\phi_i} dV
\]  

(7)

For further information on passive scalars please refer to [1].

3.4.4 COURANT-FRIEDRICHS-LEWY CONDITION

An important stability condition that will come into play due to the fine mesh used during the simulation of the liquid jet breakup is the Courant-Friedrichs-Lewy condition or CFL condition. Equation 8 is the CFL condition.

\[
C = \frac{c \Delta t}{\Delta x} \leq 1
\]

(8)

C is the Courant number. The CFL condition dictates that as the size of the cell used for the simulation becomes smaller, the time step must also become smaller otherwise the simulation will become unstable. For more information on the CFL condition, please refer to [12]
3.5 SOLVER SETTINGS

Star-CCM+ was used for the simulations to study the effect of mesh size, mesh cell type, time modeling, and turbulence modeling; determining how to pulse the nozzle; studying the spray distribution. OpenFOAM was used during the simulations to study the liquid jet breakup. In both cases, the fluid was considered to be an incompressible two-phase material. There were several different physics model setups. Table 1 is a table that lists the physics models used for each simulation conducted in Star-CCM+.

<table>
<thead>
<tr>
<th>Physics Model</th>
<th>Selection</th>
</tr>
</thead>
<tbody>
<tr>
<td>Space</td>
<td>Three Dimensional</td>
</tr>
<tr>
<td>Material</td>
<td>Eulerian Muliphase</td>
</tr>
<tr>
<td>Eulerian Multiphase Model</td>
<td>Volume of Fluid</td>
</tr>
<tr>
<td>Viscous Regime</td>
<td>Turbulent</td>
</tr>
<tr>
<td>Optional Models</td>
<td>Passive Scalar</td>
</tr>
<tr>
<td></td>
<td>Gravity</td>
</tr>
</tbody>
</table>

Table 1: Physics Models Selected For All Simulations

During the simulations investigating the effects of meshing, the physics models were held constant. Time and turbulence modeling was also held constant during the pulse width modulation and spray distribution simulations. Table 2 shows the turbulence and time models selected.

<table>
<thead>
<tr>
<th>Physics Model</th>
<th>Selection</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time</td>
<td>Implicity Unsteady</td>
</tr>
<tr>
<td>Reynolds-Average Turbulence</td>
<td>$k - \epsilon$ Turbulence</td>
</tr>
</tbody>
</table>

Table 2: Time and Turbulence Modeling For Mesh Effects, Pulse Width Modulation, and Spray Distribution simulations

Table 3 shows the selections for the simulations investigating time and turbulence modeling. Table 4 shows the physics models automatically selected by other physics models.

The Eulerian Multiphase model requires additional models for the individual phases as well as the phase interaction. Table 5 shows the selected Eulerian Multiphase
### Table 3: Turbulence and Time Modeling Selections for Turbulence and Time Modeling Simulations

<table>
<thead>
<tr>
<th>Physics models</th>
<th>Selection</th>
<th>Simulation</th>
<th>Steady State</th>
<th>Unsteady</th>
<th>$k - \varepsilon$ Turbulence</th>
<th>$k - \omega$ Turbulence</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time</td>
<td>Steady</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Unsteady</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Turbulence</td>
<td>$k - \varepsilon$ Turbulence</td>
<td>x</td>
<td>x</td>
<td></td>
<td>x</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$k - \omega$ Turbulence</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

Table 4: Automatically Selected Physics Models

<table>
<thead>
<tr>
<th>Model</th>
<th>Selected By</th>
</tr>
</thead>
<tbody>
<tr>
<td>Multiphase Interaction</td>
<td>Eulerian Multiphase Model</td>
</tr>
<tr>
<td>Gradients</td>
<td>Volume of Fluid Model</td>
</tr>
<tr>
<td>Segregated Volume Flux Based Flow</td>
<td>Volume of Fluid Model</td>
</tr>
<tr>
<td>Two-Layer All $y+$ Wall Treatment</td>
<td>$k - \varepsilon$ Turbulence Model</td>
</tr>
<tr>
<td>Realizable $k - \varepsilon$ Two-Layer</td>
<td>$k - \varepsilon$ Turbulence Model</td>
</tr>
<tr>
<td>Exact Wall Distance</td>
<td>$k - \varepsilon$ or $k - \omega$ Turbulence Model</td>
</tr>
<tr>
<td>All $y+$ Wall Treatment</td>
<td>$k - \omega$ Turbulence Model</td>
</tr>
<tr>
<td>SST (Mentor) $k - \omega$</td>
<td>$k - \omega$ Turbulence Model</td>
</tr>
</tbody>
</table>

Table 5: Eulerian Phase Models

<table>
<thead>
<tr>
<th>Model</th>
<th>Phase</th>
<th>Water</th>
<th>Air</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Equation of State</td>
<td></td>
<td>Liquid</td>
<td>Gas</td>
</tr>
<tr>
<td>Constant Density</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 6: Phase Interaction Models

<table>
<thead>
<tr>
<th>Model</th>
<th>Selection</th>
</tr>
</thead>
<tbody>
<tr>
<td>Phase Interaction Topology</td>
<td>VOF-VOF Phase Interaction</td>
</tr>
<tr>
<td>Optional Models</td>
<td>Surface Tension Force</td>
</tr>
<tr>
<td></td>
<td>Multiphase Interaction (Selected By Surface Tension Force)</td>
</tr>
</tbody>
</table>

There were different combinations of boundary conditions used for the different simulations. Tables 7, 8, and 9 show the boundary condition selections for the different simulations. The corresponding geometry for each boundary can be seen in Figure 5 and for simulations investigating mesh and physics models and the pulsed simulations and
Figure 6 for the nozzle broken into different regions. The geometry corresponding to the boundary can be seen in Figure 10 for the 20 in. simulation.

<table>
<thead>
<tr>
<th>Region</th>
<th>Selection</th>
</tr>
</thead>
<tbody>
<tr>
<td>in1</td>
<td>Stagnation Inlet</td>
</tr>
<tr>
<td>in2</td>
<td>Velocity Inlet</td>
</tr>
<tr>
<td>pipe</td>
<td>Symmetry Plane</td>
</tr>
<tr>
<td>outlet</td>
<td>Pressure Outlet</td>
</tr>
<tr>
<td>nozzle</td>
<td>nozzle (whole) Wall</td>
</tr>
<tr>
<td></td>
<td>nozzle entrance Wall</td>
</tr>
<tr>
<td></td>
<td>inner nozzle Wall</td>
</tr>
<tr>
<td></td>
<td>nozzle entrance Wall</td>
</tr>
<tr>
<td></td>
<td>outer nozzle wall</td>
</tr>
</tbody>
</table>

Table 7: Regions For Investigation of Mesh and Physics Models Simulations

<table>
<thead>
<tr>
<th>Region</th>
<th>Selection</th>
</tr>
</thead>
<tbody>
<tr>
<td>in1</td>
<td>Velocity Inlet</td>
</tr>
<tr>
<td>in2</td>
<td>Velocity Inlet</td>
</tr>
<tr>
<td>nozzle</td>
<td>Wall</td>
</tr>
<tr>
<td>pipe</td>
<td>Symmetry Plane</td>
</tr>
<tr>
<td>outlet</td>
<td>Outlet</td>
</tr>
</tbody>
</table>

Table 8: Regions For Pulse Simulations

<table>
<thead>
<tr>
<th>Region</th>
<th>Selection</th>
</tr>
</thead>
<tbody>
<tr>
<td>in1</td>
<td>Velocity Inlet</td>
</tr>
<tr>
<td>in2</td>
<td>Velocity Inlet</td>
</tr>
<tr>
<td>pipe</td>
<td>Outlet</td>
</tr>
<tr>
<td>outlet</td>
<td>Outlet</td>
</tr>
<tr>
<td>nozzle</td>
<td>Wall</td>
</tr>
</tbody>
</table>

Table 9: Regions For 20 in. Simulations

The simulations investigating the effects of mesh and physics models used pressure boundary conditions. The pressure boundary conditions were initialized using velocity boundary conditions. The simulation was run with the velocity boundary conditions for approximately 1,500 iterations, then the simulation was stopped and the regions were switched to pressure boundary conditions. The pulsed inlet was incorporated into the
simulation using a user-defined function to pulse the nozzle inlet. The pulse function was used with velocity boundary conditions as opposed to pressure boundary conditions. Velocity boundary conditions were chosen because they do not require initialization. Equation 9 is the user-defined function used to define the pulsed velocity inlet.

\[
(sin(60 \times \{\text{time}\}) \geq 0) \ ? \ 1 : 0 \tag{9}
\]

The open-source solver OpenFOAM was used to model the liquid jet breakup. The OpenFOAM solver interFOAM was used for the simulations. InterFOAM is a multiphase solver available with OpenFOAM. As discussed in Section 3.3, the mesh was created using Star-CCM+ and was imported to OpenFOAM. Table 10 shows the boundary conditions used for the simulation of the liquid jet breakup. The geometry corresponding to each boundary can be seen in Figure 12.

<table>
<thead>
<tr>
<th>Region</th>
<th>Selection</th>
</tr>
</thead>
<tbody>
<tr>
<td>in1</td>
<td>Patch</td>
</tr>
<tr>
<td>in2</td>
<td>Patch</td>
</tr>
<tr>
<td>left</td>
<td>Wall (Slip)</td>
</tr>
<tr>
<td>right</td>
<td>Wall (Slip)</td>
</tr>
<tr>
<td>front</td>
<td>Wall (Slip)</td>
</tr>
<tr>
<td>back</td>
<td>Wall (Slip)</td>
</tr>
<tr>
<td>out</td>
<td>Patch</td>
</tr>
<tr>
<td>nozzle</td>
<td>Wall</td>
</tr>
</tbody>
</table>

Table 10: Regions For Liquid Jet Breakup
4 RESULTS AND DISCUSSION

This section contains a discussion of the results from each of the simulations. The results are given in the following order:

- Mesh Size
- Mesh Type
- Time Modeling
- Turbulence Modeling
- Pulsed Nozzle
- Spray Distribution
- Droplet Modeling

4.1 MESH SIZE

A study of the effects mesh size has on simulations of spray nozzles was done. This study was done using a fine and a coarse mesh. The coarse mesh contained approximately 600,000 cells and the fine mesh contained approximately 1.2 million cells. Both simulations used a trimmed cell mesh and were simulated as unsteady, with gravity enabled, and $k - \epsilon$ turbulence modeling. The first result that was obtained was the volume fraction of water in air. Figure 14 shows the volume fraction contours for the coarse and fine mesh with the coarse mesh being on the top and the fine mesh on the bottom.

In Figure 14, less of the spray plume of the coarse mesh can be visualized using the volume fraction of water. This is due to the larger cell size in the coarse mesh. As seen in Figure 7, the cell size of the coarse mesh is much larger than the cell size of the fine mesh. To visualize the entire spray plume using the volume fraction is computationally expensive due to the small cell size. A much less computationally expensive solution to
Figure 14: Comparison of Volume Fraction of Water for Coarse and Fine Mesh

(a) Coarse Mesh

(b) Fine Mesh
visualizing the spray plume is to use a passive scalar. As discussed in Section 3.4.3, a passive scalar is similar to using a dye in the liquid which does not affect the physics of the spray. Figure 15 shows the contour plot of the passive scalar for the coarse and fine mesh. Again, the coarse mesh is on the top and the fine mesh is on the bottom.

An important result is understood by comparing the two contour plots of the passive scalar in Figure 15. The coarse mesh results in a wider spray angle compared to the fine mesh. This is likely due to the larger cell size of the mesh. In general, the simulations which use a larger cell size result in a wider spray angle. One potential explanation for this is the larger cell size does not adequately capture the interface between the two phases. This results in more of the passive scalar being distributed throughout the cell. In the case of the fine mesh, the smaller cell size does a better job of capturing the interface between the two phases resulting in a better representation of the spray plume. The last result collected was the velocity contours of the spray plume. Figure 16 is the contour plots of the velocity magnitude for the coarse mesh on the top and fine mesh on the bottom.

Similar velocities near the exit of the nozzle are observed in Figure 16. As the distance from the exit of the nozzle increases in the spray plume, the velocity contours tend to differ. Again, this can be explained by comparing the size of the cells near the exit of the nozzle. As seen in Figure 7, the cell size near the exit of the nozzle is similar for both the coarse and fine mesh. As the distance from nozzle exit increases, the cell size increases which impacts the velocity.

4.2 MESH TYPE

The next study done was an investigation into the effects of the cell type used for the mesh. As discussed in Section 3.3, the three cell types used were a trimmed cell, tetrahedral cell, and polyhedral cell mesh types. Again, the trimmed cell mesh used a volumetric control and the tetrahedral and one polyhedral mesh has the nozzle broken into
Figure 15: Comparison of Passive Scalar Between Coarse and Fine Mesh
Figure 16: Comparison of Velocity Between Coarse and Fine Mesh
multiple regions instead of one as discussed in Section 3.3. This study held the mesh size and physics models constant. We see the trends discussed in Section 4.1 hold true for the different cell types. Results for the volume fraction of water, passive scalar, and velocity were obtained for each simulation.

### 4.2.1 TETRAHEDRAL CELL MESH

Figures 17, 18, and 19 are a comparison of the contour plots for the volume fraction of water, passive scalar, and velocity between the trimmed cell and tetrahedral cell mesh with the trimmed cell on the bottom and the tetrahedral cell on the top.

Comparing the cell size between the trimmed cell and tetrahedral cell meshes in Figure 8, a significantly smaller cell size is used for the trimmed cell mesh when compared to the cell size of the tetrahedral mesh. As expected, the smaller cell size used for the trimmed cell mesh results in a better ability to visualize the spray plume using the volume fraction of water and a tighter spray angle when examining the passive scalar. Again similar velocity contours result in areas where the cell sizes are similar. Another take away from these comparisons is the tetrahedral cell mesh does not provide a well-defined spray pattern. Again, this is likely explained by the larger cell size having trouble tracking the interface between the two phases.

### 4.2.2 POLYHEDRAL CELL MESH

Figures 20, 21, and 22 are comparisons of the contours for the volume fraction of water, passive scalar, and velocity for the trimmed and polyhedral cell meshes. The trimmed cell mesh is on the bottom and the polyhedral cell is on the top. Again, similar trends are seen between the volume fraction of water, the passive scalar, and the velocity contours. The polyhedral mesh also provides a better representation of the spray plume than the tetrahedral mesh.
Figure 17: Comparison of Volume Fraction of Water Between Tetrahedral and Trimmed Cell Mesh
Figure 18: Comparison of Passive Scalar Between Tetrahedral and Trimmed Cell Mesh

(a) Tetrahedral Mesh

(b) Trimmed Cell Mesh
Figure 19: Comparison of Velocity Between Tetrahedral and Trimmed Cell Mesh
Figure 20: Comparison of Volume Fraction of Water Between Polyhedral and Trimmed Cell Mesh
Figure 21: Comparison of Passive Scalar Between Polyhedral and Trimmed Cell Mesh
Figure 22: Comparison of Velocity Between Polyhedral and Trimmed Cell Mesh
4.2.3 POLYHEDRAL CELL MESH WITH MULTIPLE REGIONS

Figures 23, 24, and 25 are the comparisons of the contour plots for the volume fraction of water, passive scalar, and velocity magnitude between the trimmed cell mesh and the polyhedral cell mesh broken into multiple regions. As expected, the polyhedral mesh broken into multiple regions provides more similar results to the trimmed cell mesh.

4.3 TIME MODELING

Next, physics models such as the treatment of time and the selection of turbulence models were investigated. First, a comparison between conducting the simulation as a steady and unsteady problem was done. A coarse and fine mesh was used when investigating the effects of time modeling. The meshes used were the same coarse and fine mesh used in Section 4.1. Results were collected for the volume fraction of water, passive scalar, and velocity magnitude. Figures 26, 27, and 28 are the comparisons of the contour plots for the volume fraction of water, passive scalar, and velocity between the unsteady simulation and the steady-state simulation using the coarse mesh. The unsteady simulation is on the bottom and the steady-state simulation is on the top.

Looking at the passive scalar and velocity contours, we see similar distributions near the exit of the nozzle. Further down and at the edges of the spray plume is where the unsteadiness starts to have more of an impact on the results. This is due to the simulation picking up the unsteadiness of the droplets whereas near the exit of the nozzle the spray could be considered as a single liquid.

Next, steady-state simulations were done using the fine mesh. Figures 29, 30, and 31 are the comparisons of the contour plots of volume fraction of water, passive scalar, and velocity between the unsteady and the steady-state simulation using the fine mesh.

Unlike the unsteady simulations, decreasing the cell size of the mesh does not improve results when conducting simulations at steady-state. The finer mesh is picking up more of the unsteadiness of the spray which significantly impacts the results.
Figure 23: Comparison of Volume Fraction of Water Between Polyhedral Broken into Multiple Regions and Trimmed Cell Mesh
Figure 24: Comparison of Passive Scalar Between Polyhedral Broken into Multiple Regions and Trimmed Cell Mesh
Figure 25: Comparison of Velocity Between Polyhedral Broken into Multiple Regions and Trimmed Cell Mesh
Figure 26: Comparison of Volume Fraction of Water Between Steady-State Simulation with Coarse Mesh and Unsteady Simulation
Figure 27: Comparison of Passive Scalar Between Steady-State Simulation with Coarse Mesh and Unsteady Simulation
Figure 28: Comparison of Velocity Between Steady-State Simulation with Coarse Mesh and Unsteady Simulation
Figure 29: Comparison of Volume Fraction of Water Between Steady-State Simulation with Fine Mesh and Unsteady Simulation
Figure 30: Comparison of Passive Scalar Between Steady-State Simulation with Fine Mesh and Unsteady Simulation
Figure 31: Comparison of Velocity Between Steady-State Simulation with Fine Mesh and Unsteady Simulation
4.4 TURBULENCE MODELING

Simulations were done to compare the different turbulence models. During these simulations, the mesh used for the base case was used. Figures 32, 33, and 32 are the comparisons of contour plots for the volume fraction of water, passive scalar, and velocity magnitude between $k - \epsilon$ and $k - \omega$ turbulence modeling with $k - \epsilon$ being on the bottom and $k - \omega$ being on the top.

Comparing Figures 32, 33, and 34, there are no major differences between the two turbulence models which suggest that turbulence modeling does not have a big effect on the simulation results. However, the simulations using $k - \omega$ turbulence modeling results in a slightly more narrow spray angle.

4.5 SUMMARY OF RESULTS

A summary of the spray angles from each of the comparison simulations can be seen in Table 11. The spray angle was calculated by taking measurements of the spray plume from the passive scalar at a slice 50 mm below the tip of the nozzle. Again, the base case simulation is the simulation that all simulations were compared to. The simulation used the fine trimmed celled mesh, $k - \epsilon$ turbulence modeling, and unsteady time modeling.

<table>
<thead>
<tr>
<th>Simulation</th>
<th>Spray Angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base Case</td>
<td>102.8deg</td>
</tr>
<tr>
<td>Fine Mesh</td>
<td>123.9deg</td>
</tr>
<tr>
<td>Tetrahedral Cell Mesh</td>
<td>119.0deg</td>
</tr>
<tr>
<td>Polyhedral Cell Mesh</td>
<td>109.9deg</td>
</tr>
<tr>
<td>Polyhedral Cell Mesh (With Multiple Regions)</td>
<td>107.0deg</td>
</tr>
<tr>
<td>Coarse Mesh Steady State</td>
<td>102.2deg</td>
</tr>
<tr>
<td>Fine Mesh Steady State</td>
<td>114.6deg</td>
</tr>
<tr>
<td>$k - \omega$ Turbulence Modeling</td>
<td>96.2deg</td>
</tr>
</tbody>
</table>

Table 11: Summary of Spray Angles
Figure 32: Comparison of Volume Fraction of Water Between $k-\omega$ and $k-\epsilon$ Turbulence Modeling

(a) $k-\omega$ Turbulence

(b) $k-\epsilon$ Turbulence
Figure 33: Comparison of Passive Scalar Between $k-\omega$ and $k-\epsilon$ Turbulence Modeling
Figure 34: Comparison of Velocity Between $k - \omega$ and $k - \epsilon$ Turbulence Modeling
4.6 PULSED NOZZLE

Once the effects of mesh size and cell type were investigated and the appropriate physics conditions were selected, simulations were done to determine how to incorporate the pulsing of the spray plume into the simulations. As discussed in Section 3.5, the method chosen was to use a user-defined function with velocity boundary conditions to pulse the velocity inlet. Figure 35 and Figure 36 are a series of screenshots of the passive scalar and velocity magnitude contours from the pulsed simulation.

The series of screenshots start at 0.025 seconds into the spray time. At 0.05 seconds the velocity is turned off and the spray plume is cut off. The spray plume moves down through the domain until fully evacuated. At 0.1 seconds the spraying process is continued and a new spray plume is created. The velocity and pressure were tracked during the simulation. Figures 37 and 38 show the velocity magnitude and absolute pressure at the entrance of the nozzle during the duration of the simulation.

Figures 37 and 38 show that the spray plume is being pulsed. The pressure is being held approximately constant during the spraying process which shows this method can simulate the pulsing of the spray plume. The pressure was slightly too high, so the velocity was reduced during further simulations.

4.7 SPRAY DISTRIBUTION

Once the method of pulsing the spray was determined, simulations could be done to compare the spray distribution from the CFD simulation to the spray distribution collected from experimental data. Figures 39 and 40 are a series of screenshots of the passive scalar and velocity contours during the simulation. The absolute pressure and velocity magnitude were also monitored during the simulation. Figures 41 and 42 are the plots of the velocity magnitude and absolute pressure at the entrance of the nozzle.

To determine the spray distribution during the simulation, the mean of the passive scalar was monitored at a slice 20 inches below the tip of the nozzle. Figure 43 is the
Figure 35: Pulsed Simulation Passive Scalar

mean of the passive scalar 20 inches below the tip of the nozzle.

The experimental data used for comparison was collected from the Raven Sprayer
Figure 36: Pulsed Simulation Velocity

Testbed. A patternator was used to collect the data to determine the spray distribution. This involved spraying water into a group of cups 20 inches below the tip of the nozzle.
The cups were measured to determine the volumetric flow rate. Figure 44 is the volumetric flow rate at a slice 20 inches below the nozzle.

Comparing Figures 43 and 44, similar trends are noticed. The simulation predicts that the highest distribution will be off of the centerline of the nozzle which agrees with the experimental data. The simulation also predicts that the edges of the spray plume have a lower distribution than the centerline of the spray plume. This also agrees with the experimental data. The shape of the spray distribution is slightly different than the experimental data. This could be due to the cell size of the mesh used during the
Figure 39: Pulsed Simulation Passive Scalar

(a) 0.025 Seconds
(b) 0.05 Seconds
(c) 0.075 Seconds
(d) 0.1 Seconds
(e) 0.125 Seconds
(f) 0.15 Seconds
(g) 0.175 Seconds
(h) 0.2 Seconds
Figure 40: Pulsed Simulation Velocity
The final topic studied while determining the capabilities of using CFD to model spray nozzles was to investigate the ability to simulate the liquid jet breakup within the spray plume. Results were obtained for the volume fraction of water, passive scalar, and velocity magnitude. Figures 45, 46, and 47 contain a series of screenshots at various time steps for the volume fraction of water, passive scalar, and velocity magnitude.

### 4.8 DROPLET MODELING

The final topic studied while determining the capabilities of using CFD to model spray nozzles was to investigate the ability to simulate the liquid jet breakup within the spray plume. Results were obtained for the volume fraction of water, passive scalar, and velocity magnitude. Figures 45, 46, and 47 contain a series of screenshots at various time steps for the volume fraction of water, passive scalar, and velocity magnitude.
Figure 43: Mean Passive Scalar 20 Inches Below the Nozzle

Figure 44: Volumetric Flow Rate [GPM] 20 Inches Below the Nozzle Collected From Raven Testbed
Figure 45: Droplet Modeling Volume Fraction of Water
Figure 46: Droplet Modeling Passive Scalar
Figure 47: Droplet Modeling Velocity Magnitude

(a) 0.002 Seconds
(b) 0.003 Seconds
(c) 0.004 Seconds
(d) 0.005 Seconds
(e) 0.006 Seconds
(f) 0.007 Seconds
(g) 0.008 Seconds
(h) 0.009 Seconds
The liquid jet breakup within the spray plume is starting to be captured within the simulation. This is seen in Figure 45. Examining Figure 45, as time progresses the spray plume starts to break away into the individual droplets. Another important takeaway from these simulations can be found by comparing Figures 45 and 46. While the passive scalar is not able to capture the individual droplets within the plume, the spray plume visualized with the passive scalar provides a good estimation of the overall shape and distribution within the spray plume. This further supports the use of using the passive scalar to visualize the spray plume in previous simulations.
5 CONCLUSIONS AND FUTURE WORK

Simulations were done to determine whether computational fluid dynamics could be used to model spray nozzles. This section contains a discussion of the conclusions taken from the simulations and potential future work. The conclusions are sorted into mesh and physics model effects, pulsed spraying and spray distribution modeling, and finally droplet modeling.

5.1 CONCLUSIONS

5.1.1 MESH AND PHYSICS MODEL EFFECTS

Investigations of the effects of mesh cell size, mesh cell type, time modeling, and turbulence modeling were done. During these simulations, one variable was changed, such as the cell size, and the simulation was compared to a base case. In the study of mesh cell size, the results suggest that mesh cell size has a significant impact on the results of the simulation. In general, when larger cell sizes are used, the spray angle is wider. The wider spray angle is likely due to the Volume of Fluid method used for the simulations. As previously discussed, the Volume of Fluid method tracks the interface between the phases. The larger cell size would have a much harder time tracking the interface than the smaller cell size which is causing the wider angle.

The trimmed cell mesh was determined to perform the best during the investigation of the different mesh types. The other cell types used were the tetrahedral and polyhedral cell types. Although the polyhedral cell mesh provided a spray angle closer to the listed value of the nozzle, the trimmed cell mesh provided a better representation of the spray plume. The better performance of the trimmed cell meshing model is due to the cell size used within the mesh. While keeping the same number of cells in the mesh, the cell size of the trimmed cell mesh is significantly smaller which results in a better representation of the spray plume. Again, this shows the importance of
the cell size during simulations of spray nozzles.

The simulations using unsteady time modeling performed better than the simulations which used steady time modeling. This is most likely due to the mesh picking up the unsteadiness of the liquid breakup during the spraying process. This is supported by the fact that when a finer mesh is used during the steady-state simulations, the results become worse. The finer mesh is picking up more of the unsteadiness of the simulation which has significant impacts on the results.

Turbulence modeling had the least significant results on the simulation of spray nozzles. The two turbulence models that were compared were the $k - \epsilon$ and $k - \omega$ turbulence models. The $k - \epsilon$ turbulence model was provided a slightly closer spray angle to the listed value of the nozzle. This resulted in the $k - \epsilon$ turbulence model being used for further simulations.

5.1.2 PULSED SPRAY AND SPRAY DISTRIBUTION

A method to pulse the spray, as opposed to spraying continuously, was chosen. The method chosen was to use a user-defined function to define a pulsed velocity inlet for the nozzle instead of a pulsed pressure inlet. The absolute pressure was monitored during these simulations. The absolute pressure was help approximately constant. This suggests that the use of a pulsed velocity inlet to simulate a pulsed pressure inlet can be used if the pressures match the operating conditions.

The pulsed velocity inlet method was used with the results from the simulations investigating mesh and physics model effects to determine whether CFD could be used to model the spray distribution of a spray nozzle. Simulation results were compared to experimental results collected using the Raven Sprayer Testbed. The results between the experiment and simulation are not an exact match but several similarities are seen between the two such as areas with a higher spray distribution. A possible explanation for the differences is the mesh cell size. These results suggest that computational fluid dynamics
can be used to simulate and study spray nozzles, but the accuracy of the results depends on the computational resources available.

### 5.1.3 DROPLET MODELING

The last topic discussed is the use of using CFD to model the liquid jet breakup. This was accomplished by using a fine mesh that was able to capture the interface between the droplets. The simulation was able to begin to capture the breakup of the droplets. This approach was very computationally expensive because the mesh needed to be fine enough to capture the interface between the droplet and the air. This results in a mesh with a large number of cells which can limit the size of the domain that is used. Another problem that can arise comes from the Courant Number. From the Courant Number, as the cell size is reduced, the time step used for the simulation must also be reduced. The need for a reduction in the time step can also cause the simulation to become computationally expensive. Again, this suggests that computational fluid dynamics can be used to simulate the liquid jet breakup but the accuracy of the results depends on the computational resources available. This simulation also supports the use of using a passive scalar to model the spray plume. While the passive scalar does not capture the droplets, the passive scalar can be used to capture the overall shape and distribution within the spray plume.

### 5.2 FUTURE WORK

This work focused on determining whether computational fluid dynamics could be used to accurately model spray nozzles. Three major factors were examined which were an investigation of mesh and physics model effects, determining a method to pulse the spray and compare simulation data to experimental data, and determine whether the liquid jet breakup within the spray plume could be modeled. By investigating these topics, this work has provided a solid footing to be expanded upon. Areas of expansion could include investigating other types of spray nozzles, improving upon the results that were compared
to experimental data, investigating whether CFD could be used to study drift, or determining a better method to model the liquid jet breakup such as investigating the use of an adaptive mesh.
APPENDIX

Step 1: Open a new file.

Figure A.1: Step 1

Select the processor settings and press OK.

Step 2: Import the geometry

Figure A.2: Step 2
Select Import Surface Mesh then select the file and follow the import dialog.

Step 3: Separate the faces into surfaces

Right click on the geometry and select Split by Patch.

Step 4: Name the faces
Click on each face and give it a name then click Create until the geometry is broken into all of the desired regions. Click Close.

**Step 5: Assign parts to regions.**

![Image of Step 5: Assign parts to regions]

Right click on part name in the tree and select Assign Parts to Regions. In the dialog select Create a Region for Each Part and Create a Boundary for Each Part Surface. Click Apply once then click Close.

**Step 6: Select the region types.**

Expand the Boundaries folder under the Region node and set the region types under properties for each region.

**Step 7: Create the mesh continuum.**

Right click on the continuum node and select New Mesh Continuum.

**Step 8: Select mesh models**

Select the desired mesh models then click Close.

**Step 9: Create the volume shapes for volumetric control**

Right click on the Volume Shape node under Tools ⇒ Volume Shape and select New Shape Block.
Step 10: Enter the information for the block.
Enter the coordinates for each block and click Create. When all blocks are created, click Close.

Step 11: Enter properties for mesh.
Enter the base size for the mesh under Mesh1 ⇒ Models. To create a new volumetric
control right click Volumetric Control under Mesh1 ⇒ Volumetric Control and select New Volumetric Control. Select the blocks in the shape field under the new Volumetric Control 1 node.

Step 12: Set the cell size for the volumetric control.

Select isotropic under Continua ⇒ Mesh 1 ⇒ Volumetric Control ⇒ Volumetric Control
1 ⇒ Mesh Conditions ⇒ Trimmer. Then set the Relative Size in the Percentage of Base field under Continua ⇒ Mesh 1 ⇒ Volumetric Control ⇒ Volumetric Control 1 ⇒ Mesh Values ⇒ Custom Size.

Step 13: Refine mesh around the nozzle.

Under Regions ⇒ Boundaries ⇒ nozzle ⇒ Mesh Conditions ⇒ Custom Surface Size,
select enable Custom Surface Size. Under the Regions ⇒ Boundaries ⇒ nozzle ⇒ Mesh Values ⇒ Surface Size node set the Relative Minimum Size and Relative Target Size in the Percentage of Base Field. Then click the Create Mesh button.

Step 14: Select the physics models.

Right click on the Physics 1 node under Continuum ⇒ Physics 1 and select Select Physics
Models. Select the desired physics models then click Close.

Step 15: Set the Eulerian Phases.

Right click Eulerian Phases under Continua ⇒ Physics 1 ⇒ Models ⇒ Eulerian Multiphase ⇒ Eulerian Phases and select New Eulerian Phase. Expand the new Phase 1 node then right click on Models and select Select Models. Select the desired models then
click Close. Repeat for all Eulerian Phases.

Step 16: Select the multiphase interaction models.

Right click Phase Interactions under Continua ⇒ Physics 1 ⇒ Models ⇒ Multiphase Interactions and select Create New Phase Interactions. Expand the new Phase Interaction 1 node and right click models. Select Select Phase Interaction Models. Select the desired models and click Close.

Step 17: Specify primary and secondary phases.


Step 18: Create a new passive scalar.

Right click the Passive Scalars node under Continuum ⇒ Physics 1 ⇒ Models ⇒ Passive Scalar and select New.

Step 19: Specify reference values and initial conditions.

Under the Continua ⇒ Physics 1 ⇒ Reference Values set make any required changes to the reference values. Under Continua ⇒ Physics 1 ⇒ Initial Conditions ⇒ Volume
Fraction set the initial condition for the Volume Fraction in the value node.

Step 20: Create the pulsed velocity field function.

Right click the Field Functions node under Tools node and select New ⇒ Scalar.

Step 21: Define the field function.

In the new Field Function 1 node, click the Definition field. Enter in the field function to
define the pulsed velocity inlet then click OK.

Step 22: Define the physics values for each boundary, initialize the solution, and run the simulation.

Under Regions ⇒ fluid ⇒ Boundaries specify the physics values for each Passive Scalar, Velocity Magnitude, and Volume Fraction for the to inlets. For the nozzle inlet change the
Method in the Velocity Magnitude node to Field Function and select the pulsed velocity inlet field function. Click the green flag to initialize the solution then click Run to run the solution.
REFERENCES


application quality and homogeneity in super-intensive olive tree canopies,”

spray generation and droplet transport,” Biosystems Engineering, vol. 92,


breakup in turbulent coaxial air flow,” International Journal of Multiphase