



Faculty of Engineering

**Design and Evaluation of Modified Laval Nozzle for Atomization
Performance**

Tang Shanshan

**Doctor of Philosophy
2025**

Design and Evaluation of Modified Laval Nozzle for Atomization Performance

Tang Shanshan

A thesis submitted

In fulfillment of the requirements for the degree of Doctor of Philosophy

(Mechanical Engineering)

Faculty of Engineering
UNIVERSITI MALAYSIA SARAWAK

2025

DECLARATION

I declare that the work in this thesis was carried out in accordance with the regulations of Universiti Malaysia Sarawak. Except where due acknowledgements have been made, the work is that of the author alone. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Tang Shanshan

.....

Signature

Name: Tang Shanshan

Matric No.: 21010368

Faculty of Engineering

Universiti Malaysia Sarawak

Date: 22 May 2025

ACKNOWLEDGEMENT

I would like to take this opportunity to those who have contributed directly or indirectly to my thesis.

First, I would like to express my deepest gratitude to my main supervisor, Associate Professor Ir Dr Mohd Danial Bin Ibrahim, and co-supervisor, Professor Ir Dr Andrew Ragai Henry Rigit, for their supervision, mentoring, and continuous encouragement through the development of this research. There were moments of hesitation and uncertainty during this research, but they tried to share with a kind and patient explanation, which had a soothing effect. With their patience and encouragement, I could finish my doctoral thesis.

Then, I would like to express my gratitude to my husband (Chaokun Wei), my parents, and my two sons (Zhangxun Wei and Zhangyu Wei). In particular, my husband provided me with material and emotional support, making my studies in Malaysia much easier. Additionally, I would like to thank the Faculty of Mechanical Engineering lecturers and my supportive friends, whose assistance greatly accelerated my progress.

Finally, I sincerely thank the Centre for Graduate Studies for the advice and support given during my study at Universiti Malaysia Sarawak and the management of the Universiti Malaysia Sarawak for making it possible for me to complete my studies here in Sarawak. Thank you all.

ABSTRACT

Particulate matter generated during the mining, processing, and transportation of coal and stone materials has become one of the primary sources of air pollution, especially fine particles with small particle sizes. The traditional nozzle requires high-pressure conditions, and the atomization droplets are large and uneven. This study aims to design a nozzle with a simple structure and good atomization performance and to study the influence of nozzle structure parameters and operating conditions on atomization performance. First, a linear Laval nozzle was designed. Then, the VOF (Volume of Fluid) model and DPM (Discrete Phase Model) of Fluent are used to simulate the internal and external fields of the nozzle and analyze the velocity, pressure, droplet size, and atomization angle. Finally, the droplet size and angle of the Laval nozzle spray were measured by experiments, and comparisons with numerical analysis were also conducted. The results show that the flow inside the linear Laval nozzle can reach supersonic speed. The optimized water pressure parameters are 0.1 MPa, air pressure of 0.5 MPa, nozzle outlet diameter of 2.5 mm, and oscillator distance of 4 mm. Compared to traditional nozzles, the water pressure is reduced by over 90%, and the D_(3,2) droplet diameter decreases by over 50%. Moreover, the spray angle increases by approximately 67.5%. Some fitted curves have been plotted, enabling the prediction of velocity and atomization angle within a specific range. The linear Laval nozzle studied in this research is easy to manufacture, exhibits excellent atomization performance, operates at low pressure, and is cost-effective. Particularly, a combination-type nozzle is proposed, which expands the atomization angle. The linear nozzles are applicable in diverse enclosed environments such as mines, construction sites, and processing plants. They can improve dust removal efficiency effectively, thus significantly contributing to human health and environmental protection.

Keywords: Laval nozzle, CFD, volume of fluid, discrete phase model, experiment, atomization performance

Reka Bentuk dan Penilaian Nozel Laval Diubah Suai untuk Prestasi Pengabusan

ABSTRAK

Zarah yang terhasil semasa perlombongan, pemprosesan, dan pengangkutan batu arang dan bahan batu telah menjadi salah satu punca utama pencemaran udara, terutamanya zarah halus dengan saiz zarah kecil. Muncung tradisional yang digunakan memerlukan keadaan tekanan tinggi, dan titisan pengabusan adalah besar dan tidak sekata. Kajian ini bertujuan untuk mereka bentuk satu muncung dengan struktur yang mudah dan prestasi pengabusan yang baik serta mengkaji pengaruh parameter struktur muncung dan keadaan pengendalian terhadap prestasi pengabusan. Pertama, satu muncung Laval linear direka. Kemudian, model VOF (isi padu bendalir) dan DPM (model fasa diskret) dalam Fluent digunakan untuk mensimulasikan medan dalaman dan luaran muncung dan menganalisis halaju, tekanan, saiz titisan, dan sudut pengabusan. Akhirnya, saiz titisan dan sudut semburan muncung Laval diukur melalui eksperimen, dan perbandingan dengan analisis berangka juga dilakukan. Keputusan menunjukkan bahawa aliran dalam muncung Laval linear dapat mencapai halaju supersonik. Parameter tekanan air yang dioptimumkan adalah 0.1MPa, tekanan udara 0.5MPa, diameter laluan muncung 2.5mm, dan jarak penggetar 4mm. Berbanding dengan muncung tradisional, tekanan air dikurangkan lebih dari 90%, dan diameter titisan $D_{(3,2)}$ berkurangan lebih dari 50%. Selain itu, sudut semburan meningkat sekitar 67.5%. Beberapa lengkung padanan telah digambarkan, membolehkan ramalan halaju dan sudut pengabusan dalam julat tertentu. Laval linear yang dikaji dalam kajian ini mudah untuk dihasilkan, mempunyai prestasi pengabusan yang cemerlang, beroperasi pada tekanan rendah, dan berkos rendah. Secara khusus, satu jenis muncung gabungan dicadangkan, yang memperluaskan sudut pengabusan. Muncung linear ini sesuai digunakan dalam pelbagai persekitaran tertutup seperti perlombongan, tapak pembinaan,

dan kilang pemrosesan. Ia boleh mengurangkan keberkesanan penyingkiran debu dengan berkesan, memberikan sumbangan yang signifikan kepada kesihatan manusia dan perlindungan alam sekitar.

Kata kunci: *Muncung Laval, CFD, isi padu bendalir, model fasa diskret, eksperimen, prestasi pengabusan*

TABLE OF CONTENTS

	Page
DECLARATION	i
ACKNOWLEDGEMENT	ii
ABSTRACT	iii
<i>ABSTRAK</i>	v
TABLE OF CONTENTS	vii
LIST OF TABLES	xiii
LIST OF FIGURES	xv
LIST OF ABBREVIATIONS	xix
LIST OF SYMBOLS	xxi
CHAPTER 1 INTRODUCTION	1
1.1 Background	1
1.2 Problem Statement	4
1.3 Research Questions	6
1.4 Research Objectives	6
1.5 Research Hypotheses	6
1.6 Scope of the Research	7
1.7 Research Gap	8

1.8	Thesis Outline	8
CHAPTER 2	LITERATURE REVIEW	10
2.1	Introduction	10
2.2	Dust and Dust Suppression Method	10
2.2.1	Definition of Dust	10
2.2.2	Dust Particle Size and Particle Size Distribution	11
2.2.3	Dust Suppression Method	16
2.2.3.1	Types of Dust Suppression Methods	16
2.2.3.2	Atomized Dust Suppression Technology	18
2.3	Jet Flow and Atomization	21
2.3.1	Jet Flow	21
2.3.2	Atomization	22
2.3.2.1	Atomization Principle	22
2.3.2.2	Supersonic Atomization Principle	25
2.4	Aerodynamics and Compressible Flow	29
2.4.1	Compressible Flow	29
2.4.2	Shock Waves and Expansion Waves	32
2.5	Nozzle	35
2.5.1	Two-phase Flow Nozzle	36
2.5.2	Supersonic Laval Nozzle	43

2.6	Factors Affecting the Effect of Dust Removal	44
2.7	Numerical Analysis Model and Experimental Measuring Tool	48
2.7.1	Numerical Analysis Model	48
2.7.2	Experimental Measuring Tool	50
2.8	Chapter Summary	52
CHAPTER 3	RESEARCH METHODOLOGY	54
3.1	Introduction	54
3.2	Research Variables	54
3.3	Research Tools	55
3.3.1	SolidWorks and ANSYS Fluent	56
3.3.2	Laser Particle Size Analyzer	57
3.4	Flow Chart	59
3.5	Research Process	60
3.5.1	Laval Nozzle Structure Design	60
3.5.1.1	Laval Nozzle Structure	60
3.5.1.2	Laval Nozzle Size Determination	60
3.5.2	Numerical Simulation of the Internal Flow Field in the Nozzle	64
3.5.3	Numerical Analysis with Varied Water Pressure	68
3.5.3.1	VOF Simulation Setup	69
3.5.3.2	DPM Simulation Setup	75

3.5.4	Numerical Analysis with Varied Air Pressure	81
3.5.5	Numerical Analysis with Varied Nozzle Outlet Diameters	81
3.5.6	Numerical Analysis with Varied Oscillator Distances	82
3.5.7	Experiments with Varied Water Pressure	82
3.5.7.1	Experimental Process	82
3.5.7.2	Experimental Equipment	85
3.5.8	Experiments with Varied Air Pressure	90
3.5.9	Mesh Independence Test and Residual Plot	89
3.6	Summary of the Chapter	91
CHAPTER 4	RESULTS, ANALYSES AND DISCUSSIONS	93
4.1	Overview	93
4.2	Verification of the Supersonic Effect of the Laval Nozzle	93
4.2.1	Numerical Simulation Results of the Internal Flow Field in the Nozzle	93
4.2.2	Comparision with Other Numerical Analysis Results	94
4.3	Results of Numerical Analysis with Varied Water Pressure	96
4.3.1	Velocity Analysis	96
4.3.2	Atomization Angle Analysis	100
4.3.3	Droplet Size Analysis	106
4.3.4	Mach Number Analysis	113
4.4	Results of Numerical Analysis with Varied Air Pressure	114

4.4.1	Velocity Analysis	114
4.4.2	Atomization Angle Analysis	117
4.5	Results of Numerical Analysis with Varied Nozzle Diameters	124
4.5.1	Velocity Analysis	124
4.5.2	Atomization Angle Analysis	128
4.6	Results of Numerical Analysis with Varied Oscillator Distances	135
4.6.1	Velocity Analysis	135
4.6.2	Atomization Angle Analysis	138
4.7	3D Model Validation	142
4.7.1	3D Model Creation	142
4.7.2	Numerical Analysis Comparison of 3D and 2D Models	142
4.8	Analysis of Experimental Results with Varied Water Pressure	144
4.8.1	Analysis of Atomization Angle and Droplet Size	144
4.8.1.1	Atomization Angle Analysis	144
4.8.1.2	Droplet Size	146
4.8.2	Comparison between Experimental Results and Numerical Analysis	152
4.8.2.1	Angles of Atomization Range	152
4.8.2.2	Sauter Mean Diameter	154
4.9	Analysis of Experimental Results with Varied Air Pressure	155
4.9.1	Atomization Angle Analysis	155

4.9.2	Comparison between Experimental Results and Simulation Results	158
4.10	Research Contribution	160
4.11	Summary of the Chapter	163
CHAPTER 5	CONCLUSION, INNOVATIONS AND RECOMMENDATIONS	164
5.1	Conclusion	164
5.2	Innovations	165
5.3	Recommendations for Future Work	166
REFERENCES		167
APPENDICES		185

LIST OF TABLES

	Page
Table 2.1 Equivalent diameter representation of irregularly shaped particles (Maroof <i>et al.</i> , 2020; Su & Yan, 2020)	12
Table 2.2 Summary of the papers on the structural improvement of dust remover	45
Table 2.3 Summary of the papers on optimization of the operating condition of dust remover	47
Table 2.4 Summary of the papers on the numerical analysis model	49
Table 3.1 DPM droplet source Settings	80
Table 3.2 The summary of numerical simulation for various parameters	82
Table 3.3 Experiments with varied air pressure	90
Table 3.4 The grid independence test with varied element size	90
Table 4.1 The maximum velocity of the mixture under varied water pressures	99
Table 4.2 Angle of atomization range under varied water pressures	105
Table 4.3 Sauter mean diameter (SMD) under varied water pressures	112
Table 4.4 The maximum velocity of the mixture under varied air pressures	116
Table 4.5 Angle of atomization range under varied air pressures	122
Table 4.6 Maximum velocity magnitude of mixture with varied nozzle outlet diameters	126
Table 4.7 Angle of atomization range with varied nozzle outlet diameters	133
Table 4.8 Maximum velocity magnitude of mixture with varied oscillator distances	137
Table 4.9 Angle of atomization range with varied oscillator distances	140
Table 4.10 Atomization angle at varied water pressures by experiment	145
Table 4.11 Droplet size characteristic parameters under varied water pressures (Air pressure is 0.5 MPa)	150

Table 4.12	A comparison between the angles of atomization range by numerical analysis and experimental atomization angles under varied water pressures	152
Table 4.13	A comparison between $D(3,2)$ by numerical analysis and $D(3,2)$ by experiments under varied water pressures	154
Table 4.14	Atomization angle under varied air pressures by experiments	157
Table 4.15	A comparison between the numerical analysis of angles of atomization range and the experimental atomization angles under varied air pressures	159
Table 4.16	The comparison of water pressure between the traditional nozzle and the optimized Laval nozzle	160
Table 4.17	The comparison of spray angle between traditional nozzles and optimized Laval nozzles	161
Table 4.18	The comparison of droplet size $D(3,2)$ between traditional nozzles and optimized Laval nozzles	162

LIST OF FIGURES

	Page
Figure 1.1 Statistics of new cases of occupational disease and pneumoconiosis in China from 2017 to 2022 (Zhao <i>et al.</i> , 2020)	2
Figure 1.2 Dust pollution (Sager, 2020)	2
Figure 1.3 Laval model (Sun <i>et al.</i> , 2023)	4
Figure 2.1 Droplet and dust (Zhu <i>et al.</i> , 2021)	11
Figure 2.2 Fine water mist dust removal (Zhou & Qin, 2021)	16
Figure 2.3 Internal mixed air-assisted nozzle (Watanawanyoo <i>et al.</i> , 2012)	37
Figure 2.4 External mixed air-assisted nozzle (Watanawanyoo <i>et al.</i> , 2012)	38
Figure 2.5 Compound air-assisted nozzle (Sarghini & De Vivo, 2019)	39
Figure 2.6 Prefilm jet nozzle (Carvalho & Heitor, 1998)	40
Figure 2.7 Bubble nozzle (Ponasse <i>et al.</i> , 1998)	41
Figure 2.8 Air whistle nozzle (Pang <i>et al.</i> , 2022b)	42
Figure 2.9 Laval structure and Laval nozzle (Bian <i>et al.</i> , 2018)	43
Figure 3.1 Spray cone angle (Sovani <i>et al.</i> , 2001)	55
Figure 3.2 Nozzle model drawn in Solidworks	56
Figure 3.3 Fluent simulation in this research	57
Figure 3.4 Laser particle size analyser (Blott <i>et al.</i> , 2004)	58
Figure 3.5 Flow chart	59
Figure 3.6 Laval structural parts drawing	62
Figure 3.7 Nozzle assembly drawing: ① Mechanical ultrasonic oscillator; ② Laval structure; ③ Main body; ④ Connecting rod; ⑤ shell	63
Figure 3.8 3D modeling with SolidWorks	63
Figure 3.9 Laval combined nozzle	64
Figure 3.10 Fluid region of the Laval structure	66

Figure 3.11	Mesh generation	67
Figure 3.12	Solution method	68
Figure 3.13	Internal and external flow fields of the nozzle	71
Figure 3.14	An enlarged view of the flow field around the nozzle	71
Figure 3.15	Boundary conditions for external flow fields	71
Figure 3.16	Boundary conditions for nozzle	72
Figure 3.17	The mesh for the internal and external fluid regions of the nozzle	73
Figure 3.18	The mesh surrounding the nozzle	74
Figure 3.19	The mesh inside the nozzle	74
Figure 3.20	The inflation of the boundary layer	74
Figure 3.21	VOF solution method	75
Figure 3.22	DPM solution methods	80
Figure 3.23	Test system diagram	84
Figure 3.24	Working principle of the test system	85
Figure 3.25	Water tank and air compressor	86
Figure 3.26	Booster pump and valve	87
Figure 3.27	Laser particle analyzer and the location during its usage	87
Figure 3.28	Water manometer and barometer	88
Figure 3.29	Data collection computer and Laval nozzle	88
Figure 3.30	Photos of the experimental site	89
Figure 3.31	(a) convergence criteria; (b) residual plot	91
Figure 4.1	Contours of Mach number	94
Figure 4.2	Flow separation occurred in the Laval nozzle diverging section (Zhang <i>et al.</i> , 2020a)	95
Figure 4.3	The maximum speed of droplets in Zhang's paper (Zhang <i>et al.</i> , 2020b)	95
Figure 4.4	The velocity magnitude of droplets in this research	96

Figure 4.5	Contours of velocity magnitude at varied water pressures	97
Figure 4.6	Scatter plot of velocity magnitude under varied water pressures	99
Figure 4.7	Contours of water volume fraction at varied water pressures	100
Figure 4.8	Angle of atomization range at varied water pressures	103
Figure 4.9	Scatter plot of the angle of atomization range under varied water pressures	106
Figure 4.10	Droplet traces colored by droplet diameter at varied water pressures	107
Figure 4.11	Statistical droplet size distribution at varied water pressures	111
Figure 4.12	Scatter plot of Sauter mean diameter (SMD) under varied water pressures	112
Figure 4.13	Contours of Mach number at 0.1 MPa water pressure	113
Figure 4.14	Contours of velocity magnitude at varied air pressures	114
Figure 4.15	Scatter plot of the maximum velocity of the mixture under varied air pressures	116
Figure 4.16	Contours of water volume fraction at varied air pressures	117
Figure 4.17	Angle of atomization range at varied air pressures	120
Figure 4.18	Scatter plot of the angle of atomization range under varied air pressures	123
Figure 4.19	Contours of velocity magnitude with varied nozzle outlet diameters	124
Figure 4.20	Scatter plot of maximum velocity magnitude of mixture with varied nozzle outlet diameters	127
Figure 4.21	Contours of water volume fraction with varied nozzle outlet diameters	128
Figure 4.22	Angle of atomization range with varied nozzle outlet diameters	131
Figure 4.23	Scatter plot of the angle of atomization range with varied nozzle outlet diameters	134
Figure 4.24	Contours of velocity magnitude with varied oscillator distances	135
Figure 4.25	Scatter plot of maximum velocity magnitude of mixture with varied oscillator distances	137
Figure 4.26	Angle of atomization range with varied oscillator distances	138

Figure 4.27	Scatter plot of the angle of atomization range with varied oscillator distances	141
Figure 4.28	Linear Laval nozzle 3D model	142
Figure 4.29	Numerical Analysis Comparison of 3D and 2D Models	143
Figure 4.30	Atomization angle at varied water pressures by experiments: (a) 0.1 MPa; (b) 0.2 MPa; (c) 0.3 MPa; (d) 0.4 MPa; (e) 0.5 MPa	144
Figure 4.31	Scatter plot of atomization angle at varied water pressures by experiment	146
Figure 4.32	Droplet size testing report: (a) Droplet size characteristic parameters; (b) Droplet size distribution graph; (c) Droplet size distribution table.	148
Figure 4.33	Droplet size testing report at varied water pressures: (a) 0.1 MPa; (b) 0.2 MPa; (c) 0.3 MPa; (d) 0.4 MPa; (e) 0.5 MPa.	149
Figure 4.34	Droplet size characteristic parameters at varied water pressures by experiment: (a) D10, D25, D50, D75, and D90; (b) D(3,2).	151
Figure 4.35	A comparison between the angles of atomization range by numerical analysis and experimental atomization angles under varied water pressures	153
Figure 4.36	a comparison between D(3,2) by numerical analysis and D(3,2) by experiments under varied water pressures	154
Figure 4.37	Atomization angle at varied air pressures by experiments: (a) 0.2 MPa; (b) 0.3 MPa; (c) 0.4 MPa; (d) 0.5 MPa; (e) 0.6 MPa	156
Figure 4.38	Scatter plot of atomization angle at varied air pressures by experiments	158
Figure 4.39	A comparison between the numerical analysis of angles of atomization range and the experimental atomization angles under varied air pressures	159
Figure 4.40	The angle between the nozzles in a combination nozzle structure	161

LIST OF ABBREVIATIONS

CAD	Computer-aided design
CFD	Computational Fluid Dynamics
CGS	Centre for Graduate Studies
Co., Ltd.	Company, Limited
DPM	Discrete Phase Model
LCN	Laval combined nozzle
LLN	Linear Laval nozzle
LPSA	Laser particle size analyzer
ME	Mechanical Engineering
mm	millimeter
MPa	Megapascal
OPT	Online Protractor tool
PDF	Probability density function
PM	Particulate Matter
Re	Reynolds number
SMD	Sauter mean diameter
TPF	Two-phase Flow
TSP	Total Suspended Particulate
μm	Micrometer
UNIMAS	Universiti Malaysia Sarawak
VOF	Volume of Fluid
We	Weber number

D ₁₀	Droplet diameter at which the cumulative distribution curve is equal to or less than 10%
D ₂₅	Droplet diameter at which the cumulative distribution curve is equal to or less than 25%
D ₅₀	Droplet diameter at which the cumulative distribution curve is equal to or less than 50%
D ₇₅	Droplet diameter at which the cumulative distribution curve is equal to or less than 75%
D ₉₀	Droplet diameter at which the cumulative distribution curve is equal to or less than 90%

LIST OF SYMBOLS

A	Windward area of fog drops (m^2)
A_m	Amplitude (m)
A_p	Projected area of the particle (m^2)
A_s	Surface area of particle (m^2)
C_d	Drag coefficient
C_p	Specific heat capacity of particles [$\text{J}/(\text{kg} \cdot \text{K})$]
$C_{p,i}$	Specific heat of volatiles precipitated [$\text{J}/(\text{kg} \cdot \text{K})$]
$D(3,2)$	Sauter mean diameter (m)
$D_{T,P}$	Thermal swimming force coefficient
E	Total energy density (J/m^3)
F	Volume fraction
F'	Body forces due to external actions (N)
F_d	Drag force (N)
$F(d)$	Cumulative distribution of particles
K_C	Wave number (m^{-1})
L_1	Length of the converging section (m)
L_2	Length of the diverging section (m)
p_p	Pressure (Pa)
R_{rep}	Repeatability error
T	Temperature (K)
T_F	Fluid temperature (K)

T_p	Particle temperature (K)
V	The volume of the particle (m ³)
C	Wave velocity (m/s)
d	Target particle size (m)
d_0	Nozzle outlet diameter (m)
d_1	Inlet diameter (m)
d_2	Diameter of the throat (m)
d_3	Diameter of the outlet (m)
d_a	Aerodynamic diameter (m)
d_i	Diameter of the i-th particle (m)
d_p	Equivalent diameter (m)
$d_{p'}$	Actual diameter of the particle (m)
d_{pi}	The size of particles in the section i (m)
f_m	Cumulative distribution of mass under sieve
f_n	Cumulative distribution of quantity under sieve
$f(x)$	Probability density function (PDF) of the particle size distribution
g	Acceleration of gravity (m/s ²)
h_{fg}	Latent heat from volatilization analysis (J/kg)
i, j	Tensor indices
k	Gas thermal conductivity [W/(m·K)]
k_p	Thermal conductivity of particles [W/(m·K)]
l	Length (m)
m	Total number of droplets

m_p	Particle mass (kg)
$m_{p,0}$	Initial mass of particles (kg)
$\dot{m}_p$	Particle mass flow rate (kg/s)
$\dot{m}_{p,0}$	Initial mass flow rate of particles (kg/s)
n	Total number of particles
n_i	The number of particles in the section i
t	Time (s)
u	Flow velocity (m/s)
u_0	Liquid exit velocity (m/s)
u_a	Air velocity (m/s)
u_l	Droplet velocity (m/s)
u_p	Particle velocity (m/s)
u_v	Fluid velocity (m/s)
x	Particle size variable
z	Times of the measurement
Φ	Viscous dissipation function [W/m ³]
α	Half angle of the diverging (°)
β	Constant
δ_{re}	Relative error
θ	Cone angle of the converging section (°)
κ	Thermal conductivity [W/(m · K)]
κ'	Geometric mean particle size (m)

λ	Wavelength (m)
λ_c	Certain critical value (m)
λ_g	Mean molecular free path of gas
μ	Dynamic viscosity of the fluid ($\text{p}_a \cdot \text{s}$)
ν	Hydrodynamic viscosity coefficient (N.s/m^2)
ρ	Density (kg/m^3)
ρ_a	Air density (kg/m^3)
ρ_l	Liquid density (kg/m^3)
ρ_p	Particle density (kg/m^3)
ρ_v	Fluid density (kg/m^3)
σ	Geometric standard deviation
ξ	Surface tension of a liquid (N/m)
w	Angular frequency of a wave (rad/s)
$\nabla \cdot (\rho \vec{u})$	The divergence of density flow [$\text{kg}/(\text{m}^3 \cdot \text{s})$]
$\nabla \cdot (\rho \vec{u} \vec{u} + P_p \vec{I})$	Divergence of momentum flux (N/m^3)
Π	Average value of Z measurements
Π_n	Value of the nTH measurement
$\overline{d_i}$	Diameter of each droplet (m)
$\overline{m_p}$	Average particle mass (kg)
$\vec{u}$	Fluid velocity vector (m/s)
$\vec{f}$	Volume forces, such as gravity (N/kg)
$\vec{\tau}$	Viscous stress tensor (N/m^2)
$\vec{I}$	Unit matrix (Tensor form of the pressure term)

CHAPTER 1

INTRODUCTION

1.1 Background

In recent years, with the rapid development of the national economy and the great demand for resources, particulate matter generated during mining, processing, and transportation of coal and stone materials has become one of the primary sources of air pollution. Especially when small particles such as PM_{2.5} and PM₁₀ diffuse into the atmosphere, it is difficult to capture and remove them again, which will cause significant harm to human health and the environment, attract important social attention, and threaten human survival (Kazem *et al.*, 2020; Salamah *et al.*, 2022).

China is rich in coal resources. As of 2020, China has a proven coal reserve of 187.55 billion tons, making it the world's largest coal storage country and producer. By 2020, China's annual coal production was 3.84 billion tons, accounting for nearly half of global coal production. Coal mining produces a lot of dust, which seriously pollutes the working environment and causes occupational diseases such as pneumoconiosis. In China, over 90% of the nearly one million people diagnosed with occupational diseases are occupational pneumoconiosis, and 80% of them work in coal mines (Shekarian *et al.*, 2021). The statistics of new occupational illness and pneumoconiosis cases in China from 2017 to 2022 are shown in Figure 1.1.

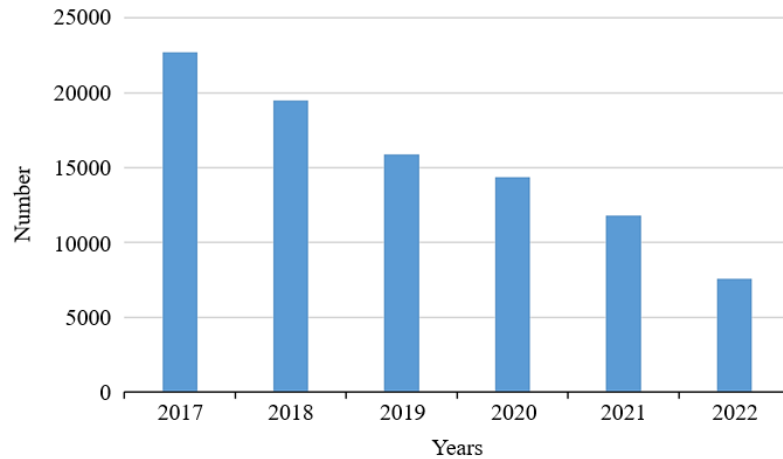


Figure 1.1: Statistics of new cases of occupational disease and pneumoconiosis in China from 2017 to 2022 (Zhao *et al.*, 2020)

Dust pollution is shown in Figure 1.2. The Chinese government promulgated the Law of the People's Republic of China on the Prevention and Control of Air Pollution on September 5, 1987, which has been amended and revised twice. The latest revision in 2018 highlighted that atmospheric environmental protection is closely related to the fundamental interests of the people and the sustainable, healthy development of the economy. It intensified efforts toward air quality governance in China and called on society to establish a principle of "breathing and striving together." (Yuan *et al.*, 2012).



Figure 1.2: Dust pollution (Sager, 2020)

According to the aerodynamic equivalent diameter, atmospheric particulate matter

can be classified into total suspended particulate matter (TSP), inhalable particulate matter (PM₁₀), and fine particulate matter (PM_{2.5}) (Li *et al.*, 2022). Among them, TSP refers to atmospheric particulate matter with a diameter less than or equal to 100 μm ; PM₁₀ refers to atmospheric particulate matter with a diameter less than or equal to 10 μm ; PM_{2.5} refers to atmospheric particulate matter with a diameter less than or equal to 2.5 μm atmospheric particulate matter (Orellano *et al.*, 2020).

In air pollution control, particulate matter—such as construction dust, road dust, transportation coal dust, and yard coal dust—produced by industrial activities in uncontained environments (e.g., coal mining, urban construction, and port transport) significantly contributes to PM_{2.5} levels. Due to its small particle size, large specific surface area, prolonged suspension time, long-distance transport, tendency to accumulate toxic substances, and ease of inhalation into the alveoli, PM_{2.5} poses a serious threat to public safety and citizens' physical and mental health. According to the Global Burden of Disease Report, PM_{2.5} is the fourth most important risk factor affecting public health in China (Peng *et al.*, 2020a).

Current status of dust removal technology: Atomized dust removal technology, Clustering-dust removal technology, Electric dust removal technology, Chemical agglomeration technique, and Negative pressure dust removal technology, among which atomized dust removal technology is the most widely used. In the process of atomized dust removal, when the size of atomized droplets is similar to the size of dust particles, the probability of dust adsorption and condensation is the largest, and the concentration of dust particles is correspondingly reduced to the minimum (Yao *et al.*, 2020).

The standards for the emission and control of particulate matter are becoming more and more stringent. Although the traditional dust removal technology has a good control effect on the total particulate matter, the removal effect of many fine particles is not ideal. The use of water mist to achieve particle control in an unorganized environment has become an important technical way to control particulate matter at present, and the generation of ultra-fine water mist by improving atomization technology to promote the collision and settlement of fine particles is a new method with great application potential (Sun *et al.*, 2023).

This study looks into the design of a Laval nozzle based on the Laval structure. Laval model is shown in Figure 1.3. Supersonic atomization technology is characterized by the outlet velocity of the atomization medium reaching the supersonic speed. The jet velocity plays a key role in the atomization process. So, compared with the general air atomization technology, the supersonic atomization technology can obtain finer droplets and more concentrated droplet size distribution.

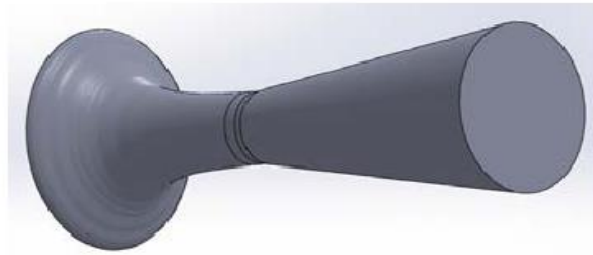


Figure 1.3: Laval model (Sun *et al.*, 2023)

1.2 Problem Statement

Dust pollution is a common environmental problem, especially in cities and regions with high industrialization (Yang *et al.*, 2016). Dust mainly comes from industrial production, transportation, construction, agricultural activities, and natural weathering. In

many cities, especially in developing countries, the concentration of dust in the air is high and exceeds environmental standards. This kind of pollution affects people's health very much. Industrial zones are often a major source of dust pollution, especially in areas surrounded by heavy industry and chemical plants. The production activities of these factories will release large amounts of dust, which will seriously impact the surrounding environment. On construction sites, construction activities also produce a lot of dust. Dust removal is essential to protect human health, reduce environmental pollution, and improve the production environment. At present, the main dust removal method is atomization dust removal.

Droplet sizes between 14 and 30 microns effectively capture respirable dust (Nie *et al.*, 2023). Traditional nozzles require high-pressure conditions and produce large and uneven atomized droplets. The removal effect of a large number of fine particles is not ideal. The jet velocity plays a key role in the atomization process. The Laval nozzle can generate smaller and more uniform droplets under low-pressure conditions. There is limited research in the field of dust removal, especially Linear Laval nozzle, which are easier to manufacture and have lower costs (Luo *et al.*, 2022; Wang *et al.*, 2022; Yang *et al.*, 2020). Water pressure, air pressure, and orifice diameter significantly impact the atomization performance of nozzles, but there is almost no research on the effect of water pressure, air pressure, or orifice diameter on Linear Laval nozzles. Additionally, the distance of the vibration chamber can affect the atomization angle, but there is also limited research in this area (Ni *et al.*, 2023; Sun *et al.*, 2024; Wang *et al.*, 2024). Most studies on Laval nozzles are experimental, with a lack of numerical analysis research, particularly in the areas of VOF (Volume of Fluid) internal and external flow field analysis and DPM (Discrete Phase Model) particle size analysis (Jing *et al.*, 2022; Ren *et al.*, 2024; Tian *et al.*, 2024).

1.3 Research Questions

Based on the issues outlined and discussed above, this study has identified the following questions to be explored:

- i. What is the nozzle structure based on the Laval principle? How do water and air pressure affect the atomization performance of the Laval nozzle?
- ii. How do the orifice diameter and oscillator distances affect the atomization performance of the Laval nozzle?
- iii. How consistent are the numerical analysis and experimental results regarding the optimized nozzle spray field?

1.4 Research Objectives

The primary goal of this study is to improve the atomization performance of the dust removal nozzle. To achieve this goal, some specific objectives are set as below:

- i. To design the Laval nozzle and evaluate the influence of water and air pressure on Laval nozzle atomization performance.
- ii. To optimize the orifice diameter and oscillator distances of the Laval nozzle to improve the efficiency of inhalable dust removal.
- iii. To compare computational and experimental results of the spray characteristics of the modified Laval nozzle.

1.5 Research Hypotheses

The following are a few of the research hypotheses that have been developed to meet these objectives.

- i. The Laval nozzle can reach supersonic speeds internally; water pressure and air pressure are important factors influencing the atomization performance of the Laval nozzle.
- ii. The orifice diameter and oscillator distances are key factors that significantly impact the atomization performance of the Laval nozzle.
- iii. The optimized Laval nozzle's numerical analysis and experimental results show the same trend.

1.6 Scope of the Research

In this research, the literature on dust removal were conducted. Incorporating relevant fluid mechanics theory, a modified Laval nozzle was designed, and a model was constructed in SolidWorks. The flow field was then simulated in Fluent to analyze the effects of varying water pressure, air pressure, orifice diameter, and vibration chamber distance. Finally, atomization performance was measured experimentally and compared with the numerical analysis results.

- i. Reviewed the literature and designed the nozzle structure.
- ii. Analyzed the flow field inside and outside the nozzle, including velocity, pressure, and atomization angle. The turbulent flow satisfies the continuity equation, is isentropically compressible, and follows the ideal gas law.
- iii. A laser particle size analyzer (DPSA) was used to analyze droplet size and distribution, and a high-speed camera was used to capture images to determine the atomization angle.
- iv. Origin was used to analyze the data and determine optimal parameters.

1.7 Research Gap

Extensive research on nozzle atomization and dust suppression worldwide covers various atomization methods, nozzle structures, and working conditions. These studies have significantly advanced the theory of atomization and dust suppression. In particular, when the size of the water mist droplets is comparable to that of suspended dust particles, the dust particles in the air can be effectively captured and settled. Theoretical studies have been widely recognized for improving dust suppression efficiency, and scholars have conducted extensive research on it and achieved many results.

However, despite these theories, there has been limited research on using Laval nozzles for dust suppression, particularly the Linear Laval Nozzle, which warrants further investigation. Therefore, this thesis builds upon previous research and focuses on the two-fluid nozzle based on the Laval structure. The nozzle's atomization performance and parameter optimization were studied through numerical analysis and experimentation. This research will provide a theoretical foundation and test data for applying Laval nozzles in dust suppression.

1.8 Thesis Outline

This thesis is categorized into five main chapters: Introduction, Literature Review, Research Methodology, Results, and Discussion, and Conclusion and Recommendation. Chapter 1 introduces the Background, Problem Statement, Research Questions, Research Objectives, Hypothesis, Outcome, Scope of Research, and Research gap. Chapter 2 reviews the current work that has been done on dust suppression. Chapter 3 provides details on the methods used in the study of dust suppression. The analysis of the data was presented and

discussed in Chapter 4. Finally, Chapter 5 provides the conclusion and recommendations of the study.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

In this chapter, the literature review includes dust and dust suppression methods, jet flow and atomization, aerodynamics and compressible flow, nozzle, factors affecting the effect of dust removal, and numerical analysis model and experimental measuring tool in sections 2.2, 2.3, 2.4, 2.5, 2.6 and 2.7, respectively. Next, the summary for this chapter is presented in section 2.8.

2.2 Dust and Dust Suppression Method

2.2.1 Definition of Dust

Dust generally refers to solid particles that can be suspended in the air for a long time. It mainly refers to those solid particles formed by mechanical impact, grinding, and rolling of solid materials. These solid particles are dispersed by air flow and suspended in the air. The particle size is mostly 0.25-20 μm , and most are 0.5-5 μm (Maring, 2003).

Respirable dust enters the trachea, bronchus, and alveoli from the larynx and may cause pneumoconiosis and other diseases. The deposition of dust particles in the respiratory system can be divided into three areas: The upper respiratory tract area (including nose, mouth, pharynx, and larynx), trachea and bronchus, and the alveolar area. The particle size with an aerodynamic diameter of more than 10 μm is generally believed to be mainly deposited in the nasopharynx, and the dust particles below 10 μm can enter the deep respiratory tract. Most of the dust deposited in the alveoli is dust particles below 5 μm , especially dust below 2 μm (Sarver *et al.*, 2021).

2.2.2 Dust Particle Size and Particle Size Distribution

Dust removal, shown in Figure 2.1, separates particulate pollutants from the gas. Dust here refers to aerosol particles suspended in a dispersed phase in a gas. Particle size is the most basic parameter to describe aerosol particles. A particle must be suspended in a gas to be called an aerosol, which means it must be small enough. It is believed that the upper limit of dust size is about $100\text{ }\mu\text{m}$, and particles of this size settle too quickly in the air (about $0.1\text{--}0.25\text{ m/s}$) to persist in aerosols. Particle sizes can be as small as nanometres, about the size of a cluster of molecules. Therefore, the size of aerosol particles ranges roughly from 0.001 to $100\mu\text{m}$ (Zhu *et al.*, 2021).

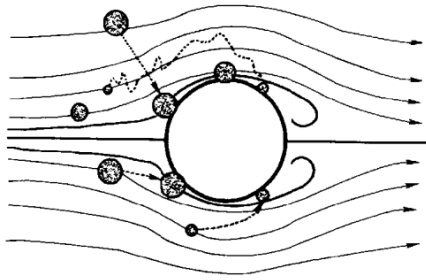


Figure 2.1: Droplet and dust (Zhu *et al.*, 2021)

Spherical particles refer to tiny particles with a spherical or approximately spherical shape. Spherical particles typically have relatively uniform surfaces with equal dimensions in any direction. Researchers or engineers often assume particles to be spherical because it simplifies mathematical modeling and analysis. Irregular particles, on the other hand, refer to tiny particles with irregular, non-uniform shapes. Their shapes could be polygons, prisms, or irregular curves. Unlike spherical particles, irregular particles typically have more complex surfaces, and their dimensions may not be equal in all directions.

Table 2.1: Equivalent diameter representation of irregularly shaped particles (Maroof *et al.*, 2020; Su & Yan, 2020)

Equivalent diameter	Definition	Computational formula
Length diameter	The diameter measured in a given direction	$d_p = l$
Average diameter	Diameters measured in multiple directions	$d_p = \frac{1}{n} \sum_{i=1}^n d_i$
Circumference diameter	The diameter of a circle with the same circumference P as the particle	$d_p = \frac{P}{\pi}$
Projected area diameter	The diameter of a circle with the same projected area A_p as the particle	$d_p = \sqrt{\frac{4A_p}{\pi}}$
Surface area diameter	The diameter of a ball with the same surface area A_s as the particle	$d_p = \sqrt{\frac{A_s}{\pi}}$
Volume diameter	The diameter of a ball with the same volume V_p as the particle	$d_p = \sqrt[3]{\frac{6V_p}{\pi}}$
Stokes diameter	The diameter of the ball with the same density ρ_p and settling velocity ν as the particle	$d_p = \sqrt{\frac{18\mu_p}{g(\rho_p - \rho_a)}}$
Aerodynamic diameter	The diameter of the ball with the same mass and standard density as the particle	$d_a = d_p \sqrt{\frac{\rho_p}{1000}}$

- l ——Length (m)
- P ——The projected perimeter of the particle (m)
- A_p ——Projected area of the particle (m^2)
- A_s ——Surface area of particle (m^2)
- V ——The volume of the particle (m^3)
- u_p ——Settling velocity of the particle (m/s)
- ρ_p ——The density of particles (kg/m^3)
- ρ_a ——Density of air (kg/m^3)
- g ——Acceleration due to gravity (m/s^2)
- n ——Total number of particles
- d_p ——Equivalent diameter (m)
- d_a ——Aerodynamic diameter (m)
- $d_{p'}$ ——Actual diameter of the particle (m)
- μ ——Dynamic viscosity of the fluid ($\text{p}_a \cdot \text{s}$)
- d_i ——Diameter of the i -th particle (m)

Spherical particles can be expressed in geometric diameters, but solid particles produced by industrial processes are usually non-spherical. The shape of irregular particles can be generalized into three categories: block, plate, and needle. The size of the irregular particle can be expressed as the equivalent diameter. There are many definitions of particle size, and the appropriate definition depends on the measurement method: for example, the particle size measured by an optical microscope is called the optical diameter. The particle size data obtained by the multistage cyclone separator or cascade impact separator is the aerodynamic diameter. For particles below $0.5 \mu\text{m}$, the diffusion separation method is called the diffusion particle size. The particle size obtained by the sedimentation method is Stokes diameter (Maroof *et al.*, 2020; Su & Yan, 2020). Table 2.1 shows an equivalent diameter representation of irregularly shaped particles.

Using "average" particle size alone is insufficient to characterize a collection of particles of different sizes. Therefore, it is necessary to use the particle size distribution to represent the dispersion degree of the particle swarm. Moreover, particle size distribution is also known as dispersion, which refers to the quantity fraction and mass fraction of particles in varied particle size ranges. The table, graph, and function methods can express particle size distribution.

i. Frequency Distribution

A histogram can be made according to the size range and the number of particles. Equation 2.1 and Equation 2.2 indicate Quantitative frequency distribution q_n and mass frequency distribution q_m (Lachs *et al.*, 2021).

$$q_n = \frac{n_i}{\sum n_i} \quad \text{Equation 2.1}$$

$$q_m = \frac{n_i d_{pi}^3}{\sum n_i d_{pi}^3} \quad \text{Equation 2.2}$$

n_i - - The number of particles in the section i ;

d_{pi} - - The size of particles in the section i (m).

ii. Density Distribution

Equation 2.3 and Equation 2.4 to indicate quantitative density distribution f_n and mass density distribution f_m (Sundell & Saylor, 2021).

$$f_n = \frac{q_n}{\Delta d_p} = \frac{dF_n}{dd_p} \quad \text{Equation 2.3}$$

$$f_m = \frac{q_m}{\Delta d_p} = \frac{dF_m}{dd_p} \quad \text{Equation 2.4}$$

f_n -- Cumulative distribution of quantity under sieve;

f_m -- Cumulative distribution of mass under sieve.

iii. Cumulative Distribution

The particle size distribution of aerosol particles can also be expressed as the cumulative distribution of particles $F(d)$ passing through or retained on a sieve. In practice, the cumulative distribution of particles passing through the sieve is often used for representation. This distribution refers to the fraction of the mass (or quantity) of all particles with a size less than a certain particle size d , relative to the total mass (or quantity), as shown in Equation 2.5 (Svensson *et al.*, 2022).

$$F(d) = \int_0^d f(x) dx \quad \text{Equation 2.5}$$

$F(d)$ -- Cumulative distribution of particles;

$f(x)$ -- Probability density function (PDF) of the particle size distribution;

x -- Particle size variable;

d -- Target particle size (m).

iv. Distribution Function.

Although the particle size distribution can be expressed in tables and graphs, in some cases, it is more convenient for mathematical analysis to be expressed in functional form. According to the actual measurement and long-term practice summary, Gregson *et al.* (2021)

found that the aerosol particles randomly generated in the production process mostly obey the lognormal distribution shown in Equation 2.6, no matter the mass density distribution function f_m or quantity density distribution function f_n (Gregson *et al.*, 2021).

$$f(d) = \frac{1}{d\sigma\sqrt{2\pi}} \exp\left(-\frac{(\ln d - \ln \kappa')^2}{2\sigma^2}\right) \quad \text{Equation 2.6}$$

κ' -- Geometric mean particle size (m);

σ -- Geometric standard deviation.

2.2.3 Dust Suppression Method

2.2.3.1 Types of Dust Suppression Methods

i. Atomized dust removal technology

In the dust removal and suppression process, when the particle size of the atomized droplet is similar to that of the dust particle, the probability of the dust being adsorbed and condensed is the largest, and the dust particle concentration is also reduced to the minimum. Fine water mist dust removal is shown in Figure 2.2.



Figure 2.2: Fine water mist dust removal (Zhou & Qin, 2021)

Atomized dust removal technology is based on this principle: water is atomized to form tiny droplets. This increases the contact area between the droplets and dust particles, promoting the condensation of atomized droplets onto the dust, thereby improving dust removal and suppression effectiveness.

ii. Agglomerative dust removal technology

In the process of removing fine particle dust, coalescence dust removal technology as a pretreatment method mainly includes acoustic coalescence technology, electric coalescence technology, and turbulence coalescence technology. As a pretreatment method of dust removal technology, aggregation technology is effective and has once attracted the attention of the dust removal community. However, there are still a series of problems, such as high energy consumption and noise in the application of aggregation technology, so the aggregation dust removal device still needs more in-depth research (Chu *et al.*, 2021).

iii. Electric dust removal technology

Electrostatic dust removal technology refers to the corona and dust poles on both ends of the high-voltage direct current. The strong electric field generates force gas ionization so that the dust particles are charged, with positive and negative ions, to the corona plate and dust plate, respectively, to achieve the effect of dust removal. Electrostatic dust removal technology mainly includes new electrostatic dust removal technology, electric bag composite dust removal technology, and wet electrostatic dust removal technology (Chanchangi *et al.*, 2020).

iv. Negative pressure dust removal technology

Negative-pressure dust removal technology uses high-pressure water ejected through a small nozzle at high speed, which collides with the atmosphere to form an atomized gas-

liquid mixture, creating a two-phase flow. At the same time, due to the negative pressure guiding effect of the ejector duct, the air is inhaled while the dust particles are inhaled together. The water mist in the air duct then condenses the inhaled dust particles. When the condensed particles move to the baffle in the air duct, they are impacted by the water flow gathered by the spray, achieving the dust removal purpose (Guo *et al.*, 2020).

v. Chemical agglomeration technology

Chemical agglomeration technology is a pretreatment method that promotes the removal of fine particles by adding chemical agglomeration agents. According to the different locations of agglomeration agents, chemical agglomeration technology can be divided into chemical agglomeration during combustion and chemical agglomeration after combustion. Chemical agglomeration after combustion refers to spraying a small amount of chemical agglomeration accelerator into soot to improve the solid bridge force and liquid bridge force between fine particles and promote the agglomeration of fine particles according to the flocculation theory. Chemical agglomeration after combustion can effectively promote the growth of fine particles and improve dust removal efficiency without changing the normal production conditions and existing dust removal facilities, so it has a good practical prospect (Porozova *et al.*, 2021).

2.2.3.2 Atomized Dust Suppression Technology

Atomized dust suppression technology mainly includes fine water mist dust removal technology, acoustic wave atomized dust removal technology, magnetized water dust removal technology, pre-charged electrospray dust removal technology, foam dust removal technology, and wetting agent dust removal technology.

i. Fine water mist dust removal technology

The dust removal technology of fine water mist is mainly based on the principle of aerodynamics. When the droplet size of fine water mist is similar to that of dust particles, fine water mist can more effectively remove dust in the air. The condensation nucleation mechanism will play a major role when the fine water mist evaporates into water vapor. The large amount of evaporation of fine water mist will quickly saturate the air humidity. The saturated water vapor will directly condense with the dust particles so that the particle size of the dust particles is increasing. When the particle size increases to 10 μ m-20 μ m, it will fall on its own or be easily removed by the general water mist (Liang *et al.*, 2022).

ii. Acoustic wave atomized dust removal technology

Acoustic atomization dust removal technology mainly starts by improving the atomization degree and the mutual binding rate of dust and spray droplets to remove respiratory dust. The technology uses an acoustic wave generator, which has the advantages of conventional nozzle atomization dust removal, which improves the spray effect and reduces water consumption. In addition, the high frequency and high energy sound wave generated by the acoustic generator atomizes the water droplets that have been broken and atomized twice so that the size of the atomized water droplets is reduced again, and the contact condensation effect between dust and atomized water droplets is improved (Kong *et al.*, 2024).

iii. Magnetized water dust removal technology

Magnetized water dust removal technology improves ordinary spray dust removal technology. It mainly uses the interaction between magnetic force and water mist droplets

to improve the atomization performance of water, reduce the size of fog droplets, and improve the efficiency of spray dust removal (Chang *et al.*, 2020).

iv. Pre-charged electrospray dust removal technology

The pre-charged electrospray dust removal technology is mainly based on the fact that most of the floating dust particles will have a certain charge due to the changes in the surrounding environment and the action of some charged particles issued in the field production process, which can make the atomized water droplets with the opposite charge of the dust particles so that the electrostatic attraction between the dust particles and the atomized water droplets. The contact probability between dust particles and atomized water drops is increased to improve the efficiency of atomized dust removal (Jin *et al.*, 2022).

v. Foam dust removal technology

The key to foam dust removal technology lies in the addition of surfactant and foaming agent in water, which generates a large number of foam groups through atomization and injection, increases the contact area between dust particles and foam, and improves the efficiency of spray dust removal (Ji *et al.*, 2022).

vi. Wetting agent dust removal technology

Wetting agent dust removal technology is the use of adding wetting agents that can reduce the water surface tension in water, improve the surface characteristics of dust particles, and enhance the affinity of dust particles for water spray, thereby improving the efficiency of spray dust removal (Bao *et al.*, 2012).

2.3 Jet Flow and Atomization

2.3.1 Jet Flow

Jet flow refers to the flow of fluid from various forms of orifice or nozzle into the same or another fluid. In water conservancy and hydropower engineering, aerospace engineering, water supply and drainage engineering, environmental engineering and chemical, metallurgy, energy, machinery, and many other fields, there will be many jet problems. If the fluid ejected from the nozzle is injected into the infinite space with the same fluid characteristics, it is called a submerged free jet. If the jet is in the laminar state, it is called a laminar jet. If the jet is turbulent, it is called a turbulent jet or turbulent jet. An impact jet refers to the impact flow of a free jet on a solid wall, which can be divided into three flow regions according to flow characteristics: free jet region, impact region, and wall jet region. Wall jet was introduced by Glauert (Mao *et al.*, 2021). A generalized wall jet impinges tangential or at a certain angle on a wall surrounded by static or moving fluid. The rotating jet is a compound flow of free jet plus rotation. The difference from the ordinary jet is that it has all-direction velocity (rotation velocity), so the fluid in the nozzle rotates to give the fluid coming out of the nozzle a tangential velocity component. In addition to the nonrotating jet's axial and radial velocity components, rotational velocity leads to radial and axial pressure gradients.

According to the number of jets, there are single, double, and even multiple jets. According to its flow characteristics, the double jet can be divided into two zones: the convergence zone and the union zone. For multiple impinging jets, in addition to the basic flow region of a single impinging jet, there are also the upper jet formation region and the upper jet region, which are formed by the collision of two or more adjacent wall jets. When the velocity of the liquid jet is large, or there is a velocity discontinuity, the aeration

phenomenon will occur; that is, the surrounding air is sucked into the jet to form a gas-liquid two-phase flow, such as the milky natural waterfall (Wu *et al.*, 2022).

2.3.2 Atomization

2.3.2.1 Atomization Principle

Atomization is the process of dispersing liquid into tiny droplets through nozzles or high-speed air streams and has been extensively studied since the early 20th century. The atomization of liquid is generally completed through a nozzle or atomizer, and the atomization process is also a competition process between internal and external forces. Among them, the external force mainly includes liquid pressure, aerodynamic force, etc., and the internal force mainly includes liquid surface tension and viscous force (Sescu, 2011). The surface tension and viscous forces of the liquid prevent the surface area of the droplet from shrinking and deformation, respectively. When the combined force of the surface tension and viscous forces of the liquid cannot resist the external force, the liquid will break into many smaller droplets. Because the small droplets formed in this initial crushing stage are unstable, many droplets will be further broken into smaller droplets under the shear action of the surrounding gas. That is, the secondary atomization process occurs (Sellens, 2016).

The condition of liquid atomization is to make the liquid into a very thin liquid film or a very fine jet by some method. The common methods are: 1) The flow channel through which the liquid flows is specially designed to realize the transformation of the liquid into a thin liquid film or a small jet, such as the flow channel into a small hole structure, the wall of the flow channel is designed to be a special groove, and the outlet of the flow channel is designed to be a flat, narrow slit. 2) The flow channel is designed as a rotating structure to

force the liquid to rotate at a high speed in the flow channel. Under centrifugal force, rotating friction, and side wall collision, the liquid film is unstable and broken into liquid filaments or droplets. The liquid filaments or droplets are further broken into small droplets after high-speed rotational motion (Alvim *et al.*, 2016). The crushing modes can be summarized as droplet crushing, liquid column crushing, and film crushing. The experiment proves that these three forms usually exist in different atomized regions during the liquid atomization stage.

(1) Droplet crushing

The atomization form of droplet crushing changes with different air flows. The spherical droplets are flattened into ellipsoids and then broken into small droplets. At this time, the droplets are usually in parallel or spiral air flows. Spherical droplets are stretched to a cigar shape and broken into small droplets, usually in a reservoir flow or a parallel hyperbola. Spherical droplets first appear on the bulge, then form a fold away from the bulk, then peel off and break into small droplets, usually in irregular airflow (Villermaux & Bossa, 2011).

(2) Liquid column crushing

Usually, the liquid column will be formed when the liquid is sprayed into the air from the nozzle. When the liquid column moves in the air, it will gradually break into new liquid columns of different shapes due to the joint action of internal and external forces. In most cases, the state and form of the jet depend on the injection pressure and injection speed of the liquid column. According to the gradual increase of the injection speed, the liquid column is divided into four forms (Bray *et al.*, 1991):

- i. Rayleigh fracturing: When the liquid jet velocity is very low, the liquid column does not break when it just leaves the nozzle, and the formation of the liquid column is almost independent of the aerodynamic force. Under the action of the liquid surface tension, the fracturing occurs after a certain distance. In this case, the fracturing length of the liquid column increases with the increase of the jet velocity.
- ii. The first type of wind-driven fracturing: in the case of a low-speed liquid jet, the liquid column does not break when it just leaves the nozzle; at this time, the effect of aerodynamic force will increase, thus enhancing the effect of surface tension of the liquid column. In this case, the fracturing length of the liquid column will decrease with the increase of the jet velocity.
- iii. The second type of wind-driven fragmentation: with the gradual increase of the jet velocity, the aerodynamic force plays a major role, and the surface of the liquid column will form unstable disturbances leading to instability; thus, jet fragmentation occurs. The broken length of the liquid column also gradually decreases.
- iv. Atomization: With the further increase of jet velocity, when it reaches a certain value, the liquid column will split and break up obviously at the moment it leaves the nozzle, and the broken length is approximately equal to zero (Rogala *et al.*, 2021).

(3) Liquid film crushing

The liquid film is formed at the outlet of the nozzle. When the external force is greater than the tension and viscous force on the surface of the liquid film, the liquid film will break into small droplets. There are three common forms of liquid film breakage (Rui *et al.*, 2023):

- i. Flanged broken: When the liquid's surface tension and viscous force are relatively large, the liquid surface deforms into a wheel-shaped liquid film.
- ii. Porous breaking: The liquid film is destroyed by moving small droplets around it, generating pores of different sizes. Under the aerodynamic force, the porous liquid film will be formed until it is finally broken.
- iii. Undulating fragmentation: The pressure generated by the liquid film increases with the increase of the turbulent kinetic energy of the liquid film, and the gradual increase of the pressure will peel the liquid sheet at the peak and valley of the film. Under external forces such as aerodynamics, the peeled liquid sheet will gradually deform into small droplets (Pang *et al.*, 2022a).

2.3.2.2 Supersonic Atomization Principle

According to the atomization principle analysis, jet velocity plays a key role in atomization. Compared with the general gas atomization technology, supersonic atomization technology is characterized by the outlet speed of the atomizing medium reaches supersonic speed, and supersonic atomization technology can obtain finer droplets and more concentrated droplet size distribution than the general gas atomization. The atomization process of a supersonic atomizing nozzle is extremely complex, and the collision and fragmentation of droplets accompany the unstable two-phase flow. In this process, environmental changes cause evaporation, condensation, mass transfer, and other phenomena. When fog droplets move in the air medium, they are affected by various internal and external forces. When the internal and external forces are unbalanced, the stable fog droplets will be distorted, unbalanced, and broken. The necessary condition for atomization is a contrast between the liquid and the surrounding gas (Mandal *et al.*, 2022).

With high relative speed, the supersonic atomizing nozzle can produce smaller droplets because of its ultra-high energy and ultra-intense turbulent pulse, which can tear the liquid film into filaments, and the broken droplets will be further atomized under the strong action of gas power.

The atomization process of a supersonic atomizing nozzle involves a complex two-phase flow. From the perspective of fluid mechanics, it can be considered that the Reynolds number (Re) and Weber number (We) are two critical factors for the splitting and breaking of fog droplets. Reynolds number is the ratio of the inertia and viscous forces of the fluid. When the Reynolds number is small, the influence of the viscous force on the flow field is greater than that of the inertial force. The disturbing force in the flow field will decrease, and the fluid tends to be stable, which is the laminar flow state. When the Reynolds number is large, the influence of inertia force is large, and the fluid is unstable, resulting in irregular turbulent flow field. Therefore, Re can be used as an indicator to measure liquid film splitting. When Re exceeds the critical value, it becomes turbulent state and liquid film splitting intensifies. The Weber number is the ratio of inertia force and surface tension of fog drops, which can be used as a measurement index of fog drop breakage. The larger the number of We , the greater the probability of breakage. When the critical value is reached, the fog drop will break. Equation 2.7 and Equation 2.8 indicate Re and We (Mousavi *et al.*, 2024).

$$Re = \frac{u_0 d_0 \rho_l}{\nu} \quad \text{Equation 2.7}$$

$$We = \frac{u_0^2 d_0 \rho_a}{\xi} \quad \text{Equation 2.8}$$

u_0 --Liquid exit velocity (m/s);

d_0 --Nozzle outlet diameter (m);

ρ_l --Liquid density (kg/m³);

ρ_a --Air density (kg/m³);

ν --Hydrodynamic viscosity coefficient (N.s/m²);

ξ --Surface tension of a liquid (N/m).

When the atomizing medium and nozzle outlet diameter are constant, it can be seen from Equation 2.7 and Equation 2.8 that Re and We both increase with the increase of liquid outlet velocity u_0 , indicating that the increase of u_0 is conducive to liquid film splitting and droplet breaking, and plays a positive role in improving atomizing performance of nozzle.

The air resistance and surface tension of droplets moving in the air are the main factors for further cracking. According to fluid dynamics, Equation 2.9 indicates the resistance of droplets moving in the air (Nguyen *et al.*, 2021).

$$F_d = \frac{1}{2} C_d \rho_a A (u_l - u_a)^2 \quad \text{Equation 2.9}$$

F_d -- Drag force (N);

C_d -- Drag coefficient, dependent on the droplet's shape and flow condition;

A -- The windward area of fog drops (m²);

u_a -- Air velocity (m/s);

u_l -- Droplet velocity (m/s);

ρ_a -- Air density (kg/m³).

From Equation 2.9, the droplets in the air movement and gas-liquid two-phase velocity difference $(u_l - u_a)$ cause droplets of F_d resistance to increase, producing positive effects on broken droplets.

In addition, in supersonic atomization, the friction and impact between the fluid and the nozzle will produce violent vibration. The gas-liquid interface and fog drops vibrate violently in atomizing the airflow field. Vibration has a great influence on fog drop breakage. When the fog droplet's oscillatory deformation reaches half its initial radius, the droplet begins to split and break. The vibrating droplet becomes unstable under the combined action of external and internal forces, leading to its rupture. From the perspective of vibration analysis, the correlation equation of linear instability analysis is (Turkyilmazoglu, 2020)

$$A_m = A_{m0} e^{wt} \quad \text{Equation 2.10}$$

$$w = \frac{\beta \rho_a (u_l - u_a + c)^2}{2 \rho_l \cdot c} K_C - \frac{2\nu}{\rho_l} K_C^2 \quad \text{Equation 2.11}$$

$$\lambda_c \approx 2\pi \left[\frac{4u_l \sqrt{\xi / \rho_l}}{\beta \cdot \rho_a (u_l - u_a)^2} \right] \quad \text{Equation 2.12}$$

$$K_C = \frac{2\pi}{\lambda}; \quad c = (\xi K_C / \rho_l)^{0.5} \quad \text{Equation 2.13}$$

A_m -- Amplitude (m);

w -- Angular frequency of a wave (rad/s);

t -- Movement time (s);

K_C -- Wave number (m^{-1}) is the reciprocal of wavelength and represents the number of wavelengths per meter length;

c -- Wave velocity (m/s);

λ -- Wavelength (m);

λ_c -- Certain critical value (m);

β -- constant.

According to Equation 2.10 to Equation 2.13, when $\lambda > \lambda_c$, w is positive, A increases exponentially and forms a slender hump. The shear action of the gas-liquid two-phase causes the slender hump to break into the initial fog drops of the same order of magnitude in diameter and wavelength. On this premise, the greater the velocity difference between the gas-liquid two phases, the greater the angular frequency of the wave w and the more conducive to the formation of tiny fog drops. That is, it is more conducive to the improvement of atomization performance.

The above analysis of the nozzle atomization principle shows from two different angles of fluid mechanics and vibration that the gas-liquid two-phase velocity difference is an essential factor affecting the supersonic atomization performance. It is a very reasonable method to increase the nozzle exit velocity and thus increase the atmospheric liquid two-phase velocity difference to improve the supersonic atomization performance through the Laval effect.

2.4 Aerodynamics and Compressible Flow

2.4.1 Compressible Flow

Aerodynamics, the study of the behavior of air as it interacts with solid objects, is a critical field in engineering, particularly in the design of vehicles, aircraft, and various propulsion systems. Compressible flow, a subset of fluid dynamics, deals with fluid flow in which the fluid density changes significantly, typically occurring at high velocities where the Mach number exceeds approximately 0.32. Compressible flow is characterized by

density, pressure, and temperature variations, which significantly influence the flow behavior.

One of the fundamental aspects of compressible flow is the impact of flow conditions on aerodynamic performance. For instance, Kharulaman et al. demonstrated that airfoils subjected to compressible flow experience enhanced lift compared to those in incompressible flow, attributed to the higher freestream velocities encountered in compressible regimes (Kharulaman *et al.*, 2019). This finding is supported by Liu et al., who established a universal theory for aerodynamic forces that applies across a range of flow conditions, from subsonic to supersonic, thereby reinforcing the notion that compressibility plays a crucial role in determining lift and drag characteristics (Liu *et al.*, 2017). Similarly, Yamaguchi and Anyoji investigated low-Reynolds-number compressible flows around rotor blades, emphasizing the unique challenges posed by these specific flow conditions (Yamaguchi & Anyoji, 2023).

The interaction of compressible flows with various geometries has also been a focal point of research. Kanai et al. utilized isogeometric discretization to model compressible-flow aerodynamics of parachutes, successfully applying this method to analyze the Orion spacecraft's drogue parachute (Kanai *et al.*, 2019). This work underscores the importance of advanced computational techniques in understanding complex flow phenomena. The aerodynamic characteristics of various systems, including vehicles and aircraft, are heavily influenced by compressible flow dynamics. Oh et al. conducted a numerical analysis of the aerodynamic characteristics of the Hyperloop system, revealing how tube pressure and temperature affect drag and flow behavior (Oh *et al.*, 2019). Their findings indicate that high

tube temperatures can reduce aerodynamic drag by altering the Mach number for a pod speed, thereby delaying choking effects.

In the context of high-speed transportation, the aerodynamic load analyses of maglev trains in evacuated tube systems have been explored by Wang *et al.*, who employed computational fluid dynamics (CFD) to investigate choked flow and shock waves (Wang *et al.*, 2023). This research highlights the critical role of compressible flow in optimizing the design and performance of advanced transportation systems.

Laval nozzles, designed to accelerate fluids to supersonic speeds, are integral to studying compressible flow. Mu and Lyu analyzed the turbulent boundary layer jet flow pattern of Laval nozzles, demonstrating their ability to produce efficient supersonic flows (Mu & Lyu, 2022). Additionally, Fukunaga *et al.* utilized rainbow schlieren deflectometry to visualize jets from Laval nozzles, providing insights into the internal flow structures and behaviors under different conditions (Fukunaga *et al.*, 2021). The influence of Laval nozzles extends beyond traditional aerospace applications. Guo's research on melt-blown processes highlighted how a Laval nozzle could enhance the manufacturing of polymer fibers by optimizing the airstream field (Guo & Zhu, 2023). This versatility illustrates the broader implications of compressible flow and nozzle design in industrial applications.

The advancement of computational techniques has significantly enhanced the understanding of aerodynamics and compressible flow. For instance, the hybrid recursive regularized lattice Boltzmann method developed by Feng *et al.* has shown promise in simulating compressible aerodynamics, particularly in transonic and supersonic regimes (Feng *et al.*, 2020). Such innovations are crucial for accurately predicting flow behaviors and optimizing designs in various engineering applications.

In summary, aerodynamics and compressible flow fields are rapidly evolving, driven by advancements in computational methods and experimental techniques. Recent literature highlights the complexities of compressible flows, the aerodynamic characteristics of various systems, and the critical role of Laval nozzles in achieving supersonic speeds. As research progresses, integrating these findings will enhance the design and performance of vehicles, aircraft, and industrial processes, paving the way for innovative applications in the future.

2.4.2 Shock Waves and Expansion Waves

The compressible flow dynamics also introduce complexities such as shock waves and Overexpansion waves. For example, shock-induced separation of turbulent boundary layers is a well-documented phenomenon significantly affecting aerodynamic performance, particularly in high-speed applications like rocketry and turbomachinery (Prabhu *et al.*, 2011).

Shock and expansion waves are fundamental phenomena in fluid dynamics, particularly relevant in compressible flow scenarios. These waves are critical in various applications. Shock waves are characterized by abrupt changes in pressure, temperature, and density, propagating faster than the local speed of sound in the medium. Various mechanisms, including explosive events, supersonic flows, and rapid phase transitions, generate them. Sivakumar *et al.* described shock waves as high-energy supersonic waves resulting from sudden energy release, leading to significant changes in the medium's properties (Sivakumar *et al.*, 2020). This unique behavior distinguishes shock waves from ordinary sound waves, which propagate at lower speeds and do not induce such drastic changes.

Recent studies have explored the effects of shock waves on materials and biological systems. For instance, Gong investigated cavitation shock wave emissions in thin liquid layers, emphasizing the need to control shock wave intensity to prevent damage during ultrasonic machining (Gong *et al.*, 2024). Similarly, Wei *et al.* examined the impact of shock-induced cavitation bubble collapse on cell membranes, highlighting shock waves' destructive and therapeutic potential in medical applications (Wei *et al.*, 2021).

Shock waves have found diverse applications across various fields. In medicine, extracorporeal shock wave therapy (ESWT) has gained popularity for treating conditions such as tendinopathy and fracture nonunions. Stania *et al.* reviewed the effectiveness of ESWT for Achilles tendinopathy, noting its potential to promote healing through controlled shock wave application (Stania *et al.*, 2019). Furthermore, Wu *et al.* explored the implications of radial extracorporeal shock waves for treating fracture nonunions, demonstrating the technique's non-invasive nature and efficacy (Wu *et al.*, 2021).

In materials science, shock waves can induce phase transformations in solids. Sivakumar *et al.* reported on the shock wave-driven solid-state phase transformation of Co_3O_4 to CoO nanoparticles, showcasing the ability of shock waves to facilitate rapid material changes under high-energy conditions (Sivakumar *et al.*, 2020). This capability is particularly valuable in synthesizing nanomaterials with tailored properties.

Expansion waves occur when a fluid expands and accelerates, decreasing pressure and density without the abrupt changes associated with shock waves. These waves are typically generated in supersonic flows when the flow area increases, allowing the fluid to expand smoothly. Recent studies have focused on the dynamics of expansion waves in various contexts. For instance, Li and Lyu investigated the interaction between shock and

instability waves in supersonic jet flows, emphasizing the significance of accurately predicting wave structures to understand jet screech frequencies (Li & Lyu, 2023). Their findings highlight the complex interplay between expansion waves and other wave types in high-speed flows.

The surrounding flow conditions also influence the behavior of expansion waves. Yu's research on the flow separation of overexpanded single-expansion ramp nozzles demonstrated how external flow fields can significantly affect the characteristics of expansion waves, particularly in the context of nozzle design (Yu *et al.*, 2023). This underscores the importance of considering environmental factors when analyzing expansion wave behavior.

Overexpansion waves occur when the pressure of the fluid exiting a nozzle is lower than the ambient pressure, leading to a phenomenon where the flow expands excessively. This condition can result in flow separation and the formation of shock waves downstream. Recent literature has explored the implications of overexpansion in various applications. For example, the study by Martelli *et al.* characterized unsteadiness in overexpanded planar nozzles, revealing how internal flow separation can lead to significant fluctuations in pressure and velocity (Martelli *et al.*, 2019). Such insights are crucial for optimizing nozzle designs in aerospace applications, where managing overexpansion effects is essential for performance. Additionally, the effects of overexpansion on jet mixing have been investigated. Rajkumar *et al.* studied the mixing characteristics of supersonic jets in the presence of co-flow streams, demonstrating how varying levels of overexpansion impact the mixing efficiency of the primary jet (Rajkumar *et al.*, 2023). Their findings indicate that

overexpansion can enhance mixing under certain conditions, which is valuable for propulsion and combustion systems applications.

2.5 Nozzle

Various fields need to use nozzle atomization of the working medium, including all kinds of combustion equipment burning liquid fuel and non-combustion equipment in the liquid, paint, water, etc. The spray function of the nozzle on the non-combustion equipment is mainly to form the liquid working medium into spray without burning. But there are also their own requirements; some requirements of the spray are particularly fine, some are thicker, and some are uniform.

Atomization is usually a fragmentation process in which high-speed liquid is injected into the relatively low-speed airflow. Typical nozzles include pressure nozzles and rotary nozzles. The role of nozzles is to spray and atomize, but the spray and atomization methods of various nozzles are different, so the application field is not completely the same. Pressure and rotating nozzles spray into a relatively slow gas environment at a higher liquid rate. In comparison, two-phase flow nozzles inject a relatively high gas rate at a lower liquid rate. They all use the dynamic action of air or flash micro-explosion to atomize. The spray in the combustion chamber of the space rocket propeller is an exception. For the special environment of low gas density and even vacuum in space, fuel, and liquid oxidizers must achieve atomization, mixing, and combustion by their impact and turbulence. The atomization mechanism of ultrasonic and electrostatic nozzles differs from those mentioned. The high-frequency pressure wave of ultrasonic nozzles makes the atomizing surface vibrate at high frequency and "shakes" the liquid into liquid drops, while the electrostatic nozzles make use of the electric pressure generated by high voltage electricity to break and atomize

the liquid (Agarwal & Trujillo, 2020; Broumand *et al.*, 2020). An ideal nozzle should have the following characteristics:

- i. It can provide good atomization in various liquid flow rates.
- ii. Rapid response to changes in liquid flow rate.
- iii. Jet has the maximum instability.
- iv. In the static gas environment, the oil beam does not have serious distortion and deformation in the circular atomization and distribution area.
- v. Require lower power drive energy.
- vi. It can meet the design requirements of atomization quality under various proportions and sizes.
- vii. It has a low cost, is small quality, is easy to manufacture, and is easy to disassemble and assemble.
- viii. The jet hole is not easy to be blocked by greasy or carbon particles attached to the nozzle
- ix. The surface of the nozzle is not easy to form glia due to heat.
- x. Tight seal, no dripping phenomenon.

2.5.1 Two-phase Flow Nozzle

(1) Air-assisted nozzle

Aerodynamic force plays a significant role in promoting liquid fragmentation and atomization, and both reverse flow double nozzle and rotary nozzle introduce air to promote atomization. Although duplexes and two-way nozzles can improve the atomization effect at low flow, a more effective method is to enhance liquid atomization with the help of airflow, especially under low injection pressure. This nozzle is called a two-phase flow nozzle, which includes an air-assisted nozzle, jet nozzle, bubble nozzle, and gas whistle nozzle.

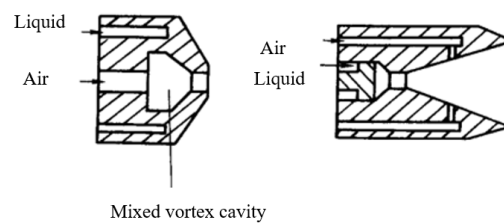


Figure 2.3: Internal mixed air-assisted nozzle (Watanawanyoo *et al.*, 2012)

Various air-assisted nozzles are widely used in industrial gas turbines, oil-fired boilers, and other industrial fields, where high-speed air flows mix with low-speed liquids inside or outside the nozzle, greatly improving liquid atomization. Figure 2.3 shows the internal mixed air-assisted nozzle. The spray cone angle is the smallest when the air velocity is maximum. As the air velocity decreases, the spray cone angle becomes wider. This nozzle is suitable for high-viscosity liquid, and a low liquid flow rate can also obtain a good atomization effect.

The external mixed air-assisted nozzle shown in Figure 2.4 can keep the spray cone Angle unchanged at any liquid flow rate, but its air utilization rate is low, requiring a larger driving force.

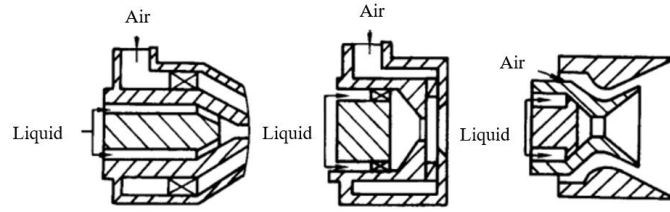


Figure 2.4: External mixed air-assisted nozzle (Watanawanyoo *et al.*, 2012)

The disadvantage of air-assisted nozzles is that they require an additional high-pressure air compressor, so they cannot be used in aircraft engines. However, it can be used in ground-based industrial engines, which require high-pressure air help only at start-up and acceleration. The main role of air power is to improve the spray at low injection pressure and low liquid flow and stop filling high-pressure air at high injection pressure and high liquid flow; that is, air power is not needed in the whole spray process. However, air assist is required across the entire liquid flow range for applications with consistently low injection pressure and low fluid flow. Strong air resistance becomes necessary, and separate internal or external mixing nozzles are no longer sufficient. As a result, the compound air-assisted nozzle comes into play. The nozzle is designed with two internal and external airways. The composite air-assisted nozzle in Figure 2.5 is used for the coal slurry spray process. The liquid spirally enters the vortex chamber, generates a counterclockwise vortex in the vortex chamber, and jets as an annular liquid film. The air in both channels enters the vortex chamber in a clockwise swirl. The inner airflow first impinges on the liquid flow, and the outer airflow again tears the liquid at the nozzle exit.

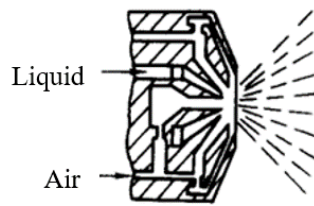


Figure 2.5: Compound air-assisted nozzle (Sarghini & De Vivo, 2019)

(2) Jet nozzle

The working principle of jet nozzles is almost the same as that of air-assisted nozzles. Both rely on the kinetic energy of air to break up liquid into droplets. The difference lies in the amount of air used and the airflow rate. A compressor or high-pressure bottle supplies the air of the air-assisted nozzle. The air injection speed is very high, characterized by a small amount of air impacting the liquid at a high speed to break it. The airflow rate of the jet nozzle is relatively slow, usually with a maximum of 120 m/s, and the flow rate also depends on the gas pressure in the combustion chamber. The airflow of the nozzle is large to ensure a good atomization effect. Jet nozzles have many advantages compared to pressure nozzles, especially in applying a high-pressure combustion system, which can obtain fine atomized oil droplets at lower pump pressure. The airflow of the jet nozzle is large, and the mixing of atomized oil droplets is sufficient to form a completely burned low, bright blue flame, and the carbon smoke emission is very low. Because of these remarkable advantages, jet nozzles are widely used in aviation, Marine, and industrial gas turbines. The most used jet nozzle is the prefilm nozzle, where the fuel is first injected into a continuous liquid film and then broken into small droplets under high-speed airflow. Figure 2.6 shows a prefilm jet nozzle for gas turbines. Fuel is ejected through a ring nozzle to form a ring film jet. Two air streams are incorporated: one enters from the center of the nozzle and is redirected by a

vortex generator to form a rotating vortex that acts on the inner surface of the annular liquid film; the other airflow surrounds the outside of the annular liquid film and acts on the outer ring surface of the annular liquid film in the form of an eddy current. Two rotating air streams simultaneously act on the liquid film outside the nozzle, causing it to crack and atomize. The disadvantage of this nozzle is that the atomization quality is poor when the airflow velocity is low.

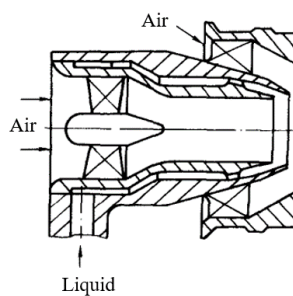


Figure 2.6: Prefilm jet nozzle (Carvalho & Heitor, 1998)

(3) Bubble nozzle

The two-phase flow nozzles have a common characteristic: the continuous liquid is first ejected into a circular or liquid film jet and then broken into droplets under high-speed airflow. There is also a method using gas atomization, that is, the gas is directly dissolved into the liquid in the upstream area of the nozzle, the formation of bubbles in the liquid, and the bubble flows in the mixing chamber. The bubble accelerates, deforms, and expands in the two-phase flow process, extrudes the liquid into a filament shape at the nozzle outlet, and expands sharply at a very short distance from the nozzle outlet due to increased internal and external pressure difference. The flash micro-explosion of dissolved gas "explodes" the liquid into very small droplets. This nozzle is called the bubble nozzle, shown in Figure 2.7, and its advantages are:

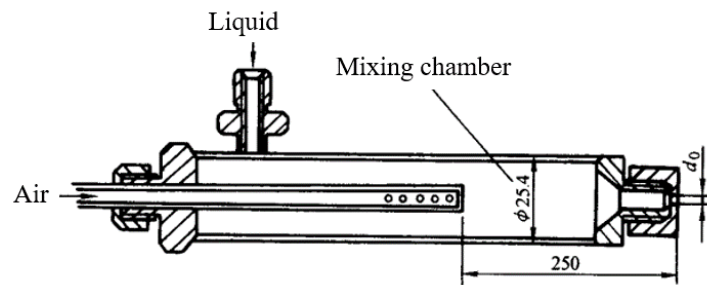


Figure 2.7: Bubble nozzle (Ponasse *et al.*, 1998)

- i. Very little gas medium is used, and a good atomization effect can be achieved with a small gas-liquid ratio (less than 15%).
- ii. The injection pressure of both gas and liquid is low.
- iii. The diameter of the jet hole is large, which helps avoid clogging due to coking.
- iv. Atomization quality is less affected by liquid viscosity.
- v. High reliability, easy maintenance, and low cost.

The disadvantage is that an additional air supply device is required. Bubble nozzles can be used in boilers burning heavy residuals with high boiling points, poor volatility, high viscosity, and surface tension. The flow characteristics of the nozzle and the formation of bubbles inside it depend on factors such as the size of each nozzle component, the nozzle's geometry, the properties of the liquid, and the injection pressure of the gas-liquid mixture. Meanwhile, atomization quality is influenced by the nozzle's flow characteristics, the bubble formation inside the nozzle, and the fragmentation occurring outside the nozzle. If the nozzle of the bubble nozzle is too large, the pressure in the mixing chamber will be too low. When the pressure ratio between the mixing chamber and the gas supply pressure falls below the critical pressure ratio, the gas entering the chamber will reach the speed of sound, generating

a high-frequency whistle. This phenomenon serves as an excitation source that promotes the fragmentation and atomization of the liquid (Ma *et al.*, 2019; Tsuge *et al.*, 2006).

(4) Air whistle nozzle

Figure 2.8 shows the air whistle nozzle, where liquid and air are introduced into the nozzle, respectively. The converter is installed in the air nozzle, making the cross-sectional area of the gas channel smaller. When the high-pressure air passes, ultrasonic airflow will be generated, and high-frequency whistles will be emitted. When the ultrasonic air flow acts on the liquid jet, droplet atomization will occur immediately, and there is no transition process of round jet or liquid film jet. The sound frequency is about 10 kHz, and the atomized droplet diameter is about 50 μm at the flow rate of 1.25 L/s.

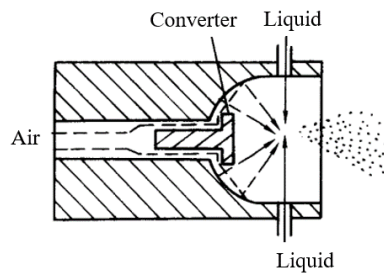
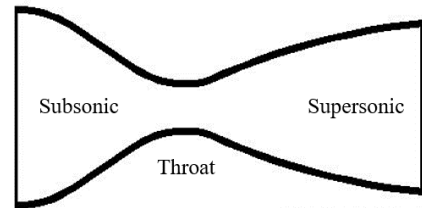


Figure 2.8: Air whistle nozzle (Pang *et al.*, 2022b)

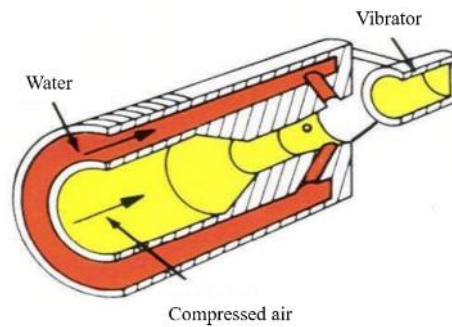
The disadvantage of the nozzle is that the size of atomized droplets varies from large to small, which is not easy to control. Wilcox and Tate (1965) systematically studied the nozzle and showed that the sound range had no significant effect on the atomization quality throughout the spray process (Wilcox & Tate, 1965). Therefore, Topp and Eisenklam (1972) proposed that a gas-whistle nozzle is a form of air-assisted nozzle that promotes liquid cracking employing aerodynamic action at the gas-liquid interface (Topp & Eisenklam, 1972).

2.5.2 Supersonic Laval Nozzle

The ultrasonic nozzle has good atomization performance, but its structure is complex. The supersonic Laval nozzle in Figure 2.9 has good atomization performance, a simple structure, and convenient operation.



(a) Laval structure



(b) Laval nozzle

Figure 2.9: Laval structure and Laval nozzle (Bian *et al.*, 2018)

The Laval nozzle is a specially designed nozzle that comprises three components: a converging section, a throat section, and a diverging section. Laval nozzles are specifically engineered to increase the velocity of high-pressure gases to achieve supersonic speeds.

Laval nozzle advantages:

- i. Liquid and compressed air are mixed inside the nozzle and then atomized again by a tight mechanical ultrasonic oscillator to produce fine spray droplets.

- ii. A large aperture can prevent clogging.
- iii. The working process can produce a self-cleaning effect due to high-frequency vibration.

Some scholars analyzed the compressibility of gas flow under steady-state conditions. Li et al. (2021) studied the jet behavior of oxygen inside and outside Laval nozzles under different pressure and ambient temperature conditions and obtained the quantitative relationship between jet core length and pressure. Yang et al. (2024) believe the nozzle with a Laval structure could generate supersonic airflow, significantly increasing atmospheric liquid's two-phase velocity difference. The numerical simulation method is used to study the flow field of the supersonic nozzle, which shows that the airflow in the nozzle undergoes the expansion process of reducing pressure, increasing speed, and decreasing temperature. Chen et al. (2020) concluded that with the increase of inlet pressure, the shock waves in the supersonic nozzle atomization flow field kept moving outward, and the acceleration characteristic curve of the inner nozzle was determined (Chen *et al.*, 2020; Li *et al.*, 2021; Yang *et al.*, 2024).

The Laval nozzle is widely used in various fields, such as rocket propulsion systems, metallurgy, internal combustion engine oil atomization, and natural gas dehydration. However, applying Laval nozzles in dust removal is rare, especially linear Laval nozzles. So, more studies must be done on Laval nozzles for dust removal.

2.6 Factors Affecting the Effect of Dust Removal

The following are some methods to improve the effect of dust removal.

(1) Structural improvement

Numerous researchers have optimized the structure of dust remover. A summary of the papers on the structural improvement of dust remover is listed in Table 2.2.

Table 2.2: Summary of the papers on the structural improvement of dust remover

Reference	Improvement of structure
(Chen <i>et al.</i> , 2018)	Design wet roller-shutter dust-collecting net
(Liu <i>et al.</i> , 2019)	A nozzle with an orifice diameter of 2.4 mm
(Xu <i>et al.</i> , 2019)	The diameter of the nozzle pore was 2.4 mm
(Peng <i>et al.</i> , 2020a)	Length of the conical cover: 100 mm; The angle of the conical cover: 15.6°; Nozzle installation position: 10 mm
(Xu <i>et al.</i> , 2020)	A large nozzle diameter(2.4mm); Suction ports on both sides of the ejector roller
(Arya <i>et al.</i> , 2020)	Design a flooded-bed dust Scrubber system
(Zhang <i>et al.</i> , 2020b)	Propose the optimized opening mode of hydraulic support spraying
(Peng <i>et al.</i> , 2022a)	Develop a type of negative pressure jet dust-removal water curtain; The distance between the nozzle opening and the air suction port $L_s = 6\text{cm}$
(Chen <i>et al.</i> , 2024)	The diameter of the diverging segment, Throat diameter, and diameter of the shrinkage section, with outlet diameter is 2 mm. (Curved Laval Nozzle)

Chen et al. (2018) designed wet dust-collecting nets applied to a 7m fully mechanized mining face (Chen *et al.*, 2018). Liu et al. (2019), Xu et al. (2019), and Xu et al. (2020) simulated three nozzle diameters (1.6mm, 2.0mm, 2.4mm). They determined that the nozzle diameters of 2.4mm provided the most significant dust suppression effect (Liu *et al.*, 2019;

Xu *et al.*, 2019, 2020). Peng *et al.* (2020a) and Peng *et al.* (2022a) instituted external spray negative pressure mist curtain dust suppression devices and negative pressure jet dust removal water curtain technology. At the same time, nozzle installation position, nozzle type, and the distance between the nozzle opening and the air suction port were optimized, respectively (Peng *et al.*, 2020a; Peng *et al.*, 2022a). Arya *et al.* (2020) proposed a flooded-bed dust scrubber system (Arya *et al.*, 2020). Zhang *et al.* (2020b) optimized the opening mode of hydraulic support spraying (Zhang *et al.*, 2020b). Chen *et al.* (2024) investigated the influence of structural parameters of curved Laval nozzles on atomization performance (Chen *et al.*, 2024). Many studies focus on the nozzle exit diameter as the key structural parameter.

(2) Operating environment

A summary of the papers on optimizing the operating environment of dust remover is listed in Table 2.3. Liu *et al.* (2019) optimized the sprinkling pressure of an external spraying system to achieve a spraying pressure of 5 MPa (Liu *et al.*, 2019). On the other hand, Xu *et al.* (2020) and Peng *et al.* (2022b) proposed a spray dust-settling device and negative pressure jet dust-removal water curtain technology, respectively, and the conclusion was that 6 MPa had excellent atomization and dust suppression efficacy (Peng *et al.*, 2022b; Xu *et al.*, 2020). However, Xu *et al.* (2019) stated that the dust suppression effect was evident when the spraying pressure reached 8 MPa (Xu *et al.*, 2019). Similarly, Shi *et al.* (2019) simulated coalbed water infection and concluded that an injection pressure of 8 MPa was generally acceptable (Shi *et al.*, 2019).

Table 2.3: Summary of the papers on optimization of the operating condition of dust remover

Reference	Parameter	Optimized result
(Liu <i>et al.</i> , 2019)	Water pressure: 1/1.5/2/2.5/3/3.5/4/4.5/5/5.5 MPa	Spraying pressure of 5 MPa
(Xu <i>et al.</i> , 2019)	Spray pressure of 1/2/3/4/5/6/7/8 MPa	Spray pressure of 8 MPa
(Shi <i>et al.</i> , 2019)	Water injection pressure: 2/4/6/8/10/12 MPa	Water injection pressure of 8 MPa
(Xu <i>et al.</i> , 2020)	Spray pressure: 1/2/3/4/5/6/7/8 MPa	Spray pressure of 6 MPa
(Zhou <i>et al.</i> , 2021)	Relative humidity: 35%/45%/55%/65%/75%/85%/95 %	Relative humidity of 75% - 85%
(Peng <i>et al.</i> , 2022b)	Pw: 2/4/6 MPa	Pw of 6 MPa
(Tian <i>et al.</i> , 2024)	Air pressure: 0.3/0.4/0.5 MPa (Curved Laval Nozzle)	Air pressure of 0.5 MPa
(Chen <i>et al.</i> , 2024)	Water pressure of 0.3/0.4/0.5/0.6/0.7/0.8 MPa Air pressure of 0.2/0.3/0.4/0.5/0.6 MPa (Curved Laval Nozzle)	Water pressure of 0.3 Mpa Air pressure of 0.4 MPa

Considering air humidity conditions in the fully mechanized mining face, Zhou *et al.* (2021) pointed out that the optimal relative humidity was between 75% and 85% (Zhou *et al.*, 2021). Tian *et al.* (2024) investigated the air pressure of curved Laval nozzles and concluded that the optimal air pressure is 0.5 MPa (Tian *et al.*, 2024). Chen *et al.* (2024) studied the water and air pressure of curved Laval nozzles, determining that the optimal air pressure is 0.4 MPa, while the optimal water pressure is 0.3 MPa (Chen *et al.*, 2024). The

research above indicates that optimizing the operating environment, particularly the spraying pressure, is important.

The optimization of dust collectors primarily focuses on two aspects: structure and operating environment. Structurally, optimizing the nozzle exit diameter has proven highly effective. Regarding the operating environment, pressure optimization plays a crucial role. Traditional nozzles operate at high pressures, typically above 5 MPa, while Laval nozzles work at lower pressures, offering enhanced safety and energy savings. Current research has mainly explored curved Laval nozzles, with limited studies on linear Laval nozzles.

2.7 Numerical Analysis Model and Experimental Measuring Tool

2.7.1 Numerical Analysis Model

Fluent is the main software for numerical analysis. The computational models include the VOF and DPM models, with the k- ϵ model commonly used for turbulence. Table 2.4 summarizes the papers on the numerical analysis models.

Peng et al. (2019) created a new wind-assisted centralized spraying dedusting device for dust control, utilizing the DPM and k- ϵ models in Fluent software (Peng *et al.*, 2019). Similarly, Peng et al. (2022b) investigated a synergistic blowing-spraying dust removal technology installed on hydraulic supports using the DPM and k- ϵ models in Fluent software (Peng *et al.*, 2022b). Additionally, Peng et al. (2020b) studied a full-section water curtain device to prevent PM_{2.5} diffusion pollution, employing the DPM and k- ϵ models in Fluent software (Peng *et al.*, 2020b).

Zhang et al. (2020c) designed a new nozzle by integrating existing supersonic aerodynamic atomization nozzles with a Laval nozzle and a probe structure. They found that the new nozzle achieved a larger atomization angle and smaller atomized particle size. They

utilized Fluent software's DPM and k- ϵ models (Zhang *et al.*, 2020c). On the other hand, Yin *et al.* (2019) and Ma *et al.* (2020) explored external spray systems for shearers. Yin *et al.* (2019) designed a multi-stage atomization system with a fan for the shearer, using Fluent software's DPM and k- ϵ models (Yin *et al.*, 2019). Similarly, Ma *et al.* (2020) investigated the nozzle types of an external spraying device for coal cutters and optimized the nozzle calibers, employing Fluent's DPM and k- ϵ models (Ma *et al.*, 2020).

However, Zhai *et al.* (2020) and Nie *et al.* (2023) utilized negative pressure suction to capture dust-laden airflow. Zhai *et al.* (2020) designed an ejector precipitator, optimizing the nozzle parameters and the liquid-to-gas ratio. They used Fluent software's VOF and k- ϵ models (Zhai *et al.*, 2020). Similarly, Nie *et al.* (2023) studied a wind-assisted spray device, integrating macroscopic and microscopic perspectives and employing Fluent's DPM and k- ϵ models (Nie *et al.*, 2023).

Table 2.4: Summary of the papers on the numerical analysis model

Reference	Software	Computational models	Turbulence models
(Yin <i>et al.</i> , 2019)	Fluent	DPM	Standard k- ϵ model
(Peng <i>et al.</i> , 2019)	Fluent	DPM	Realizable k- ϵ model
(Zhang <i>et al.</i> , 2020c)	COMSOL	DPM	One-equation Spalart–Allmaras model
(Peng <i>et al.</i> , 2020b)	Fluent	DPM	Standard k- ϵ model
(Zhai <i>et al.</i> , 2020)	Fluent	VOF	Standard k- ϵ model
(Ma <i>et al.</i> , 2020)	Fluent	DPM	Realizable k- ϵ model
(Peng <i>et al.</i> , 2022b)	Fluent	DPM	Realizable k- ϵ model
(Nie <i>et al.</i> , 2023)	Fluent	DPM	Realizable k- ϵ model

In previous research, Fluent software was primarily used. Due to the need to analyze particle size characteristics, the DPM model was frequently employed. The k- ϵ model was widely applied, including the standard and realizable versions.

2.7.2 Experimental Measuring Tool

The twin-fluid spray technology involves mixing gas and liquid inside the nozzle, utilizing the high-speed gas flow's kinetic energy and shear force to break the liquid into fine droplets. Twin-fluid nozzles, as the carriers of this technology, have gained significant attention in fields such as dust suppression, agricultural spraying, and fuel atomization due to their advantages of producing small droplet sizes, rapid mixing, and relatively low energy consumption for spraying (Chen *et al.*, 2022). The spray process of twin-fluid nozzles is a complex, multiphase, dynamic process where the characteristics of the spray particles change over time and space. These factors lead to challenges in practical applications, such as uneven droplet size distribution, large droplet sizes, and difficulty controlling the spray effect. Quantitative analysis of the spatial characteristics of spray particles is critical to addressing these issues (Dumitrescu *et al.*, 2018). The rapid development of advanced electronic and optical testing instruments has facilitated in-depth research into these challenges, with tools such as the Phase Doppler Particle Analyzer (PDPA) and laser particle analyzer playing a significant role (Nie *et al.*, 2023; Yu *et al.*, 2020).

Wang *et al.* (2020) used a Malvern droplet size analyzer to study the atomization performance of six X-swirl nozzles with different outlet diameters. They found that when the water supply pressure exceeded 4 MPa, a nozzle with an outlet diameter of 1.2 mm was most suitable (Wang *et al.*, 2020). Wang *et al.* (2019a) also used a Malvern droplet size analyzer to investigate an internal-mixing air-assisted atomizer nozzle, determining the optimal nozzle outlet diameter (Wang *et al.*, 2019a). Zhu *et al.* (2020) employed a BT-1600

particle size image analysis system to study foam technology for dust suppression, noting a significant improvement in dust suppression efficiency after device modifications (Zhu *et al.*, 2020). Wang *et al.* (2019b) used a Malvern droplet size analyzer to research an X-type swirl pressure nozzle, establishing a multivariate nonlinear predictive mathematical model for the Sauter mean diameter (Wang *et al.*, 2019b). Peng *et al.* (2019) used a phase Doppler laser interferometer (PDI) to study a novel wind-assisted centralized spraying dedusting device, achieving good dust suppression results at a water pressure of 8 MPa and air pressure of 1 MPa (Peng *et al.*, 2019). Ma *et al.* (2020) analyzed a pressure round-mouth nozzle with an X-shaped core using a PDI200MS phase Doppler laser interferometer (PDI), finding optimal dust suppression at a spray pressure of 8 MPa (Ma *et al.*, 2020). Han *et al.* (2020) utilized a Malvern droplet size analyzer to study an internal mixing air atomizing nozzle, showing that droplet size and velocity increased with water supply pressure (Han *et al.*, 2020). Yang *et al.* (2019) used a phase Doppler laser interferometer (PDI) to study hybrid nozzles with an X-shaped diversion core, identifying the best atomization performance at a spray pressure of 8 MPa (Yang *et al.*, 2019). Zhai *et al.* (2020) used a 3D particle dynamic analyzer to research the ejector precipitator installed at the low-level caving coal hydraulic support, concluding that a water supply pressure of 12 MPa was beneficial for dust removal (Zhai *et al.*, 2020). Xu *et al.* (2019) used a phase Doppler laser interferometer to study pressure swirling nozzles, finding optimal atomization performance at a spray pressure of 8 MPa. This review shows that ordinary nozzles operate at high pressures, generally above 5 MPa (Xu *et al.*, 2019). Zhang *et al.* (2020) conducted particle size tests using a Malvern droplet size analyzer, comparing curved Laval nozzles and ordinary nozzles. They found that curved Laval nozzles had better atomization performance and required lower water pressure (Zhang *et al.*, 2020a). Chen *et al.* (2024) researched sonic blasting nozzles using a

HELOS-Vario laser particle size analyzer. Compared to traditional nozzles, sonic boom atomizing nozzles consume less water (Chen *et al.*, 2024). Tian *et al.* (2023) used Particle Image Velocimetry and a Malvern laser particle size analyzer to study supersonic siphon atomization nozzles, finding that the atomization cone angle ranges from 57° to 65° under a pressure of 0.3-0.6 MPa (Tian *et al.*, 2023). Yuan *et al.* (2022) investigated the impact of water and air pressure on the atomization performance of curved Laval nozzles using a Malvern droplet size analyzer (Yuan *et al.*, 2022). Yang *et al.* (2016) used the OMEGA DP-02 Laser Particle Analyzer (manufactured by Zhuhai OMEGA Instrument Co., Ltd.) to measure curved Laval nozzle' droplet sizes and distribution (Yang *et al.*, 2016).

2.8 Chapter Summary

Sections 2.2 to 2.7 have provided detailed discussions on the main dust suppression methods and atomized dust removal methods. Due to their small particle size, respirable dust particles (PM10, PM2.5) are harmful and difficult to remove. When the size of atomized droplets is similar to that of dust particles, the probability of dust adsorption and condensation reaches its maximum, reducing the concentration of dust particles to its minimum. However, droplet size should not be too small, as excessively small droplets evaporate easily. Droplets with sizes between 14 and 30 microns are effective for capturing respirable dust. Based on the discussion in Sections 2.5 and 2.6, traditional nozzles operate under high pressure, producing large and uneven droplet sizes. In contrast, Laval nozzles require lower operating pressure, generating smaller and more uniform droplets, making them a promising new technology in dust removal. The linear Laval nozzle, which is easier to manufacture but has been less studied, holds significant research potential. Water pressure, air pressure, and nozzle outlet diameter greatly affect the atomization performance of the

nozzle. Chapter 3 will further elaborate on the linear Laval nozzle's structural design and atomization performance.

CHAPTER 3

RESEARCH METHODOLOGY

3.1 Introduction

In this chapter, the research methodology includes the research variables, research tools, flow chart, and research process in sections 3.2, 3.3, 3.4, and 3.5, respectively. Next, the conclusion for this chapter is presented in section 3.6.

3.2 Research Variables

Variables are attributes or characteristics used in scientific research for measurement, observation, or manipulation. They are fundamental concepts in scientific research used to describe and quantify different aspects of the research subject. Variables can be categorized into different types, such as:

Independent Variable: The researcher intentionally manipulates or controls the independent variable. It is the manipulated variable in the study, and its values are changed to observe its effects on other variables. For example, in this study, the nozzle structure, combination configuration, nozzle size, liquid pressure, liquid flow rate, and gas pressure.

Dependent Variable: The dependent variable is measured or observed in the study, and the independent variable influences its values. The dependent variable is the outcome or response of the study, and the researcher focuses on the effects of the independent variable on the dependent variable. For example, in this study, the atomization performance.

Controlled Variable: The controlled variable is kept constant in the study. By controlling other variables that may influence the dependent variable, researchers can better

determine the effects of the independent variable on the dependent variable. For example, temperature, humidity, and fluid properties in this study.

Mediating Variable: The mediating variable explains the relationship between the independent and dependent variables. It mediates or carries the influence between the independent variable and the dependent variable, aiding in understanding the mechanism of this relationship. Mediating variables are often crucial in research to explain how the independent variable affects the dependent variable. For example, in this study, the cone angle of atomization is shown in Figure 3.1, along with spray velocity, penetration distance, and droplet size.

Defining and controlling the variables is crucial in research design and data analysis. This ensures the reliability and effectiveness of the study's outcomes.

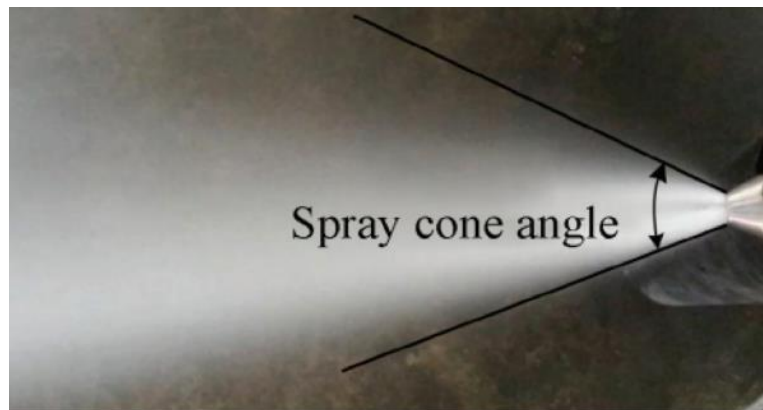


Figure 3.1: Spray cone angle (Sovani *et al.*, 2001)

3.3 Research Tools

Research tools include software such as SolidWorks (2023), ANSYS Fluent (2022 R1), Origin (2021), and equipment like the Laser Particle Size Analyzer and Nozzle Atomization System.

3.3.1 SolidWorks and ANSYS Fluent

The nozzle models are built with SolidWorks. SolidWorks, shown in Figure 3.2, is a leading 3D computer-aided design (CAD) software developed by Dassault Systems. SolidWorks is used to create static parts and assemblies and conduct motion simulations and structural analysis. It offers powerful tools for simulating and evaluating design performance, motion, stress, and vibrations. This helps engineers validate and optimize designs, ensuring product reliability and performance in real-world usage.

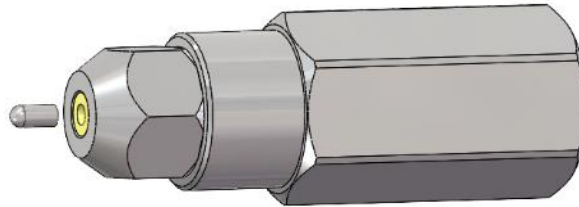


Figure 3.2: Nozzle model drawn in Solidworks

In this study, Fluent software was used to simulate the flow field. Figure 3.3 shows an example of Fluent simulation. Fluent software is a widely used computer simulation software for computational fluid dynamics (CFD). It was developed by ANSYS, an American company, and is part of the ANSYS suite, which simulates and analyzes fluid dynamics problems.

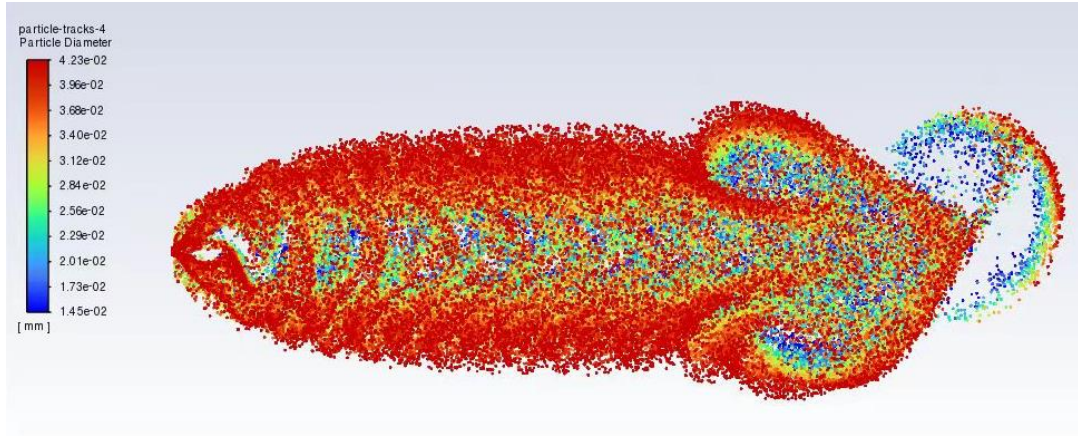


Figure 3.3: Fluent simulation in this research

The design of the traditional nozzle is based on experiments; the analysis period is longer, and the experiment cost is higher. CFD (Computational Fluid Dynamics) is an effective numerical simulation method for studying fluid dynamics, greatly reducing the experimental time and cost.

3.3.2 Laser Particle Size Analyzer

The laser particle size analyzer shown in Figure 3.4 measures droplet size and droplet distribution. A laser particle size analyzer is a specialized instrument used to measure and analyze the size distribution of particles suspended in a liquid or gas medium. It utilizes laser diffraction technology to determine the particle size based on the scattering pattern produced when a laser beam passes through the particle suspension.

The laser particle size analyzer emits a laser beam into the particle sample. As the laser beam encounters the particles, it scatters in various directions. The scattered light is then collected by detectors positioned at different angles. The instrument calculates the

particle size distribution by analyzing the intensity and pattern of the scattered light. Key features and capabilities of laser particle size analyzers include the following:

- i. **Measurement Range:** Laser particle size analyzers can measure particle sizes ranging from nanometres to millimeters, depending on the specific instrument's capabilities.
- ii. **High Accuracy:** These analyzers provide precise and accurate particle size measurements, allowing for detailed analysis of the particle population.
- iii. **Wide Application Range:** Laser particle size analyzers are widely used in pharmaceuticals, chemicals, food and beverages, cosmetics, environmental monitoring, and research laboratories.
- iv. **Automated Operation:** Many modern laser particle size analyzers offer automated sample handling, measurement, and data analysis, streamlining the measurement process and improving efficiency.
- v. **Software and Data Analysis:** These analyzers often come with dedicated software that enables users to analyze and interpret the particle size data, generate size distribution plots, and export results for further analysis.



Figure 3.4: Laser particle size analyser (Blott *et al.*, 2004)

3.4 Flow Chart

Figure 3.5 presents the flowchart for this research. First, a literature review is conducted to determine the nozzle structure and dimensions. Next, Fluent is used to simulate the nozzle's internal and external flow fields, and the nozzle is manufactured for experimentation. Then, the test results are compared with the simulation results, and the parameters are adjusted accordingly. Finally, the optimal nozzle structure and operating conditions are determined.

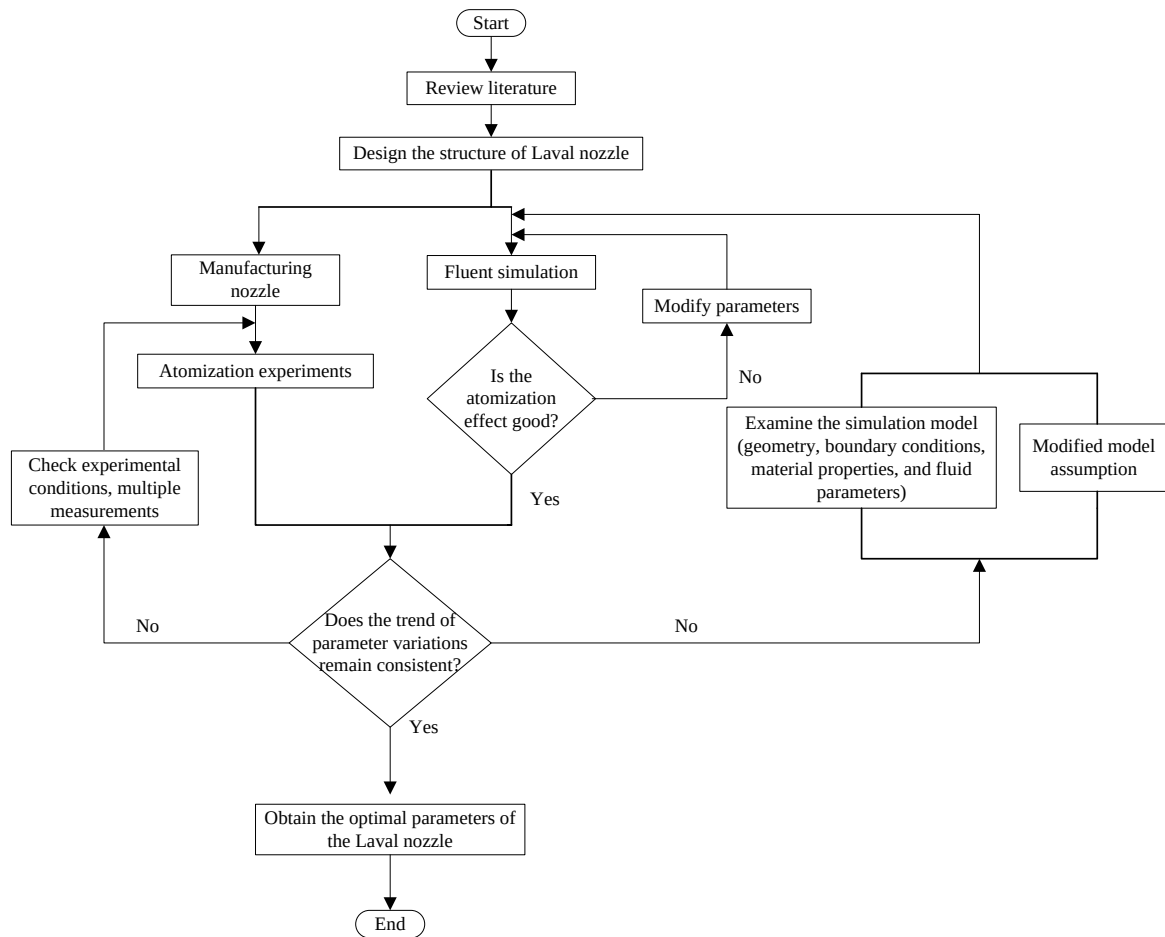


Figure 3.5: Flow chart

3.5 Research Process

This section consists of the following: Laval nozzle structure design, Verification of supersonic flow inside the nozzle, Numerical analysis method, and experimental method. The flow is characterized as an isentropic compressible flow that follows the ideal gas law and exhibits turbulent behavior.

3.5.1 Laval Nozzle Structure Design

3.5.1.1 Laval Nozzle Structure

The Laval nozzle comprises a subsonic converging section, a critical throat section, and a supersonic diverging section. The general principle of Laval nozzle design is that the pressure energy of the airflow must be effectively converted into kinetic energy. Currently, the internal construction of the Laval nozzle consists of two types: cone and curve (Tsai *et al.*, 1999). The converging section of the curved structure can be designed with the Vitosinski curve. The manufacturing cost of the curved surface is high, and the Linear surface is easy to process (Gortyshov *et al.*, 2007). Therefore, the cone is selected for the converging section in this research.

3.5.1.2 Laval Nozzle Size Determination

i. Diverging section

If the diameter of the throat is d_2 , the length of the diverging section and the half angle of the diverging are L_2 and α , respectively. Then the diameter at the outlet d_3 is shown in Equation 3.1. The length of the diverging segment L_2 is shown in Equation 3.2 (Durif, 2022).

$$d_3 = 2L_2 \tan \alpha + d_2 \quad \text{Equation 3.1}$$

$$L_2 = (d_3 - d_2) / 2 \tan \alpha \quad \text{Equation 3.2}$$

Because the airflow completely works in the supersonic range, the friction irreversible loss is too large if the diverging section is too long. If the diverging section is too short and the section diverging is too large, the airflow will be separated from the wall, resulting in eddy current loss, which is unfavorable to energy conversion. As a rule of thumb, the diverging half angle is generally $8^\circ - 15^\circ$. According to the literature, the outlet airflow velocity of the Laval nozzle gradually increases within the diverging half-angle range of 8° to 10° . It tends to stabilize from 11° to 15° (Yang *et al.*, 2016). Therefore, a diverging half-angle of 10° is considered the most optimal.

ii. Converging section

The performance of the converging section depends on the ratio of the inlet area to the outlet area of the converging section and the shape of the converging section. The diameter at the inlet is d_1 . The diameter of the throat is d_2 . In order to accelerate the airflow statically, the cone angle θ of the converging section is generally 30° to 60° (Mil'man *et al.*, 2023), and the length of the converging section is shown in Equation 3.3. L_1 is usually (3 - 5) times d_2 (throat diameter).

$$L_1 = \frac{(d_1 - d_2)}{2} * \cot \frac{\theta}{2} \quad \text{Equation 3.3}$$

Where,

d_1 -- Inlet diameter;

d_2 -- Throat diameter;

θ -- Cone angle of the converging section;

L_1 -- Length of the converging section.

iii. Throat section

The throat is the critical position where the airflow changes from subsonic to supersonic and is the most important part of the entire nozzle design. In practical applications, the nozzle flow rate should not be too low, which requires a throat diameter generally between 1–2 mm.

The final size of Laval's structure is shown in Figure 3.6. The units in the figure are in millimeters. The nozzle assembly drawing is shown in Figure 3.7. The common nozzle material is 304 stainless steel (304SS) or 316 stainless steel (316SS). 3D modeling with SolidWorks is shown in Figure 3.8.

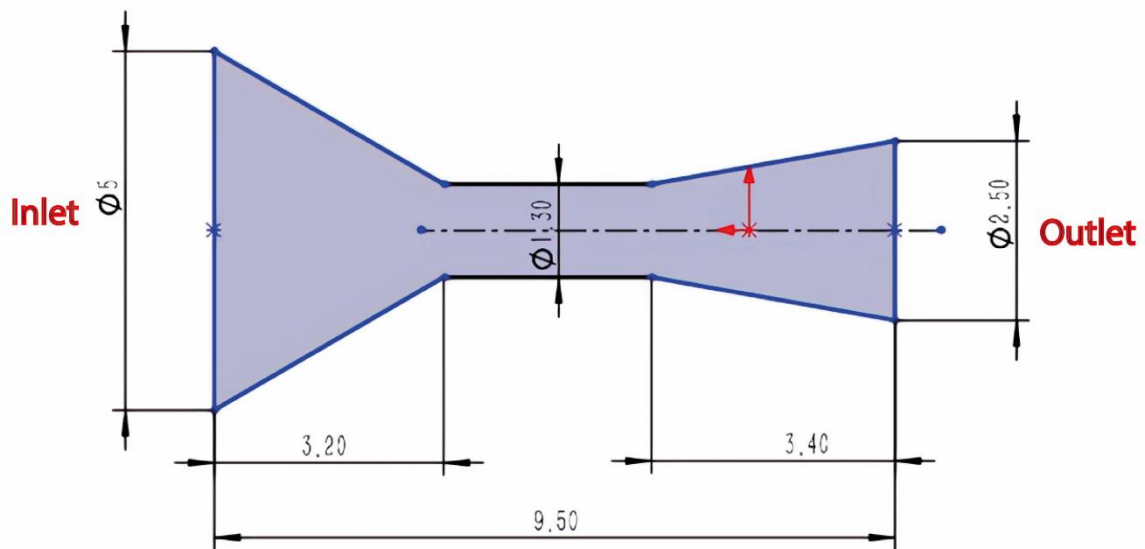


Figure 3.6: Laval structural parts drawing

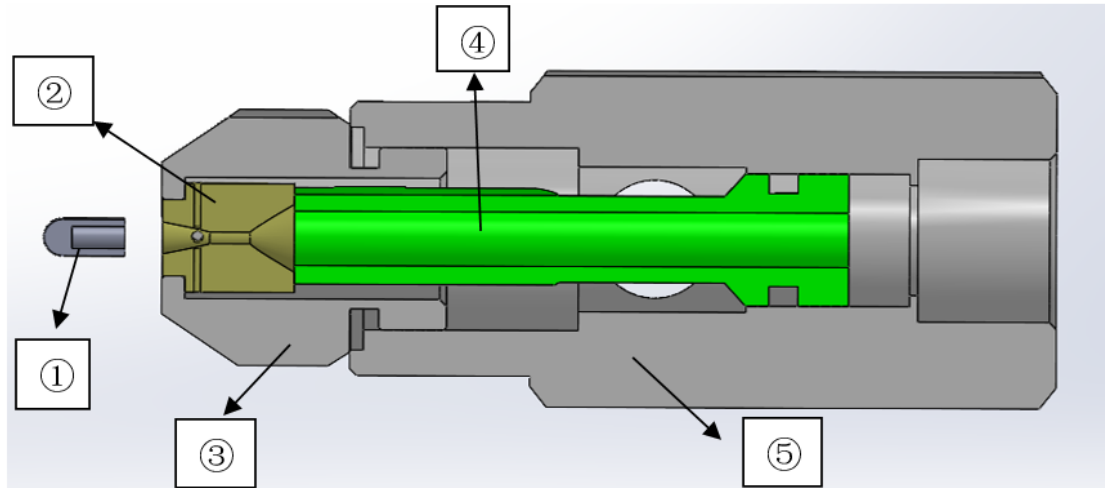


Figure 3.7: Nozzle assembly drawing: ①Mechanical ultrasonic oscillator; ②Laval structure; ③Main body; ④Connecting rod; ⑤shell

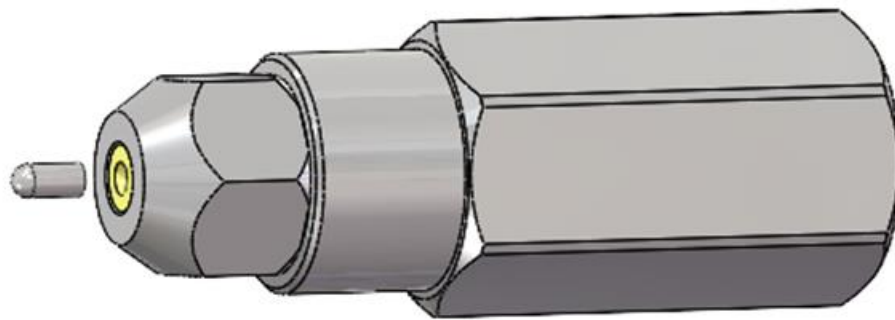


Figure 3.8: 3D modeling with SolidWorks

Two atomization processes: The first atomization occurs when the air and water are mixed at the core. Then, the high-speed gas-liquid mixing flow impinges on the Mechanical ultrasonic oscillator at supersonic speed to produce ultrasonic waves. The high-frequency vibration energy of the ultrasonic wave atomizes the liquid droplet again.

In the case of high dust concentration, the combined nozzle shown in Figure 3.9 can be used, which has a higher dust removal efficiency.

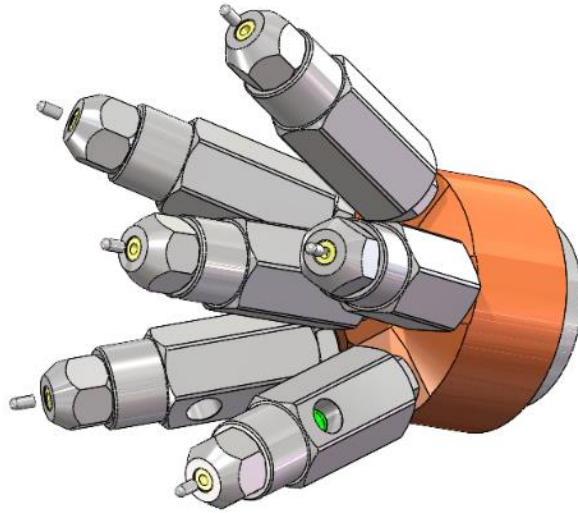


Figure 3.9: Laval combined nozzle

3.5.2 Numerical Simulation of the Internal Flow Field in the Nozzle

Compressed air of 0.5 MPa enters the converging section of the Laval nozzle from the inlet, accelerates to the throat, and continues to accelerate in the diverging section. In this section, the flow is simulated by Fluent to observe the internal flow of the Laval nozzle and verify that flow velocity reaches sonic speed in the nozzle throat.

i. Governing equation

According to fluid mechanics, the gas flow inside the nozzle follows three governing equations shown in Equation 3.43.4 to 3.6 (Balabel *et al.*, 2011).

Continuity equation

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0 \quad \text{Equation 3.4}$$

Navier-Stokes equation

$$\frac{\partial(\rho \vec{u})}{\partial t} + \nabla \cdot (\rho \vec{u} \vec{u} + P_p \vec{I}) = \rho \vec{f} + \nabla \cdot \vec{\tau} \quad \text{Equation 3.5}$$

Energy equation

$$\frac{\partial(\rho E)}{\partial t} + \nabla \cdot [(\rho E + P_p) \vec{u}] = \nabla \cdot (\kappa \nabla T) + \rho \vec{u} \cdot \vec{f} + \Phi \quad \text{Equation 3.6}$$

ρ -- Density (kg/m³);

t -- Time (s);

$\vec{u}$ -- Fluid velocity vector (m/s);

$\nabla \cdot (\rho \vec{u})$ -- The divergence of density flow [kg/(m³.s)];

P_p -- Pressure (Pa);

$\vec{I}$ -- Unit matrix (Tensor form of the pressure term);

$\vec{f}$ -- Volume forces, such as gravity (N/kg);

$\vec{\tau}$ -- Viscous stress tensor (N/m²);

$\nabla \cdot (\rho \vec{u} \vec{u})$ -- Divergence of momentum flux (N/m³)

E -- Total energy density (J/m³);

κ -- Thermal conductivity [W/(m · K)];

T -- Temperature (K);

Φ -- Viscous dissipation function [W/m³].

ii. Domain and boundary conditions

The fluid region of the Laval structure is shown in Figure 3.10.

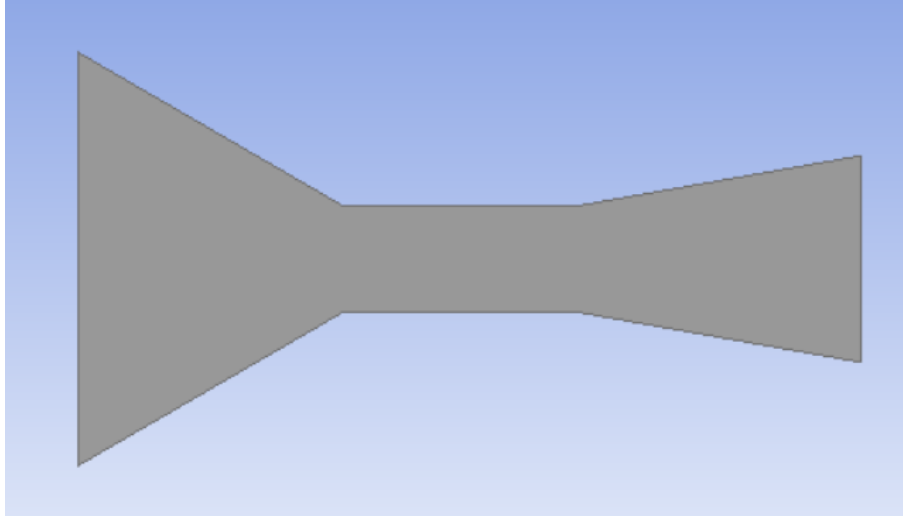


Figure 3.10: Fluid region of the Laval structure

The fluid region model is imported into the Fluent Mesh module, and the operating pressure is set to 101,325 Pa. The boundary condition settings are shown in Figure 3.11. The blue boundary in the figure represents the fluid inlet. Select 'Pressure Inlet' and set the pressure to 0.5 MPa. The red boundary represents the fluid outlet. Select 'Pressure Outlet' and set the pressure to 0 MPa.

iii. Meshing and Setting up

The meshing shown in Figure 3.11 is performed using the Fluent Mesh module. The domains are composed of 9788 unstructured triangle elements and 7131 nodes. Element size is 0.004m. The inflation option is total thickness. The number of Layers is 5, the Growth rate is 1.05, and the maximum thickness is 0.0005. The average orthogonal quality is an indicator used to assess the mesh quality, evaluating the orthogonality and uniformity of the mesh cells. The higher the value, the better the mesh quality and the more reliable the simulation results. This study's average orthogonal quality is 0.96113, indicating excellent mesh quality.

The grid is imported into Fluent for solution. The gas flows through the Laval nozzle at high velocity, constituting compressible flow. The gas material is chosen as the ideal gas, and the energy equation is activated. The solution method is Pressure-Velocity coupling, as shown in Figure 3.12. The turbulence model is the standard k- ϵ model with standard wall functions. The First-Order Upwind Scheme is selected for its strong stability and computational efficiency, making it well-suited for preliminary simulations.

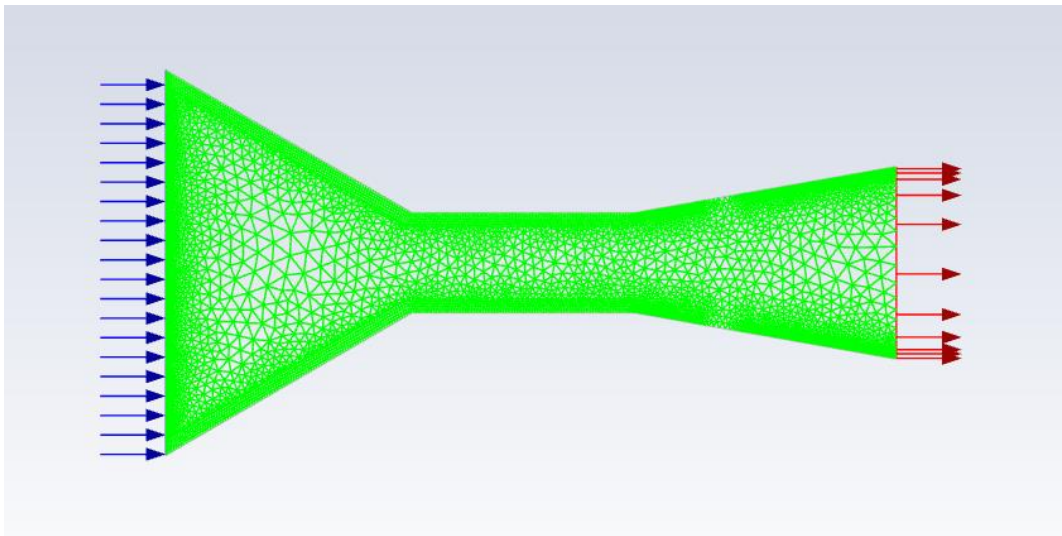


Figure 3.11: Mesh generation

Task Page

Solution Methods 

Pressure-Velocity Coupling

Scheme
 Coupled

Flux Type
 Rhie-Chow: distance based ☐ Auto Select

Spatial Discretization

Gradient
 Green-Gauss Cell Based

Pressure
 Standard

Density
 First Order Upwind

Momentum
 First Order Upwind

Turbulent Kinetic Energy
 First Order Upwind

Turbulent Dissipation Rate
 First Order Upwind

Energy
 First Order Upwind

Transient Formulation
☐ Non-Iterative Time Advancement
☐ Frozen Flux Formulation
☒ Pseudo Time Method

Figure 3.12: Solution method

3.5.3 Numerical Analysis with Varied Water Pressure

The inlet pressure should not be excessively high, as it can easily lead to outlet disturbances. Additionally, high pressure imposes higher demands on equipment and nozzle manufacturing processes. Therefore, in practical usage, priority is given to selecting medium

and low-pressure conditions. Furthermore, Laval nozzles exhibit high internal airflow velocities, and medium and low water pressures can achieve excellent atomization effects. Based on the literature and field application data, water pressures of 0.1, 0.2, 0.3, 0.4, and 0.5 MPa are selected, with an air pressure of 0.5 MPa, a nozzle outlet diameter of 2.5 mm, and a nozzle-to-oscillator distance of 4 mm (Chen *et al.*, 2024).

3.5.3.1 VOF Simulation Setup

For the Laval gas-liquid two-phase flow nozzle, liquid entering the nozzle cavity is dispersed by high-speed airflow. The gas-liquid mixture is expelled from the nozzle outlet after the droplet breakup process. This process involves the flow of gas and liquid phases and can be simulated using the VOF (Volume of Fluid) model to analyze the flow fields inside and outside the nozzle. The VOF model is a type of Eulerian multiphase flow model. In this model, if the volume fraction of a certain fluid within a computational cell is set to 0, it indicates the absence of that fluid in that cell. Conversely, if the volume fraction is set to 1, it denotes the presence of only that fluid in the cell.

i. Governing equation

The governing equations used in the VOF (Volume of Fluid) model are as shown in Equation 3.7, Equation 3.8, and Equation 3.9 (Liovic *et al.*, 2001).

Continuity equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad \text{Equation 3.7}$$

Momentum equation

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial P_p}{\partial x_i} + \frac{\partial \mu}{\partial x_j} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \rho g_i + F'_j \quad \text{Equation 3.8}$$

Volume fraction equation

$$\frac{\partial F}{\partial t} + u_i \frac{\partial F}{\partial x_i} = 0 \quad \text{Equation 3.9}$$

Where,

$$\rho = F \rho_1 + (1-F) \rho_2; u_i = F u_{i1} + (1-F) u_{i2} \quad \text{Equation 3.10}$$

ρ -- Density (kg/m³);

P_p —Pressure (N);

t -- Time (s);

μ -- Dynamic viscosity of the fluid (p_a · s)

u -- Flow velocity (m/s);

F' -- Body forces due to external actions (N);

F -- Volume fraction;

i, j -- Tensor indices;

g -- Acceleration of gravity (m/s²).

ii. Domain and boundary conditions

The air pressure is 0.5 MPa, and the water pressure varies from 0.1 MPa to 0.5 MPa. The external flow field measures 1500mm in length and 600mm in width. Figure 3.13 illustrates both the internal and external flow fields of the nozzle. The nozzle outlet diameter is 2.5 mm, with the oscillator positioned 4 mm away from the nozzle. Figure 3.14 provides an enlarged view of the flow field around the nozzle.



Figure 3.13: Internal and external flow fields of the nozzle

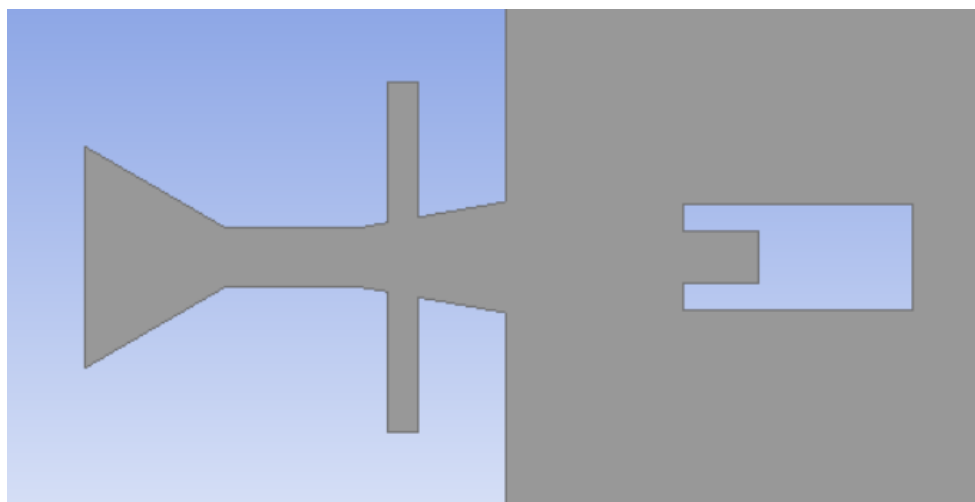


Figure 3.14: An enlarged view of the flow field around the nozzle

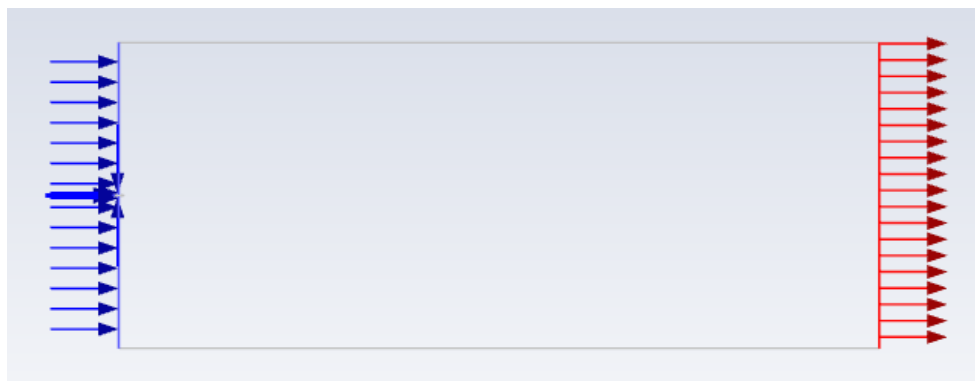


Figure 3.15: Boundary conditions for external flow fields

The operating pressure is 101325 Pa. Figure 3.15 illustrates the boundary conditions for external flow fields. On the left side of the external flow field, it is set as a Pressure-far-field, with the pressure being 0 MPa. It is set as a pressure outlet on the right side, with the pressure being 0 MPa. The acoustic wave model employed is Non-Reflecting.

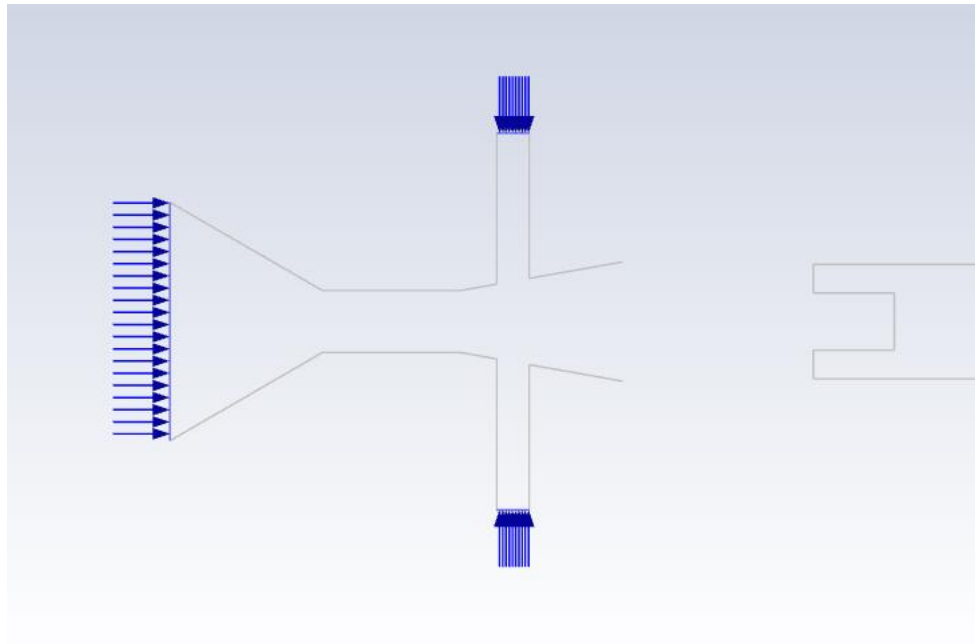


Figure 3.16: Boundary conditions for nozzle

Air is the primary phase, and water is the secondary phase. The boundary conditions of the nozzle section are set, as shown in Figure 3.16. The left side is the air inlet, and the type is set as a pressure inlet. The volume fraction of water is 0. The pressure is 0.5MPa. They act as water inlets on the top and bottom sides, set as pressure inlets. The water volume fraction is set to 1, with pressure settings ranging from 0.1 to 0.5 MPa. All other boundaries are treated as walls, including the surface of the oscillator.

iii. Meshing and setting up

(1) Meshing

The meshing was performed using the Fluent-Mesh module. The domains are composed of 137482 unstructured triangle elements and 73864 nodes. Element size is 0.004 m. The mesh for the nozzle's internal and external fluid regions is shown in Figure 3.17.

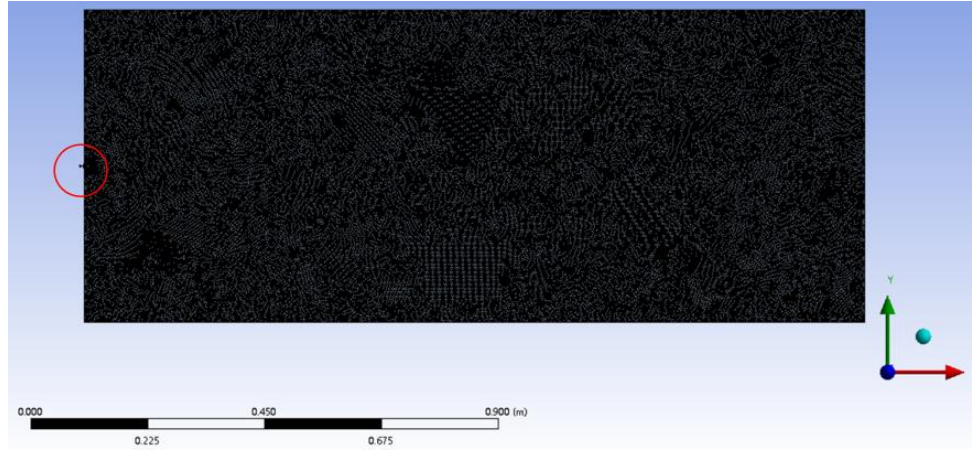


Figure 3.17: The mesh for the internal and external fluid regions of the nozzle

The mesh surrounding the nozzle is depicted as shown in Figure 3.18. The mesh inside the nozzle is depicted as shown in Figure 3.19. The boundary layer's inflation is illustrated as shown in Figure 4.4. The inflation option is total thickness. The number of Layers is 5, the Growth rate is 1.05, and the maximum thickness is 0.0005. The average orthogonal quality is excellent, with 0.9586.

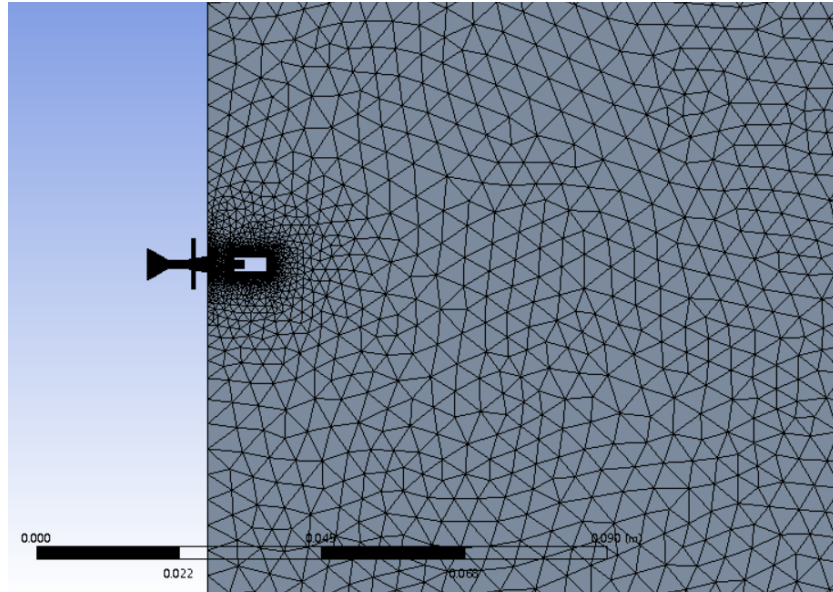


Figure 3.18: The mesh surrounding the nozzle

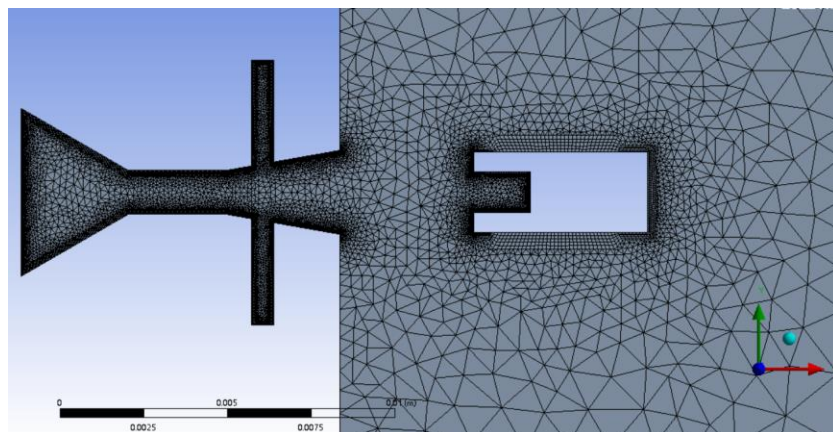


Figure 3.19: The mesh inside the nozzle

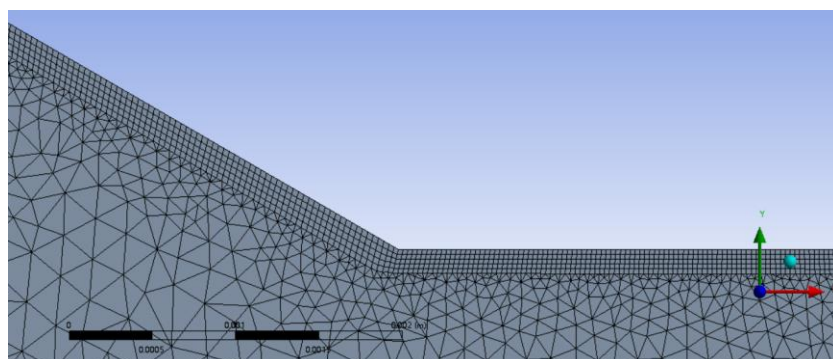


Figure 3.20: The inflation of the boundary layer

(2) Setting up

The mesh, properly partitioned, is imported into Fluent for solving. The materials selected are ideal gas and liquid water, with the energy equation activated. The pressure-velocity coupling method adopts the SIMPLE algorithm, as shown in Figure 3.21. The turbulence model chosen is the standard k- ϵ model, with standard wall functions.

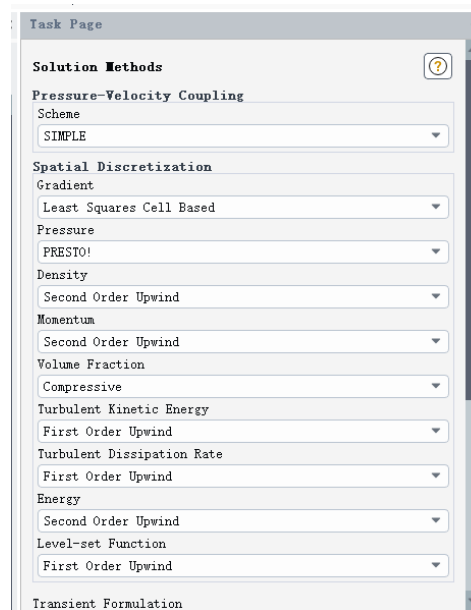


Figure 3.21: VOF solution method

A numerical analysis of the velocity, atomization angle, Mach number, and liquid phase volume fraction in the internal and external flow fields of the Laval nozzle is conducted under varying water pressures of 0.1, 0.2, 0.3, 0.4, and 0.5 MPa using Fluent's VOF model. This analysis aims to determine the optimal water pressure.

3.5.3.2 DPM Simulation Setup

The Discrete Phase Model (DPM) is used in Fluent to simulate the flow of particles. The DPM model assumes that particles are discrete point-like objects and simulates processes such as particle transport, collision, and deposition by tracking the movement of these particles in the fluid. This model is typically suitable for situations where there is a strong interaction between particles and the fluid, such as particle transport in air flows or particle sprays. It defines gas as the continuous phase and liquid droplet as the discrete phase. The DPM model can simulate droplet trajectories and display particle sizes (Peng et al., 2019).

In Fluent, users can simulate various types of particle behavior by setting parameters such as initial position, velocity, and size of the particles. Additionally, users can couple fluid flow conditions, boundary conditions, etc., with particle simulations to obtain information such as particle trajectories, concentration distributions, etc., within the fluid (Han et al., 2020; Ma et al., 2020; Yang et al., 2019).

i. Governing equation

Using the Lagrangian discrete phase model allows for a more accurate simulation of droplet atomization, providing a more realistic depiction of the process of droplet fragmentation and trajectory. The trajectory of the droplets is determined by solving the differential equations of particle forces in the Lagrangian coordinate system, which are then transformed into equilibrium equations in the Cartesian coordinate system shown in Equation 3.11 (Nie et al., 2023; Yin et al., 2019; Yu et al., 2020; Zhai et al., 2020).

$$\begin{aligned}
\frac{du_{p,x}}{dt} &= F_D(u_{v,x} - u_{p,x}) + \frac{g_x(\rho_p - \rho_v)}{\rho_p} - D_{T,P} \frac{1}{m_p T_F} \frac{\partial T_F}{\partial x} \\
\frac{du_{p,y}}{dt} &= F_D(u_{v,y} - u_{p,y}) + \frac{g_y(\rho_p - \rho_v)}{\rho_p} - D_{T,P} \frac{1}{m_p T_F} \frac{\partial T_F}{\partial y} \\
\frac{du_{p,z}}{dt} &= F_D(u_{v,z} - u_{p,z}) + \frac{g_z(\rho_p - \rho_v)}{\rho_p} - D_{T,P} \frac{1}{m_p T_F} \frac{\partial T_F}{\partial z}
\end{aligned}$$

Equation 3.11

Where,

- m_p ——Particle mass (kg);
- $D_{T,P}$ ——Thermal swimming force coefficient;
- ρ_v ——Fluid density (kg/m³);
- ρ_p ——Particle density (kg/m³);
- u_v ——Fluid velocity (m/s);
- u_p ——Particle velocity (m/s);
- T_F ——Fluid temperature (K).

The formula for the drag force F_D and the thermal swimming force coefficient $D_{T,P}$ during the movement of the droplet per unit mass is shown in Equation 3.12 to 3.16.

$$F_D = \frac{18\nu}{\rho_p d_p^2} \frac{C_D R_e}{24} \quad \text{Equation 3.12}$$

$$C_D = \frac{24}{R_e} \left(1 + 0.186 R_e^{0.6526} \right) + \frac{0.44 R_e}{7185.3535 + R_e} \quad \text{Equation 3.13}$$

$$R_e = \frac{d_p \rho_v |u_p - u_v|}{\nu} \quad \text{Equation 3.14}$$

$$D_{T,P} = \frac{7.02\pi d_p \nu^2 (k/k_p + 4.36\lambda_g/d_p)}{\rho (1 + 6.84\lambda_g/d_p) (1 + 2k/k_p + 8.72\lambda_g/d_p)} \quad \text{Equation 3.15}$$

$$k = \frac{15\nu R}{4} \quad \text{Equation 3.16}$$

- ν ——Hydrodynamic viscosity[kg/(m·s)]
 d_p ——Particle diameter (m)
 k ——Gas thermal conductivity [W/(m·K)]
 k_p ——Thermal conductivity of particles [W/(m·K)]
 λ_g ——Mean molecular free path of gas

In the discrete phase model, there is an interaction between the continuous and discrete phases. The movement of atomized droplets in the continuous phase is affected by the continuous phase, and there is momentum transfer between the continuous and discrete phases (Yin *et al.*, 2019). When the droplets pass through the continuous phase, the change value of particle momentum is shown in Equation 3.17.

$$\begin{aligned}
 P_x &= \sum \left(\frac{3\beta\nu C_D \text{Re}}{4\rho_p d_p^2} (u_{v,x} - u_{p,x}) + \frac{g_x(\rho_p - \rho_v)}{\rho_p} - D_{T,p} \frac{1}{m_p T_F} \frac{\partial T_F}{\partial x} \right) m'_p \Delta t \\
 P_y &= \sum \left(\frac{3\beta\nu C_D \text{Re}}{4\rho_p d_p^2} (u_{v,y} - u_{p,y}) + \frac{g_y(\rho_p - \rho_v)}{\rho_p} - D_{T,p} \frac{1}{m_p T_F} \frac{\partial T_F}{\partial y} \right) m'_p \Delta t \\
 P_z &= \sum \left(\frac{3\beta\nu C_D \text{Re}}{4\rho_p d_p^2} (u_{v,z} - u_{p,z}) + \frac{g_z(\rho_p - \rho_v)}{\rho_p} - D_{T,p} \frac{1}{m_p T_F} \frac{\partial T_F}{\partial z} \right) m'_p \Delta t
 \end{aligned}
 \tag{Equation 3.17}$$

When there is no chemical reaction between the discrete and continuous phases, the heat exchange is shown in Equation 3.18.

$$Q = \left[\frac{\overline{m_p}}{m_{p,0}} c_p \Delta T_p + \frac{\Delta m_p}{m_{p,0}} \left(-h_{fg} + \int_{T_{ref}}^{T_p} c_{p,i} dT \right) \right] m'_{p,0}
 \tag{Equation 3.18}$$

Here,

- $\dot{m}_p$ — Particle mass flow rate (kg/s)
 $\dot{m}_{p,0}$ — Initial mass flow rate of particles (kg/s)
 $m_{p,0}$ — Initial mass of particles (kg)
 C_p — Specific heat capacity of particles [J/(kg · K)]
 h_{fg} — Latent heat from volatilization analysis (J/kg)
 $C_{p,i}$ — Specific heat of volatiles precipitated [J/(kg · K)]
 $\overline{m}_p$ — Average particle mass (kg);
 T_p — Particle temperature (K).

When coupling between the discrete and continuous phases, the relationship for particle mass variation is given in Equation 3.19.

$$M = \frac{\Delta m_p}{m_{p,0}} \dot{m}_{p,0} \quad \text{Equation 3.19}$$

ii. Domain and Meshing

Domain and Meshing, like VOM, will not be introduced here.

iii. Boundary conditions and setting up

The boundary conditions for DPM differ from those of the VOF model. The gas inlet, pressure outlet, exhaust, and wall conditions are the same as in the VOF model. The water inlet of DPM is designed as a particle source, and the particle material is liquid water. An ideal gas is chosen for the gas phase with the energy equation enabled. Pressure-velocity coupling is implemented using the Coupled algorithm, as shown in Figure 3.22. The turbulence model employed is the standard k-ε model with standard wall functions. Diameter Distribution follows the Rosin-Rammler distribution. Droplet parameters are set as indicated in Table 3.1 (Yang *et al.*, 2023).

Table 3.1: DPM droplet source Settings

Water Pressure	0.1MPa	0.2MPa	0.3MPa	0.4MPa	0.5MPa
Velocity magnitude(m/s)	2.89	5.37	7.16	9.71	12.84
Total flow rate(kg/s)	0.0010479	0.001949	0.0026008	0.0035258	0.0046638
Maximum particle diameter(mm)	0.04	0.09	0.105	0.125	0.15
Minimum particle diameter(mm)	0.005	0.002	0.001	0.002	0.002
Mean particle diameter(mm)	0.02	0.028	0.031	0.04	0.05

Task Page

Solution Methods ?

Pressure-Velocity Coupling

Scheme
Coupled

Flux Type
Rhie-Chow: distance based ☐ Auto Select

Spatial Discretization

Gradient
Green-Gauss Cell Based

Pressure
Standard

Density
First Order Upwind

Momentum
First Order Upwind

Turbulent Kinetic Energy
First Order Upwind

Turbulent Dissipation Rate
First Order Upwind

Energy
First Order Upwind

Transient Formulation

First Order Implicit

☐ Non-Iterative Time Advancement

☐ Frozen Flux Formulation

☐ Warped-Face Gradient Correction

Figure 3.22: DPM solution methods

Under varied water pressures of 0.1, 0.2, 0.3, 0.4, and 0.5 MPa, numerical analysis using Fluent's DPM will be conducted to investigate the spray shape and droplet sizes in the outflow field of the Laval nozzle.

3.5.4 Numerical Analysis with Varied Air Pressure

With a water pressure of 0.1 MPa, a nozzle outlet diameter of 2.5 mm, and a nozzle-to-oscillator distance of 4 mm, air pressures of 0.2, 0.3, 0.4, 0.5, and 0.6 MPa are set. VOF is utilized for numerical analysis to investigate the Laval nozzle's spray shape, velocity, atomization cone angle, and liquid volume fraction. The modeling of the Laval nozzle's internal and external flow domains, meshing, and parameter settings remains consistent with Section 3.5.3, with adjustments made to the air and water pressure boundary conditions.

3.5.5 Numerical Analysis with Varied Nozzle Outlet Diameters

According to literature, the Mach number achievable by accelerating airflow in a supersonic nozzle is determined by the ratio of the exit area to the throat area (Gerhardt et al., 2022). Therefore, studying the impact of varied outlet diameters on the Laval nozzle is significant. With an air pressure of 0.5 MPa, water pressure of 0.1 MPa, and a nozzle-to-oscillator distance of 4 mm, varied nozzle outlet diameters of 1.5, 2.0, 2.5, 3.0, and 3.5 mm are configured. VOF is utilized for numerical analysis to investigate the Laval nozzle's spray shape, velocity, atomization cone angle, and liquid volume fraction. The modeling of the internal and external flow domains of the Laval nozzle, meshing, and parameter settings remains consistent with Section 3.5.3, with the only alteration being the change in the nozzle outlet diameter.

3.5.6 Numerical Analysis with Varied Oscillator Distances

The high-frequency vibration of the oscillator leads to secondary atomization of the gas-liquid mixture, and the distance from the oscillator to the nozzle is known to affect the flow field. Under an air pressure of 0.5 MPa and water pressure of 0.1 MPa, with a nozzle outlet diameter of 2.5 mm, the distance from the oscillator to the nozzle varies. The distances are 2 mm, 3 mm, 4 mm, 5 mm, and 6 mm. The study employs the VOF model to investigate the spray velocity and atomization angle under varied oscillator distances. Modifying the Laval nozzle's internal and external flow domains, meshing, and parameter settings remains consistent with Section 3.5.3.

Based on the literature and field application data, the summary of numerical simulations for various parameters is presented in Table 3.2.

Table 3.2: The summary of numerical simulation for various parameters

Varied Water Pressure (MPa)	Varied Air Pressure (MPa)	Varied Nozzle Diameter (mm)	Varied Oscillator Distance (mm)
0.1	0.2	1.5	2
0.2	0.3	2	3
0.3	0.4	2.5	4
0.4	0.5	3	5
0.5	0.6	3.5	6

3.5.7 Experiments with Varied Water Pressure

3.5.7.1 Experimental Process

This study used spray test equipment to measure the flow field of two-fluid spray. The test equipment mainly comprises a laser particle size analyzer test system and a two-fluid spray system. China OMEC Corporation manufactures the laser particle size analyzer used. Figure 3.23 illustrates the schematic diagram of the spray test apparatus. The spray system comprises a booster pump, an air compressor, a water tank, pressure regulating valves, a barometer, a water manometer, and a Laval nozzle.

During the experiment, the desired operating parameters were first adjusted. Subsequently, the water pump and air compressor were activated, initiating the spray from the two-fluid nozzle. Simultaneously, the timer was started, and after approximately 10 seconds, when the flow field stabilized, data collection began. The laser particle size analyzer test system automatically recorded and processed the collected data. Each set of measurements lasted approximately 20 seconds. The ambient temperature during the experiment is maintained at room temperature, with a relative humidity range of 60% to 65%.

Experimental measurements were conducted on a nozzle with an outlet diameter of 2mm and an oscillator distance of 4mm, operating at an air pressure of 0.5 MPa. The study investigated the droplet size and atomization angle at varied water pressures (0.1, 0.2, 0.3, 0.4, and 0.5 MPa) while observing the spray morphology.

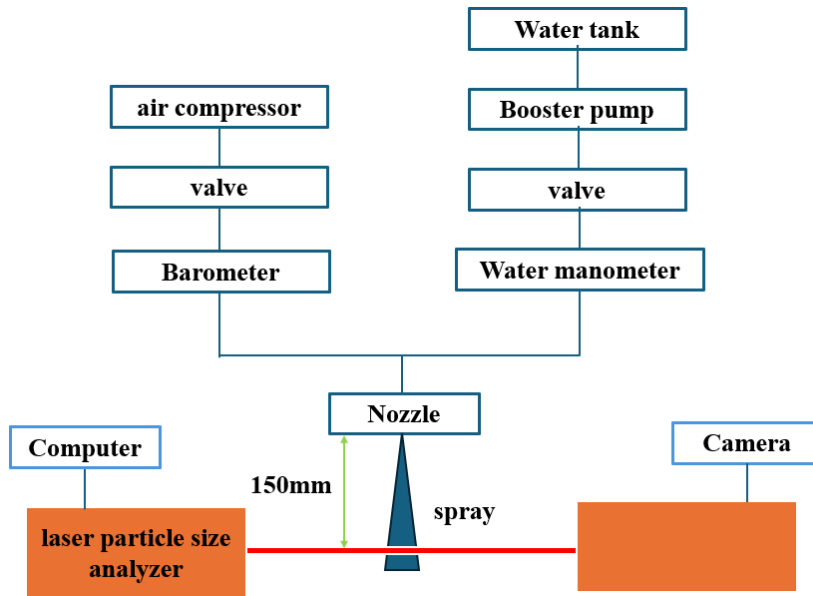


Figure 3.23: Test system diagram

Figure 3.24 shows the working principle of the test system. The air compressor and booster pump power the system, with pressure controlled by valves, and the operating parameters are monitored by barometer and water manometer. Compressed air and water enter the Laval nozzle through pipes. A high-definition camera captures the spray morphology to measure the spray cone angle. The laser particle size analyzer measures droplet size and distribution.

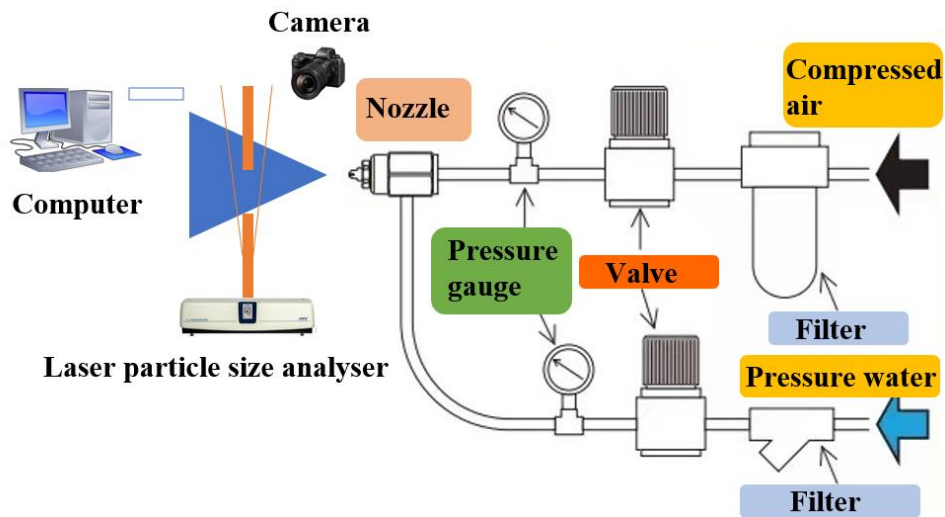


Figure 3.24: Working principle of the test system

3.5.7.2 Experimental Equipment

Figure 3.25 depicts a water tank and an air compressor. The compressor is a Jaguar screw air compressor ES-10, with a rated exhaust pressure of 0.8 MPa and a rated exhaust volume of 1 m³/min. The air compressor is the primary power source for the compressed air in the two-fluid spray system. Equipped with an air reservoir, it provides a stable and ample compressed air supply for the two-fluid spray system.

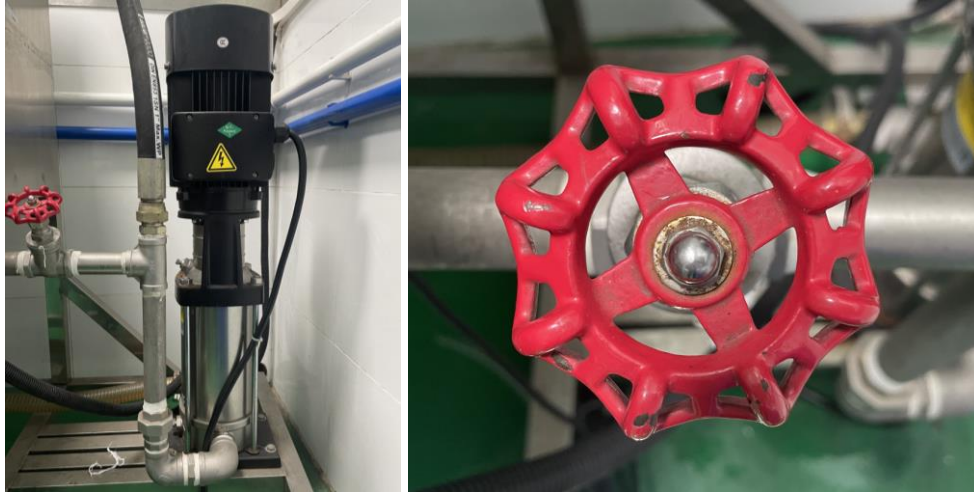


(a) Water tank

(b) Air compressor

Figure 3.25: Water tank and air compressor

Figure 3.26 illustrates a booster pump and a valve. The pump is a lightweight vertical multistage centrifugal pump, model CDLS2-13LS WSC. It has a rated flow of $2 \text{ m}^3/\text{h}$ and a rated pressure of 2.2 MPa. The pump serves as the primary power source for the pressurized water supply in the two-fluid spray system, ensuring an ample supply of pressurized water for the system.



(a) Booster pump

(b) Valve

Figure 3.26: Booster pump and valve



(a) Laser particle size analyzer

(b) The location during its usage

Figure 3.27: Laser particle analyzer and the location during its usage

Figure 3.27 shows the Laser particle size analyser its location during its usage. The OMEGA DP-02 Laser Particle size Analyzer (manufactured by Zhuhai OMEGA Instrument Co., Ltd.) is utilized for measuring droplet sizes. The measurement range for droplet sizes is 0.5 to 1500 μm , with a measurement error of less than 3%. The light source is a He-Ne laser with a power of 2 mW and a wavelength of 0.6328 μm . The diameter of the measurement spot is approximately 8 to 10 mm.



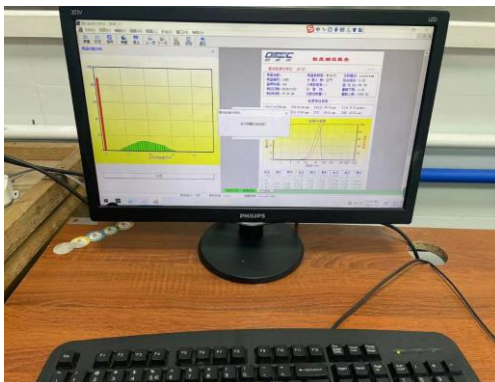
(a) Water manometer



(b) Barometer

Figure 3.28: Water manometer and barometer

Figure 3.28 displays a water manometer and a barometer. The water manometer is a Red Flag water pressure gauge, model YTN-100, with a 0-1 MPa measurement range. The barometer is an ADECO company's gas source processor, model GAR60025SJ, with a pressure regulation range of 0-2 MPa.



(a) Data collection computer



(b) Laval Nozzle

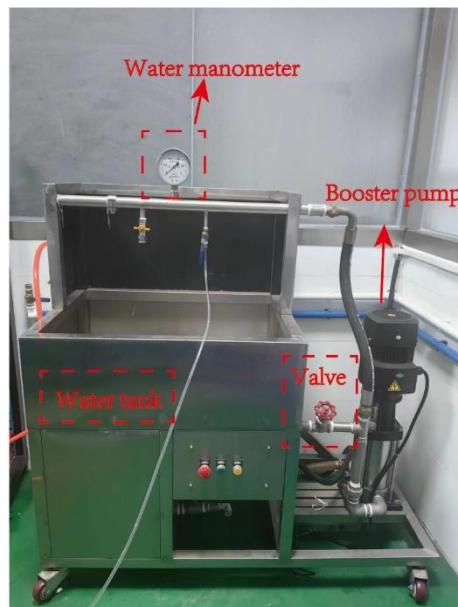
Figure 3.29: Data collection computer and Laval nozzle

Figure 3.29 shows the data collection computer and laser particle analyzer. The computer is connected to the laser particle analyzer to collect and analyze experimental data. The Laval nozzle has an outlet diameter of 2.5 mm, with the oscillator positioned 4mm away from the nozzle outlet.



(a) Measuring particle size

(b) Photographing the spray morphology



(c) Experimental control console

Figure 3.30: Photos of the experimental site

Figure 3.30 displays photographs of the experimental site. (a) The laser is positioned 150mm away from the nozzle. (b) A camera is used to capture the spray morphology, and the Online Protractor is used to measure the atomization angle. (c) The image shows the experimental control console.

3.5.8 Experiments with Varied Air Pressure

The experimental parameters were consistent with the simulation parameters, maintaining a water pressure of 0.1 MPa and setting air pressures to 0.2 MPa, 0.3 MPa, 0.4 MPa, 0.5 MPa, and 0.6 MPa.

Table 3.3: Experiments with varied air pressure

Water Pressure	Air Pressure
0.1 MPa	0.2 MPa
	0.3 MPa
	0.4 MPa
	0.5 MPa
	0.6 MPa

In this research, the experiment measured the droplet size and atomization angle of a nozzle with an outlet diameter of 2.5 mm and an oscillator distance of 4 mm at a water pressure of 0.1 MPa and varying air pressures (0.2, 0.3, 0.4, 0.5, and 0.6 MPa), as shown in Table 3.3. Additionally, the spray morphology was observed.

3.5.9 Grid (in CFD) Independence Test and Residual Plot

This section examines the impact of the total number of grids on computational results. The grid independence test, conducted under conditions of 0.5 MPa air pressure and 0.1 MPa water pressure, is presented in Table 3.4. As shown in Table 3.4, variations in the number of grids have some effect on the computational results within a certain range. However, the

overall deviation is within 0.5%, meeting the requirements for grid independence. Considering computational accuracy and efficiency, the total number of grids is ultimately set at 137,482 for the mesh division.

Table 3.4: The grid independence test with varied element size

Number	Element Size (m)	Elements	Nodes	(VOF) Maximum Velocity Magnitude (m/s)	DPM/D (3,2) (m)	Percentage of Differences
1	0.005	93770	51904	509	1.520529 e-5	0.39%
2	0.004	137482	73864	510	1.519298e-5	0%
3	0.003	228900	119746	511	1.517333e-5	0.13%

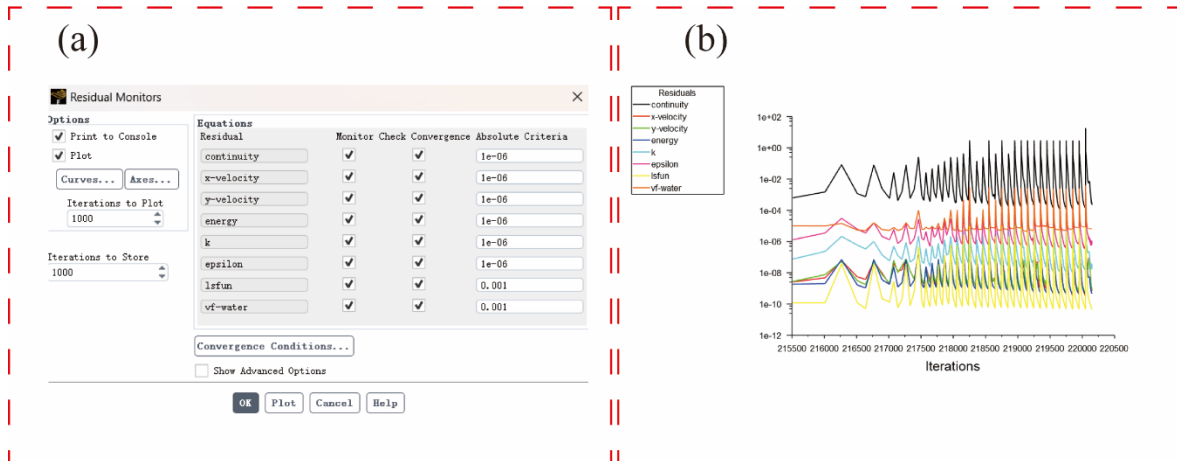


Figure 3.31: (a) convergence criteria; (b) residual plot

In the transient simulation conducted in this study, which focuses on high-speed gas–liquid two-phase flow with the Laval nozzle, the residual criterion was set to 10^{-6} , as shown in Figure 3.31. Although the continuity residual at each time step converged to below 10^{-4} , which does not reach the preset threshold of 10^{-6} , it still meets the widely accepted

engineering accuracy requirements for transient simulations. During the convergence process, key flow variables such as velocity and pressure showed a stable trend and exhibited negligible changes in subsequent iterations, indicating that the solution had reached a steady and stable state. Mass flux monitoring revealed that the difference between inlet and outlet mass flow rates was within 2%, thereby satisfying the mass conservation requirements under engineering tolerances. Additionally, the residuals of the momentum, turbulence, and energy equations all met or exceeded the specified convergence criteria, and the coupled solution showed good consistency without any localized non-physical anomalies. The simulation results were in good agreement with theoretical trends, and repeated simulations produced consistent outcomes, further confirming the reliability of the solution.

In summary, the numerical solution achieved good convergence and physical reliability, making it a sound basis for the subsequent analysis of atomization performance.

3.6 Summary of the Chapter

This chapter combined numerical analysis and experiments to study the atomization performance of the linear Laval nozzle and optimize the nozzle parameters and operating conditions. This part includes research variables, research tools, and specific research steps. Specific research steps include linear Laval nozzle structure design, verification of the existence of internal supersonic airflow, numerical analysis of the flow field characteristics of the linear Laval nozzle under varied water pressure, varied air pressure, varied outlet diameter, and varied oscillator distance using Fluent's VOF model and DPM model. Finally, the droplet size and atomization angle of spray under varied water pressure and atomization angle under varied air pressure were measured, and a comparison with numerical analysis was conducted.

CHAPTER 4

RESULTS, ANALYSES AND DISCUSSIONS

4.1 Overview

In this chapter, the results, analysis, and discussion include verification of the supersonic effect of the Laval nozzle, results of numerical analysis with varied water pressure, results of numerical analysis with varied air pressure, results of numerical analysis with varied nozzle diameters, results of numerical analysis with varied oscillator distances, 3D model validation, analysis of experimental results with varied water pressure, analysis of experimental results with varied air pressure, research contribution in sections 4.2 through 4.10, respectively.

4.2 Verification of the Supersonic Effect of the Laval Nozzle

4.2.1 Numerical Simulation Results of the Internal Flow Field in the Nozzle

The airflow velocity in the nozzle directly determines the atomization effect, so the airflow velocity of the supersonic nozzle should not be less than one times the speed of sound.

Figure 4.1 shows the contours of the Mach number inside the Laval nozzle.

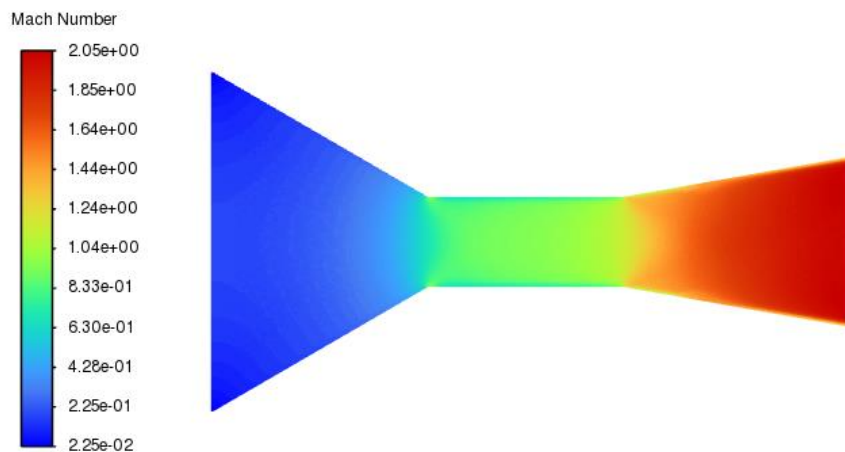


Figure 4.1: Contours of Mach number

In the initial section of the jet, the color of the contours of the Mach number is blue, indicating lower velocities. As the flow field contracts, the velocity gradually increases. Upon reaching the throat section, the color of the contours of the Mach number changes to green, indicating a Mach number of approximately 1, at which point the velocity reaches the speed of sound. After passing through the throat section, the velocity cloud map turns red. The expansion wave in the diverging section continues accelerating the fluid, reaching a Mach number of 2.05 and achieving supersonic speed. Furthermore, the airflow remains stable. There are no shock waves within the entire diverging section, and the airflow remains attached to the walls, resulting in no vortex loss or fluid separation.

4.2.2 Comparison with Other Numerical Analysis Results

As shown in Figure 4.2, in some previous studies, flow separation occurred in the Laval nozzle diverging section, which quickly generated vortices and resulted in enormous energy loss. No flow separation occurred in this study in the diverging section, and the numerical simulation results were better (Zhang *et al.*, 2020a). As shown in Figure 4.3 and Figure 4.4, in some studies, the maximum speed of droplets is 350 m/s. This study's speed is as high as 500 m/s, which is more conducive for dust removal (Zhang *et al.*, 2020b).

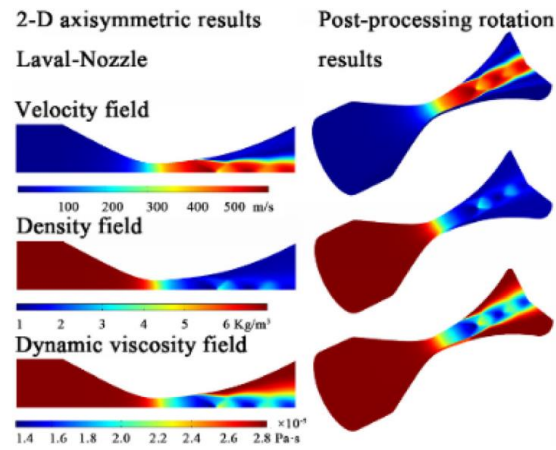


Figure 4.2: Flow separation occurred in the Laval nozzle diverging section (Zhang *et al.*, 2020a)

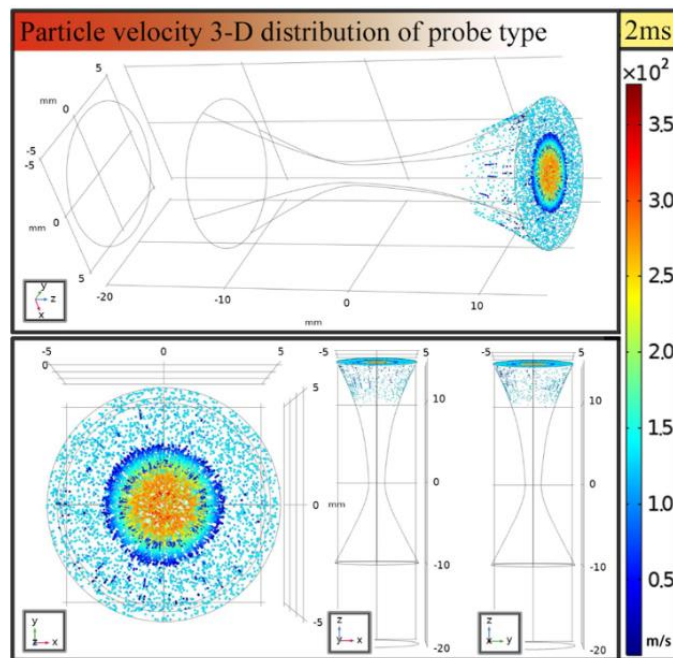


Figure 4.3: The maximum speed of droplets in Zhang's paper (Zhang *et al.*, 2020b)



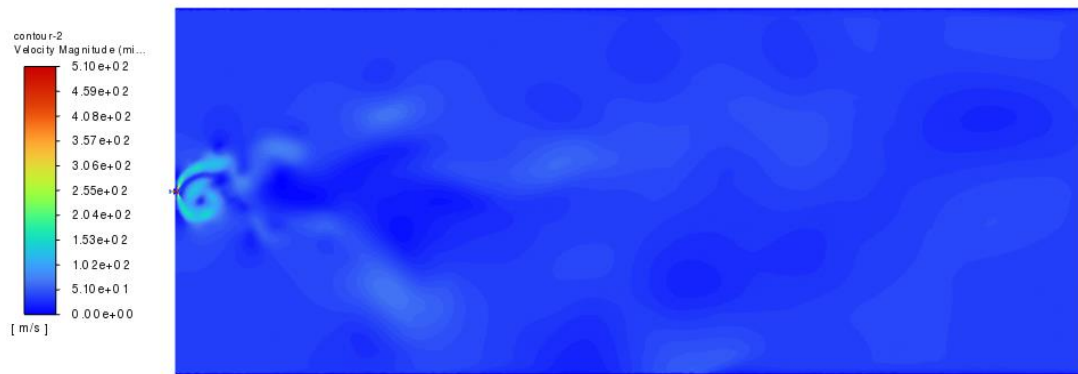
Figure 4.4: The velocity magnitude of droplets in this research

4.3 Results of Numerical Analysis with Varied Water Pressure

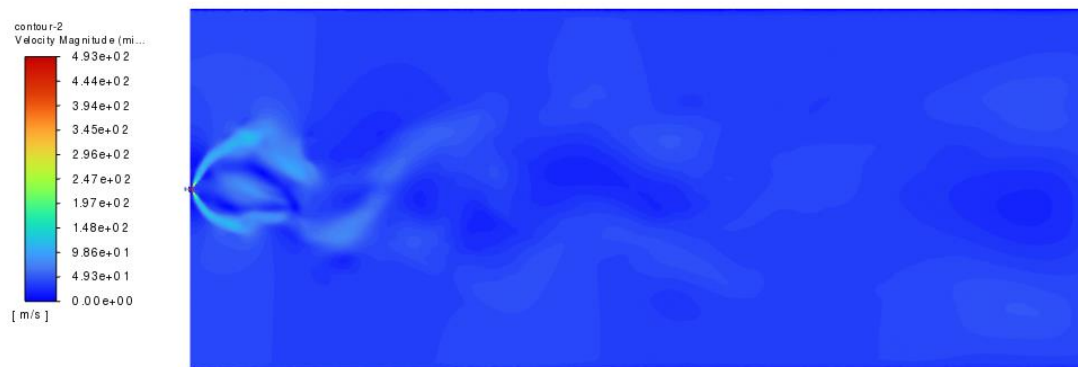
At an air pressure of 0.5 MPa, the effects of different water pressures on spray velocity, atomization angle, and droplet size were analyzed. The study examined how changes in water pressure influenced these spray characteristics.

4.3.1 Velocity Analysis

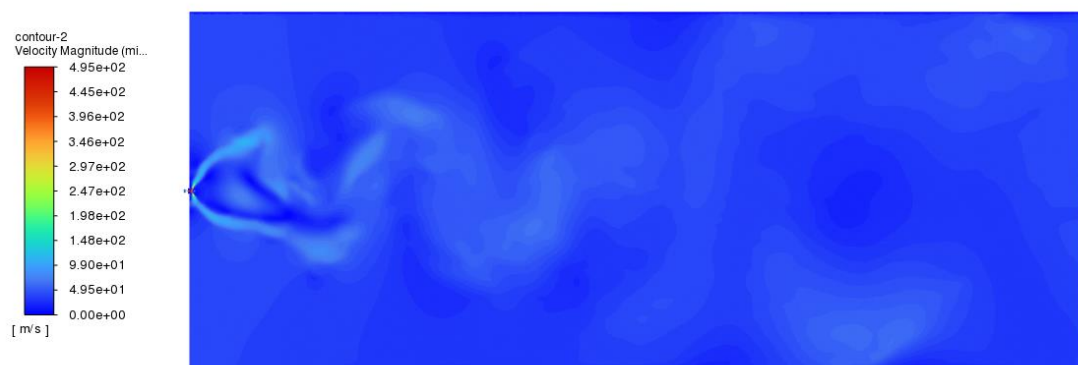
The mechanism of spray dust removal is to mist water into tiny droplets, which are sprayed into the air and in contact with floating dust. The main factor affecting the dust-trapping effect of water droplets is the relative speed of water droplets and dust particles, which determines the contact effect between dust and water droplets. The high speed of water droplets leads to sizeable kinetic energy, and the collision with dust particles is conducive to overcoming the surface tension of water and wetting and capturing dust particles more effectively. Moreover, it is generally believed that the greater the nozzle exit velocity, the more stable the flow field.



(a) Contours of velocity magnitude at 0.1 MPa water pressure

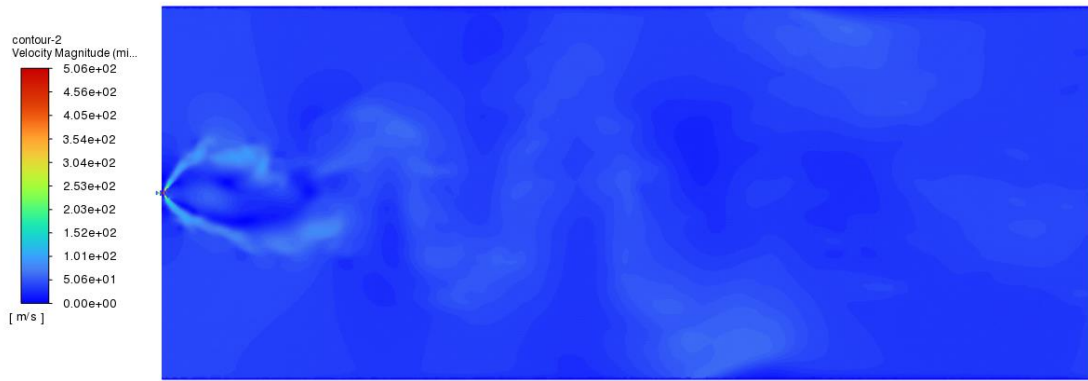


(b) Contours of velocity magnitude at 0.2 MPa water pressure

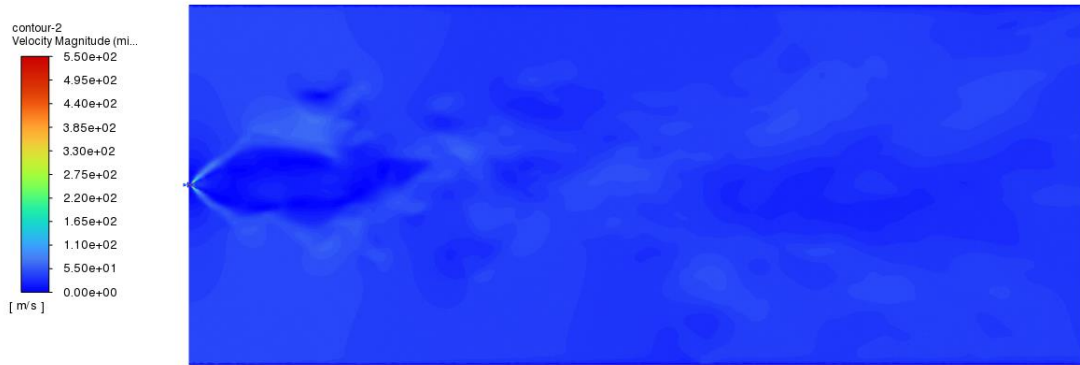


(c) Contours of velocity magnitude at 0.3 MPa water pressure

Figure 4.5: Contours of velocity magnitude at varied water pressures



(d) Contours of velocity magnitude at 0.4 MPa water pressure



(e) Contours of velocity magnitude at 0.5 MPa water pressure

Figure 4.5 (continued): Contours of velocity magnitude at varied water pressures

Figure 4.5 illustrates contours of velocity magnitude under varied water pressures. Water pressure has a significant impact on velocity magnitude as well as the morphology of the flow field. Velocity magnitude is higher near the nozzle, gradually diminishing in the outer flow field. The presence of the oscillator increases the atomization angle and creates vortices near the nozzle outlet, which facilitates atomization. Under varied water pressures, the maximum velocity magnitude of the mixture varies, as shown in Table 4.1.

Table 4.1: The maximum velocity of the mixture under varied water pressures

Air pressure is 0.5MPa. Water pressure (MPa)	Maximum velocity magnitude of mixture (m/s)
0.1	510
0.2	493
0.3	495
0.4	506
0.5	550

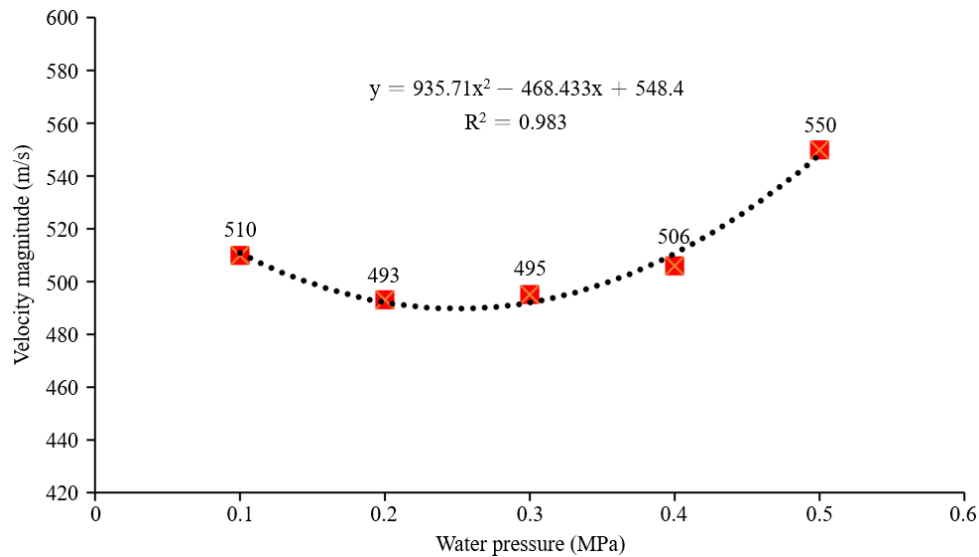


Figure 4.6: Scatter plot of velocity magnitude under varied water pressures

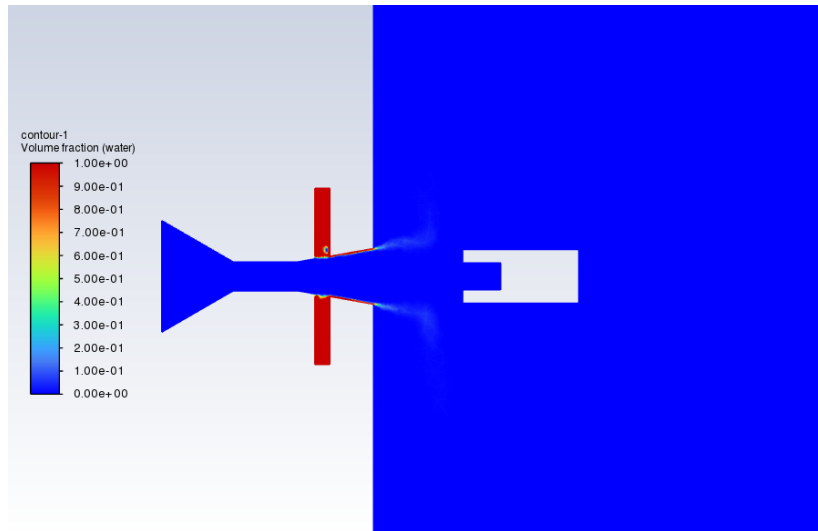
The scatter plot shown in Figure 4.6, with water pressure on the x-axis and velocity magnitude on the y-axis, illustrates a decrease and an increase in velocity magnitude as water pressure varies from 0.1 MPa to 0.5 MPa. At a water pressure of 0.5 MPa, the velocity magnitude reaches its maximum, measuring 550 m/s. Then, at a water pressure of 0.1 MPa, the velocity magnitude is 510 m/s. The minimum velocity magnitude is observed at a water pressure of 0.2 MPa, measuring 493 m/s. The fitting curve for velocity magnitude and water

pressure is given by the equation $y = 935.71 x^2 - 468.433x + 548.4$, $R^2 = 0.983$, demonstrating a good fit.

The velocity magnitude of the gas-liquid mixture decreases first and then increases, possibly because when the gap between water pressure and air pressure is relatively large, increasing water pressure will hinder air movement. When the water pressure is close to the air pressure, it promotes the movement of the air. According to the literature, the greater the droplet outlet velocity, the better the bonding effect with dust (Xie *et al.*, 2023). So, the water pressure of 0.1 MPa and 0.5 MPa is better.

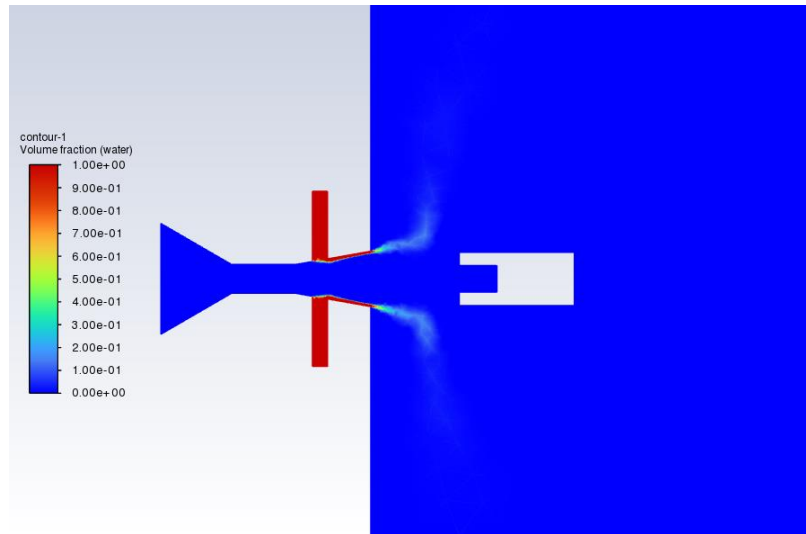
4.3.2 Atomization Angle Analysis

The atomization angle determines the distribution of fluid space to a large extent. The larger the atomization angle, the larger the influence range and the better the atomization performance.

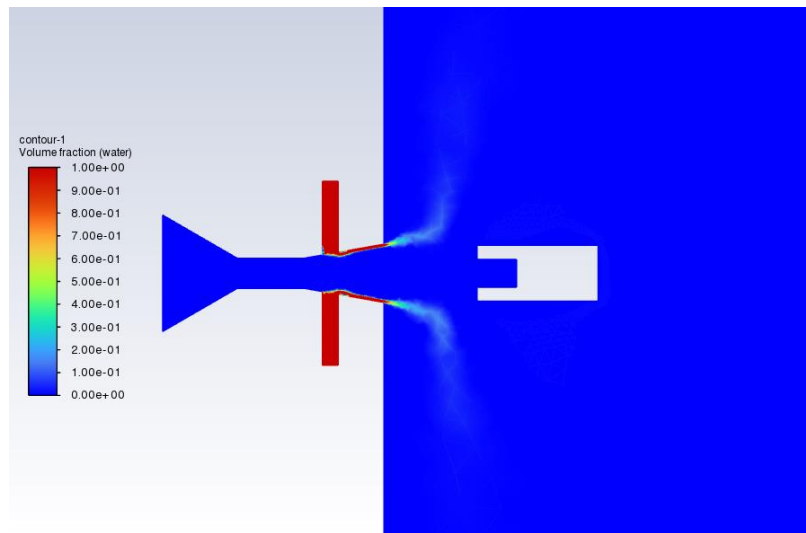


(a) Contours of water volume fraction at 0.1MPa water pressure

Figure 4.7: Contours of water volume fraction at varied water pressures

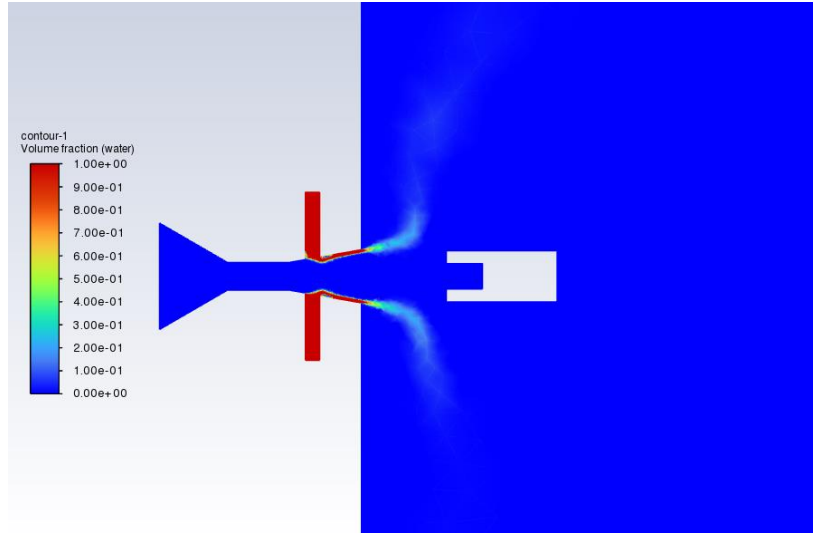


(b) Contours of water volume fraction at 0.2MPa water pressure

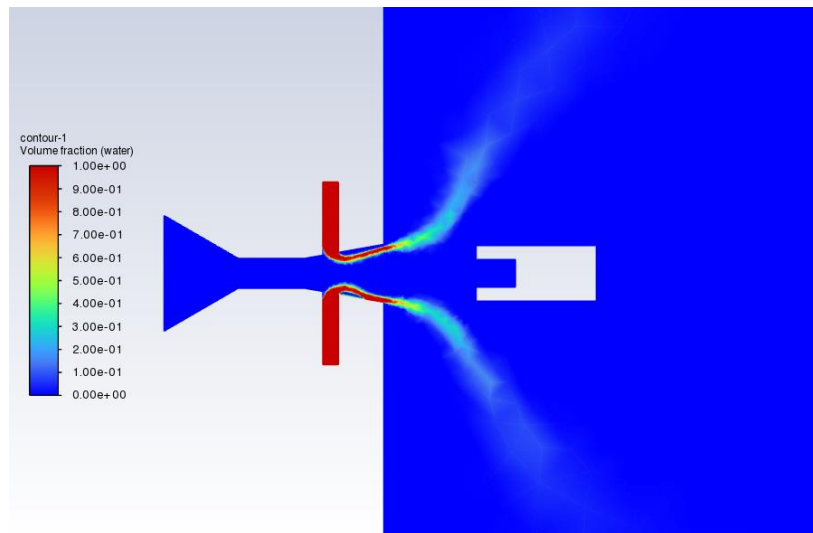


(c) Contours of water volume fraction at 0.3MPa water pressure

Figure 4.7 (continue): Contours of water volume fraction at varied water pressures



(d) Contours of water volume fraction at 0.4MPa water pressure

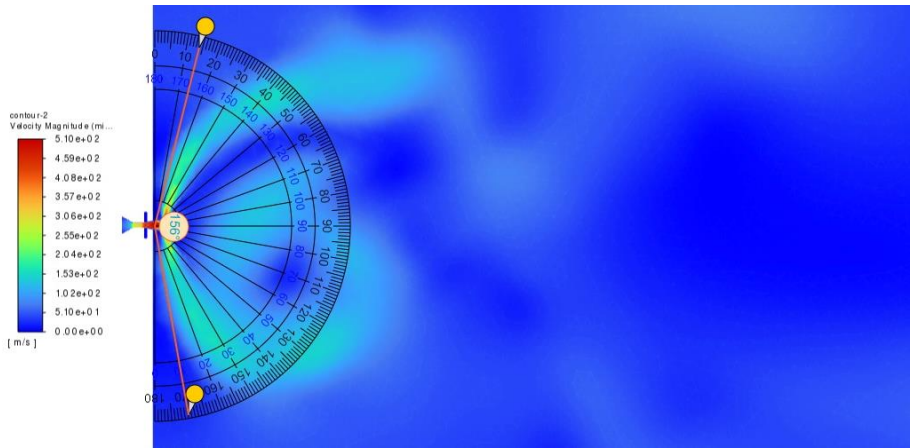


(e) Contours of water volume fraction at 0.5MPa water pressure

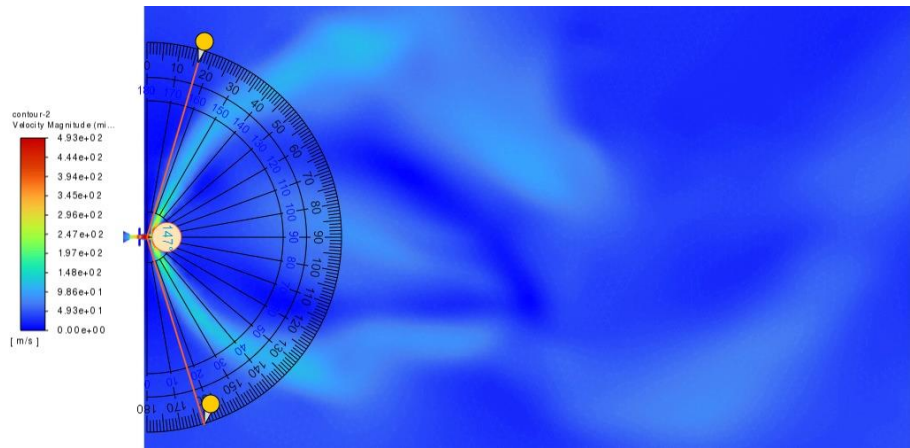
Figure 4.7 (continue): Contours of water volume fraction at varied water pressures

Figure 4.7 shows the Contours of water volume fraction under varied water pressures. The figure shows that due to atomization, the water volume fraction is lower in the outer flow field, especially at lower water pressures. Therefore, in this research, the angle of the

contours of velocity magnitude is measured to compare the atomization range under varied water pressures.

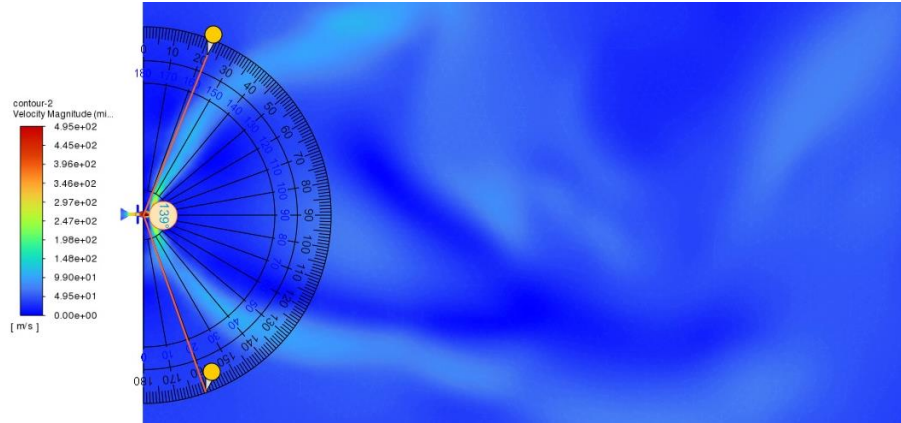


(a) Angle of atomization range at 0.1MPa water pressure

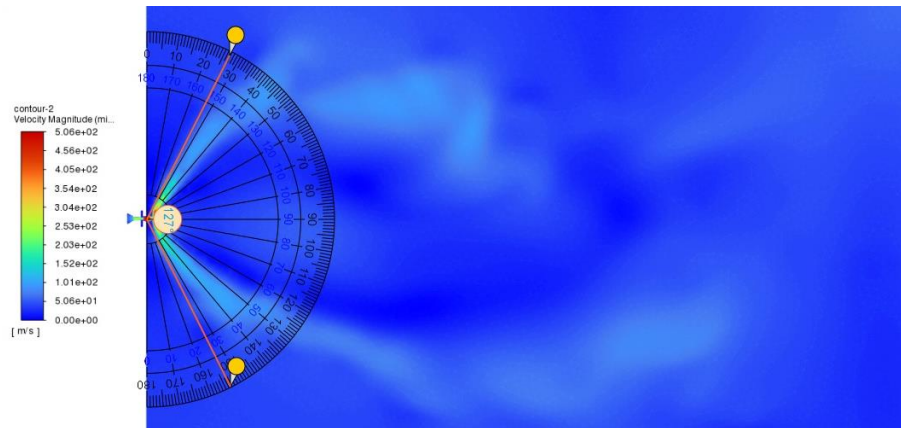


(b) Angle of atomization range at 0.2MPa water pressure

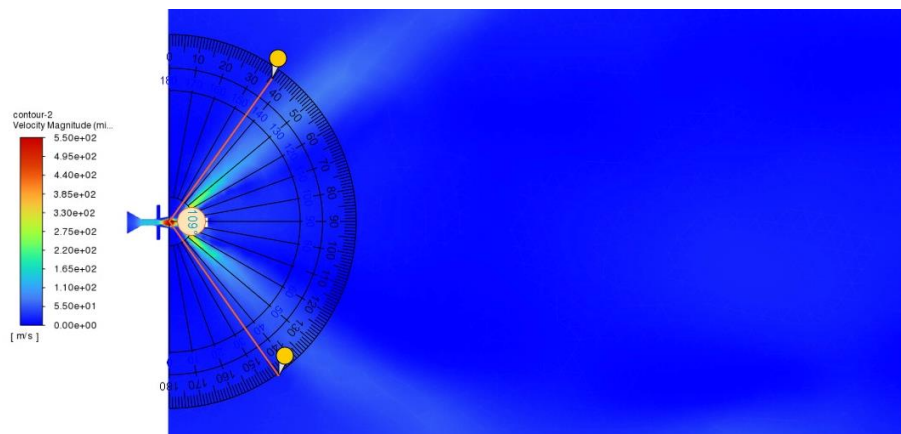
Figure 4.8: Angle of atomization range at varied water pressures



(c) Angle of atomization range at 0.3MPa water pressure



(d) Angle of atomization range at 0.4MPa water pressure



(e) Angle of atomization range at 0.5MPa water pressure

Figure 4.8 (continue): Angle of atomization range at varied water pressures

The angle of the atomization range was measured using the Online Protractor tool (OPT) shown in Figure 4.8, which, to some extent, reflects the size of the atomization angle. It is evident from Figure 4.8 that as the water pressure increases, the angle of the atomization range decreases. Higher pressure leads to an increase in jet velocity, resulting in droplets with higher kinetic energy. These high-energy droplets follow a more linear trajectory in the air, experiencing less resistance and deviation. So, the droplets move forward in a straight line under higher pressure conditions without quickly deviating, resulting in a smaller atomization angle. The data regarding the angle of atomization range under varied water pressures is listed in Table 4.2.

Table 4.2: Angle of atomization range under varied water pressures

Air pressure is 0.5MPa. Water pressure (MPa)	Angle of atomization range (°)
0.1	156
0.2	147
0.3	139
0.4	127
0.5	109

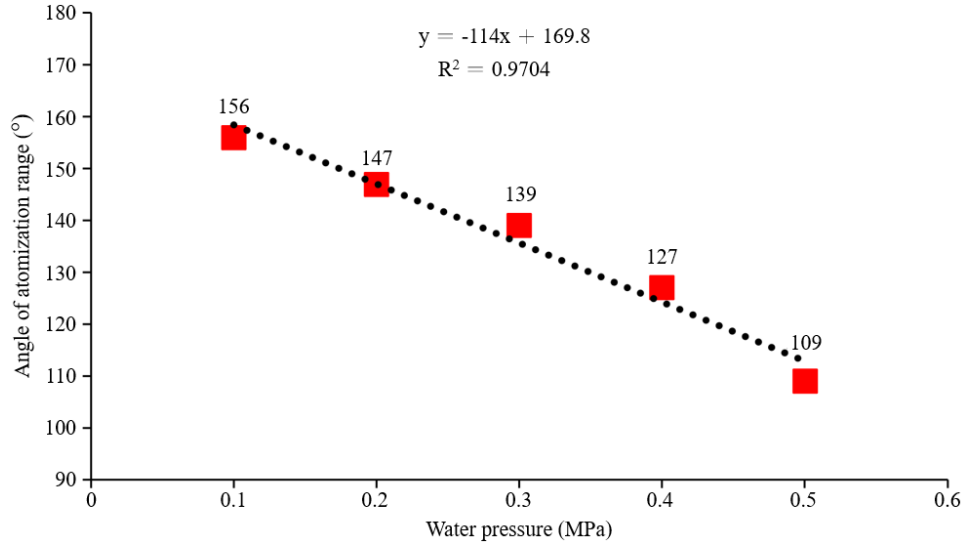


Figure 4.9: Scatter plot of the angle of atomization range under varied water pressures

The scatter plot shown in Figure 4.9 with water pressure on the x-axis and angle of atomization range on the y-axis illustrates a decrease in the angle of atomization range as water pressure varies from 0.1 MPa to 0.5 MPa. At a water pressure of 0.1 MPa, the angle of atomization range is at its maximum, measuring 156°, whereas at 0.5 MPa, it decreases to 109°. Therefore, the optimal atomization angle is observed at 0.1 MPa. The fitting curve for the relationship between the angle of atomization range and water pressure is given by $y = -114x + 169.8$, $R^2 = 0.9704$, showing a very good fit.

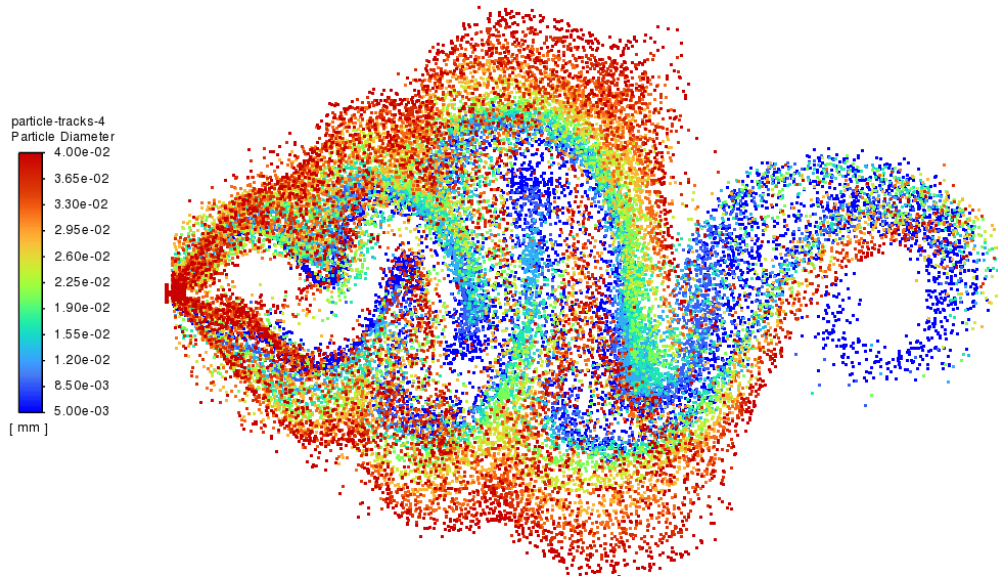
4.3.3 Droplet Size Analysis

In practical theory and application, the Sauter mean diameter (SMD) is commonly used to represent the droplet size in a fog. The Sauter mean diameter denotes the surface area-weighted mean diameter and is denoted by $D_{3,2}$. Its mathematical expression, as given by Equation 4.1, is:

$$D(3,2) = (\sum_{i=1}^m n_i \bar{d}_i^2 \cdot \bar{d}_i) / (\sum_{i=1}^m n_i \bar{d}_i^2) \quad \text{Equation 4.1}$$

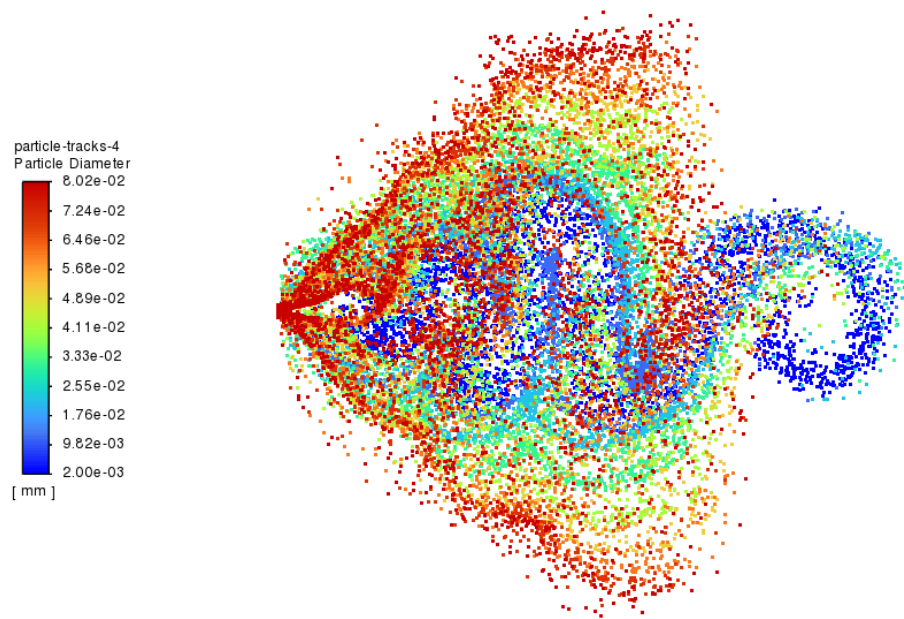
$\bar{d}_i$ represents the diameter of each droplet, and m is the total number of droplets.

The relationship $n_i \bar{d}_i^2$ is directly proportional to the total surface area of particles in the i size interval. Therefore, $D(3,2)$ represents the average particle diameter for the surface area, known as the surface area mean diameter or the Sauter mean diameter.

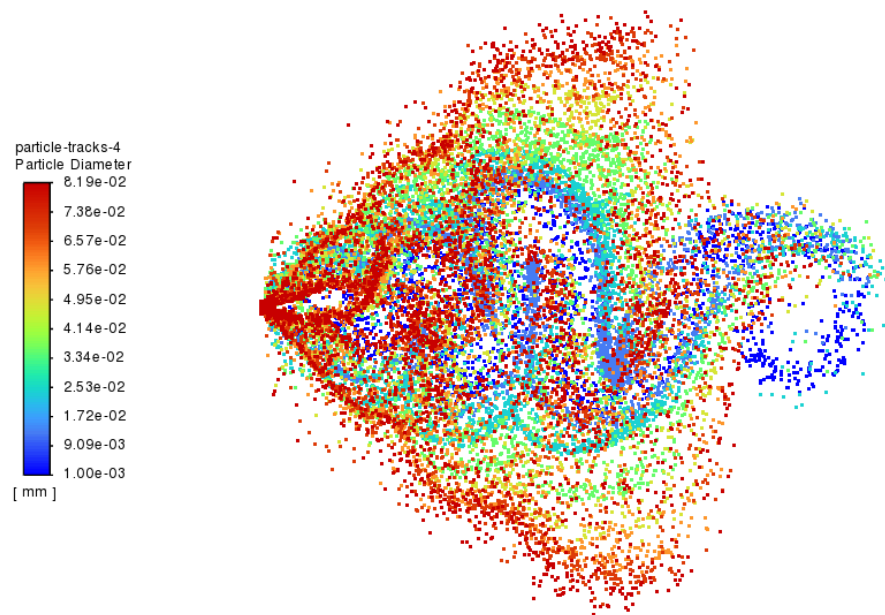


(a) Droplet traces colored by droplet diameter at 0.1 MPa water pressure

Figure 4.10: Droplet traces colored by droplet diameter at varied water pressures

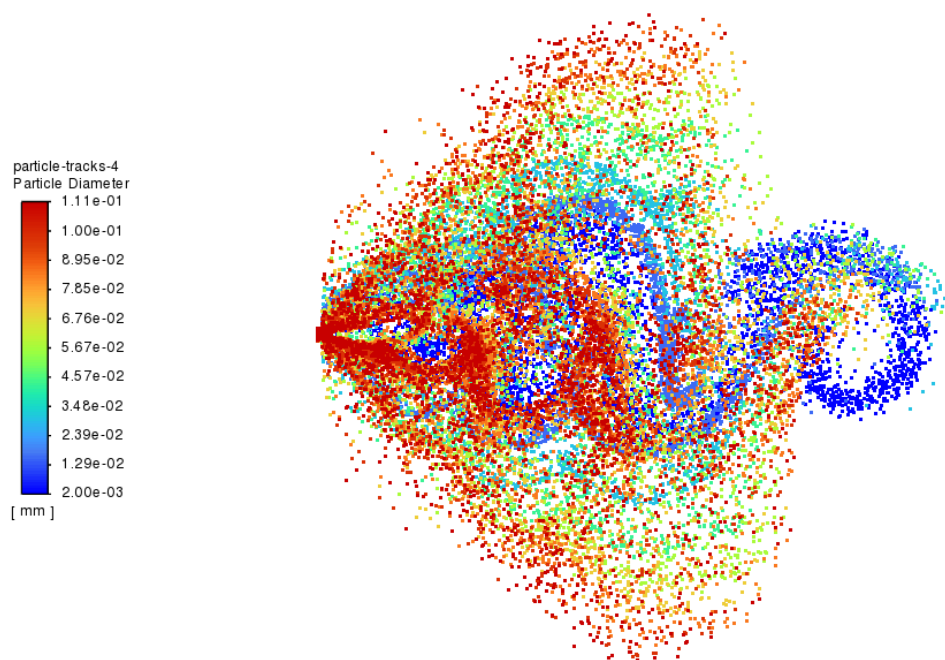


(b) Droplet traces colored by droplet diameter at 0.2 MPa water pressure

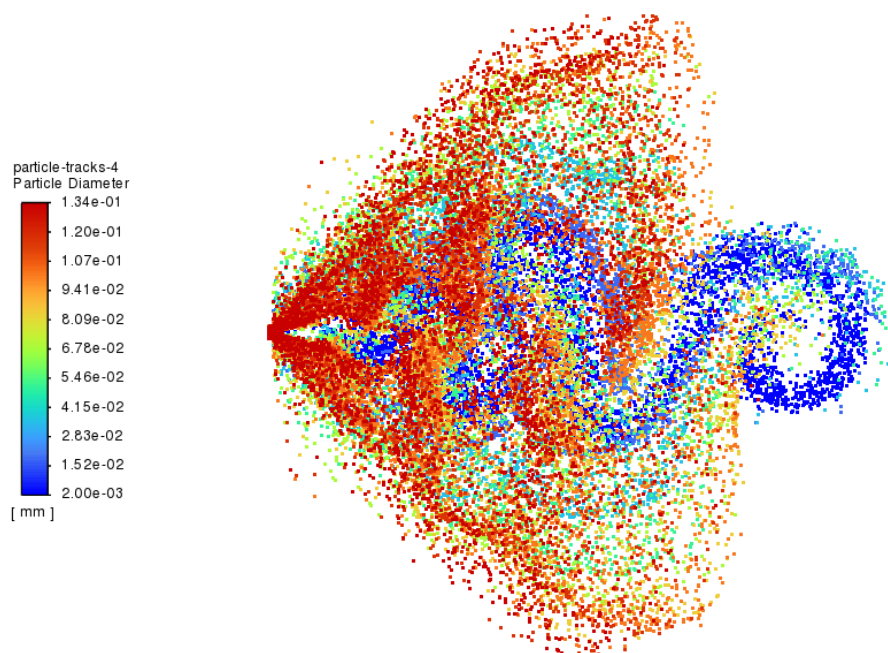


(c) Droplet traces colored by droplet diameter at 0.3 MPa water pressure

Figure 4.10 (continue): Droplet traces colored by droplet diameter at varied water pressures



(d) Droplet traces colored by droplet diameter at 0.4 MPa water pressure



(e) Droplet traces colored by droplet diameter at 0.5 MPa water pressure

Figure 4.10 (continue): Droplet traces colored by droplet diameter at varied water

From Figure 4.10, it can be observed that droplet traces colored by droplet diameter exhibit similar shapes under varied pressures. Droplets in the middle are smaller, while those on the sides are larger. When the liquid passes through the nozzle, the high-speed airflow shears and impacts the liquid, dispersing it into fine droplets. Due to the direct effect of the airflow, smaller droplets are more likely to form in the central region. In contrast, the force exerted on the liquid in the peripheral region is relatively weaker, forming larger droplets. Additionally, the airflow's aerodynamic effects influence the droplets' trajectory. The strong impact of the airflow in the central region makes it easier for smaller droplets to remain stable and disperse. The lower airflow velocity in the peripheral region causes larger droplets to drift outward due to greater inertia.

Because there is a vibrator in the front, fewer droplets form in the center. As the water pressure increases, the droplets' inertia becomes greater, reducing the influence of the gas, allowing them to maintain their forward motion, and causing the particles to concentrate toward the center. Additionally, as the water pressure increases, the water flow rate also rises, leading to more droplets. So Figure 4.10 (a) shows that no droplets form in the central area just after the outlet, and the central area is wider at 0.1 MPa compared to higher water pressures.

As water pressure increases, droplet diameters also increase. Increasing water pressure means more liquid is sprayed out, which increases the collision and impact between liquid droplets. This collision leads to the formation of larger droplets. On the other hand, increasing water pressure reduces the atomization angle, shrinking the space for droplet distribution, thereby increasing the probability of collision and impact and enlarging the

droplet diameter. In Fluent results reports, statistical droplet size distribution can be displayed. The Sauter mean diameter (SMD) under varied pressures is shown in Figure 4.11.

(a)	Overall surface diameter	(D_21): 1.213387e-05	(m)
	Overall volume diameter	(D_31): 1.357754e-05	(m)
	Overall Sauter diameter	(D_32): 1.519298e-05	(m)
	Overall De Brouckere diameter	(D_43): 1.773705e-05	(m)
(b)	Overall surface diameter	(D_21): 1.172466e-05	(m)
	Overall volume diameter	(D_31): 1.556002e-05	(m)
	Overall Sauter diameter	(D_32): 2.064999e-05	(m)
	Overall De Brouckere diameter	(D_43): 2.495938e-05	(m)
(c)	Overall surface diameter	(D_21): 9.429700e-06	(m)
	Overall volume diameter	(D_31): 1.439233e-05	(m)
	Overall Sauter diameter	(D_32): 2.196668e-05	(m)
	Overall De Brouckere diameter	(D_43): 2.677897e-05	(m)
(d)	Overall surface diameter	(D_21): 1.156934e-05	(m)
	Overall volume diameter	(D_31): 1.762769e-05	(m)
	Overall Sauter diameter	(D_32): 2.685853e-05	(m)
	Overall De Brouckere diameter	(D_43): 3.447520e-05	(m)
(e)	Overall surface diameter	(D_21): 1.223134e-05	(m)
	Overall volume diameter	(D_31): 2.073445e-05	(m)
	Overall Sauter diameter	(D_32): 3.514886e-05	(m)
	Overall De Brouckere diameter	(D_43): 4.498832e-05	(m)

Figure 4.11: Statistical droplet size distribution at varied water pressures

Figure 4.11 illustrates the statistical droplet size distribution under varied water pressures. In practical theory and application, the Sauter mean diameter (SMD) is commonly used to represent droplet sizes. Comparing at an air pressure of 0.5 MPa, the Sauter mean diameters under varied water pressures (0.1/0.2/0.3/0.4/0.5 MPa) vary. The smallest SMD is observed at a water pressure of 0.1 MPa. The specific values are shown in Table 4.3.

Table 4.3: Sauter mean diameter (SMD) under varied water pressures

Air pressure is 0.5MPa. Water pressure (MPa)	D(3,2) (Sauter mean diameter) (m)
0.1	1.519298e-5
0.2	2.064999e-5
0.3	2.196668e-5
0.4	2.685863e-5
0.5	3.514886e-5

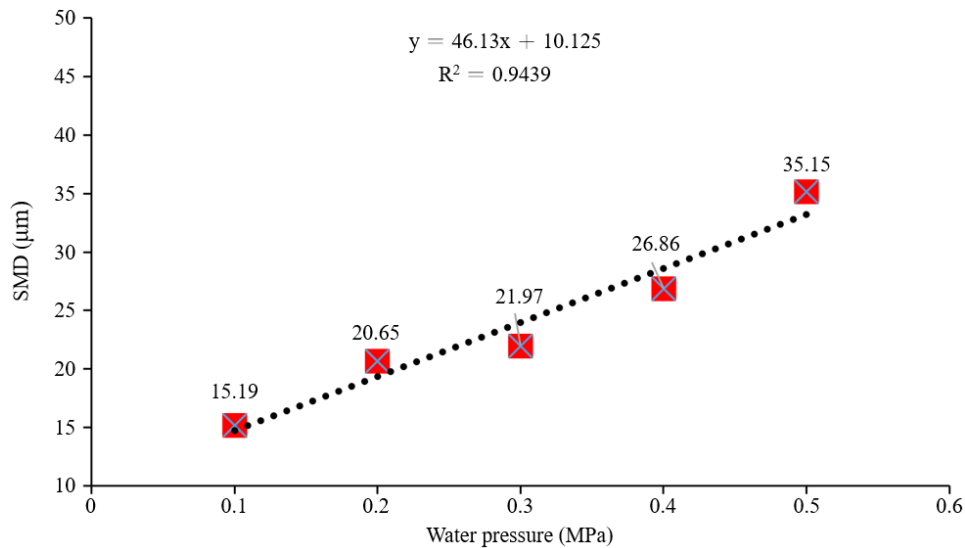


Figure 4.12: Scatter plot of Sauter mean diameter (SMD) under varied water pressures

The scatter plot shown in Figure 4.12 with water pressure on the x-axis and Sauter mean diameter (SMD) on the y-axis illustrates an increase in Sauter mean diameter (SMD) as water pressure varies from 0.1 MPa to 0.5 MPa. At a water pressure of 0.1 MPa, the Sauter mean diameter (SMD) is at its minimum, measuring 15.19 μm , whereas at 0.5 MPa, it increases to 35.15 μm . Therefore, the optimal Sauter mean diameter (SMD) is observed at 0.1 MPa. The fitting curve for the relationship between Sauter mean diameter (SMD) and water pressure is given by $y = 46.13x + 10.125$, $R^2 = 0.9439$, showing a good fit.

4.3.4 Mach Number Analysis

As shown in Figure 4.13, when the water pressure is 0.1 MPa, the Mach number is 1.98. Many points in the outflow field with relatively high Mach numbers indicate the presence of numerous vortices, which are conducive to atomization.

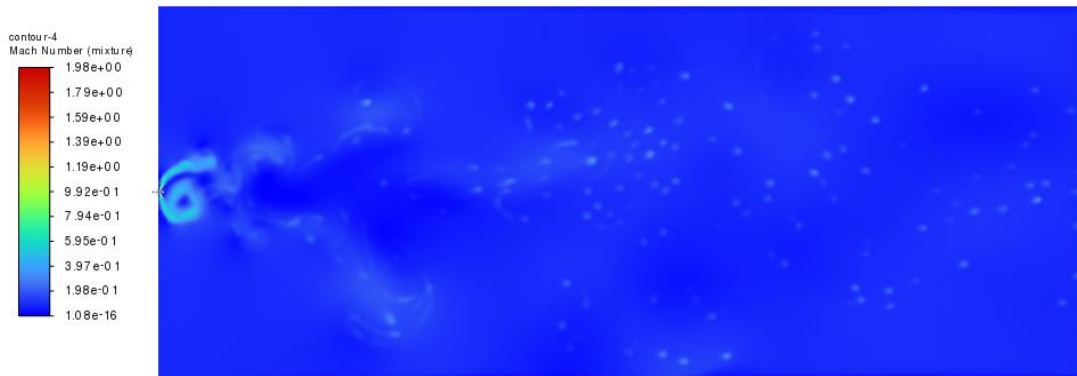


Figure 4.13: Contours of Mach number at 0.1 MPa water pressure

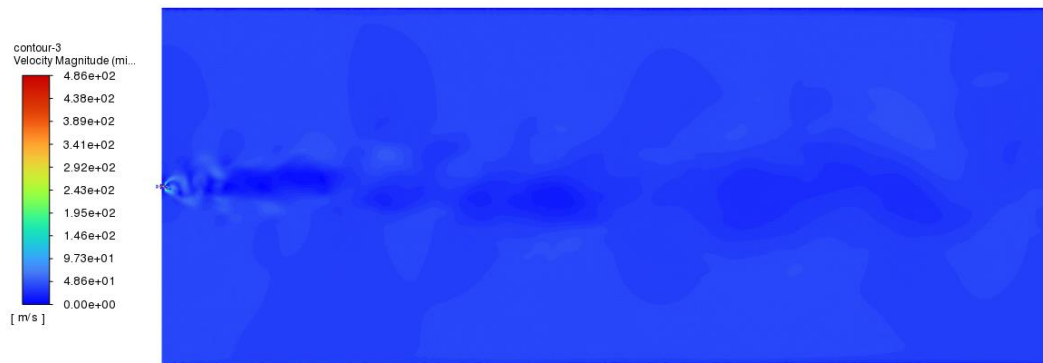
In summary, at an air pressure of 0.5 MPa, spray velocities are optimal at both 0.1 MPa and 0.5 MPa water pressures. At 0.1 MPa water pressure, the atomization cone angle is the largest, and the droplet size is the smallest. The Mach number plot indicates that at 0.1

MPa water pressure, the presence of vortices in the spray field aids atomization. Therefore, at an air pressure of 0.5 MPa, a water pressure of 0.1 MPa is optimal.

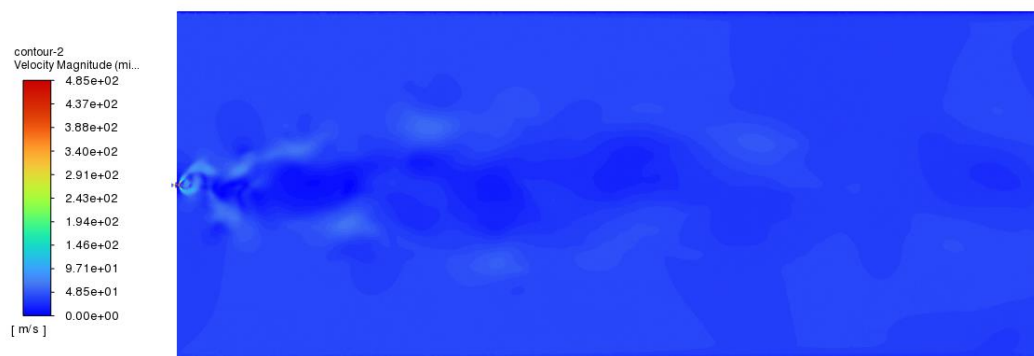
4.4 Results of Numerical Analysis with Varied Air Pressure

With a nozzle outlet diameter of 2.5 mm and an oscillator distance of 4 mm, this research analyzed the flow field velocity and atomization angle at a water pressure of 0.1 MPa across various air pressures (0.2, 0.3, 0.4, 0.5, and 0.6 MPa).

4.4.1 Velocity Analysis



(a) Contours of velocity magnitude at 0.2 MPa air pressure

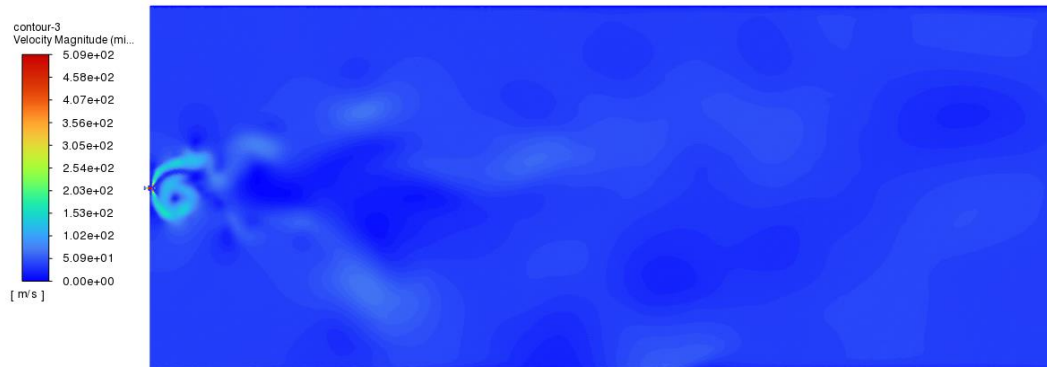


(b) Contours of velocity magnitude at 0.3 MPa air pressure

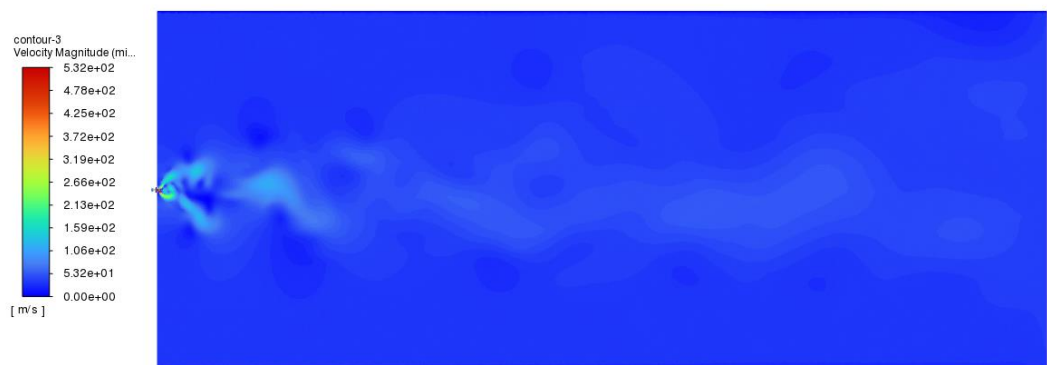
Figure 4.14: Contours of velocity magnitude at varied air pressures



(c) Contours of velocity magnitude at 0.4 MPa air pressure



(d) Contours of velocity magnitude at 0.5 MPa air pressure



(e) Contours of velocity magnitude at 0.6 MPa air pressure

Figure 4.14 (continue): Contours of velocity magnitude at varied air pressures

From Figure 4.14, the maximum velocities in the flow field under varied air pressures are listed in Table 4.4.

Table 4.4: The maximum velocity of the mixture under varied air pressures

Water pressure is 0.1MPa. Air pressure (MPa)	Maximum velocity of mixture (m/s)
0.2	486
0.3	485
0.4	497
0.5	509
0.6	532

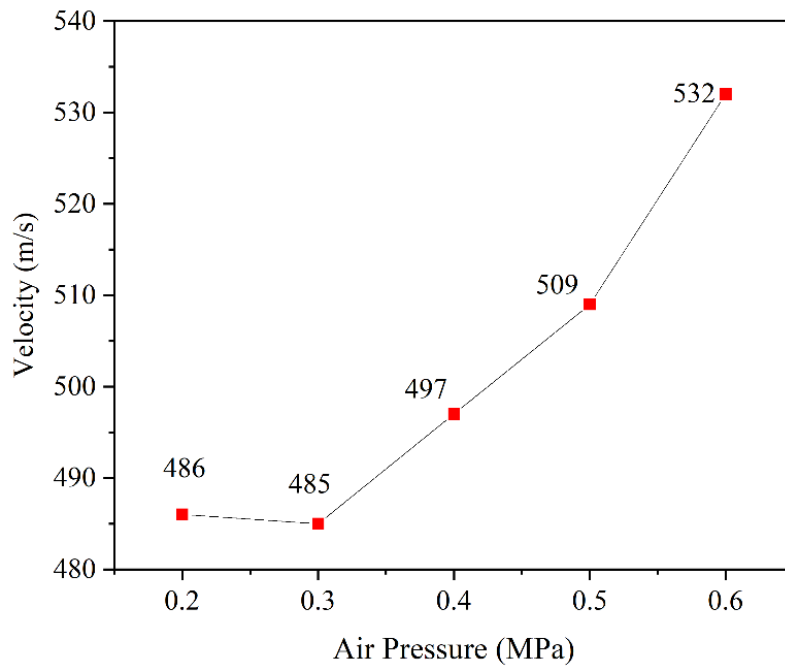
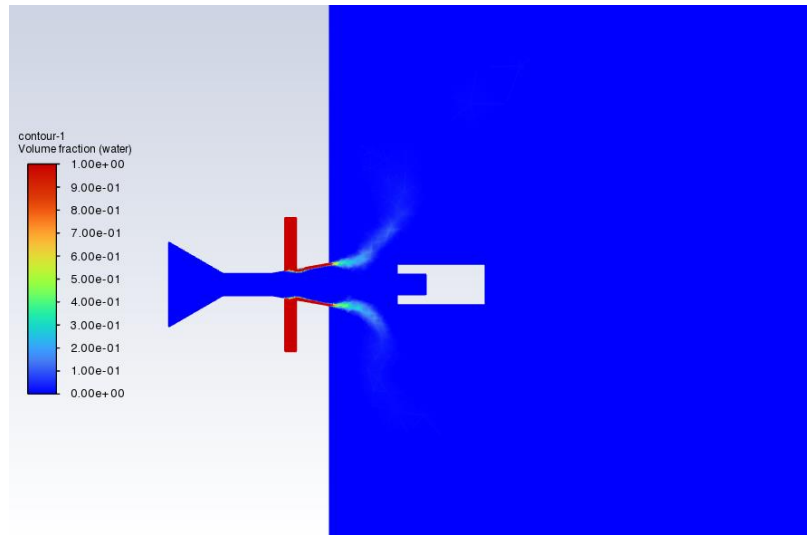


Figure 4.15: Scatter plot of the maximum velocity of the mixture under varied air pressures

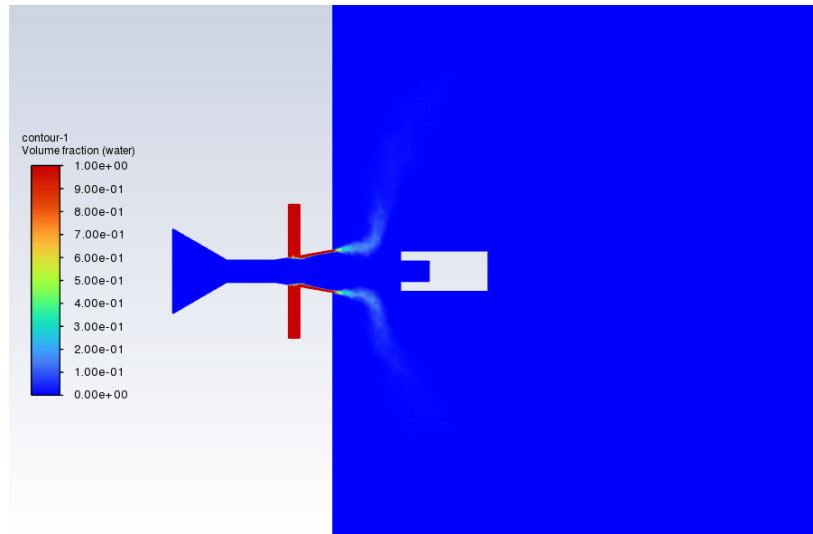
The scatter plot shown in Figure 4.15 with air pressure on the x-axis and maximum velocity of the mixture on the y-axis illustrates an increasing trend of the maximum velocity of the mixture as air pressure varies from 0.2 MPa to 0.6 MPa. Between 0.2 MPa and 0.3 MPa, the velocity stabilizes. At an air pressure of 0.3 MPa, the maximum velocity of the mixture is at its minimum, measuring 485 m/s, with a gradual increase observed beyond 0.3 MPa. The velocity reaches its maximum value of 530 m/s at 0.6 MPa. When the air pressure increases, the velocity of the air inside the nozzle also increases, and the air can provide greater kinetic energy when mixing with the liquid. So, the speed of the air-liquid mixture also increases accordingly.

4.4.2 Atomization Angle Analysis

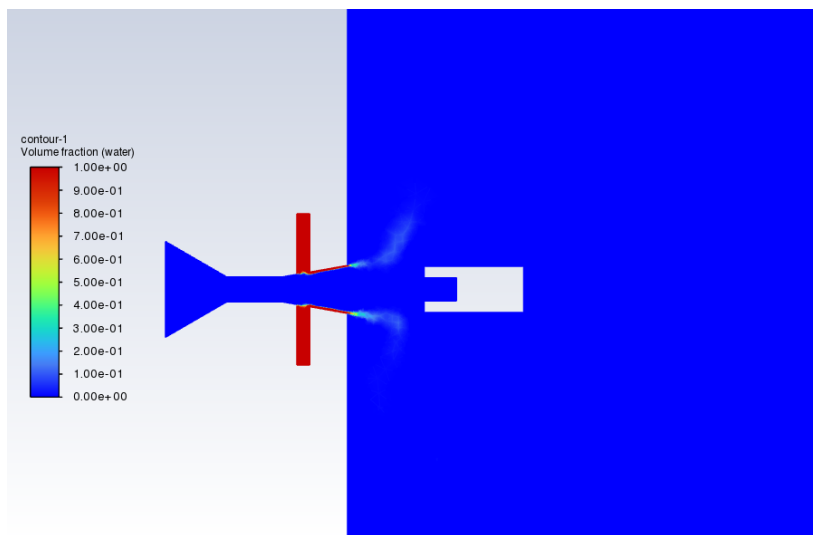


(a) Contours of water volume fraction at 0.2 MPa air pressure

Figure 4.16: Contours of water volume fraction at varied air pressures

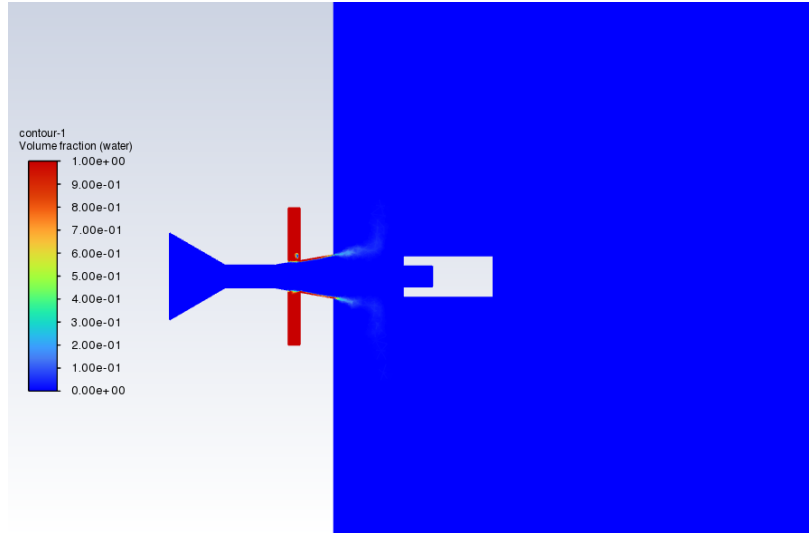


(b) Contours of water volume fraction at 0.3 MPa air pressure

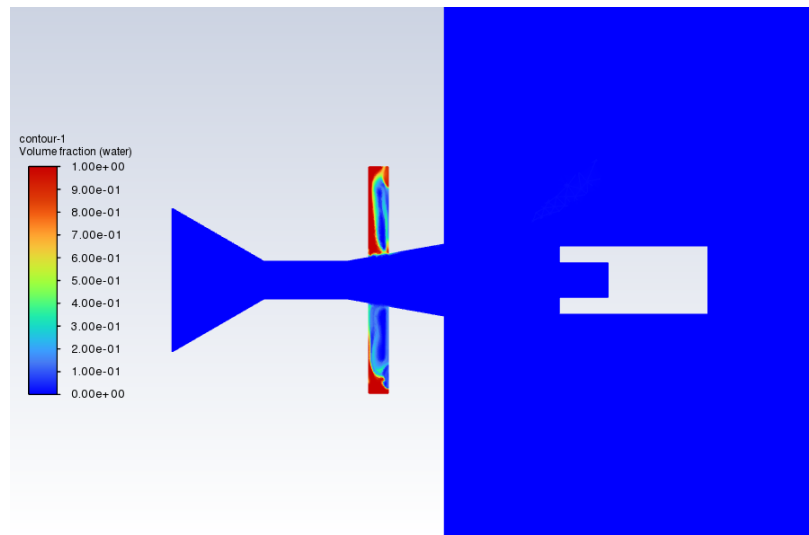


(c) Contours of water volume fraction at 0.4 MPa air pressure

Figure 4.16 (continue): Contours of water volume fraction at varied air pressures



(d) Contours of water volume fraction at 0.5 MPa air pressure

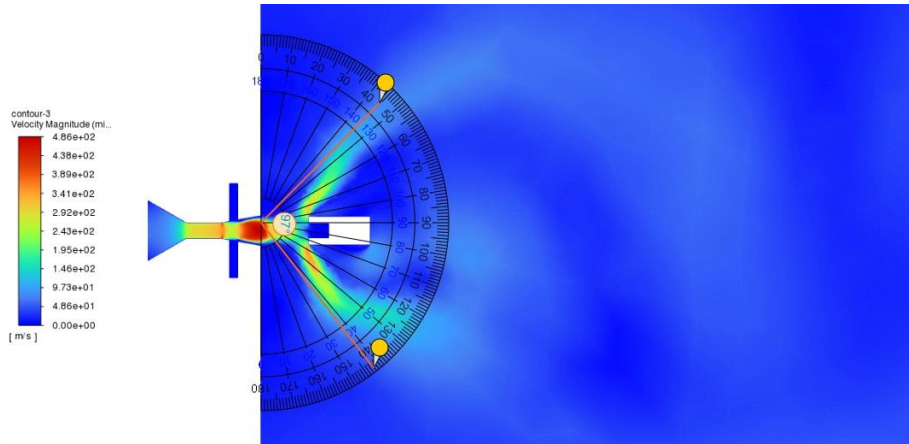


(e) Contours of water volume fraction at 0.6 MPa air pressure

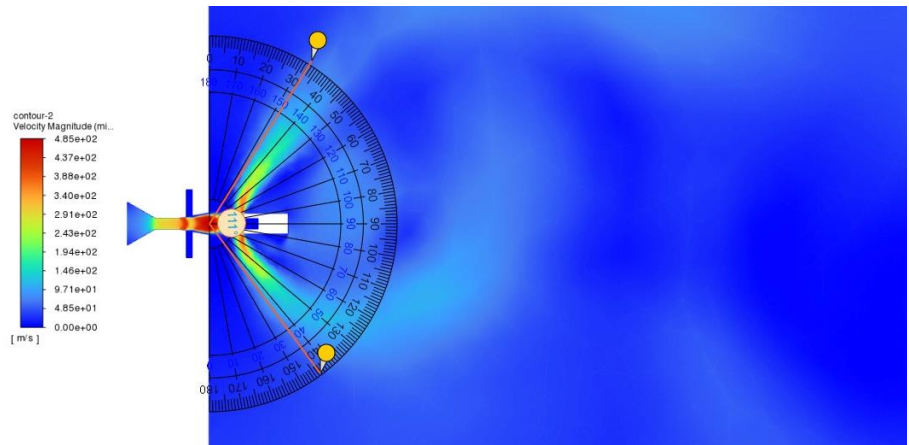
Figure 4.16 (continue): Contours of water volume fraction at varied air pressures

Figure 4.16 depicts the contours of water volume fraction at varied air pressures. It can be observed from Figure 4.16 that as air pressure increases, the water volume fraction decreases. When the water pressure is 0.1 MPa and the air pressure reaches 0.6 MPa, water is difficult to enter the nozzle interior. Therefore, excessively high air pressure should be

avoided. The water shape is irregular, making the measurement challenging and prone to significant error. Therefore, in the study, the angle measuring the velocity of the air-water mixture was selected, which is easier to measure and less subjective.

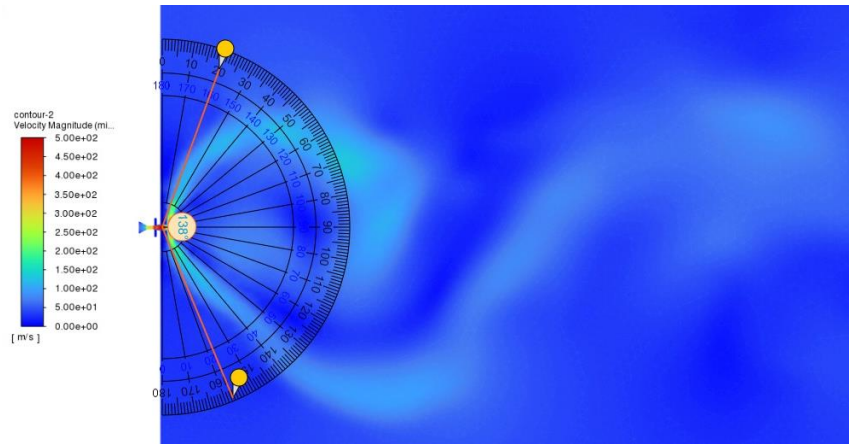


(a) Angle of atomization range at 0.2 MPa air pressure

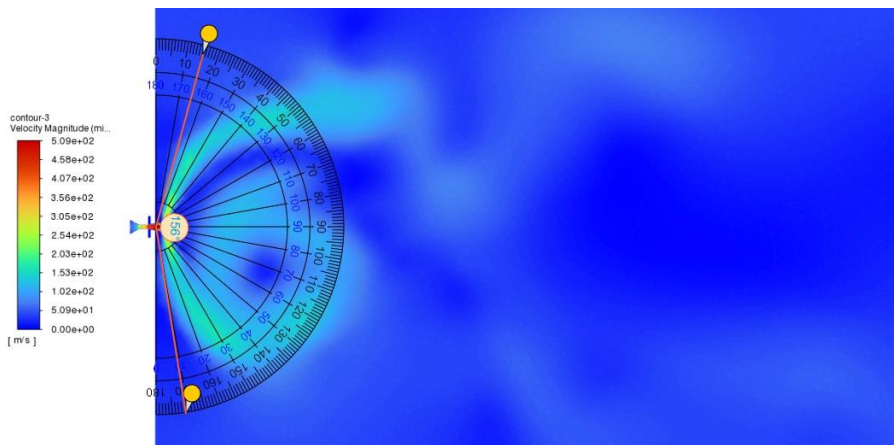


(b) Angle of atomization range at 0.3 MPa air pressure

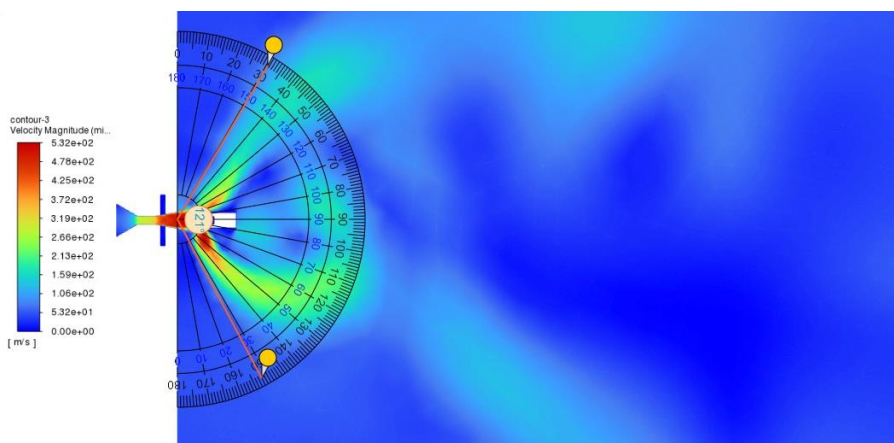
Figure 4.17: Angle of atomization range at varied air pressures



(c) Angle of atomization range at 0.4 MPa air pressure



(d) Angle of atomization range at 0.5 MPa air pressure



(e) Angle of atomization range at 0.6 MPa air pressure

Figure 4.17 (continue): Angle of atomization range at varied air pressures

As seen in Figure 4.17, the change in air pressure significantly influences the angle of the atomization range. The angle of the atomization range increases and then decreases. This trend is due to the increase in air pressure, which increases the momentum of the mixture, leading to increased collision with the oscillator and an increase in the atomization angle. However, when the momentum reaches a certain level, inertial forces dominate, causing the droplets to tend to move forward, thereby reducing the atomization angle. The values are shown in Table 4.5.

Table 4.5: Angle of atomization range under varied air pressures

The water pressure is 0.1MPa. Air pressure (MPa)	The angle of atomization range (°)
0.2	97
0.3	111
0.4	138
0.5	156
0.6	121

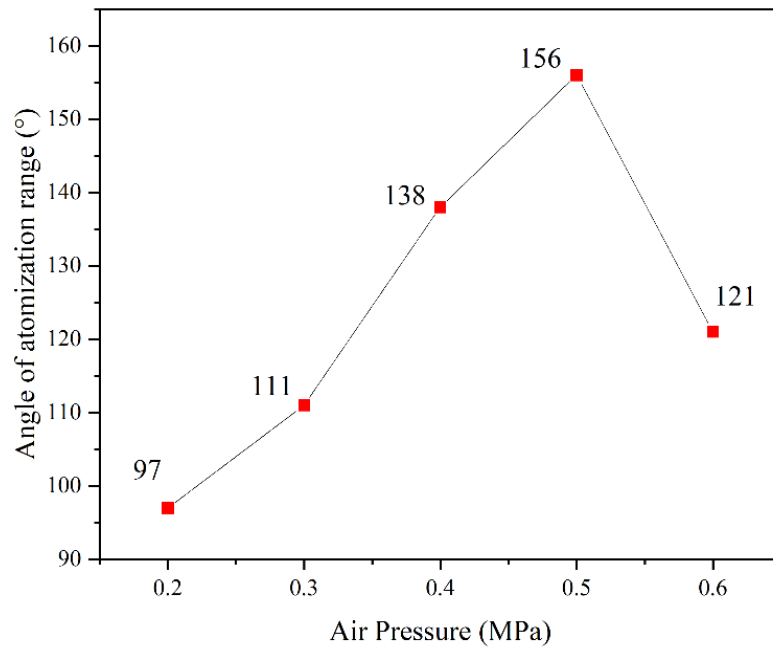


Figure 4.18: Scatter plot of the angle of atomization range under varied air pressures

The scatter plot shown in Figure 4.18 with air pressure on the x-axis and angle of atomization range on the y-axis illustrates the trend of the angle of atomization range as air pressure varies from 0.2 MPa to 0.6 MPa. Figure 4.18 shows that the angle of the atomization range increases as the pressure increases with lower air pressure; at an air pressure of 0.5 MPa, the angle of the atomization range is the largest, reaching 156°. Subsequently, as the air pressure increases, the angle of atomization range rapidly decreases. At 0.6 MPa, it drops to 121°. Therefore, the best angle of atomization range is achieved at 0.5 MPa air pressure.

In summary, at air pressures of 0.6 MPa, water has difficulty entering the nozzle interior. Comparing the remaining air pressures of 0.2 MPa, 0.3 MPa, 0.4 MPa, and 0.5 MPa, it is observed that at 0.4 MPa and 0.5 MPa, the velocity is higher. Additionally, at a water pressure of 0.5 MPa, the angle of atomization range is noticeably larger. Therefore, after optimization, the selected air pressure is 0.5 MPa.

4.5 Results of Numerical Analysis with Varied Nozzle Diameters

When the water pressure is 0.1 MPa, air pressure is 0.5 MPa, and the oscillator distance is 0.4 mm, the research focuses on the flow field velocity and atomization angle for varying nozzle outlet diameters.

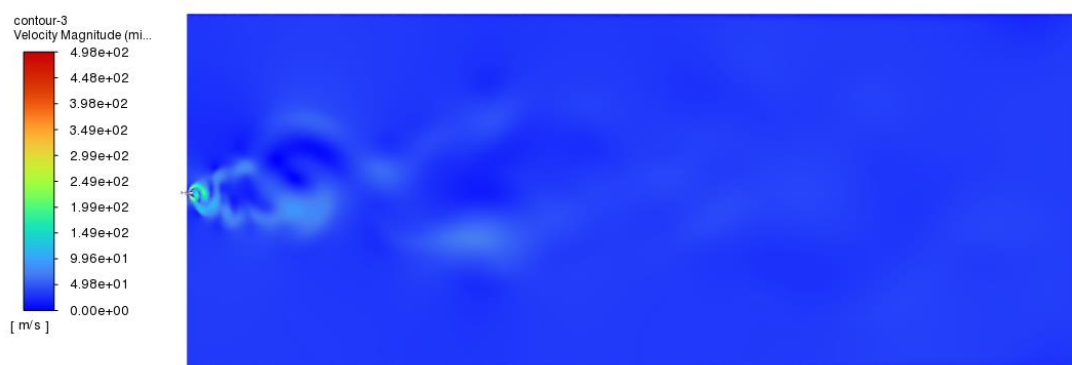
4.5.1 Velocity Analysis

Figure 4.19 shows the velocity contour of the nozzle's internal and external flow fields at a water pressure of 0.1 MPa, air pressure of 0.5 MPa, and a vibrator distance of 4 mm, with varying nozzle outlet diameters. It can be seen from the figure that the nozzle outlet diameter affects the velocity magnitude.

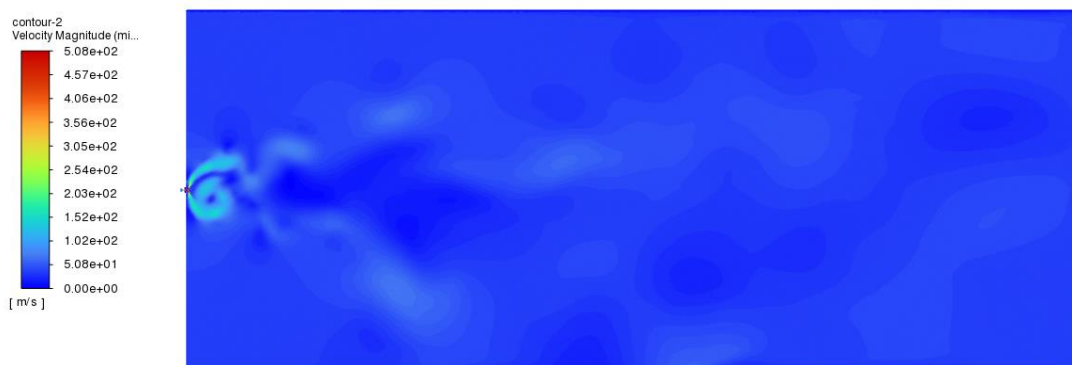


(a) Contours of velocity magnitude with a nozzle outlet diameter of 1.5 mm

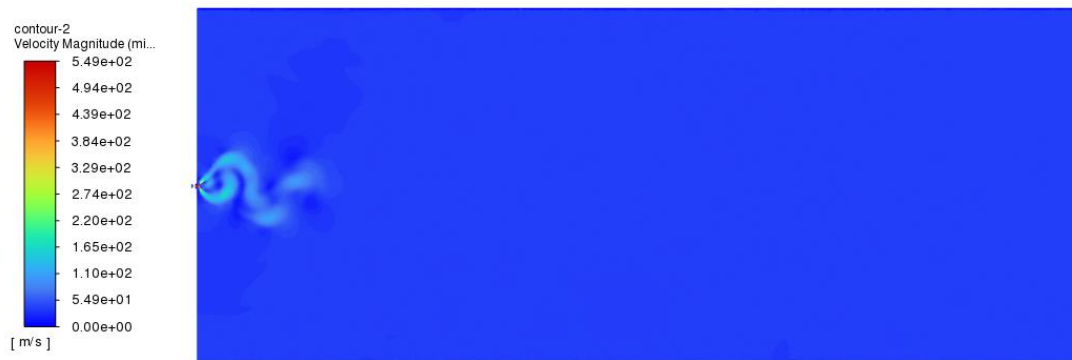
Figure 4.19: Contours of velocity magnitude with varied nozzle outlet diameters



(b) Contours of velocity magnitude with a nozzle outlet diameter of 2.0 mm



(c) Contours of velocity magnitude with a nozzle outlet diameter of 2.5 mm



(d) Contours of velocity magnitude with a nozzle outlet diameter of 3.0 mm

Figure 4.19 (continue): Contours of velocity magnitude with varied nozzle outlet diameters



(e) Contours of velocity magnitude with a nozzle outlet diameter of 3.5 mm

Figure 4.19 (continue): Contours of velocity magnitude with varied nozzle outlet diameters

Table 4.6: Maximum velocity magnitude of mixture with varied nozzle outlet diameters

Nozzle outlet diameter (mm)	Maximum velocity magnitude of mixture (m/s)
1.5	526
2.0	498
2.5	508
3.0	549
3.5	555

As depicted in Figure 4.19, the nozzle outlet's diameter significantly affects the flow field's morphology and velocity magnitude. Regions with higher velocities are concentrated inside the nozzle and near its outlet. A distinct vortex at the nozzle outlet indicates significant turbulence at this location. With a nozzle outlet diameter of 1.5 mm, the jet is more concentrated near the axis, with less apparent vortices in the external flow field. In contrast,

with a diameter of 2.5 mm, there are more vortices in the external flow field. The maximum velocities of the mixture for varied nozzle outlet diameters are listed in Table 4.6.

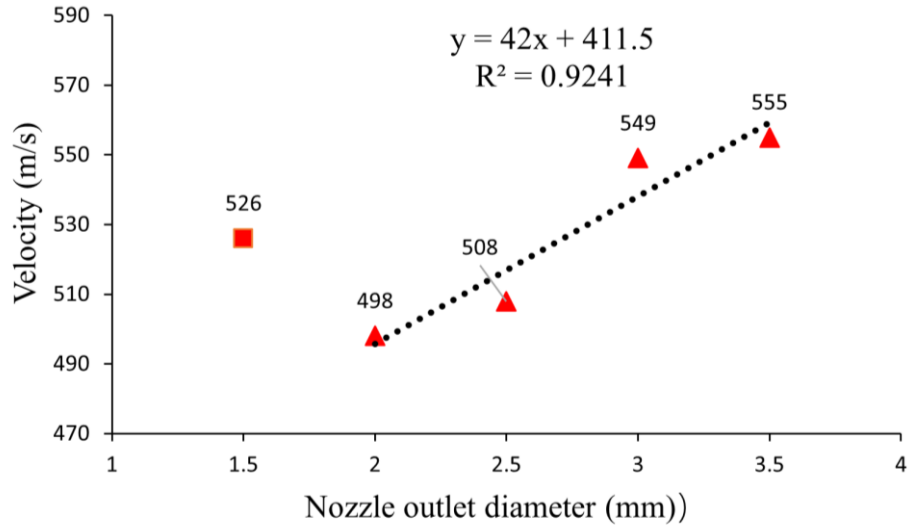
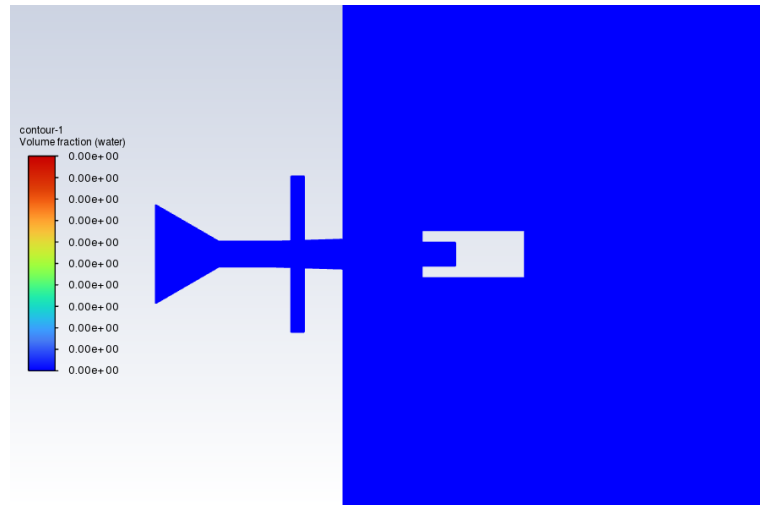


Figure 4.20: Scatter plot of maximum velocity magnitude of mixture with varied nozzle outlet diameters

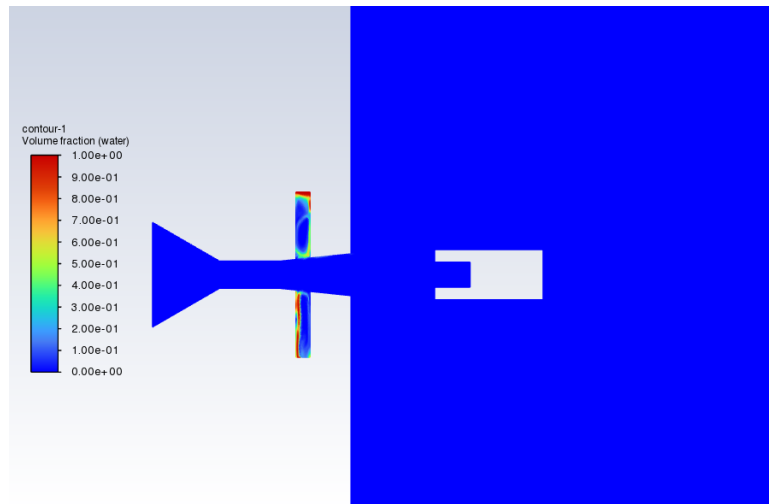
The scatter plot shown in Figure 4.20 with the nozzle outlet diameter on the x-axis and the maximum velocity magnitude of the mixture on the y-axis illustrates a decrease and an increasing trend of maximum velocity magnitude as the nozzle outlet diameter varies from 1.5 mm to 3.5 mm. Figure 4.20 shows that as the nozzle outlet diameter increases from 1.5 mm to 2.0 mm, the maximum velocity magnitude of the mixture decreases. Conversely, as the nozzle outlet diameter further increases from 2.0 mm to 3.5 mm, the maximum velocity magnitude of the mixture increases. The curve fitting the relationship between the maximum velocity magnitude of the mixture and nozzle outlet diameter (2.0 mm to 3.5 mm) is represented by the equation $y = 42x + 411.5$.

4.5.2 Atomization Angle Analysis

Figure 4.21 shows the volume fraction contour of water at a water pressure of 0.1 MPa, air pressure of 0.5 MPa, and a vibrator distance of 4 mm, with varying nozzle outlet diameters. It can be seen from the figure that the nozzle outlet diameter affects the water volume fraction.

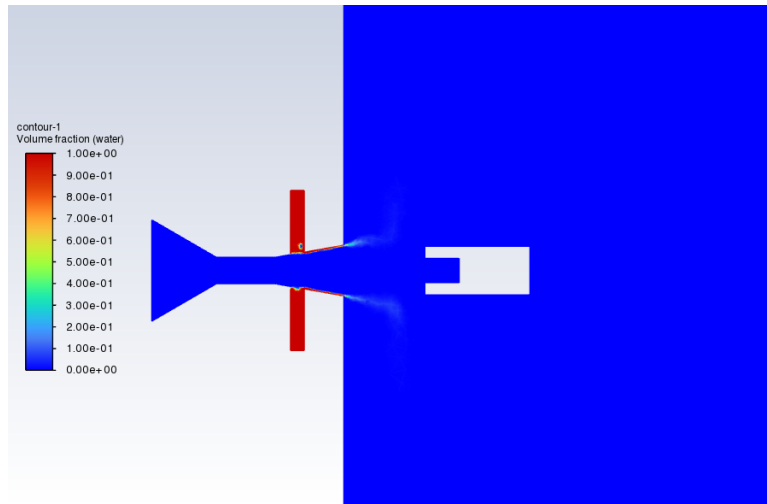


(a) Contours of water volume fraction with a nozzle outlet diameter of 1.5 mm

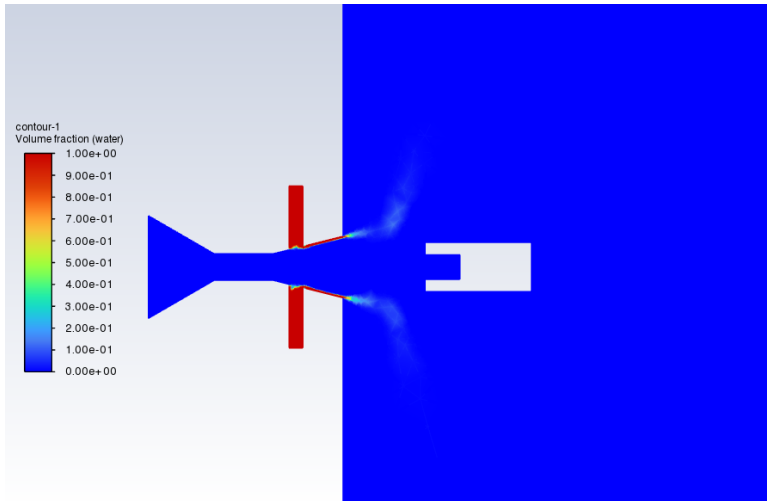


(b) Contours of water volume fraction with a nozzle outlet diameter of 2 mm

Figure 4.21: Contours of water volume fraction with varied nozzle outlet diameters

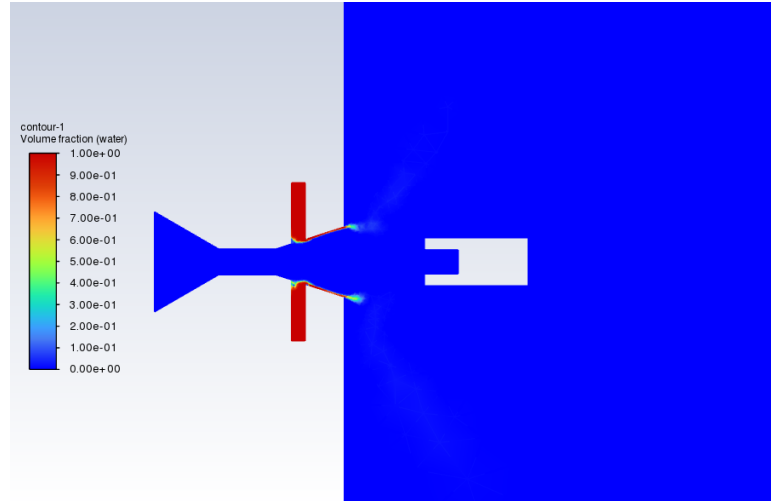


(c) Contours of water volume fraction with a nozzle outlet diameter of 2.5 mm



(d) Contours of water volume fraction with a nozzle outlet diameter of 3 mm

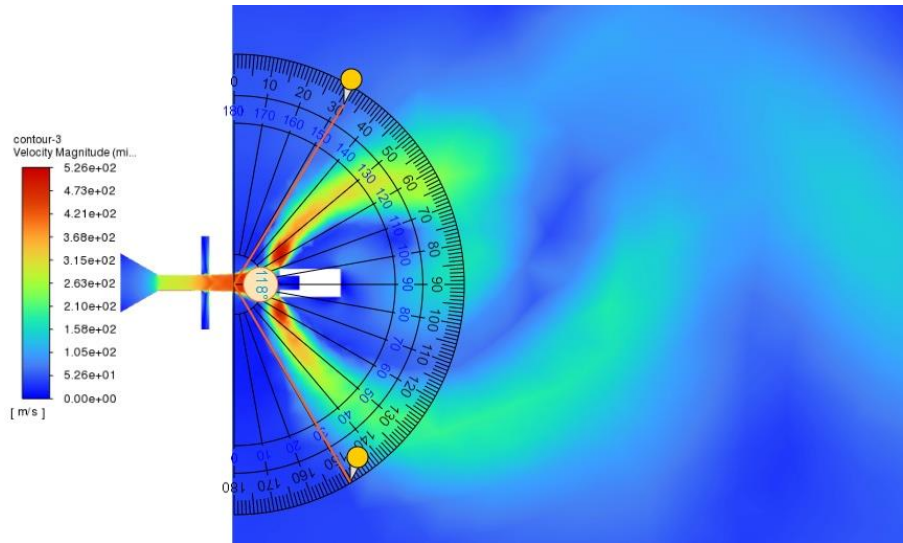
Figure 4.21 (continue): Contours of water volume fraction with varied nozzle outlet diameters



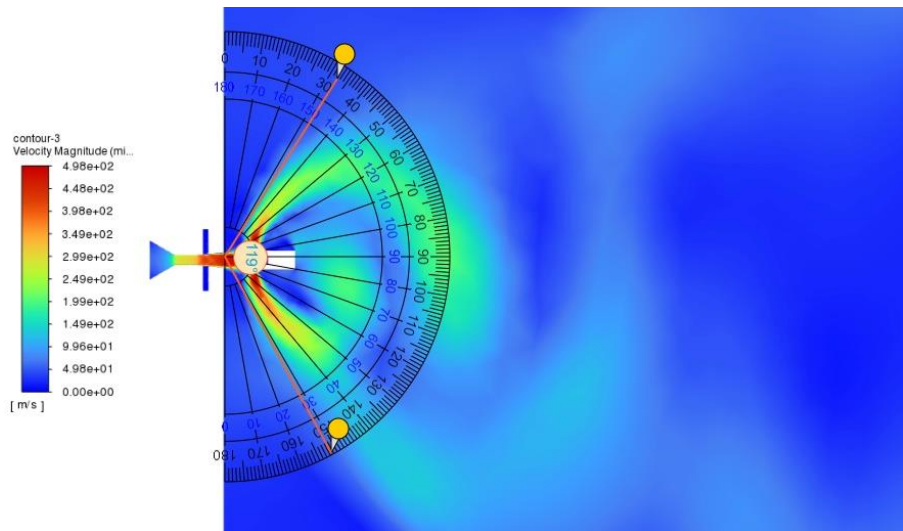
(e) Contours of water volume fraction with a nozzle outlet diameter of 3.5 mm

Figure 4.21 (continue): Contours of water volume fraction with varied nozzle outlet diameters

From Figure 4.21, it can be observed that the nozzle outlet diameter has a significant impact on the volume fraction of water. When the nozzle outlet diameter is 1.5 mm, no liquid remains inside the nozzle; the water is expelled by the air and cannot enter. With a nozzle outlet diameter of 2 mm, it is difficult for liquid to enter the nozzle, and only a very small amount of water can enter the interior of the nozzle. Therefore, at an air pressure of 0.5 MPa and water pressure of 0.1 MPa, nozzles with outlet diameters of 1.5 mm and 2 mm cannot be used. With a nozzle outlet diameter of 2.5 mm, water can flow into the nozzle normally. Due to the complexity of measuring atomization angles from water volume fraction contours, it is more practical to determine the spray range using velocity contours.

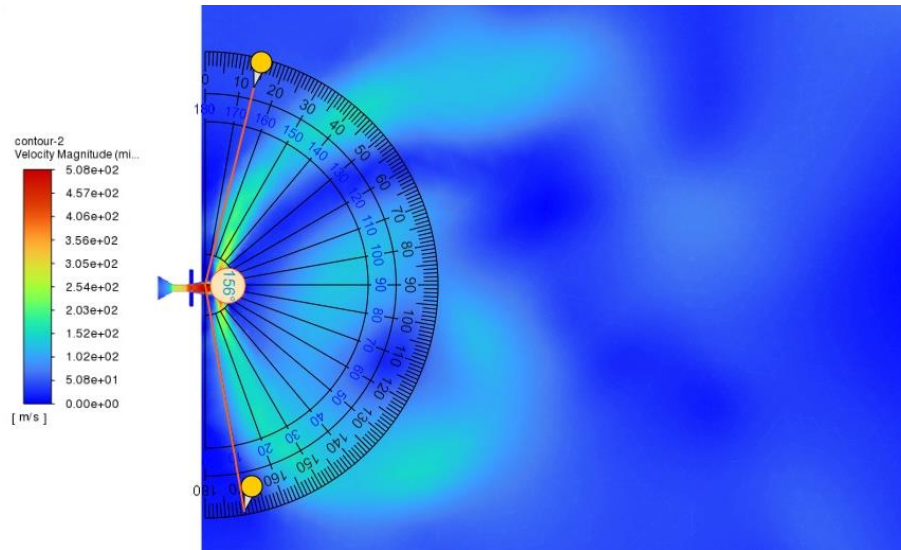


(a) Angle of atomization range with a nozzle outlet diameter of 1.5 mm

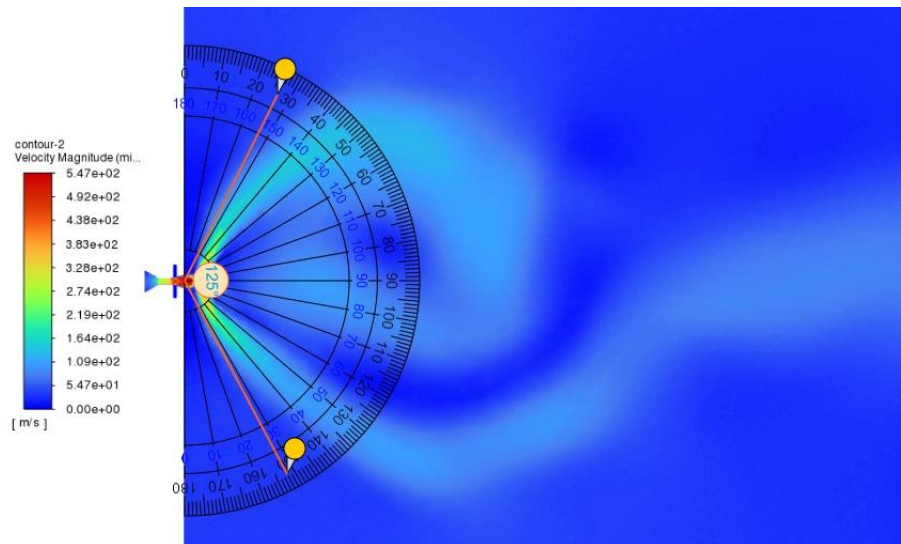


(b) Angle of atomization range with a nozzle outlet diameter of 2 mm

Figure 4.22: Angle of atomization range with varied nozzle outlet diameters

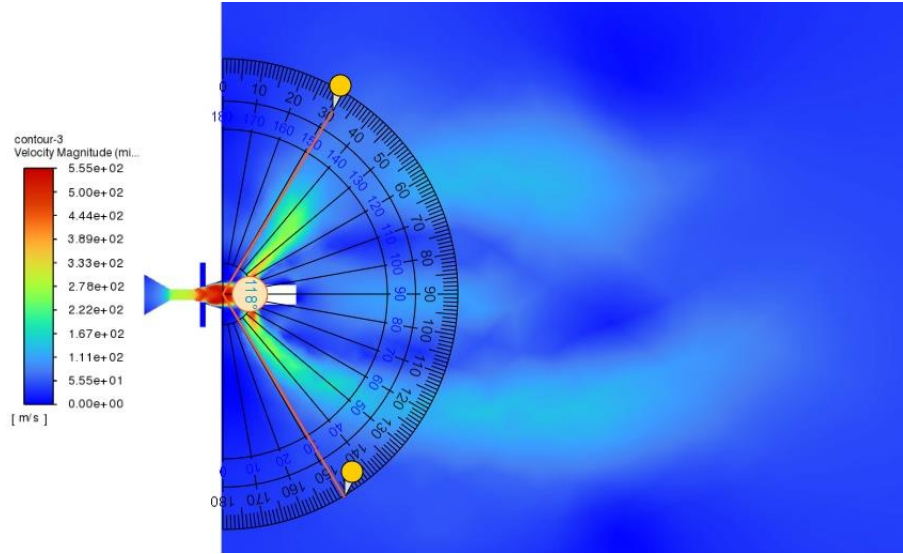


(c) Angle of atomization range with a nozzle outlet diameter of 2.5 mm



(d) Angle of atomization range with a nozzle outlet diameter of 3 mm

Figure 4.22 (continue): Angle of atomization range with varied nozzle outlet diameters



(e) Angle of atomization range with a nozzle outlet diameter of 3.5 mm

Figure 4.22 (continue): Angle of atomization range with varied nozzle outlet diameters

As illustrated in Figure 4.22, it is evident that variations in the nozzle outlet diameter exert a significant influence on the angle of the atomization range. The corresponding values are presented in Table 4.7.

Table 4.7: Angle of atomization range with varied nozzle outlet diameters

Nozzle outlet diameter (mm)	The angle of atomization range (°)
1.5	118
2.0	119
2.5	156
3.0	125
3.5	118

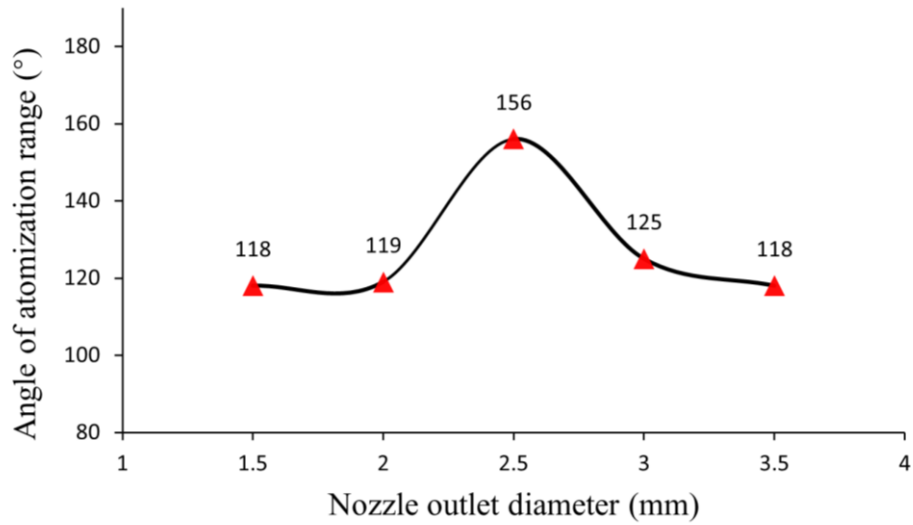


Figure 4.23: Scatter plot of the angle of atomization range with varied nozzle outlet diameters

The scatter plot shows the nozzle outlet diameter on the x-axis and the angle of atomization range on the y-axis; Figure 4.23 shows that the nozzle outlet diameter affects the angle of atomization range. When the nozzle outlet diameters are 1.5 mm, 2 mm, and 3.5 mm, the angles of the atomization range are approximately the same. With a nozzle outlet diameter of 3 mm, it is slightly higher. However, with a nozzle outlet diameter of 2.5 mm, the atomization range angle is significantly higher than the others, reaching 156°.

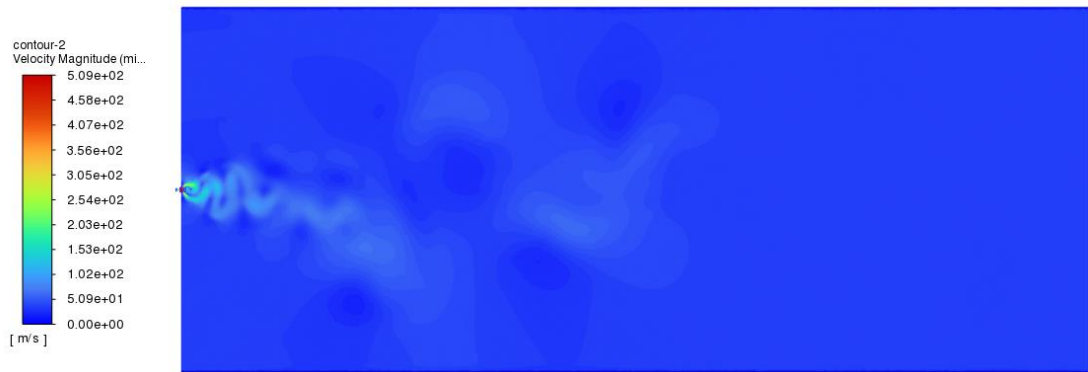
In summary, at an air pressure of 0.5 MPa and water pressure of 0.1 MPa, when the nozzle outlet diameter is 1.5 mm, water cannot enter the nozzle interior. Similarly, with a diameter of 2 mm, water faces difficulty entering, making both these diameters unsuitable. Comparatively, among diameters 3 mm and 3.5 mm, the diameter of 2.5 mm exhibits a notable increase in the angle of atomization range despite a slightly lower velocity.

Additionally, opting for a 2.5 mm diameter allows for the conservation of water flow. Therefore, the nozzle outlet diameter of 2.5 mm is preferred.

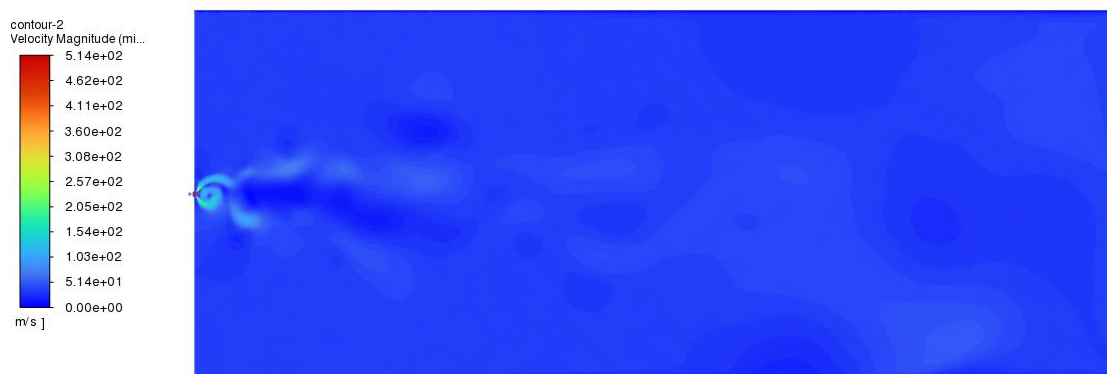
4.6 Results of Numerical Analysis with Varied Oscillator Distances

When the water pressure is 0.1 MPa, air pressure is 0.5 MPa, and the nozzle outlet diameter is 2.5 mm, the research focused on the flow field velocity and atomization angle for varied oscillator distances.

4.6.1 Velocity Analysis

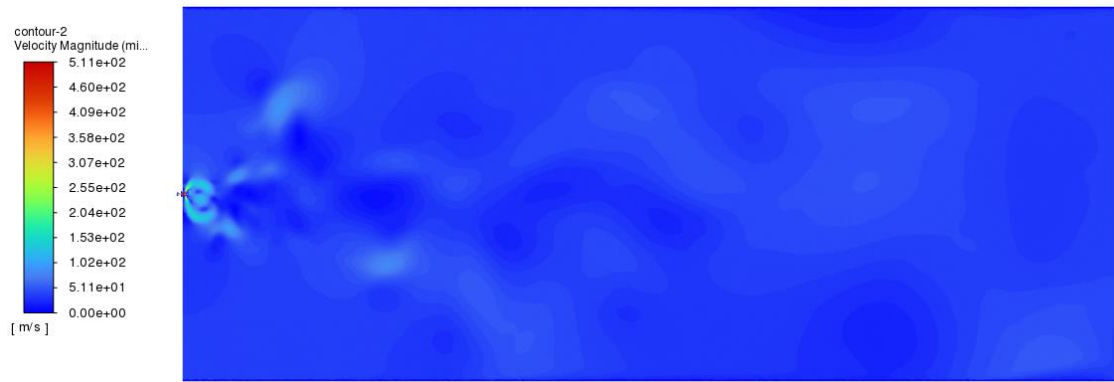


(a) Contours of velocity magnitude with oscillator distance of 2 mm

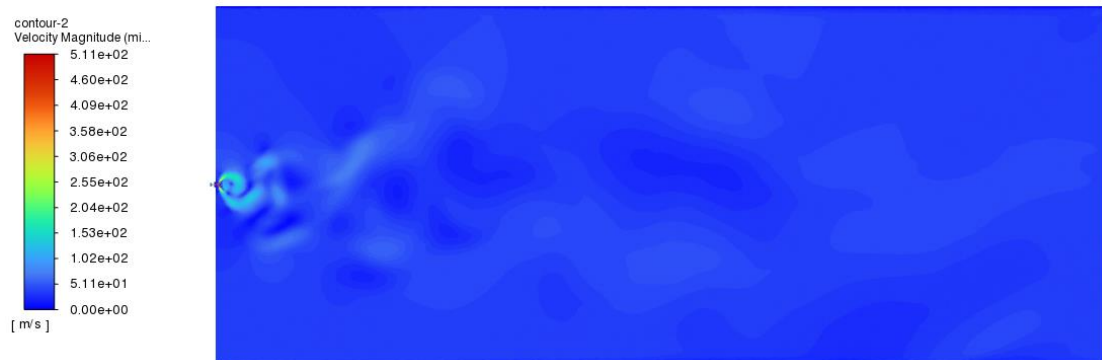


(b) Contours of velocity magnitude with oscillator distance of 3 mm

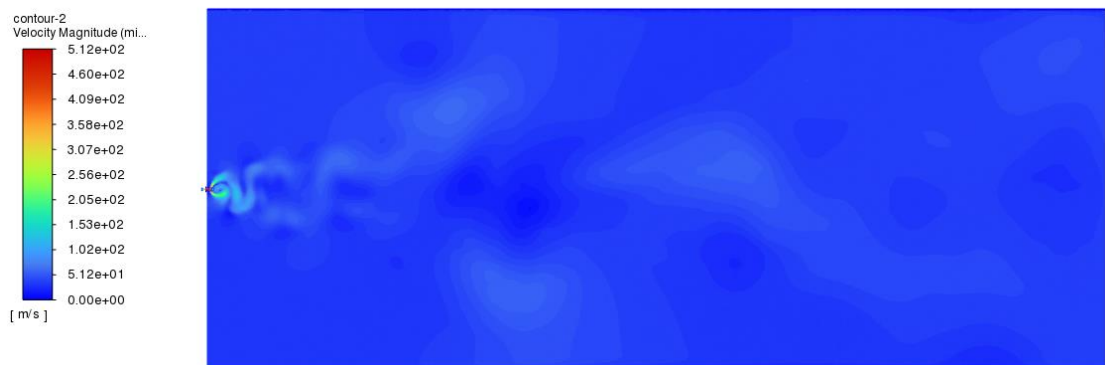
Figure 4.24: Contours of velocity magnitude with varied oscillator distances



(c) Contours of velocity magnitude with oscillator distance of 4 mm



(d) Contours of velocity magnitude with oscillator distance of 5 mm



(e) Contours of velocity magnitude with oscillator distance of 6 mm

Figure 4.24 (continue): Contours of velocity magnitude with varied oscillator distances

Based on Figure 4.24, it is evident that varying oscillator distances affect the morphology of the flow field with minimal impact on velocity magnitude. Maximum velocity magnitude for varied vibration distances is provided in Table 4.8.

Table 4.8: Maximum velocity magnitude of mixture with varied oscillator distances

Oscillator Distance (mm)	Maximum velocity magnitude of mixture (m/s)
2	509
3	514
4	511
5	511
6	512

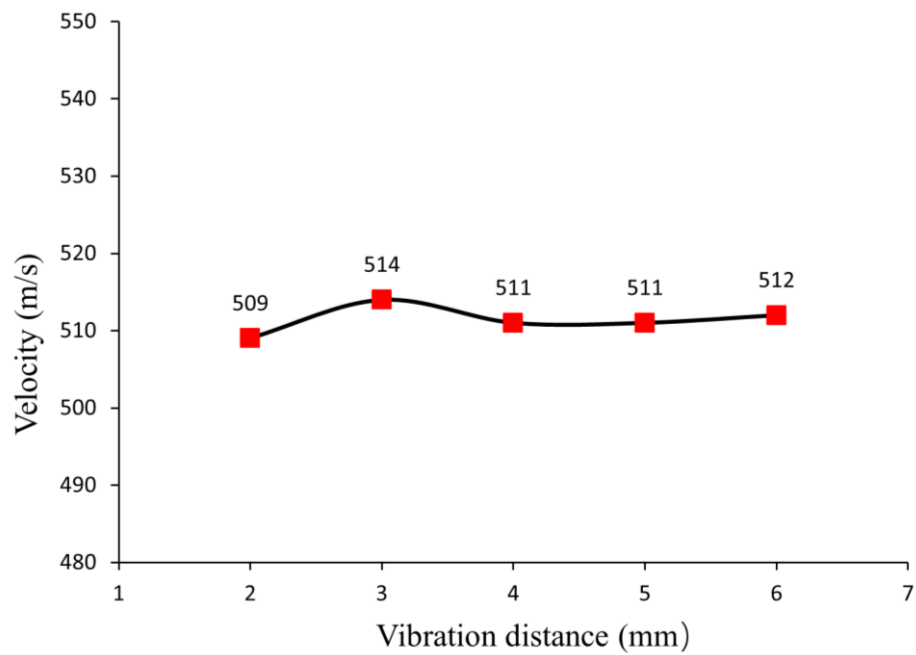
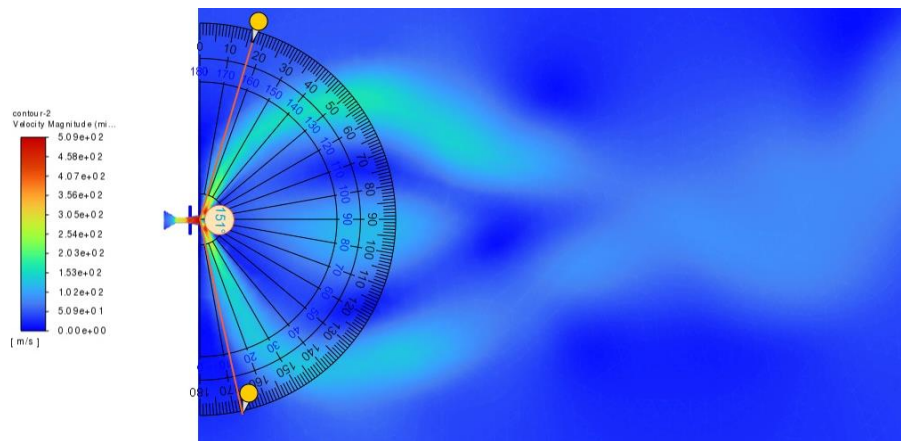


Figure 4.25: Scatter plot of maximum velocity magnitude of mixture with varied oscillator distances

The scatter plot is shown in Figure 4.25 with oscillator distance on the x-axis and maximum velocity magnitude of the mixture on the y-axis. From Figure 4.25, it can be observed that the variation in oscillator distance has minimal impact on the maximum velocity of the flow field. This could be attributed to the oscillator being located outside the nozzle, thus exerting less influence on velocity.

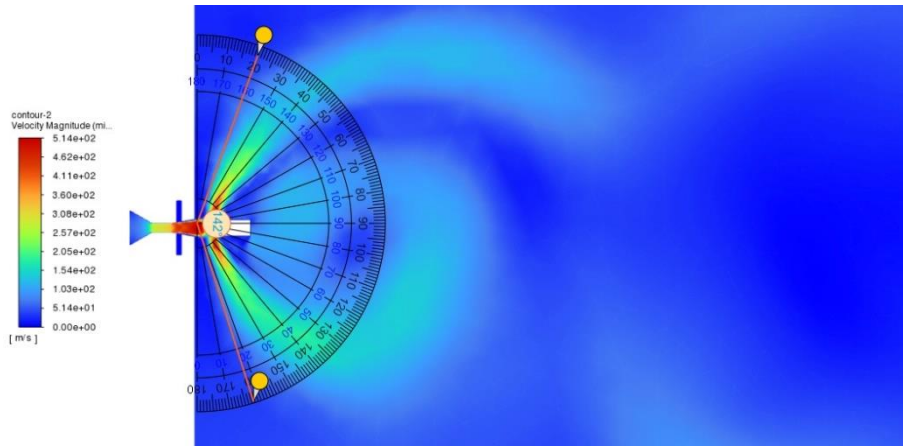
4.6.2 Atomization Angle Analysis

When measuring the atomization angle, significant measurement errors occur due to the irregular distribution of the liquid phase in the water volume fraction contours. Here, the atomization angle is determined through contours of velocity magnitude.

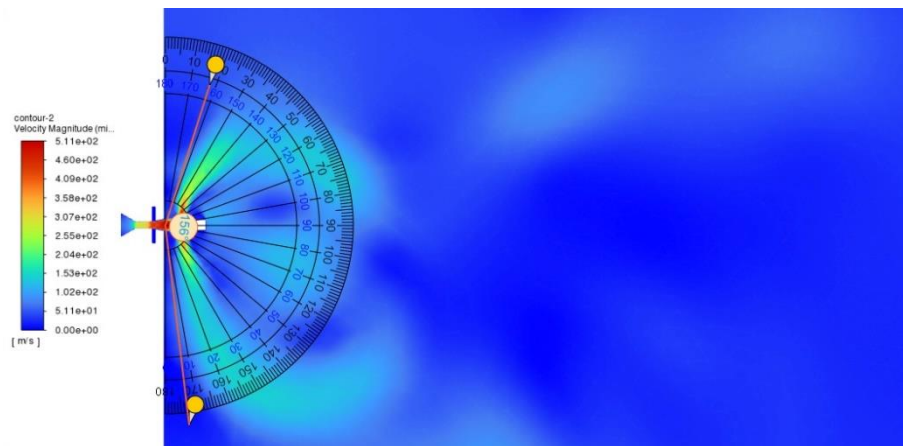


(a) Angle of atomization range with oscillator distance of 2 mm

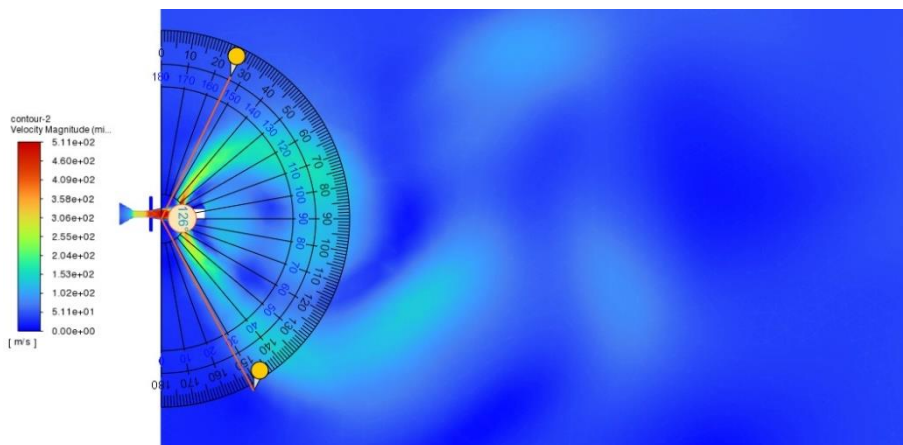
Figure 4.26: Angle of atomization range with varied oscillator distances



(b) Angle of atomization range with oscillator distance of 3 mm

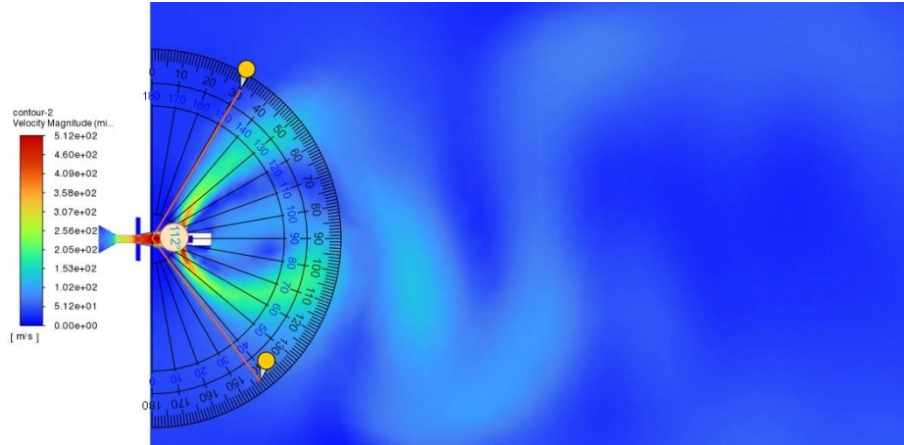


(c) Angle of atomization range with oscillator distance of 4 mm



(d) Angle of atomization range with oscillator distance of 5 mm

Figure 4.26 (continue): Angle of atomization range with varied oscillator distances



(e) Angle of atomization range with oscillator distance of 6 mm

Figure 4.26 (continue): Angle of atomization range with varied oscillator distances

From Figure 4.26, it is evident that oscillator distance significantly impacts the angle of the atomization range. This is because the gas-liquid mixture emitted from the nozzle will first collide with the oscillator, so the subsequent motion is related to the distance between the nozzle outlet and the oscillator. The angle of atomization range at varied oscillator distances is presented in Table 4.9.

Table 4.9: Angle of atomization range with varied oscillator distances

Oscillator Distance (mm)	The angle of atomization range (°)
2	151
3	142
4	156
5	126
6	112

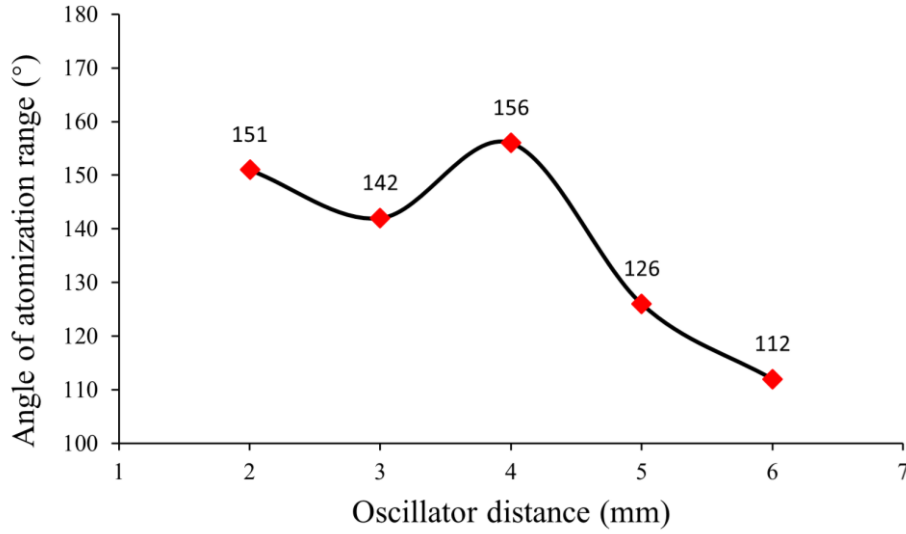


Figure 4.27: Scatter plot of the angle of atomization range with varied oscillator distances

The scatter plot in Figure 4.27 illustrates the oscillator distance on the x-axis and the angle of the atomization range on the y-axis. As shown in Figure 4.27, the angle of the atomization range is noticeably larger at oscillator distances of 2 mm and 4 mm compared to 3 mm, 5 mm, and 6 mm. The largest angle of the atomization range occurs at an oscillator distance of 4 mm.

In conclusion, the distance of the oscillator has a minimal impact on velocity magnitude but a significant effect on the angle of atomization range. The angles at oscillator distances of 2 mm and 4 mm are notably superior to 3 mm, 5 mm, and 6 mm. Additionally, both the velocity (511 m/s) and the angle of atomization range (156 °) at an oscillator distance of 4 mm are superior to those at a distance of 2 mm (velocity: 509 m/s, angle of atomization range: 151 °). Therefore, the optimized oscillator distance is chosen as 4 mm.

4.7 3D Model Validation

4.7.1 3D Model Creation

2D numerical analysis offers high efficiency, while 3D numerical analysis provides higher accuracy but at a slower speed. To verify the reliability of the 2D model, this study simulated a set of 3D models for comparison. The simulation conditions were set as follows: water pressure of 0.1 MPa, air pressure of 0.5 MPa, an outlet diameter of 2.5 mm, and a nozzle-to-vibrator distance of 4 mm. The constructed model is shown in Figure 4.28, with the same solution settings as the 2D model.

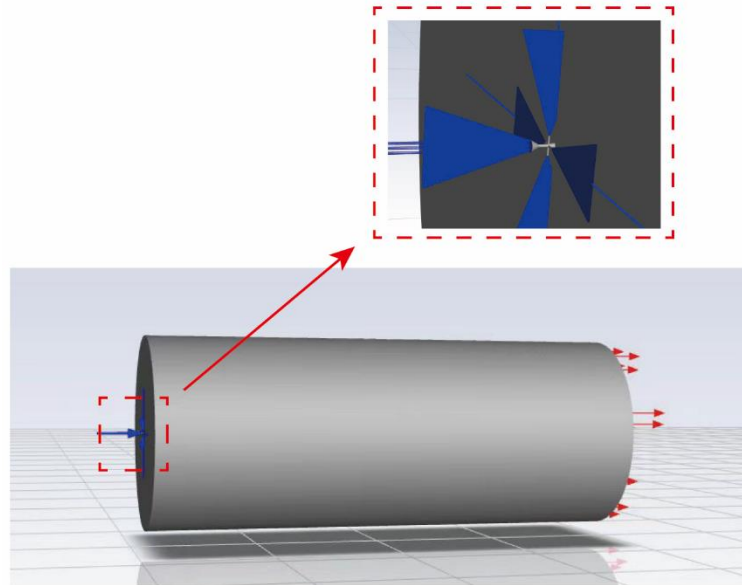


Figure 4.28: Linear Laval nozzle 3D model

4.7.2 Numerical Analysis Comparison of 3D and 2D Models

Figure 4.29 shows the velocity contour and water volume fraction contour from the 2D and 3D numerical simulations. It can be seen from the figure that the simulation results are similar.

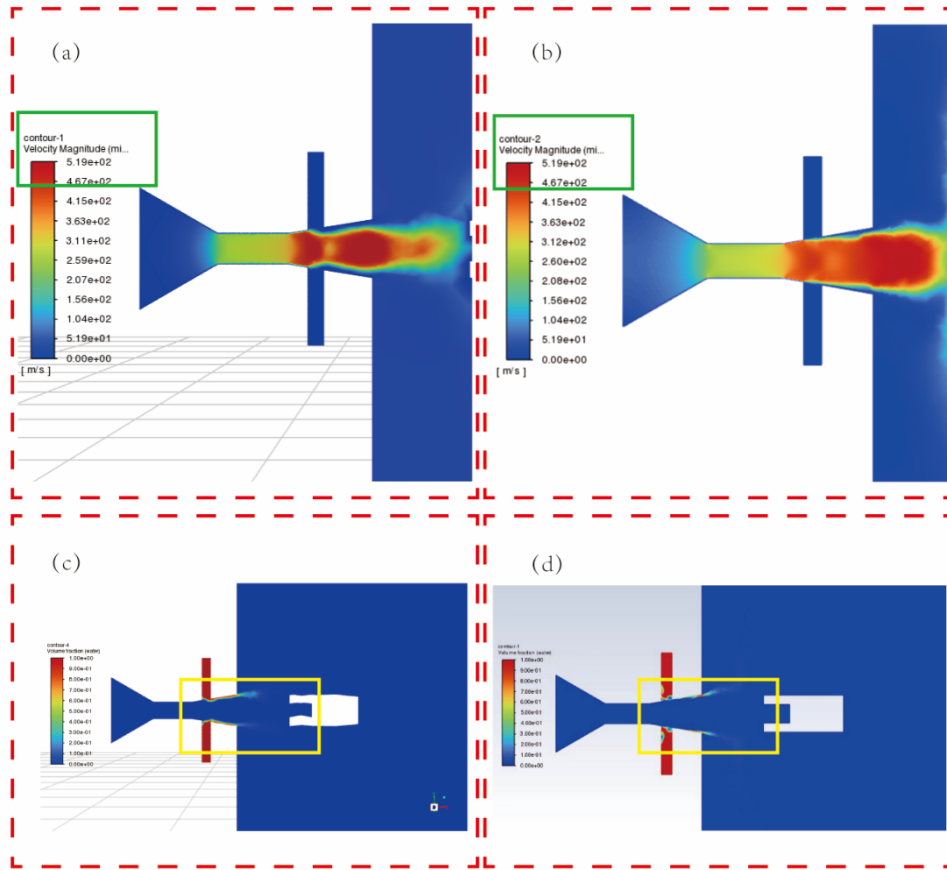


Figure 4.29: Numerical Analysis Comparison of 3D and 2D Models

In Figures (a) and (b), the flow velocity reaches sonic speed at the throat and continues to increase, with a maximum velocity of 519 m/s. The water volume fraction contours from the 2D and 3D numerical simulations show similar patterns. Most of the water moves forward along the contour of the expansion section, while the central region has a lower water volume fraction. Since the 3D model yields results similar to the 2D model but with slower computation speed, 2D numerical analysis was chosen for this study due to its higher efficiency.

4.8 Analysis of Experimental Results with Varied Water Pressure

4.8.1 Analysis of Atomization Angle and Droplet Size

The air pressure is 0.5 MPa, the diameter of the nozzle outlet is 2.5 mm, and the oscillator distance is 4 mm. The experiment aims to measure the atomization angle and droplet size under varied water pressures (0.1 MPa, 0.2 MPa, 0.3 MPa, 0.4 MPa, and 0.5 MPa).

4.8.1.1 Atomization Angle Analysis

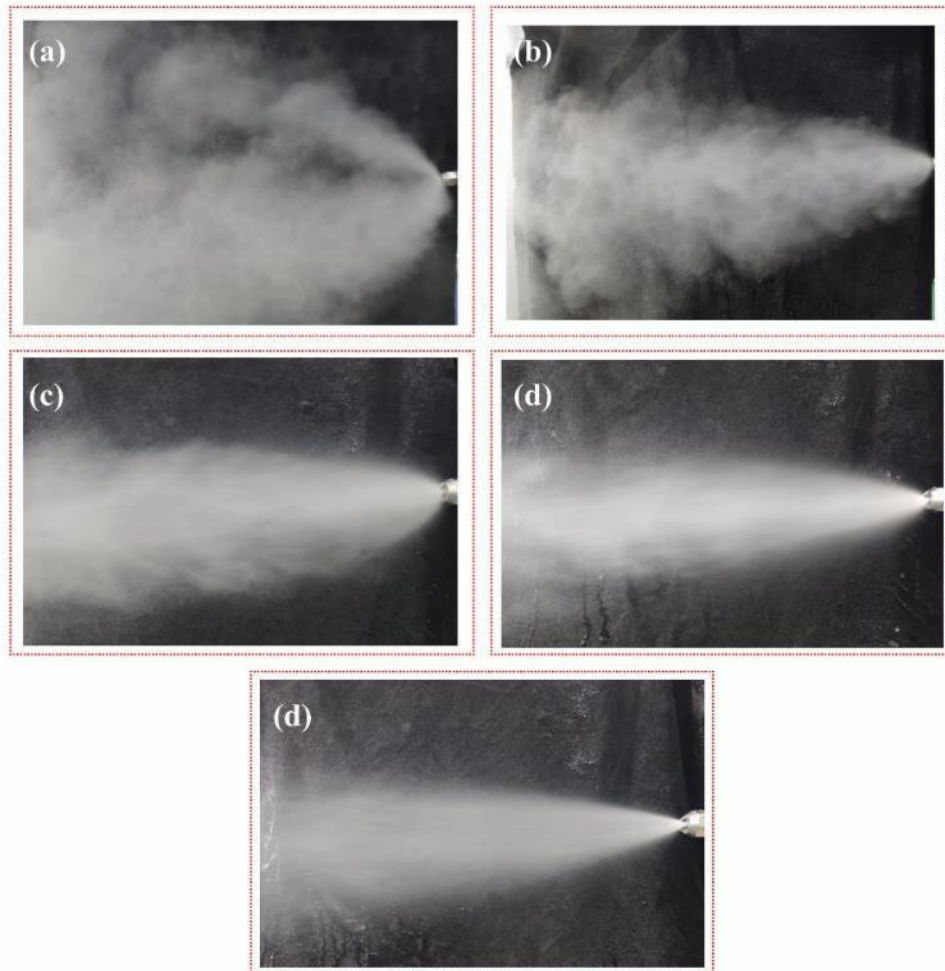


Figure 4.30: Atomization angle at varied water pressures by experiments: (a) 0.1 MPa; (b) 0.2 MPa; (c) 0.3 MPa; (d) 0.4 MPa; (e) 0.5 MPa

The experimental results of spray under varied water pressures are shown in Figure 4.30. The atomization effect is very good at a water pressure of 0.1 MPa. With higher pressures, droplets begin to fall. Water pressure has a significant impact on the atomization angle. The atomization angles, measured using CAD, are provided in Table 4.10.

Table 4.10: Atomization angle at varied water pressures by experiment

Air pressure is 0.5MPa, Water Pressure (MPa)	Atomization Angle (°)
0.1	104
	101
	105
0.2	68
	70
	67
0.3	56
	60
	58
0.4	46
	49
	47
0.5	37
	40
	38

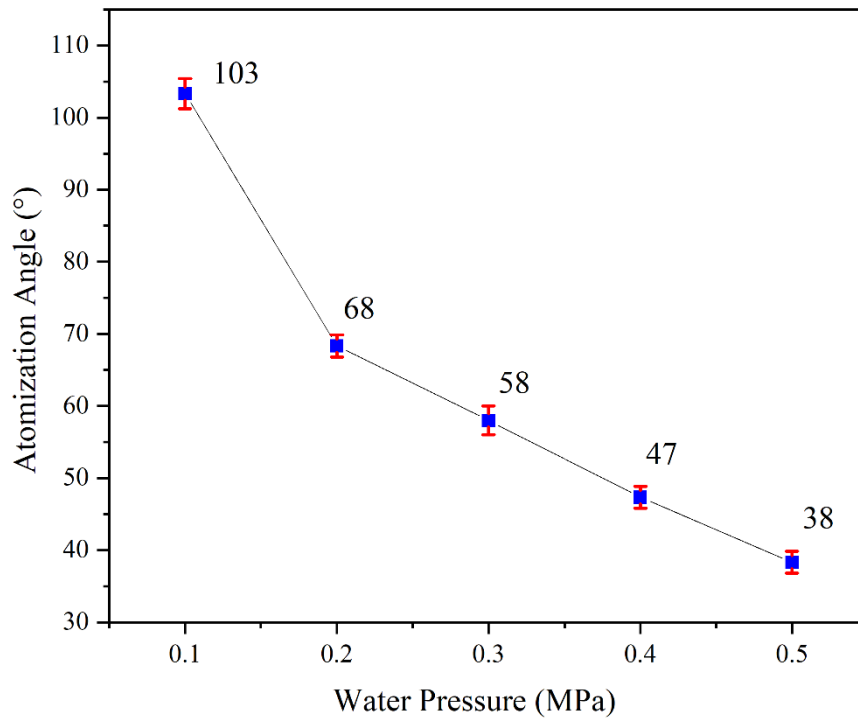


Figure 4.31: Scatter plot of atomization angle at varied water pressures by experiment

The scatter plot shown in Figure 4.31 with water pressure on the x-axis and atomization angle on the y-axis illustrates a decrease in atomization angle as water pressure varies from 0.1 MPa to 0.5 MPa. At a water pressure of 0.1 MPa, the atomization angle is at its maximum, with 103°, then rapidly decreases. At 0.2 MPa, it decreases to 68°. The atomization angle at 0.1MPa water pressure is significantly higher than at others. Therefore, the optimal atomization angle is observed at 0.1 MPa.

4.8.1.2 Droplet Size

Repeatability refers to the relative error in results obtained by the instrument from multiple measurements of the same sample. The smaller the relative error, the better the repeatability.

Suppose an instrument performs Z measurements of a specific physical quantity and obtains n measurement values ($\Pi_1, \Pi_2, \dots, \Pi_n$). In that case, the average of these Z measurements is calculated as Equation 4.2 (Bartlett & Frost, 2008).

$$\Pi = \sum_{i=1}^n \Pi_i / n \quad \text{Equation 4.2}$$

The repeatability error is calculated as Equation 4.3.

$$R_{rep} = \sqrt{\sum_{i=1}^n (\Pi_i - \Pi)^2 / (n-1)} \quad \text{Equation 4.3}$$

The relative error is calculated as Equation 4.4.

$$\delta_{re} = \left(\frac{R_{rep}}{\Pi} \right) * 100\% \quad \text{Equation 4.4}$$

As shown in Figure 4.32, the droplet size testing report includes droplet size characteristic parameters, droplet size distribution graph, and droplet size distribution table. D_{10} represents the droplet diameter at which the cumulative distribution curve is equal to or less than 10%; D_{25} represents the droplet diameter at which the cumulative distribution curve is equal to or less than 25%; D_{50} represents the droplet diameter at which the cumulative distribution curve is equal to or less than 50%; D_{75} represents the droplet diameter at which the cumulative distribution curve is equal to or less than 75%; D_{90} represents the droplet diameter at which the cumulative distribution curve is equal to or less than 90%. Figure 4.32 shows the percentage and cumulative percentage of different particle size ranges.

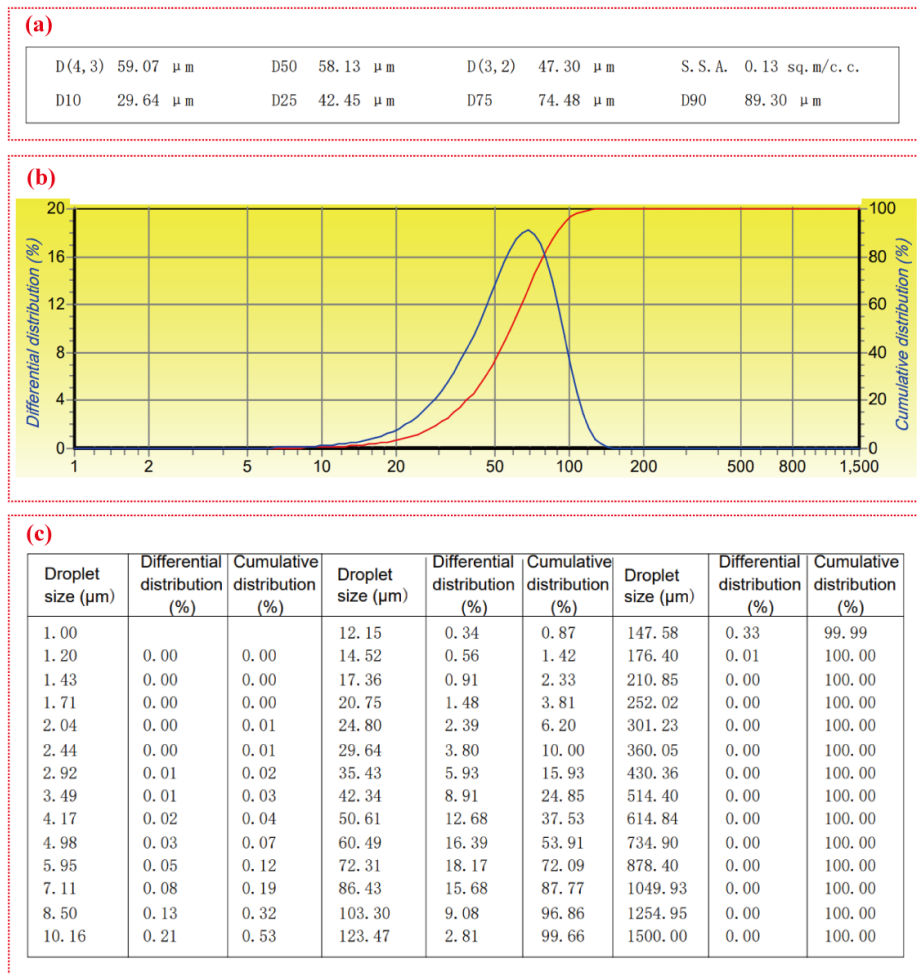


Figure 4.32: Droplet size testing report: (a) Droplet size characteristic parameters; (b) Droplet size distribution graph; (c) Droplet size distribution table.

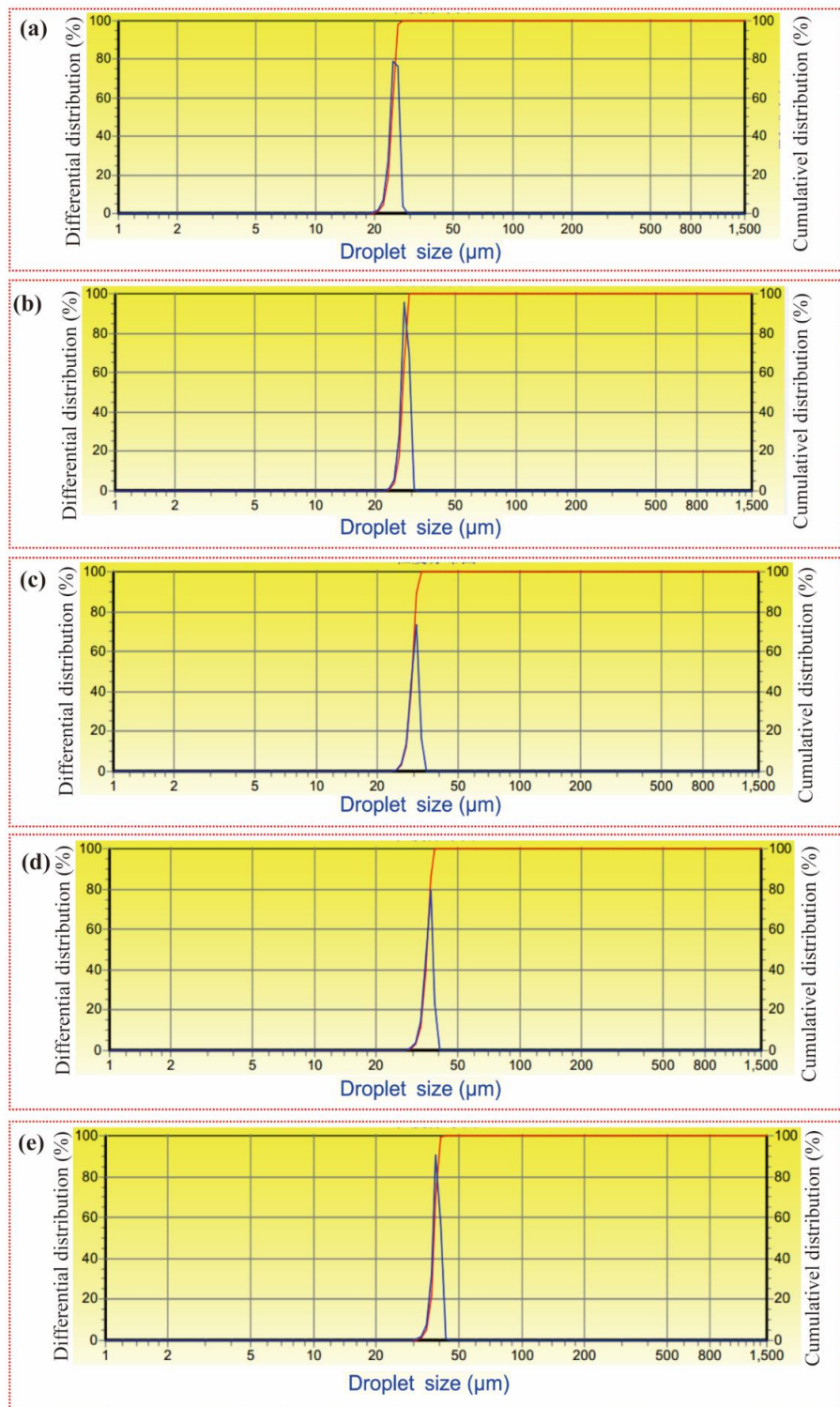
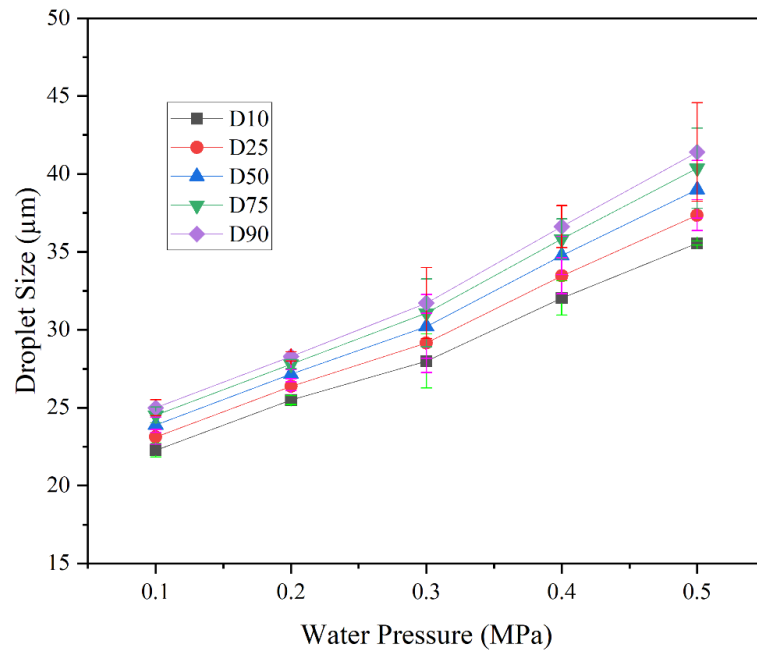


Figure 4.33: Droplet size testing report at varied water pressures: (a) 0.1 MPa; (b) 0.2 MPa; (c) 0.3 MPa; (d) 0.4 MPa; (e) 0.5 MPa.

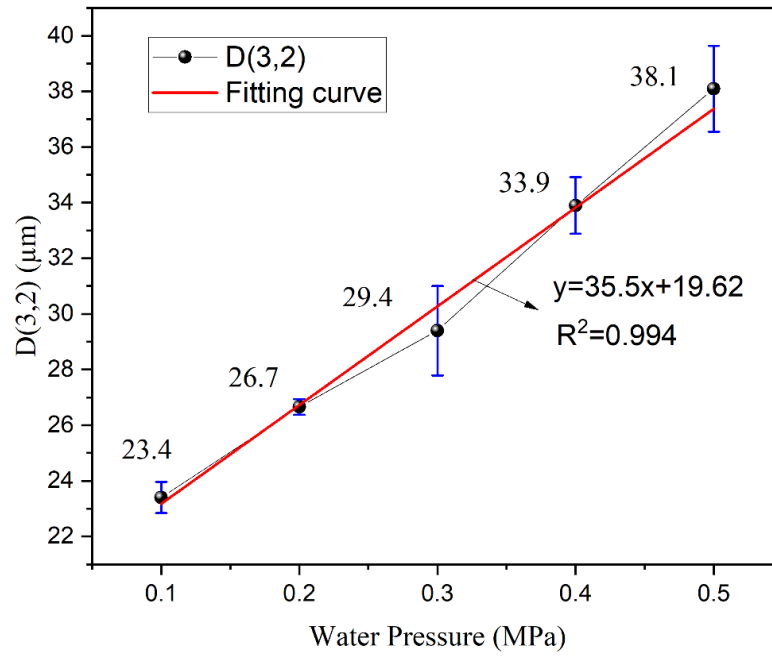
Figure 4.33 presents droplet size testing reports conducted at an air pressure of 0.5MPa with varied water pressures. The impact of water pressure on atomized droplet size is notable, influencing both droplet size and distribution. The primary droplet size characteristic parameters are outlined in Table 4.11.

Table 4.11: Droplet size characteristic parameters under varied water pressures (Air pressure is 0.5 MPa)

Water Pressure (MPa)	D(3,2) (μM)	D10 (μM)	D25 (μM)	D50 (μM)	D75 (μM)	D90 (μM)
0.1	23.73	22.57	23.47	24.28	24.95	25.44
	23.72	22.47	23.3	24.05	24.66	25.11
	22.75	21.77	22.6	23.36	23.97	24.43
0.2	26.62	25.43	26.31	27.1	27.74	28.22
	26.96	25.8	26.11	27.48	28.12	28.6
	26.39	25.23	26.68	26.9	27.54	28.03
0.3	28.7	27.24	28.37	29.39	30.22	30.84
	31.23	29.98	31.32	32.55	33.55	34.3
	28.26	26.76	27.78	28.7	29.45	30.01
0.4	34.52	32.42	33.79	35.03	36.05	36.81
	34.45	32.89	34.43	35.84	36.99	37.86
	32.73	30.8	32.17	33.42	34.44	35.2
0.5	37.12	35.42	36.75	37.96	38.94	39.67
	39.88	35.68	38.5	41.14	43.35	45.05
	37.29	35.58	36.81	37.93	38.83	39.5



(a) D10, D25, D50, D75, and D90



(b) D(3,2)

Figure 4.34: Droplet size characteristic parameters at varied water pressures by experiment: (a) D10, D25, D50, D75, and D90; (b) D(3,2).

Figure 4.34 shows the droplet size characteristic parameters trend as water pressure varies from 0.1 MPa to 0.5 MPa by experiment with droplet size characteristic parameters on the x-axis and droplet size on the y-axis. Figure 4.34 shows that the corresponding droplet size characteristic parameters also increase as the water pressure increases. The droplet size is minimal at a water pressure of 0.1 MPa, with $D(3,2)$ measuring 23.4 μm . The fitting curve for the relationship between droplet size and water pressure is given by $y = 35.5x + 19.62$, $R^2=0.994$, showing a good fit.

In summary, the atomization angle is maximized at a water pressure of 0.1 MPa, and the droplet size is minimized. Therefore, a water pressure of 0.1 MPa represents the optimal pressure value within the 0.1 to 0.5 MPa range.

4.8.2 Comparison between Experimental Results and Numerical Analysis

4.8.2.1 Angles of Atomization Range

At an air pressure of 0.5 MPa, a comparison between the angles of atomization range by numerical analysis and experimental atomization angles under varied water pressures is shown in Table 4.12.

Table 4.12: A comparison between the angles of atomization range by numerical analysis and experimental atomization angles under varied water pressures

Water pressure (MPa)	Air pressure of 0.5MPa, Angle of atomization range by numerical analysis (°)	Air pressure of 0.5MPa, Atomization Angle by experiments (°)
0.1	156	103
0.2	147	68
0.3	139	58
0.4	127	47
0.5	109	38

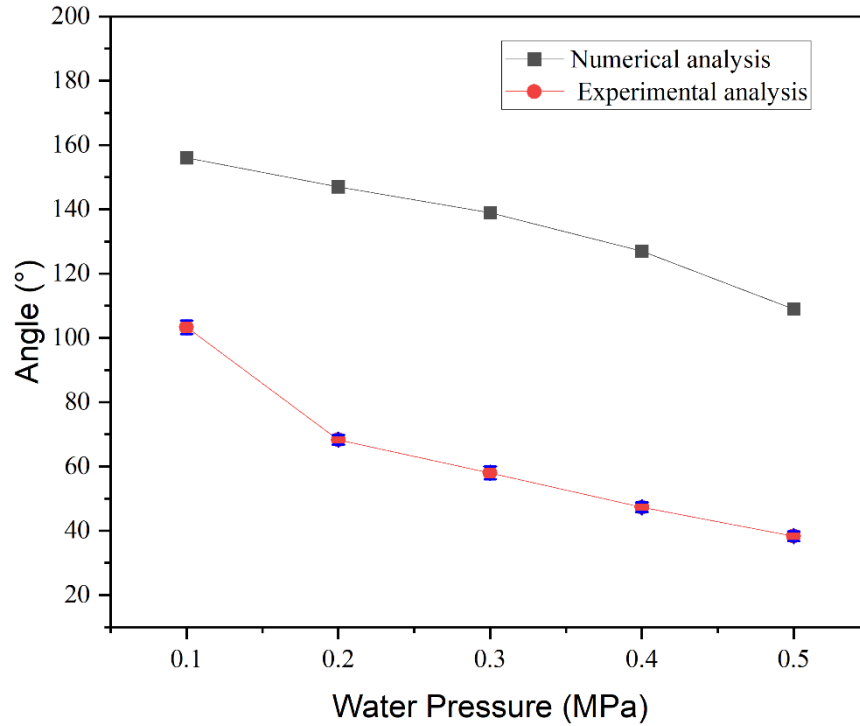


Figure 4.35: A comparison between the angles of atomization range by numerical analysis and experimental atomization angles under varied water pressures

At an air pressure of 0.5 MPa, the numerical analysis of angles of atomization range and the experimental atomization angles under varied water pressures are shown in Figure 4.35. Because the shape of the water volume fraction is not irregular in the numerical simulation, it is challenging to measure the atomization angle accurately. Therefore, the angle measured represents the atomization range of the air-liquid mixture. As a result, there is some discrepancy between the numerical analysis and experimental data, but the trends in their variations are consistent. Both curves show a decreasing trend as the pressure increases, with the maximum angle at a water pressure of 0.1 MPa.

4.8.2.2 Sauter Mean Diameter

At an air pressure of 0.5 MPa, a comparison between $D(3,2)$ by numerical analysis and $D(3,2)$ by experiments under varied water pressures is shown in Table 4.13.

Table 4.13: A comparison between $D(3,2)$ by numerical analysis and $D(3,2)$ by experiments under varied water pressures

Water pressure (MPa)	Air pressure of 0.5 MPa, $D(3,2)$ by numerical analysis (μm)	Air pressure of 0.5 MPa, $D(3,2)$ by experiments (μm)
0.1	15.2	23.4
0.2	20.6	26.7
0.3	22.0	29.4
0.4	26.9	33.9
0.5	35.1	38.1

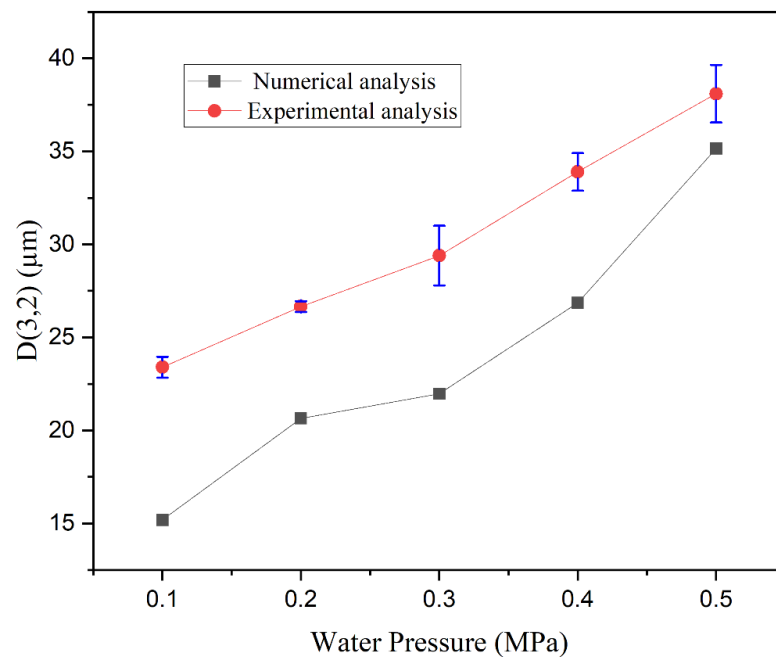


Figure 4.36: a comparison between $D(3,2)$ by numerical analysis and $D(3,2)$ by experiments under varied water pressures

Figure 4.36 compares the numerical analysis and experimental results of $D(3,2)$ under varied water pressures at an air pressure of 0.5 MPa. As the water pressure increases, $D(3,2)$ increases. Both curves exhibit the same trend, and $D(3,2)$ is the smallest at a water pressure of 0.1 MPa. Overall, the numerically analyzed droplet sizes are smaller than the experimental data. The discrepancies may stem from measurement errors of the experimental instruments and numerical analysis errors caused by model simplifications and boundary conditions. Another reason is that the simulated droplet size $D(3,2)$ represents the overall value, while the droplet size $D(3,2)$ measured in the experiment corresponds to a specific location. This difference between the overall and local measurements leads to a resulting discrepancy.

4.9 Analysis of Experimental Results with Varied Air Pressure

4.9.1 Atomization Angle Analysis

The water pressure is 0.1 MPa, the nozzle outlet diameter is 2.5 mm, and the oscillator distance is 4 mm. The experiment aims to measure the atomization angle under varying air pressures (0.2 MPa, 0.3 MPa, 0.4 MPa, 0.5 MPa, and 0.6 MPa).

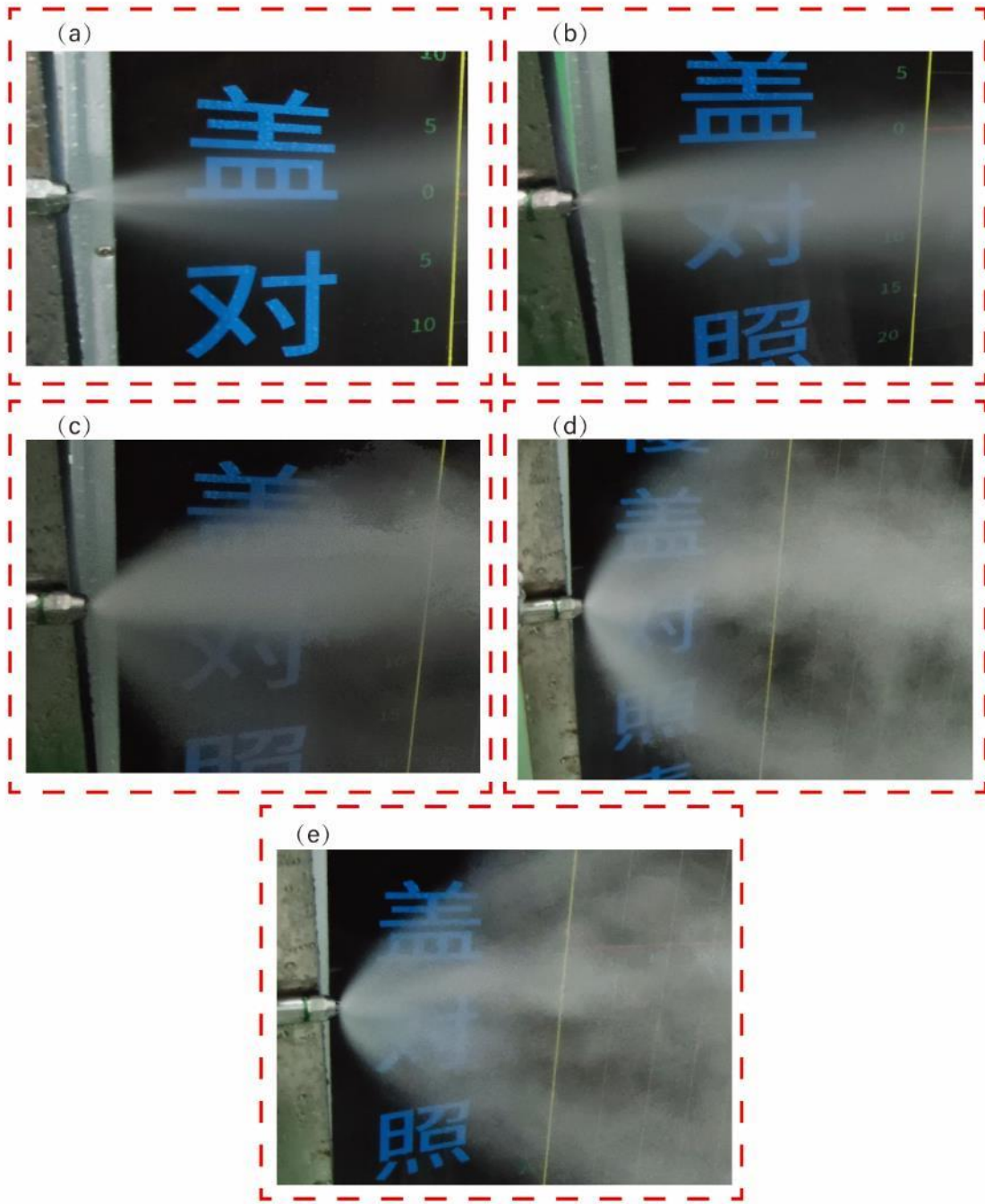


Figure 4.37: Atomization angle at varied air pressures by experiments: (a) 0.2 MPa; (b) 0.3 MPa; (c) 0.4 MPa; (d) 0.5 MPa; (e) 0.6 MPa

Figure 4.37 shows that air pressure significantly affects the atomization angle at a water pressure of 0.1 MPa. The values are presented in Table 4.14.

Table 4.14: Atomization angle under varied air pressures by experiments

Water pressure is 0.1MPa, Air Pressure (MPa)	Atomization Angle (°)
0.2	39
	41
	43
0.3	52
	53
	55
0.4	85
	89
	87
0.5	104
	103
	106
0.6	89
	90
	87

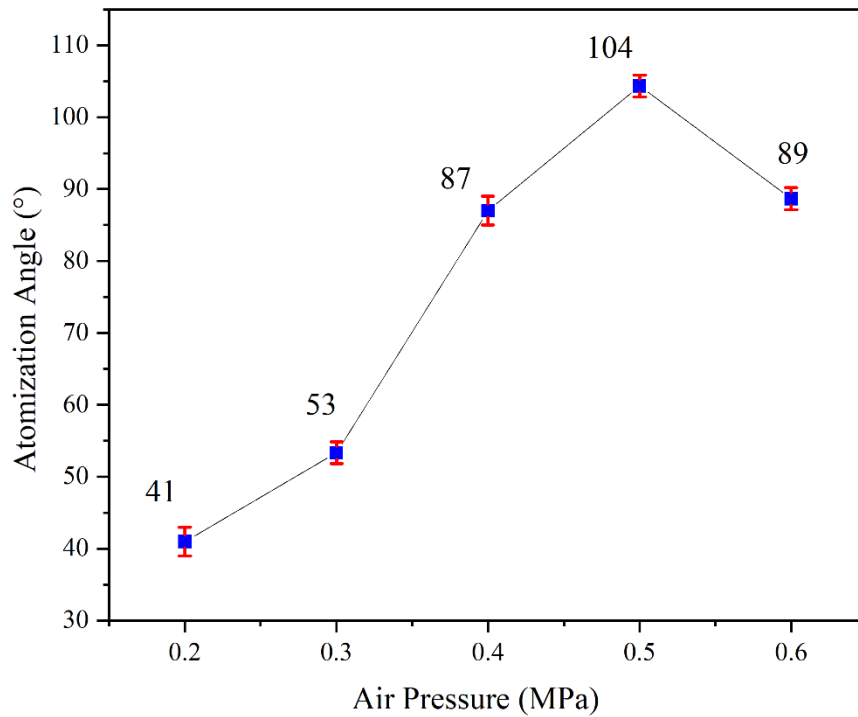


Figure 4.38: Scatter plot of atomization angle at varied air pressures by experiments

The scatter plot shown in Figure 4.38 with air pressure on the x-axis and atomization angle on the y-axis illustrates a trend of atomization angle as air pressure varies from 0.2 MPa to 0.6 MPa. Figure 4.38 shows that the atomization angle is significantly affected by air pressures with a water pressure of 0.1 MPa. At an air pressure of 0.5 MPa, the average atomization angle is at its maximum, measuring 104°.

4.9.2 Comparison between Experimental Results and Simulation Results

The comparison between the numerical analysis of angles of atomization range and the experimental atomization angles at varied air pressures under a water pressure of 0.1MPa is shown in Table 4.15.

Table 4.15: A comparison between the numerical analysis of angles of atomization range and the experimental atomization angles under varied air pressures

Air Pressure (MPa)	Water pressure of 0.1 MPa, angle of atomization range by numerical analysis (°)	Water pressure of 0.3 MPa, experimental atomization angle (°)
0.2	97	41
0.3	111	53
0.4	138	87
0.5	156	104
0.6	121	89

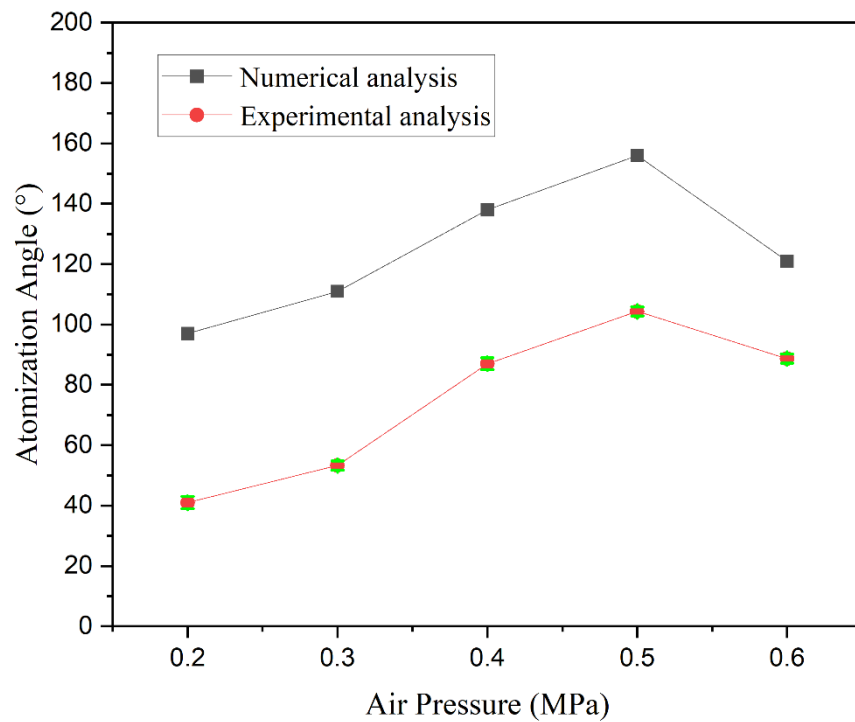


Figure 4.39: A comparison between the numerical analysis of angles of atomization range and the experimental atomization angles under varied air pressures

Figure 4.39 compares the angles of the atomization range by numerical analysis and experimental atomization angles under varied air pressures. The trend of angle variation with air pressure is consistent; it initially increases, then decreases, reaching its maximum at an air pressure of 0.5 MPa. Overall, the angles derived from numerical analysis are larger than the experimental data. The discrepancies may result from measurement errors in the experimental instruments and numerical analysis errors due to model simplifications and boundary condition assumptions.

4.10 Research Contribution

Table 4.16 compares water pressure between the traditional nozzle and the optimized Laval nozzle. Traditional pressure-rotating nozzles typically operate at water pressures exceeding 5 MPa. Such high-pressure demands increase energy consumption, substantial water usage, elevated costs, and potential safety hazards. In contrast, the optimized Laval nozzle utilized in this study operates at a water pressure of 0.1 MPa, representing a reduction of over 90% compared to traditional nozzles. This results in energy efficiency, cost-effectiveness, and improved safety measures.

Table 4.16: The comparison of water pressure between the traditional nozzle and the optimized Laval nozzle

Author & Time	Nozzle type	Water Pressure (MPa)
(Xu <i>et al.</i> , 2019)	Pressure swirling nozzle	8
(Liu <i>et al.</i> , 2019)	Press-swirl nozzle	5
This research	Laval nozzle	0.1

Table 4.17 compares the spray angle between the traditional nozzle and the optimized Laval nozzle. Typically, traditional nozzles have a maximum spray angle of around 60°,

while the optimized Laval nozzle in this research can achieve a maximum spray cone angle of 103° . This represents an increase of approximately 67.5% in the spray angle compared to traditional nozzles.

Table 4.17: The comparison of spray angle between traditional nozzles and optimized Laval nozzles

Author & Time	Nozzle type	Spray Angle ($^\circ$)
(Zhang <i>et al.</i> , 2020a)	Ultrasonic nozzle	61.5
(Amoli <i>et al.</i> , 2022)	A spray nozzle	60
This research	Laval nozzle	103

A larger spray angle allows for a more extensive dispersion of droplets, covering a larger area and facilitating a more uniform distribution, thereby enhancing dust removal efficiency. Furthermore, if a combination nozzle, as shown in Figure 4.40, is used, the spray angle can be further increased by 52° , reaching up to 155° .

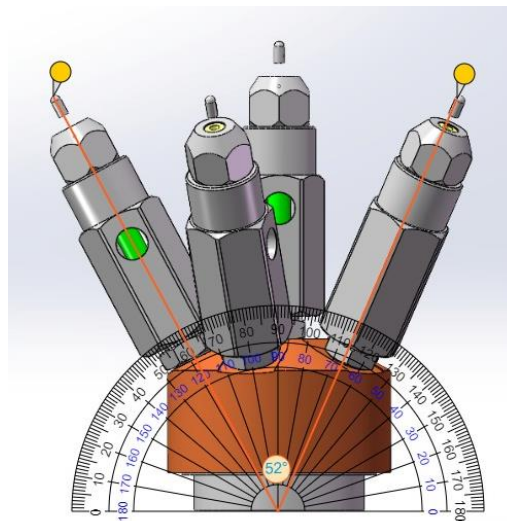


Figure 4.40: The angle between the nozzles in a combination nozzle structure

Table 4.18: The comparison of droplet size D(3,2) between traditional nozzles and optimized Laval nozzles

Author & Time	Nozzle type	Droplet size D(3,2) (μm)
(Liu <i>et al.</i> , 2019)	Press-swirl nozzle	90
(Xu <i>et al.</i> , 2020)	pressure-swirl nozzle	51
This research	Laval nozzle	23.4

Table 4.18 compares droplet size D (3,2) between traditional pressure-swirl and optimized Laval nozzles. Droplets generated by traditional pressure-swirl nozzles typically exceed 50μm in diameter, while the optimal D (3,2) achieved with the Laval nozzle used in this research is only 23.4 μm, representing a reduction of over 50%. When the diameter of droplets is similar to that of dust particles, the dust removal efficiency is significantly improved. Therefore, the small-diameter droplets produced by the Laval nozzle are advantageous for enhancing respirable dust removal efficiency.

In summary, water pressure is reduced by over 90% compared to traditional nozzles, and D (3,2) decreases by over 50%. Moreover, the spray angle increases by approximately 67.5% with the optimized Laval nozzle, leading to significantly enhanced atomization performance. The linear Laval nozzle studied in this research is easy to manufacture, exhibits excellent atomization performance, operates at low pressure, and is cost-effective. A combination-type nozzle is proposed, which expands the atomization angle and enhances the dust removal range. These nozzles produce dry mist due to the small droplet size. Compared to traditional water mist dust removal methods, they are more energy-efficient and environmentally friendly, as they do not require large amounts of water resources and do not lead to groundwater accumulation or secondary pollution. Dry mist dust removal does

not involve water, thus avoiding safety hazards such as slipping and electrical equipment failures caused by water mist. The linear nozzles are applicable in diverse enclosed environments such as mines, construction sites, and processing plants. They can reduce dust removal efficiency effectively, significantly contributing to human health and environmental protection.

4.11 Summary of the Chapter

This section presents the results of numerical analysis and experiments. The numerical analysis examines the influence of water pressure, air pressure, nozzle outlet diameter, and oscillator distance on fluid velocity and atomization angle. Additionally, it investigates the impact of water pressure on droplet size distribution and droplet size. The experiments focus on the influence of water pressure on atomization angle, droplet size, and distribution, as well as the effect of air pressure on atomization angle. The trend of numerical analysis and experiments is similar. The optimized parameters are as follows: water pressure of 0.1 MPa, air pressure of 0.5 MPa, nozzle outlet diameter of 2.5 mm, and oscillator distance of 4mm. Compared to traditional nozzles, the water pressure is reduced by over 90%, and the $D_{(3,2)}$ droplet diameter decreases by over 50%. Moreover, the spray angle increases by approximately 67.5% with the modified Laval nozzle, leading to significantly enhanced atomization performance.

CHAPTER 5

CONCLUSION, INNOVATIONS AND RECOMMENDATIONS

5.1 Conclusion

This research focuses on modified Laval nozzles, examining the effects of operating conditions (air pressure and water pressure) and structural parameters (outlet diameter and oscillator distance) on the atomization performance of linear Laval nozzles through numerical analysis and experimental studies.

The design of linear Laval structures was carried out to verify that the internal airflow velocity reaches supersonic speeds. A combined Laval nozzle structure is proposed for applications involving high dust concentrations. The effects of operating conditions, including water and air pressure, on the atomization performance of Laval nozzles were studied using FLUENT numerical analysis. Water pressures of 0.1, 0.2, 0.3, 0.4, and 0.5 MPa were selected, and the VOF model was used to analyze the velocity, pressure, and atomization angle inside and outside the nozzle. The DPM model was applied to simulate droplet distribution characteristics and droplet sizes. Numerical analysis results indicated that the best atomization performance was achieved at a water pressure of 0.1 MPa. Similarly, air pressures of 0.2, 0.3, 0.4, 0.5, and 0.6 MPa were evaluated. At the optimal water pressure of 0.1 MPa, the maximum atomization angle was observed when the air pressure was 0.5 MPa.

This study also investigated the effects of different nozzle structures on the atomization performance of the Laval nozzle, focusing on parameters such as outlet diameter

and oscillator distance. The VOF model was employed to analyze internal and external flow velocity, pressure distribution, and atomization angle. Nozzle outlet diameters of 1.5, 2.0, 2.5, 3.0, and 3.5 mm were evaluated. The numerical results indicated that at a water pressure of 0.1 MPa and an air pressure of 0.5 MPa, the optimal atomization performance was achieved with an outlet diameter of 2.5 mm. Under the same conditions—water pressure of 0.1 MPa, air pressure of 0.5 MPa, and outlet diameter of 2.5 mm—oscillator distances of 2, 3, 4, 5, and 6 mm were also tested, with the largest atomization angle observed at a distance of 4 mm.

Experimental studies were conducted to investigate the effects of air and water pressure on the atomization performance of the nozzle, primarily using a laser particle size analyzer. At an air pressure of 0.5 MPa, the droplet size distribution and atomization angle were measured at water pressures of 0.1, 0.2, 0.3, 0.4, and 0.5 MPa. The experimental results indicated that at a water pressure of 0.1 MPa, the smallest Sauter Mean Diameter (SMD) of droplets and the largest atomization angle were observed. Similarly, at a water pressure of 0.1 MPa, the atomization angle was measured at air pressures of 0.2, 0.3, 0.4, 0.5, and 0.6 MPa, with the largest atomization angle observed at an air pressure of 0.5 MPa. The trends observed in the numerical analysis and experimental results showed good consistency. Additionally, fitting curves were plotted to predict velocity, droplet size, and atomization angle.

5.2 Innovations

The main innovations of this research include investigating the atomization performance of linear Laval nozzles, which are more cost-effective and easier to manufacture than curved ones. Numerical analysis of the internal and external flow fields of

these nozzles was conducted using FLUENT software, combining the VOF model and DPM, and validated through experimental results. Additionally, a combined nozzle structure was designed, and fitting curves were plotted to predict velocity, droplet size, and atomization angle within a specific range.

5.3 Recommendations for Future Work

This study systematically investigated the atomization performance of linear Laval nozzles under varying structural parameters and operating conditions through numerical simulations and experimental analysis. The findings provide valuable insights into optimizing nozzle design and operational settings. However, several areas warrant further research. Future work could explore the interactions among multiple parameters influencing atomization, as this study primarily employed a single-variable approach; techniques such as orthogonal experimental design or response surface methodology may yield more comprehensive insights.

Additionally, the accuracy of the fitted curves—such as the comparison between the experimentally measured Sauter Mean Diameter (SMD) under a water pressure of 1.5 MPa and the fitting curve $y=46.13x+10.125$ —should be further validated through both numerical and experimental methods. The impact of bionic structures on atomization performance, including their position, shape, and quantity, also represents a promising avenue for future research. Finally, applying the research outcomes to industrial scenarios such as mining dust suppression, agricultural spraying, and combustion systems will help assess the engineering feasibility and practical effectiveness of the proposed nozzle designs.

REFERENCES

- Agarwal, A., & Trujillo, M. F. (2020). The effect of nozzle internal flow on spray atomization. *International Journal of Engine Research*, 21(1), 55–72.
- Alvim, I. D., Stein, M. A., Koury, I. P., Dantas, F. B. H., & Cruz, C. L. de C. V. (2016). Comparison between the spray drying and spray chilling microparticles contain ascorbic acid in a baked product application. *LWT-Food Science and Technology*, 65, 689–694.
- Amoli, B. S., Ajarostaghi, S. S. M., Sedighi, K., & Delavar, M. A. (2022). Evaporative pre-cooling of a condenser airflow: Investigation of nozzle cone angle, spray inclination angle and nozzle location. *Energy Sources, Part A. Recovery, Utilization, and Environmental Effects*.
- Arya, S., Sottile, J., & Novak, T. (2020). Numerical modeling of a flooded-bed dust scrubber integrated into a longwall shearer. *Mining, Metallurgy & Exploration*, 37(4), 1105–1119.
- Balabel, A., Hegab, A., Nasr, M., & El-Behery, S. M. (2011). Assessment of turbulence modeling for gas flow in two-dimensional convergent–divergent rocket nozzle. *Applied Mathematical Modelling*, 35(7), 3408–3422.
- Bao, J., Yang, L., Guo, W., Song, S., & Geng, J. (2012). Improving the removal of fine particles in the WFGD system by adding wetting agents. *Energy & Fuels*, 26(8), 4924–4931.
- Bartlett, J., & Frost, C. (2008). Reliability, repeatability and reproducibility: Analysis of measurement errors in continuous variables. *Ultrasound in Obstetrics and*

- Gynecology: The Official Journal of the International Society of Ultrasound in Obstetrics and Gynecology*, 31(4), 466–475.
- Bian, J., Cao, X., Yang, W., Edem, M. A., Yin, P., & Jiang, W. (2018). Supersonic liquefaction properties of natural gas in the Laval nozzle. *Energy*, 159, 706–715.
- Blott, S. J., Croft, D. J., Pye, K., Saye, S. E., & Wilson, H. E. (2004). Particle size analysis by laser diffraction. *Geological Society, London, Special Publications*, 232(1), 63–73.
- Bray, C. J., Spooner, E. T., & Thomas, A. V. (1991). Fluid inclusion volatile analysis by heated crushing, on-line gas chromatography; applications to Archean fluids. *Journal of Geochemical Exploration*, 42(1), 167–193.
- Broumand, M., Khan, M. S., Yun, S., Hong, Z., & Thomson, M. J. (2020). The role of atomization in the spray combustion of a fast pyrolysis bio-oil. *Fuel*, 276, 118035.
- Carvalho, I., & Heitor, M. (1998). Liquid film break-up in a model of a prefilming airblast nozzle. *Experiments in Fluids*, 24(5), 408–415.
- Chanchangi, Y. N., Ghosh, A., Sundaram, S., & Mallick, T. K. (2020). Dust and PV Performance in Nigeria: A review. *Renewable and Sustainable Energy Reviews*, 121, 109704.
- Chang, P., Zhao, Z., Xu, G., Ghosh, A., Huang, J., & Yang, T. (2020). Evaluation of the coal dust suppression efficiency of different surfactants: A factorial experiment. *Colloids and Surfaces A: Physicochemical and Engineering Aspects*, 595, 124686.
- Chen, D., Nie, W., Cai, P., & Liu, Z. (2018). The diffusion of dust in a fully-mechanized mining face with a mining height of 7 m and the application of wet dust-collecting nets. *Journal of Cleaner Production*, 205, 463–476.

- Chen, J., Jiang, W., Han, C., & Liu, Y. (2020). Study on supersonic swirling condensation characteristics of CO₂ in Laval nozzle. *Journal of Natural Gas Science and Engineering*, 84, 103672.
- Chen, M., Baby, R., Dillard, S., Lee, Y. T., & Ekkad, S. (2022). Design and test a converging and de Laval nozzle using additive manufacturing. *Frontiers in Aerospace Engineering*, 1.
- Chen, X., Bai, X., Fan, C., Ge, S., Deng, C., & Ma, G. (2024). Experimental investigation of atomization characteristics of sonic blasting nozzle based on De-Laval dynamics. *Energy*, 297, 131188.
- Chu, F., Zhang, K., Shen, H., Liu, M., Huang, W., Zhang, X., Liang, E., Zhou, Z., Lei, L., Hou, J., & others. (2021). Influence of satellite and agglomeration of powder on the processability of AlSi10Mg powder in Laser Powder Bed Fusion. *Journal of Materials Research and Technology*, 11, 2059–2073.
- Dumitrescu, O., Gherman, B., & Tipa, T. (2018). Development of a Laval nozzle for a cold gas propulsion system. *IOP Conference Series: Materials Science and Engineering*, 400, 042016.
- Durif, O. (2022). Design of de Laval nozzles for gas-phase molecular studies in uniform supersonic flow. *Physics of Fluids*, 34(1), 013605.
- Feng, Y., Guo, S., Jacob, J., & Sagaut, P. (2020). Grid refinement in the three-dimensional hybrid recursive regularized lattice Boltzmann method for compressible aerodynamics. *Physical Review E*, 101(6), 063302.
- Fukunaga, R., Islam, M. M., Awata, Y., Nakao, S., & Miyazato, Y. (2021). Application of rainbow schlieren deflectometry for jets from round Laval nozzles followed by

- cylindrical ducts. *Journal of Flow Control, Measurement & Visualization*, 09(02), 15–27.
- Gerhardter, H., Prieler, R., Zausinger, S., & Hochenauer, C. (2022). Investigating the advantages of Laval blasting nozzles in combination with injector-type sandblasters using efficient numerical methods. *Surface and Coatings Technology*, 445, 128699.
- Gong, T., Zhu, X., Ye, L., & Fu, Y. (2024). Numerical study of cavitation shock wave emission in the thin liquid layer by power ultrasonic vibratory machining. *Scientific Reports*, 14(1), 16956.
- Gortyshov, Yu. F., Tonkonog, V. G., & Kochenkov, A. G. (2007). Structure of a two-phase flow of boiling water at low-head adiabatic efflux through the laval nozzle. *Russian Aeronautics (Iz VUZ)*, 50(3), 248–252.
- Gregson, F. K., Watson, N. A., Orton, C. M., Haddrell, A. E., McCarthy, L. P., Finnie, T. J., Gent, N., Donaldson, G. C., Shah, P. L., Calder, J. D., & others. (2021). Comparing aerosol concentrations and particle size distributions generated by singing, speaking and breathing. *Aerosol Science and Technology*, 55(6), 681–691.
- Guo, D., & Zhu, Z. (2023). Influence of a meltblown die with a Laval airstream channel on the manufacturing process of a polymer fiber based on an orthogonal test and simulation analysis. *ACS Omega*, 8(51), 48742–48755.
- Guo, L., Nie, W., Yin, S., Liu, Q., Hua, Y., Cheng, L., Cai, X., Xiu, Z., & Du, T. (2020). The dust diffusion modeling and determination of optimal airflow rate for removing the dust generated during mine tunneling. *Building and Environment*, 178, 106846.
- Han, H., Wang, P., Li, Y., Liu, R., & Tian, C. (2020). Effect of water supply pressure on atomization characteristics and dust-reduction efficiency of internal mixing air atomizing nozzle. *Advanced Powder Technology*, 31(1), 252–268.

- Ji, M., Sun, Z., & Guo, H. (2022). The application of the foam technology to the dust control for reducing the dust injury of coal mines. *Applied Sciences*, 12(21), 10878.
- Jin, Y., Wu, C., Sun, P., Wang, M., Cui, M., Zhang, C., & Wang, Z. (2022). Electrification of water: From basics to applications. *Droplet*, 1(2), 92–109.
- Jing, D., Ren, S., Ge, S., Zhang, T., Wang, Z., & Liu, H. (2022). Research and application of dust reduction technology for supersonic spiral an atomizers. *Powder Technology*, 405, 117537.
- Kanai, T., Takizawa, K., Tezduyar, T. E., Tanaka, T., & Hartmann, A. (2019). Compressible-flow geometric-porosity modeling and spacecraft parachute computation with isogeometric discretization. *Computational Mechanics*, 63, 301–321.
- Kazem, H. A., Chaichan, M. T., Al-Waeli, A. H., & Sopian, K. (2020). A review of dust accumulation and cleaning methods for solar photovoltaic systems. *Journal of Cleaner Production*, 276, 123187.
- Kharulaman, L., Aabid, A., Mehaboobali, F. A. G., & Khan, S. A. (2019). Research on flows for NACA 2412 airfoil using computational fluid dynamics method. *Int. J. Eng. Adv. Technol*, 9(1), 5450–5456.
- Kong, D., Wang, Y., Tsubata, T., Kurosawa, M. K., & Aoyagi, M. (2024). Atomization characteristics of 9.6 MHz directional surface acoustic wave for 1-micron spray system. *Sensors and Actuators A: Physical*, 365, 114911.
- Lachs, L., Sommer, B., Cant, J., Hodge, J. M., Malcolm, H. A., Pandolfi, J. M., & Beger, M. (2021). Linking population size structure, heat stress and bleaching responses in a subtropical endemic coral. *Coral Reefs*, 40, 777–790.

- Li, B., & Lyu, B. (2023). Acoustic emission due to the interaction between shock and instability waves in two-dimensional supersonic jet flows. *Journal of Fluid Mechanics*, 954, A35.
- Li, B., Wu, J., & Liu, Y. (2021). Numerical study on subsonic-supersonic Laval nozzle using MacCormack scheme. *Journal of Physics: Conference Series*, 2012(1), 012096.
- Li, M., Fan, Y., Zhang, X., Wang, H., Cui, Y., Xue, K., & Gan, W. (2022). Experimental study on structure and performance of dedusting filter material modified by reducing graphene oxide spraying. *E3S Web of Conferences*, 356, 05069.
- Liang, H., Erkan, N., Zhou, Q., Porcheron, E., Gelain, T., & Suzuki, S. (2022). Effect of vessel size scale on the aerosol spray scavenging efficiency with water mist. *Journal of Aerosol Science*, 159, 105853.
- Liovic, P., Liow, J.-L., & Rudman, M. (2001). A volume of fluid (VOF) method for the simulation of metallurgical flows. *ISIJ International*, 41(3), 225–233.
- Liu, L., Wu, J., Su, W., & Kang, L. (2017). Lift and drag in three-dimensional steady viscous and compressible flow. *Physics of Fluids*, 29(11).
- Liu, Z., Nie, W., Peng, H., Yang, S., Chen, D., & Liu, Q. (2019). The effects of the spraying pressure and nozzle orifice diameter on the atomizing rules and dust suppression performances of an external spraying system in a fully-mechanized excavation face. *Powder Technology*, 350, 62–80.
- Luo, H., Wang, Q., Guan, Q., Ma, Y., Ni, F., Yang, E., & Zhang, J. (2022). Heavy metal pollution levels, source apportionment and risk assessment in dust storms in key cities in Northwest China. *Journal of Hazardous Materials*, 422, 126878.

- Ma, J., Li, J., Zhou, P., Song, Y., Chai, L., & Zhou, C. Q. (2019). A viewpoint on the dynamics of bubble formation from a submerged nozzle. *European Journal of Mechanics-B/Fluids*, 78, 276–283.
- Ma, Q., Nie, W., Yang, S., Xu, C., Peng, H., Liu, Z., Guo, C., & Cai, X. (2020). Effect of spraying on coal dust diffusion in a coal mine based on a numerical simulation. *Environmental Pollution*, 264, 114717.
- Mandal, S., Sadeghianjahromi, A., & Wang, C.-C. (2022). Experimental and numerical investigations on molten metal atomization techniques—A critical review. *Advanced Powder Technology*, 33(11), 103809.
- Mao, B., Zhao, C., Chen, H., Wang, Q., & Sun, J. (2021). Experimental and modeling analysis of jet flow and fire dynamics of 18650-type lithium-ion battery. *Applied Energy*, 281, 116054.
- Maring, H. (2003). Mineral dust aerosol size distribution change during atmospheric transport. *Journal of Geophysical Research*, 108(D19), 8592.
- Maroof, M. A., Mahboubi, A., Noorzad, A., & Safi, Y. (2020). A new approach to particle shape classification of granular materials. *Transportation Geotechnics*, 22, 100296.
- Martelli, E., Ciottoli, P., Saccoccio, L., Nasuti, F., Valorani, M., & Bernardini, M. (2019). Characterization of unsteadiness in an overexpanded planar nozzle. *AIAA Journal*, 57(1), 239–251.
- Mil'man, O. O., Goldin, A. S., Shifrin, B. A., Perov, V. B., Serezhkin, L. N., Ptakhin, A. V., Krylov, V. S., & Kartuesova, A. Yu. (2023). Experimental investigation into performance of Laval nozzles for reaction turbines. *Thermal Engineering*, 70(12), 1083–1101.

- Mousavi, M., Dimakopoulos, Y., & Tsamopoulos, J. (2024). Elasto-visco-plastic flows in benchmark geometries: I. 4 to 1 Planar Contraction. *Journal of Non-Newtonian Fluid Mechanics*, 105218.
- Mu, G., & Lyu, Q. (2022). Analysis of turbulent boundary layer jet flow pattern of Laval nozzle. *Journal of Physics: Conference Series*, 2228(1), 012038.
- Nguyen, H. N. G., Zhao, C.-F., Millet, O., & Selvadurai, A. (2021). Effects of surface roughness on liquid bridge capillarity and droplet wetting. *Powder Technology*, 378, 487–496.
- Ni, W., Sun, R., Liu, G., Ma, F., Fan, C., Xie, C., & Kang, Y. (2023). Numerical study on the nucleation law of water vapor condensation in Laval nozzle. *2023 3rd International Conference on New Energy and Power Engineering (ICNEPE)*, 264–268.
- Nie, W., Cha, X., Bao, Q., Peng, H., Xu, C., Zhang, S., Zhang, X., Ma, Q., Guo, C., Yi, S., & Jiang, C. (2023). Study on dust pollution suppression of mine wind-assisted spray device based on orthogonal test and CFD simulation. *Energy*, 263, 125590.
- Oh, J.-S., Kang, T., Ham, S., Lee, K.-S., Jang, Y.-J., Ryou, H.-S., & Ryu, J. (2019). Numerical analysis of aerodynamic characteristics of hyperloop system. *Energies*, 12(3), 518.
- Orellano, P., Reynoso, J., Quaranta, N., Bardach, A., & Ciapponi, A. (2020). Short-term exposure to particulate matter (PM₁₀ and PM_{2.5}), nitrogen dioxide (NO₂), and ozone (O₃) and all-cause and cause-specific mortality: Systematic review and meta-analysis. *Environment International*, 142, 105876.

- Pang, M., Zhang, T., Ji, X., Wu, J., & Song, S. (2022a). Measurement of the coefficient of seepage characteristics in pore-crushed coal bodies around gas extraction boreholes. *Energy*, 254, 124276.
- Pang, Y., Zhang, G., Gu, H., Yuan, D., Chen, Y., Wu, L., Yao, H., & Lin, C. (2022b). Flow and sound field characteristics of a Hartmann whistle with flow-sound-separation feature. *Applied Acoustics*, 201, 109103.
- Peng, H., Cheng, W., Guo, Y., Xu, C., Guo, C., Ma, Q., Liu, Z., & Yang, S. (2020a). Study on the spray field distribution of the roadway full-section water curtain device and its effect on the settlement of PM_{2.5}. *Process Safety and Environmental Protection*, 143, 101–113.
- Peng, H., Nie, W., Cai, P., Liu, Q., Liu, Z., & Yang, S. (2019). Development of a novel wind-assisted centralized spraying dedusting device for dust suppression in a fully mechanized mining face. *Environmental Science and Pollution Research*, 26(4), 3292–3307.
- Peng, H., Nie, W., Liu, Z., Xiu, Z., Yang, S., Xu, C., Ma, Q., & Guo, C. (2020b). Optimization of external spray negative-pressure mist-curtain dust suppression devices for roadheaders based on a multi-factor orthogonal experiment. *Journal of Cleaner Production*, 275, 123603.
- Peng, H., Nie, W., Zhang, S., Cheng, W., Liu, Q., Guo, C., Ma, Q., Zhou, Z., Xu, C., Hua, Y., & Zhang, H. (2022a). Research on negative pressure jet dust-removal water curtain technology for coal mine cleaner production. *Fuel*, 310, 122378.
- Peng, H., Nie, W., Zhang, X., Xu, C., Meng, X., Cheng, W., Liu, Q., & Hua, Y. (2022b). Research on the blowing-spraying synergistic dust removal technology for clean environment in large-scale mechanization coal mine. *Fuel*, 324, 124508.

- Ponasse, M., Dupre, V., Aurelle, Y., & Secq, A. (1998). Bubble formation by water release in nozzle—II. Influence of various parameters on bubble size. *Water Research*, 32(8), 2498–2506.
- Porozova, S., Kul'met'eva, V., Pozdeeva, T. Y., & Shokov, V. (2021). Role of nanopowder agglomerates in forming the structure and properties of ceramic materials. *Russian Journal of Non-Ferrous Metals*, 62, 226–232.
- Prabhu, S., Mohadeen, A., & others. (2011). Analysis of material behavior for friction in a nozzle for turbomachinery and high-speed vehicles. *Materials Sciences and Applications*, 2(10), 1485–1490.
- Ren, S., Jing, D., Ge, S., Chen, Y., & Chang, P. (2024). Investigate the optimum design of atomizing nozzles for coal dust suppression by using multifactor level response surface methodology. *Process Safety and Environmental Protection*, 190, 245–261.
- Rogala, M., Ferdynus, M., Gawdzińska, K., & Kochmański, P. (2021). The influence of different length aluminum foam filling on mechanical behavior of a square thin-walled column. *Materials*, 14(13), 3630.
- Rui, M., Chen, T., Zhou, M., Zheng, X., Hu, Y., Jin, D., Hu, Y., & Mao, Y. (2023). Study on physical prediction model of liquid film development and dry out point in helical tube. *Frontiers in Energy Research*, 11, 1224006.
- Sager, M. (2020). Urban soils and road dust- Civilization effects and metal pollution- A review. *Environments*, 7(11), 98.
- Salamah, T., Ramahi, A., Alamara, K., Juaidi, A., Abdallah, R., Abdelkareem, M. A., Amer, E.-C., & Olabi, A. G. (2022). Effect of dust and methods of cleaning on the performance of solar PV module for different climate regions: Comprehensive review. *Science of The Total Environment*, 827, 154050.

- Sarghini, F., & De Vivo, A. (2019). Effect of process parameters on microencapsulation of active compounds by co-flow air-assisted microfluidic extrusion. *CET Journal-Chemical Engineering Transactions*, 75.
- Sarver, E., Keleş, Ç., & Afrouz, S. G. (2021). Particle size and mineralogy distributions in respirable dust samples from 25 US underground coal mines. *International Journal of Coal Geology*, 247, 103851.
- Sellens, R. W. (2016). *A prediction of the droplet size and velocity distribution in a spray from first principles* [Master's Thesis]. University of Waterloo.
- Sescu, C. (2011). *Experimental and computational study on liquid atomization by slinger injector* [PhD Thesis]. University of Toledo.
- Shekarian, Y., Rahimi, E., Rezaee, M., Su, W.-C., & Roghanchi, P. (2021). Respirable coal mine dust: A review of respiratory deposition, regulations, and characterization. *Minerals*, 11(7), 696.
- Shi, Y., Wang, P., Liu, R., Tan, X., & Zhang, W. (2019). Numerical simulation and engineering application of coalbed water injection. *Mathematical Problems in Engineering*, 2019, 1–12.
- Sivakumar, A., Devi, S. R., Dhas, S. S. J., Kumar, R. M., Bharathi, K. K., & Dhas, S. M. B. (2020). Switchable phase transformation (orthorhombic–hexagonal) of potassium sulfate single crystal at ambient temperature by shock waves. *Crystal Growth & Design*, 20(11), 7111–7119.
- Sovani, S., Chou, E., Sojka, P., Gore, J., Eckerle, W., & Crofts, J. (2001). High pressure effervescent atomization: Effect of ambient pressure on spray cone angle. *Fuel*, 80(3), 427–435.

- Stania, M., Juras, G., Chmielewska, D., Polak, A., Kucio, C., & Krol, P. (2019). Extracorporeal shock wave therapy for Achilles tendinopathy. *BioMed Research International*, 2019(1), 3086910.
- Su, D., & Yan, W. (2020). Prediction of 3D size and shape descriptors of irregular granular particles from projected 2D images. *Acta Geotechnica*, 15(6), 1533–1555.
- Sun, L., Ge, S., Chen, X., & Liu, S. (2023). Study on atomization and dust reduction mechanisms of AEO-9-Charged solution. *Energies*, 16(6), 2800.
- Sun, W., Pen, J., Niu, L., Jiang, Z., Hu, W., & Pan, L. (2024). Sensitivity analysis of nucleation and droplet growth models in the prediction of natural gas non-equilibrium condensation in Laval nozzle. *Applied Thermal Engineering*, 236, 121562.
- Sundell, K. E., & Saylor, J. (2021). Two-dimensional quantitative comparison of density distributions in detrital geochronology and geochemistry. *Geochemistry, Geophysics, Geosystems*, 22(4), e2020GC009559.
- Svensson, D. N., Messing, I., & Barron, J. (2022). An investigation in laser diffraction soil particle size distribution analysis to obtain compatible results with sieve and pipette method. *Soil and Tillage Research*, 223, 105450.
- Tian, Z., Mu, X., Deji, J., Shaocheng, G., Xiangxi, M., Shuli, Z., & Xiaowei, Z. (2023). Influence of aerodynamic pressure on dust removal by supersonic siphon atomization. *Energy*, 282, 128727.
- Tian, Z., Xinsheng, M., Shaocheng, G., Sheng, L., Shuaishuai, R., Shuli, Z., & Shuang, T. (2024). Supersonic coaxial aerodynamic atomization dust removal technology. *Powder Technology*, 444, 120076.

- Topp, M. N., & Eisenklam, P. (1972). Industrial and medical uses of ultrasonic atomizers. *Ultrasonics*, 10(3), 127–133.
- Tsai, S. C., Luu, P., Childs, P., & Tsai, C. S. (1999). Ultrasound-modulated twin-fluid atomization of a liquid jet. *IEEE Transactions on Ultrasonics, Ferroelectrics, and Frequency Control*, 46(1), 139–146.
- Tsuge, H., Tezuka, Y., & Mitsudani, M. (2006). Bubble formation mechanism from downward nozzle- Effect of nozzle shape and operating parameters. *Chemical Engineering Science*, 61(10), 3290–3298.
- Turkyilmazoglu, M. (2020). Single phase nanofluids in fluid mechanics and their hydrodynamic linear stability analysis. *Computer Methods and Programs in Biomedicine*, 187, 105171.
- Vasanthakumar, P., Suseela Moorthi, A. K., & Rathakrishnan, E. (2024). Supersonic jet mixing in the presence of two annular co-flow streams. *International Journal of Turbo & Jet-Engines*, 41(1), 103–110.
- Villermaux, E., & Bossa, B. (2011). Drop fragmentation on impact. *Journal of Fluid Mechanics*, 668, 412–435.
- Wang, L., Zhai, Z., Chen, J., Chen, G., Gao, F., & Dong, J. (2024). Effect of inlet water vapor mass fraction on flow characteristics in Laval nozzle. *International Journal of Chemical Reactor Engineering*, 22(7), 743–757.
- Wang, P., Han, H., Liu, R., Gao, R., & Wu, G. (2020). Effect of outlet diameter on atomization characteristics and dust reduction performance of X-swirl pressure nozzle. *Process Safety and Environmental Protection*, 137, 340–351.
- Wang, P., Shi, Y., Zhang, L., & Li, Y. (2019a). Effect of structural parameters on atomization characteristics and dust reduction performance of internal-mixing air-

- assisted atomizer nozzle. *Process Safety and Environmental Protection*, 128, 316–328.
- Wang, P., Tian, C., Liu, R., & Wang, J. (2019b). Mathematical model for multivariate nonlinear prediction of SMD of X-type swirl pressure nozzles. *Process Safety and Environmental Protection*, 125, 228–237.
- Wang, X., Zheng, J., Rao, Y., Zhang, Y., Deng, Z., & Hu, X. (2023). Aerodynamic load analyses of less-emission HTS maglev train in evacuated tube transport system. *Frontiers in Energy Research*, 11, 1218137.
- Wang, Z., Zhou, W., Jiskani, I. M., Luo, H., Ao, Z., & Mvula, E. M. (2022). Annual dust pollution characteristics and its prevention and control for environmental protection in surface mines. *Science of The Total Environment*, 825, 153949.
- Watanawanyoo, P., Mochida, H., Furukawa, T., Nakamura, M., & Hirahara, H. (2012). Experimental study on the spray characteristics of an air assisted atomizer with internal mixing chamber. *European Journal of Scientific Research*, 84(4), 507–521.
- Wei, T., Gu, L., Zhou, M., Zhou, Y., Yang, H., & Li, M. (2021). Impact of shock-induced cavitation bubble collapse on the damage of cell membranes with different lipid peroxidation levels. *The Journal of Physical Chemistry B*, 125(25), 6912–6920.
- Wilcox, R. L., & Tate, R. W. (1965). Liquid atomization in a high intensity sound field. *AIChE Journal*, 11(1), 69–72.
- Wu, H., Zhang, F., Zhang, Z., & Hou, L. (2022). Atomization and droplet dynamics of a gas-liquid two-phase jet under different mass loading ratios. *International Journal of Multiphase Flow*, 151, 104043.
- Wu, W., Maffulli, N., Furia, J. P., Meindlhumer, L., Sternecker, K., Milz, S., & Schmitz, C. (2021). Exposure of zebra mussels to radial extracorporeal shock waves:

- Implications for treatment of fracture nonunions. *Journal of Orthopaedic Surgery and Research*, 16, 1–9.
- Xie, Z., Zhao, Z., Li, D., Li, F., Zhang, C., Huang, C., & Xiao, Y. (2023). Experimental study on the atomization characteristics and dust removal efficiency of a fan-shaped nozzle for purifying working environment. *Science of The Total Environment*, 894, 164994.
- Xu, C., Nie, W., Liu, Z., Peng, H., Yang, S., & Liu, Q. (2019). Multi-factor numerical simulation study on spray dust suppression device in coal mining process. *Energy*, 182, 544–558.
- Xu, C., Nie, W., Yang, S., Peng, H., Liu, Z., Ma, Q., Guo, C., & Liu, Q. (2020). Numerical simulation of the multi-index orthogonal experiments on the spray dust-settling devices. *Powder Technology*, 371, 217–230.
- Yamaguchi, T., & Anyoji, M. (2023). Numerical study on low-Reynolds compressible flows around Mars helicopter rotor blade airfoil. *Journal of Flow Control, Measurement & Visualization*, 11(2), 30–48.
- Yang, C., Bo, C., & Wanlu, J. (2016). Analysis and experiment on atomizing characteristics of supersonic nozzle based on Laval effect. *Transactions of the Chinese Society of Agricultural Engineering*, 32(19), 8.
- Yang, D., Jin, P., Qi, Y., & Cai, G. (2023). Effect of inlet mass flow rate oscillation on simplex swirl injector flow based on VOF-to-DPM model. *Acta Astronautica*, 211, 447–460.
- Yang, Q., Yang, S., Yu, H., & Wang, H. (2024). Impact of Laval nozzle structure on the flow characteristics of supersonic gas-solid two-phase flow. *Powder Technology*, 119657.

- Yang, S., Liu, J., Bi, X., Ning, Y., Qiao, S., Yu, Q., & Zhang, J. (2020). Risks related to heavy metal pollution in urban construction dust fall of fast-developing Chinese cities. *Ecotoxicology and Environmental Safety*, 197, 110628.
- Yang, S., Nie, W., Lv, S., Liu, Z., Peng, H., Ma, X., Cai, P., & Xu, C. (2019). Effects of spraying pressure and installation angle of nozzles on atomization characteristics of external spraying system at a fully-mechanized mining face. *Powder Technology*, 343, 754–764.
- Yao, H., Wang, H., Li, Y., & Jin, L. (2020). Three-dimensional spatial and temporal distributions of dust in roadway tunneling. *International Journal of Coal Science & Technology*, 7(1), 88–96.
- Yin, W., Zhou, G., & Gao, D. (2019). Simulation analysis and engineering application of distribution characteristics about multi-stage atomization field for cutting dust in fully mechanized mining face. *Advanced Powder Technology*, 30(11), 2600–2615.
- Yu, H., Cheng, W., Xie, Y., & Peng, H. (2020). Spray dedusting scheme under hybrid ventilation at a fully mechanized excavation face. *Environmental Science and Pollution Research*, 27(8), 7851–7871.
- Yu, Y., Mao, Y., Yu, T., Yang, Y., Xu, S., & Liang, S. (2023). The influence of external flow field on the flow separation of overexpanded single-expansion ramp nozzle. *Aerospace*, 10(11), 958.
- Yuan, X., Wang, P., Wu, G., Chen, S., & Li, S. (2022). Investigation of optimal selection and atomizing characteristics of nozzles for ultrasonic dry-mist dust suppression system. *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects*, 44(3), 7046–7061.

- Yuan, Y., Liu, S., Castro, R., & Pan, X. (2012). PM 2.5 Monitoring and mitigation in the cities of China. *Environmental Science & Technology*, 46(7), 3627–3628.
- Zhai, G., Zhang, W., Li, Y., Lu, X., & Hu, W. (2020). Experimental research and numerical simulation of ejector precipitator in a fully mechanized mining face. *Arabian Journal for Science and Engineering*, 45(11), 9815–9833.
- Zhang, G., Zhou, G., Song, S., Zhang, L., & Sun, B. (2020a). CFD investigation on dust dispersion pollution of down/upwind coal cutting and relevant countermeasures for spraying dustfall in fully mechanized mining face. *Advanced Powder Technology*, 31(8), 3177–3190.
- Zhang, T., Jing, D., Ge, S., Wang, J., & Chen, X. (2020b). Supersonic antigravity aerodynamic atomization dusting nozzle based on the Laval nozzle and probe jet. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, 42(6), 335.
- Zhang, T., Jing, D., Ge, S., Wang, J., Chen, X., & Ren, S. (2020c). Dust removal characteristics of a supersonic antigravity siphon atomization nozzle. *Advances in Mechanical Engineering*, 12(12), 168781402097768.
- Zhao, W., Li, H., Kou, Z., Liao, P., He, X., Shao, G., & He, Y. (2020). Epidemiological characteristics of pneumoconiosis in Gansu Province from 2010 to 2018. *Zhonghua Lao Dong Wei Sheng Zhi Ye Bing Za Zhi= Zhonghua Laodong Weisheng Zhiyebing Zazhi= Chinese Journal of Industrial Hygiene and Occupational Diseases*, 38(10), 746–749.
- Zhou, G., Zhang, L., Liu, R., Sun, B., Kong, Y., & Huang, Z. (2021). Numerical simulation investigation for the pollution characteristics of dust particles in the fully mechanized mining face under different air humidity conditions. *Journal of Environmental Chemical Engineering*, 9(6), 106861.

- Zhou, Q., & Qin, B. (2021). Coal dust suppression based on water mediums: A review of technologies and influencing factors. *Fuel*, 302, 121196.
- Zhu, X., Wang, H., Wang, D., Xu, C., Zhou, W., & Zhu, Y. (2020). Improved foam application at the tunnel face with large ventilation volume and low pressure supplied water. *Tunnelling and Underground Space Technology*, 95, 103139.
- Zhu, X., Yu, W., Li, F., Liu, C., Ma, J., Yan, J., Wang, Y., & Tian, R. (2021). Spatio-temporal distribution and source identification of heavy metals in particle size fractions of road dust from a typical industrial district. *Science of the Total Environment*, 780, 146357.

APPENDICES

Appendix A: Journal Publications

1. Shanshan Tang, Mohd Danial Ibrahim, Andrew Ragai Henry Rigit, Wei Zhang, & Chaokun Wei. (2024). A Systematic Review of Dust Suppression Methods by Experiment Based on Intelligent Technology in the Coal Mines. *Journal of Electrical Systems*, 20(3), 882-893. <https://doi.org/10.52783/jes.3019>.
2. Shanshan Tang, Mohd Danial Ibrahim, Andrew Ragai Henry Rigit, Wei Zhang, & Chaokun Wei. (2025). A Systematic Review of Coal Mine Dust Suppression Methods Based on Numerical Simulations and Experimental Investigations. *International Journal of Engineering and Technology Innovation*. <https://doi.org/10.46604/ijeti.2024.14818>. (EI) (Accepted)
3. Shanshan Tang, Mohd Danial Ibrahim, Andrew Ragai Henry Rigit, Wei Zhang, & Chaokun Wei. (2024). Analysis of Atomization Performance of Linear Laval Nozzle under Varied Water Pressures Based on VOF and DPM Models. *International Journal of Engineering and Technology Innovation*, 14(4), 335–354. <https://doi.org/10.46604/ijeti.2024.13615>. (EI)
4. Shanshan Tang, Mohd Danial Ibrahim, Andrew Ragai Henry Rigit, Wei Zhang, & Chaokun Wei. (2024). Experimental analysis of the effect of water pressure on the atomization performance of a Linear Laval nozzle and comparison with numerical analysis. *revista Matéria*, 29(4), e20240460. <https://doi.org/10.1590/1517-7076-RMAT-2024-0460>. (SCIE)