Indian Journal of Ecology (2024) 51(2): 401-409
DOI: https://doi.org/10.55362/IJE/2024/4250

Manuscript Number: 4250
NAAS Rating: 5.38

Performance Characterization of Spray Nozzles Based on CFD Simulation

Shaik Nasreen, Arun Kumar and T.P. Singh

Department of Farm Machinery and Power Engineering
G. B. Pant University of Agriculture and Technology, Pantnagar-263 145, India
E-mail: nasreensk16@gmail.com

***Abstract*:** Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems related to fluid flow. The study conducted a comprehensive analysis of the design and operational parameters of plant protection equipment, with a focus on factors such as operating pressure, height, nozzle type, fluid density, and viscosity. The investigation utilized Computational Fluid Dynamics (CFD) and the ANSYS FLUENT software to simulate the behavior of lime juice and distilled water mixtures across various proportions. Both hollow cone and flat fan assemblies exhibited increased discharge rates (0.223 to 0.254 L/min for flat fan, 0.222 to 0.359 L/min for hollow cone) and wider swath widths (372 to 558 mm for hollow cone, 254 to 385 mm for flat fan) as pressure increased from 1.5 to 2.5 kg/cm^2. Droplet size decreased with pressure for flat fan (284-263 microns) and hollow cone (336-278 microns), while spray angles increased 50 to 69^0 for hollow cone and 35 to 51^0 for flat fan with pressure 0 .5 to 2.5 kg/cm^2 and height of 400 to 500 mm. The results emphasize the need to balance environmental impact and biological efficacy in optimizing plant protection equipment performance.

Keywords: ANSYS FLUENT, CFD, Hallow cone, Flat fan

India is the fifth-largest global producer of agrochemicals and fourth largest exporter after United States, Japan and China, with the industry ranking 13th in Asia. The global pesticide consumption in 2019 was approximately 4.19 million metric tons, where China was by far the largest pesticide-consuming country (1.76 million metric tons), followed by the United States (408 thousand tons), Brazil (377 thousand tons), and Argentina (204 thousand tons) (Fernández 2021). The per hectare consumption of pesticides in India is amongst the lowest in the world and stands at 0.6 kg/ha against 5-7 kg/ha in the United Kingdom and 13 kg/ha in China (Pushpendra and Nitish Rattan 2018). Paddy accounts for 26-28% of agrochemicals, followed by cotton (18-20%). The states Maharashtra, Punjab, Uttar Pradesh, Telengana, Haryana and West Bengal, account for over 70% of agrochemicals used in India (FCCI 2021).

Pest management is crucial to reduce economic damage caused by pests, and various pest control methods include chemical, biological, mechanical, physical, legal, and cultural approaches. Chemical control is the only method for controlling pests, insects, and weeds, and can be applied through spraying or dusting. Droplet size is an important factor affecting drift, as good coverage is essential for agrochemical that comes into contact with disease-causing organisms. Fine- to medium-sized droplets are preferred for insecticides and fungicides due to better coverage. The distribution of spray coverage and deposition on the crop surface depends on factors such as droplet size, droplet density, Physio-chemical properties of spray liquid, crop surface characteristics, and meteorological conditions. Computational Fluid Dynamics (CFD) modelling can help overcome these challenges and provide more robust and accurate results.

MATERIAL AND METHODS

Liquid properties: In this study a mixture of organic lime juice and distilled water in different proportions was used. Six samples have been used with ambient temperature at 22ºC. The sample is pure organic lime juice (L100) water (W0), L10 is 10% of organic citric acid mixed with 90% water, L20, L30 and, L40, L50 to determine viscosity and density of liquid.

Density of a lime juice: The Pycnometer is cleaned, dried, and weight is determined (w_1). A specific gravity bottle is filled with distilled water (w_2), and the temperature of the mixture is. The weight is recorded (w_3) and cleaned.

$$\rho = \frac{m}{v}$$

Where-ρ= density in gm/ml, m= mass (gm), v=volume (ml)

The density of lime juice (ρ_2) = mass of liquid / mass of equal volume of distilled water

$$\frac{\rho 2}{\rho 1}=\frac{W3-W1}{W2-W1}$$

The density of distilled water at room temperature is ρ_1 0.997 gm/ml standard.

Viscosity of lime juice: The Ostwald viscometer method is used to measure the viscosity coefficient based on Poiseuille's law. The rate of flow of liquid through a capillary tube with viscosity coefficient, η, can be expressed as

$$\eta = \frac{\pi p r^4}{8 v l}$$

Where, v= vol. of liquid (in ml), t= flow time (in sec.) through capillary

r= radius of the capillary (in cm), l= length of the capillary (in cm)

P= hydrostatic pressure (in dyne/sq.cm), η = viscosity coefficient (in poise)

The hydrostatic pressure of a liquid is determined by P = ρgh, where h is the column height and ρ is the liquid's density. The viscosity coefficients η1 and η2 are used to study the liquids' densities and flow times through the same capillary.

$$\frac{\eta 1}{\eta 2} = \frac{\rho 1 t 1}{\rho 2 t 2}$$

The process involves creating various water-lime juice mixtures, filling a viscometer with the mixture, sucking it through a capillary tube, and recording the time of flows. The viscosity of distilled water at 20°C is 1.0020 cP.

Dimensional modelling of existing self-propelled intra canopy boom sprayer: The self-propelled intra boom sprayer was designed using Solid Works 2016 to understand fluid flow dynamics from the spray tank to nozzles (Fig. 1). The model includes components like flat fan and hollow cone nozzles, which were analysed using ANSYS work. Cylindrical spray tank with capacity of 118 litres with height of 600mm and a diameter of 500mm was used as a reservoir for chemical solution during spraying. The nozzles are used for specific insecticide and herbicide applications, with fan angles of 65°, 80°, and 110°. In current study flat fan nozzle TP8001VK and a hollow cone tip, TXA8002VK (Fig. 2) which were converted to 3-Dimension models and attached to the sprayer boom at 50cm spacing were used as shown in (Fig. 3).

The detailed front and side views of the model are provided.

Description of computational fluid dynamics in fluid flow simulation: Computational Fluid Dynamics (CFD) is a crucial tool in engineering design and analysis, particularly for thermal applications. Its key requirement is the ability to simulate flows along nozzles, which can be challenging due to features like eddy location, is more accurate than empirical methods used in design. Accurate simulation of flow through the nozzle is essential for predicting pressure and velocity patterns. CFD deals with the dynamic behavior of fluids and is governed by partial differential equations, which are often difficult to obtain analytically. CFD provides qualitative and quantitative solutions for predicting fluid flows through numerical methods, mathematical modeling, and software tools like solvers, pre- and post-processing tools. It enables engineers and researchers to perform numerical experiments, such as CFD simulation. ANSYS workbench is a graphical user interface that allows users to use these tools from a single place, assessing pre-processor, solver, and post-processor tools.

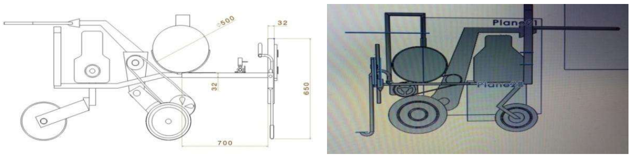


Fig. 1. Dimensions of the designed parts

CFD analysis by finite elemental method in ANSYS FLUENT14.0 Version: The steps were as follows

1. The tank and nozzle assembly model is evaluated using Solid Works 2016 design, and the research area employs ANSYS Workbench 14 version, which includes over 40 tools. The present of study area use ANSYS CFD FLUENT.
2. Double click the "Fluid flow FLUENT", Go to "Geometry" and right click "Import geometry" then Go to "Browse" and select the file modeled in solid works.
3. The text emphasizes the importance of proper geometry meshing in FEA Analysis, which is crucial for accurate results. The software solves each element for converging to a specific solution, with a higher number of elements resulting in better results.
4. The polygonal mesh is used for tank and nozzle assembly, and the model is divided into equal parts with 80118 nodes and 17324 elements**.**
5. **"**Name selection" Input is given as Tank; Output is given as nozzle tip i.e., hallow cone and flat fan nozzles at 400 mm, 500mm height.

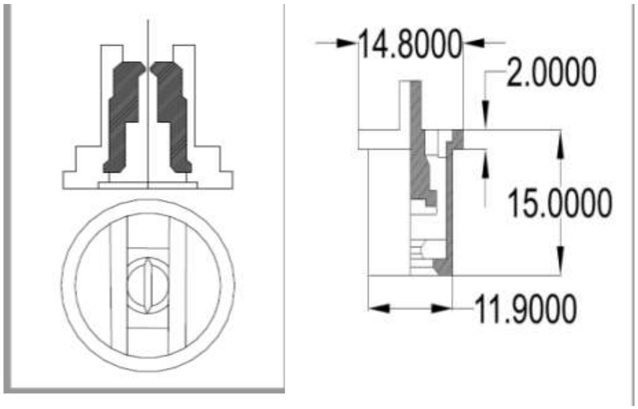


Fig. 2. 2-D views of spray nozzle flat fan and hollow cone

General settings of ANSYS CFD analysis: The Solver was taken as pressure based, time as steady and space as 3D. Velocity formulation as absolute, to analyze flow from tank to nozzle in ANSYS CFD, follow these steps: upload the model, select "General" option, click "pressure based" and "Absolute" under velocity, and select the standard k-ε turbulent model. This model provides results on pressure volume, velocity, eddy viscosity contours, discharge rate, swath, spray angle, and pressures for flat fan and hollow cone nozzles.

Standard K-Epsilon equation used in the ANSYS for the simulation of CFD: FLUENT's standard k-ε turbulent model is the default turbulence model, derived for flows with high Reynolds numbers. It solves a second transport equation for dissipation rate and is suitable for flows with nearly iso-tropic turbulence and local equilibrium energy cascade. This model was used for simulation of turbulent flow through a spray nozzle, based on kinetic energy (k) and dissipation rate (ε). Based on fully turbulent conditions, this model is reliable, accurate, and simple, with fast convergence, making it widely used for flow problem study.

Governing Equations for *k–ε* turbulent model: For turbulent kinetic energy *k*

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right] + 2\mu_t E_{ij}E_{ij} - \rho\varepsilon$$

For dissipation **ε**

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t E_{ij}E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

Where u_i represents velocity component in corresponding direction

E_{ij} represents component of rate of deformation, μ_t represents eddy viscosity.

Where σ_k, σε, $C_{1\varepsilon}$ and $C_{2\varepsilon}$ are constants .The values have been arrived at by numerous iterations of data fitting for a wide range of turbulent flows. They are as follows:

C_μ=0.09 σ_k =1.00 σ_ε=1.30 $C_{1\varepsilon}$=1.44 $C_2\varepsilon$ =1.92

The eddy viscosity is μ_t computed by combining *k* and ε as follows:

$$\mu_t = \rho\, C_\mu k^2/\varepsilon$$

The material panel displays the default material as nitrogen, oxygen, and water. To create a simulation material, double click the mixture template. The solid Galvanized iron was used for tank, and properties were calculated in ANSYS work bench. The fluid was lime water (30%) and distilled water (70%), and the properties of the fluid were calculated. The model is then defined with set boundary conditions given (Table 1) and equation used for analysis.

Click on "Solution" Initialize the "Hybrid initialization". Write the number of iterations and then click "Run calculation". We have given 10 iteration values in the initialization of solution and it is a scalar "0" and scalar "1" value ranging from 1.00000e+00 to 499987e+00. Results can be obtained from the graphic display and report in FLUENT. Results can be displayed in terms of contour, velocity vector, viscosity, eddy viscosity, pressure, volume rendering, and eddy viscosity rendering in the tank and nozzles.

Experimental parameters: This is given in Table 2.

Spray angle and swath width: The spray angle of the nozzle was calculated with the spray swath width and the height of nozzle. It is described as the angle subtended at the final orifice.

The spray angle of the nozzle was calculated using the formula

$$W=2h\tan\frac{\theta}{2}$$

Where, W is the width of spray cone, mm, h is the height of the spray, mm, θ is the spray angle in degrees

Discharge rate: The amount of liquid ejects from the nozzle in unit time, represented in litre per minute. The relationship between pressure and discharge and pressure was studied.

RESULTS AND DISCUSSION

The properties of liquid calculated in laboratory by using viscometer and pycnometer: The analysis shows (Table 3)

Table 1. Boundary conditions used in ANSYS work bench

Boundary conditions parameters	Values
Fluid	Lime (30%) + Distilled water (70%)
Pressure	1.5 kg/cm^2(147099.75 Pa) 2.5 kg/cm^2(245166 Pa)
Density	1006.06 kg/m^3
Viscosity	1.19 Centi poise
Material of tank	GI sheet
Nozzle material	Ceramics
Domain	FFF

Table 2. Experimental plan for CFD simulation from tank to the nozzle

Independent parameters		Dependent parameters
Nozzles	1. Flat fan nozzle 2. Hallowcone nozzle	1. Eddy viscosity 2. Pressure contour
Height	1. 400 mm 2. 500 mm	3. Density contour 4. Pressure volume rendering
Pressure	1.5 Kg/cm^2(147099.75 Pa) 2.5 Kg/cm^2(245166 Pa)	5. Eddy viscosity rendering 6. Density volume rendering 7. Discharge rate
Liquid	L30W70	8. Swath width
density	1006.60kg/m^3	9. Droplet size
viscosity	1.19 Cp (0.0011 Pa sec)	10. Spray Angle

an increase in viscosity and density with an increase in lime juice concentration.

Run calculation: The graphs display the variation in velocity and k-ε turbulent equation during analysis iterations. A scaled residual factor of continuity is used, with values ranging from 1e+02 to 1e+16, indicating the path acquired during operation. This allows for the calculation of any factor value at any time interval for as shown in Figure 4 (a) Flat fan nozzle at 400 mm height and 1.5 kg/cm^2 pressure, (b) Flat fan nozzle at 500 mm height and 1.5 kg/cm^2 pressure, (c) Flat fan nozzle at 400 mm height and 2.5 kg/cm^2 pressure, (d) Flat fan nozzle at 500 mm height and 2.5 kg/cm^2 pressure, (e)Hallow cone nozzle 400 mm height and 1.5 kg/cm^2 pressure, (f) Hollow cone nozzle 500 mm height and 1.5 kg/cm^2 pressure, (g) Hollow cone nozzle 400 mm height and 2.5 kg/cm^2 and (h) Hollow cone nozzle 500 mm height and 2.5 kg/cm^2 pressure.

Post Processing of ANSYS FLUENT Pictorial Charts

Pressure contours of fluid flow from tank (inlet) to the nozzle (outlet): The chart shows an increase in pressure from the tank to the nozzle outlet due to the flow of fluid. The maximum and minimum pressure values are represented in the color chart. The pressure applied at the boundary condition was 1.5 kg/cm^2 and 2.5 kg/cm^2. The pressure range for flat fan nozzle was 6.26 e+006 Pa (0.63 kg/cm^2) to 2.463e+005 Pa (2.5 kg/cm^2) from (Fig5 (a), (b), (c), (d)). For hollow cone nozzle, the values ranged from 0 Pa to 2.504 e+005 Pa (2.5 kg/cm^2) from (Fig. 5 (e), (f), (g), (h)).

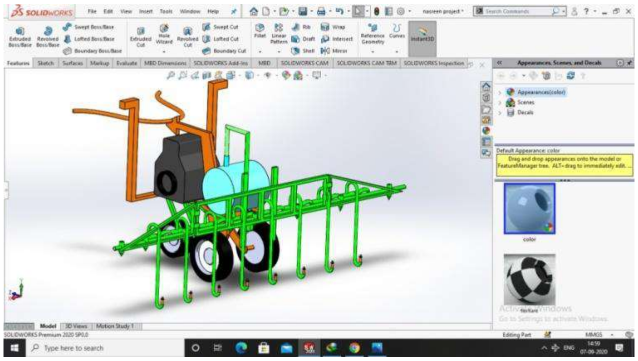
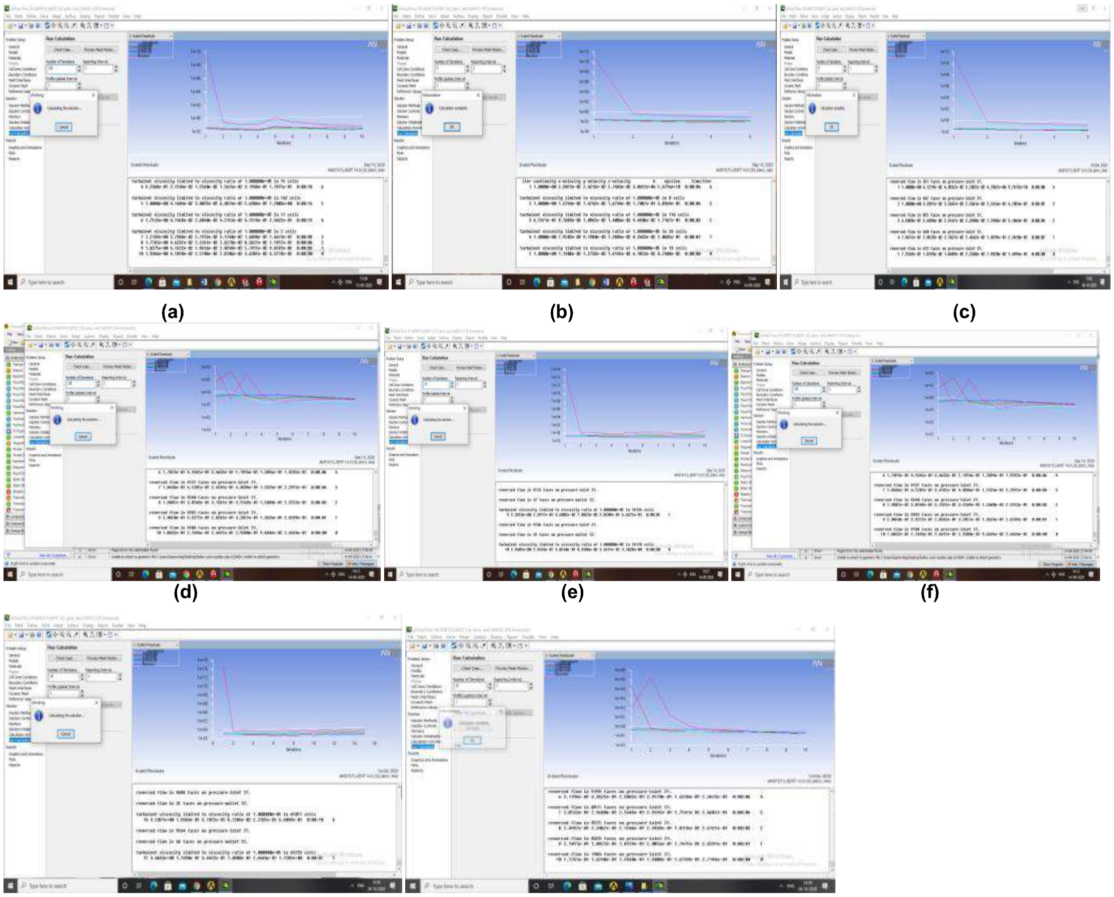


Fig. 3. 3-Dimensional assembled model of self- propelled intra canopy boom sprayer



(a) (b) (c)

(d) (e) (f)

(g) (h)

Fig. 4. Initialization of the analysis of ANSYS FLUENT

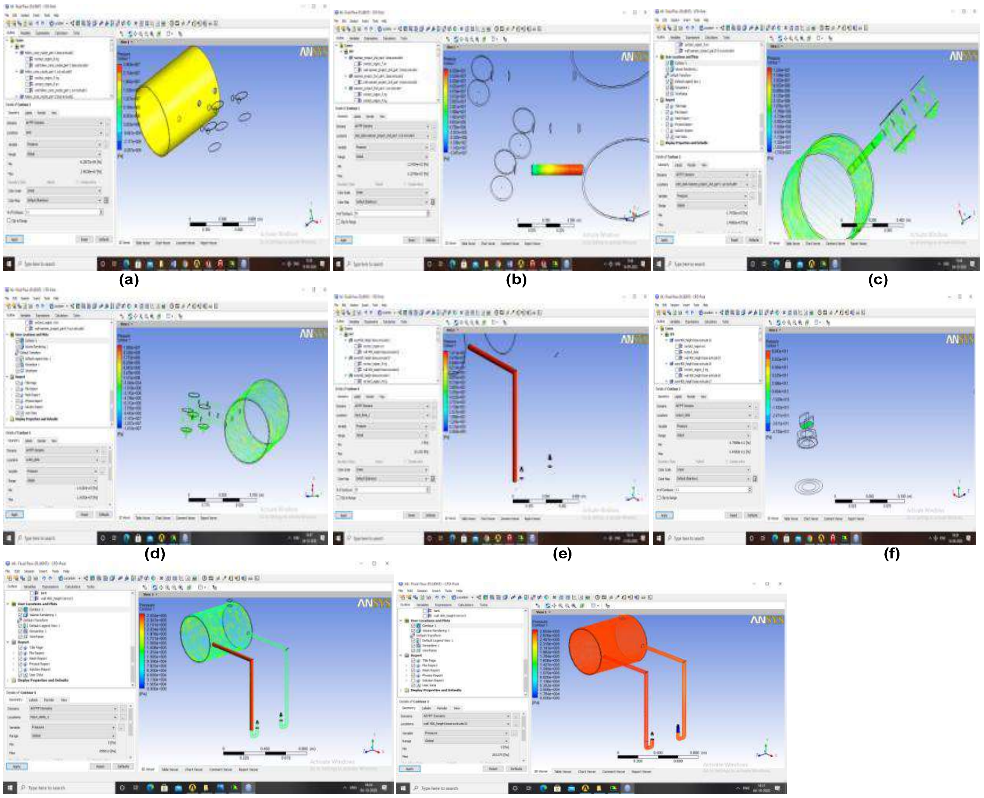

(a) (b) (c)
(d) (e) (f)
(g) (h)

Fig. 5. Pressure contour of the analysis of ANSYS FLUENT from tank to the nozzles

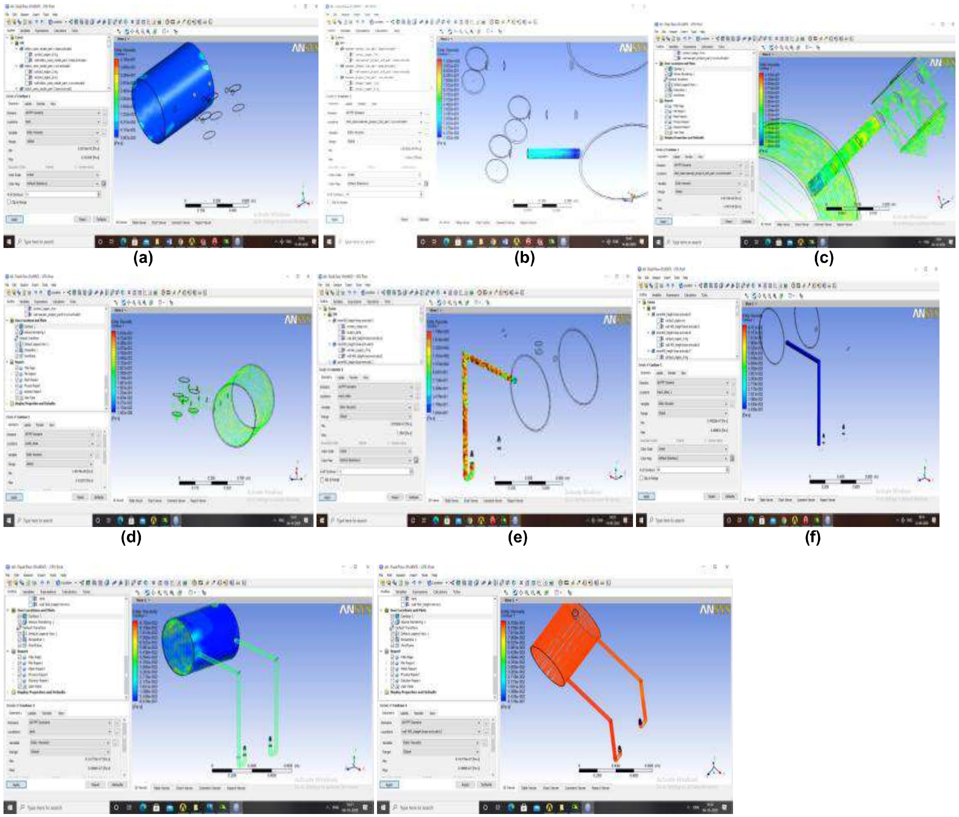

(a) (b) (c)
(d) (e) (f)
(g) (h)

Fig. 6. Eddy viscosity contour of the analysis of ANSYS FLUENT from tank to the nozzles

Eddy viscosity contours of fluid flow from tank (inlet) to the nozzle (outlet): The k-ε turbulent model indicates that the generation of eddies in the tank leads to high viscosity of fluid flow through the nozzles, a coefficient relating to average shear stress and fluid density within a turbulent flow as shown in Figure 6.

Density contours of fluid flow from tank (inlet) to the nozzle (outlet): The density of the fluid flow from tank (inlet) to the nozzles (outlet) was same in all the cases (1.225 kg/m^3) (Fig. 7).

Pressure volume rendering contours of fluid flow from tank (inlet) to the nozzle (outlet): Volume rendering is a technique used to visualize densely spaced three-dimensional data as clouds of various opacity and colors, allowing us to trace the discharge rate, spray width, and droplet size of the flow from the outlet (Fig. 8).

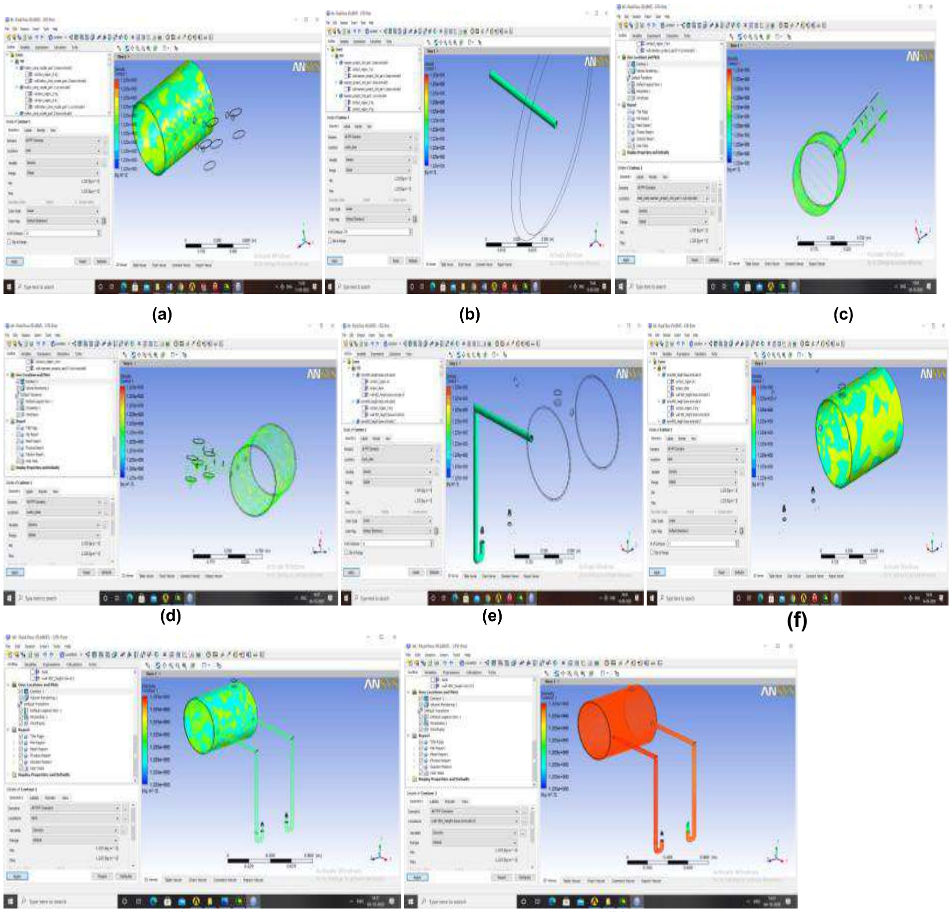
Eddy viscosity volume rendering contours of fluid flow from tank (inlet) to the nozzle (outlet): The eddy viscosity volume showed that viscosity of liquid flow from tank (inlet) to the nozzle (outlet) increases due to the turbulence created by the k–ε turbulent equation (Fig. 9).

Table 3. Properties of liquid calculated in laboratory by using Viscometer and Pycnometer

Lime juice + distilled water (%)	Viscosity (cP)	Density (kg/m^3)
L100W0	1.9	1019.32
L10W90	1.03	995.33
L20W80	1.16	998.72
L30W70	1.19	1006.60
L40W40	1.24	1010.56
L50W50	1.5	1017.35

Table 4. Effect of swath width of flat fan and hollow cone nozzles at different pressures and height

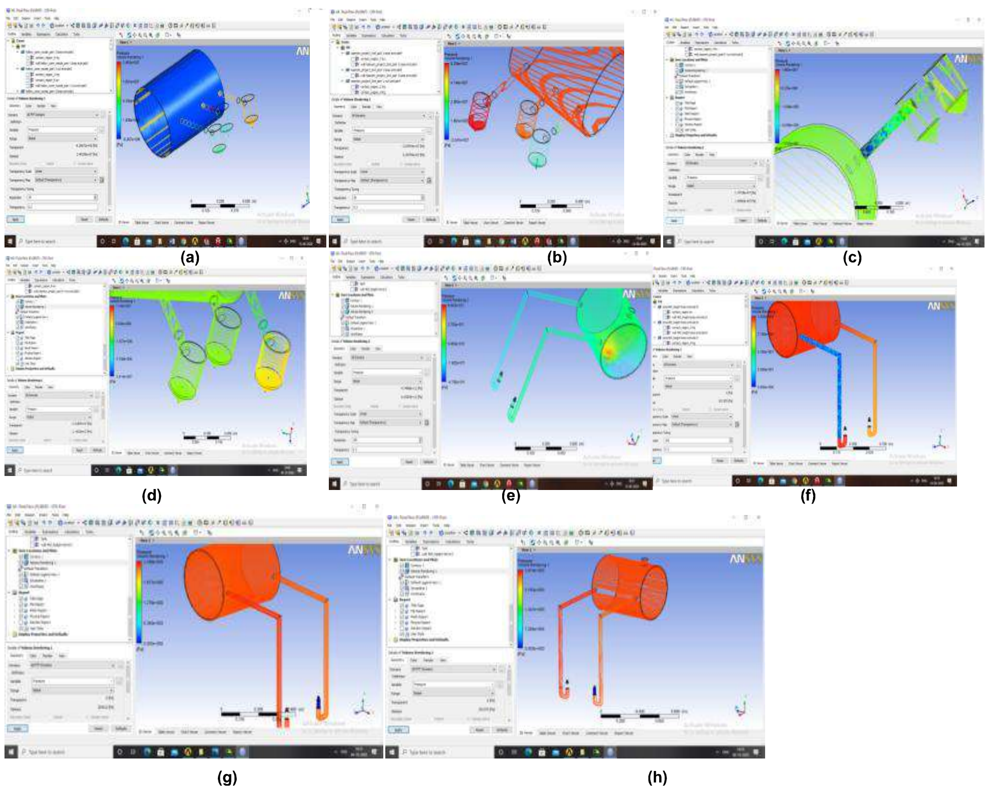
Nozzle type	Operating pressure (kg/cm^2)	Swath width at different heights (mm)	
		400	500
Flat fan nozzle	1.5	254	275
	2.5	356	385
Hallow cone nozzle	1.5	372	384
	2.5	548	558


(a) (b) (c)

(d) (e) (f)

(g) (h)

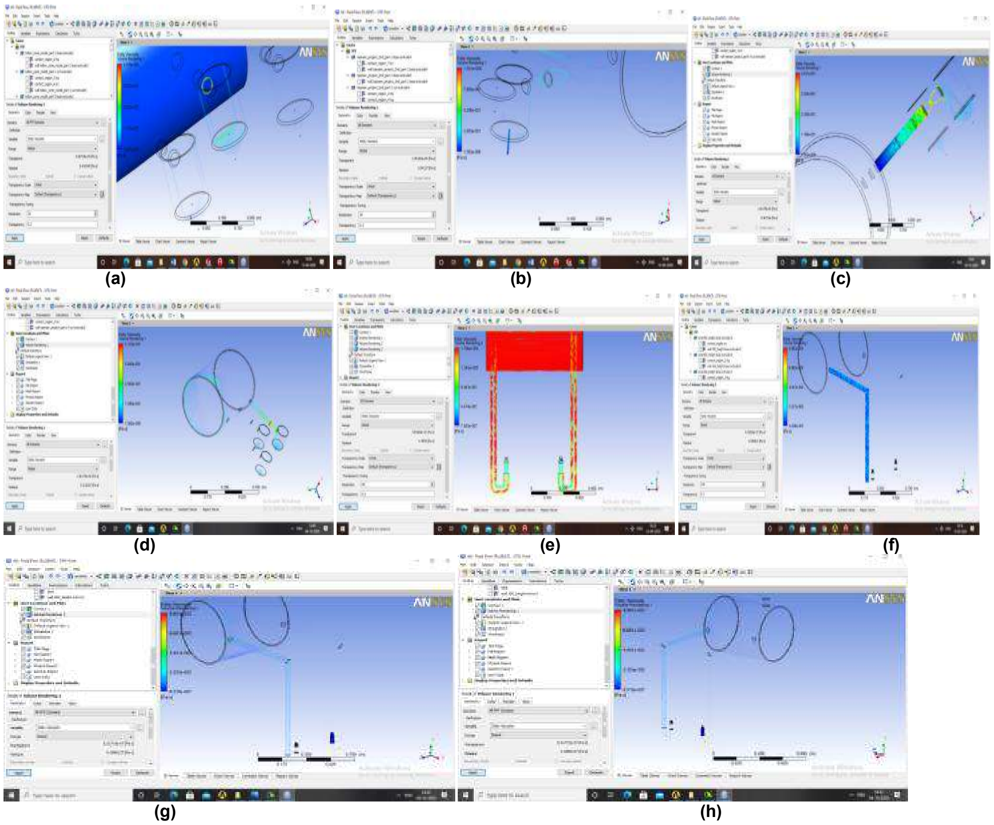
Fig. 7. Density contour of the analysis of ANSYS FLUENT from tank to the nozzles


(a) (b) (c)

(d) (e) (f)

(g) (h)

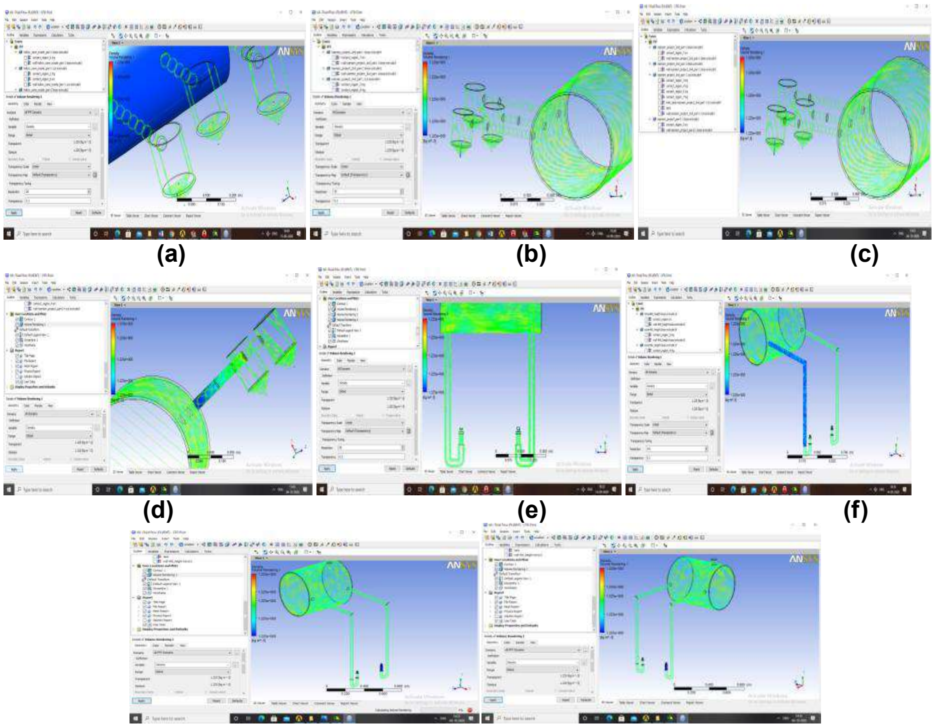
Fig. 8. Pressure volume rendering contour of the analysis of ANSYS FLUENT from tank to the nozzles


(a) (b) (c)

(d) (e) (f)

(g) (h)

Fig. 9. Eddy viscosity volume rendering contour of the analysis of ANSYS FLUENT from tank to the nozzle


(a) (b) (c)

(d) (e) (f)

(g) (h)

Fig. 10. Density volume rendering contour of the analysis of ANSYS FLUENT from tank to the nozzles

Density volume rendering contours of fluid flow from tank (inlet) to the nozzle (outlet): The density of the fluid flow from tank (inlet) to the nozzles (outlet) is same in all the cases (Fig. 10).

The parameter of nozzle obtained in ANSYS CFD FLUENT analysis is given in Table 4 to 7

Table 5. Effect of operating pressure on discharge rate of nozzles

Nozzle type	Operating pressure (kg/cm^2)	Discharge rate (lit/min)
Flat fan nozzle	1.5	0.223
	2.5	0.254
Hollow cone nozzle	1.5	0.222
	2.5	0.359

Table 6. Effect of operating pressure and height on spray angle of nozzles

Nozzle type	Operating pressure (Kg/cm^2)	Spray angle (degrees) at different heights	
		400 (mm)	500 (mm)
Flat fan nozzle	1.5	35	38
	2.5	48	51
Hollow cone nozzle	1.5	50	51.2
	2.5	68	69

Table 7. Effect of operating pressure on droplet size

Nozzle type	Operating pressure (Kg/cm^2)	Droplet size (microns)
Flat fan nozzle	1.5	284
	2.5	263
Hallow cone nozzle	1.5	336
	2.5	278

CONCLUSION

The study used 3D visualizations to analyze the influence of sprayer design on performance. Computational Fluid Dynamics (CFD) simulations proved valuable for optimizing settings, revealing that pressure, viscosity, and nozzle arrangements significantly affected spray conditions. The model offers potential for extended analysis, including factors like wind speeds and directions, to enhance understanding of sprayer design impact.

REFERENCES

American Society of Agricultural and Biological Engineering. 2009. Spray Nozzle Classification by Droplet Spectra, USA, and ANSI/ASAE **572**.1.

Babasaheb G and Singh KO 2015. Comparative performance of tractor operated boom type field sprayer on cotton crop. *International Journal of Agricultural Engineering* **8**(1): 85-91.

Badules J, Vidal M, Bone A, Gil E and Javier Garcia-Ramos F 2020. Biphasic computational fluid dynamics modelling of the mixture in an agricultural sprayer tank, *Molecules* **25**: 1870; DOI:10.3390/molecules 25081870.

Balachand CH and Shridar B 2016. Spray pattern analysis system for motorized knapsack mist blower. *International Journal of Agricultural Sciences***.** ISSN: 0975-3710& E-ISSN: 0975-9107, Volume 8, Issue 53, 2016 pp.-2800-2802.

Czaczyk Z 2012. Spray classification for selected flat fan nozzles. *Journal of Plant Protection Research.* DOI: 10.2478/v10045-012-0027-2

Dekeyser D, Duga AT, Verboven P, Endalew AM, Hendrickx N and Nuyttens D 2012. Assessment of orchard sprayer using laboratory experiment and computational fluid dynamics modelling. doi.org/10.1016/j.biosystemseng.2012.11.013.

Fariqin AB 2015. *Evaluation of granular fertilizer boom sprayer performance via computer simulation.* Thesis University TUN Hussien ONN Malaysia.

Fernandez L 2021. Global pesticide use by country Statista. Available online at:https://www.statista.com/statistics/ 1263069/ global-pesticide-use-by-country/(accessed May31, 2022).

Foque D and Nuyttens D 2011. Effects of nozzle type and spray angle on spray deposition in ivy pot plant. *Pest Management Sci*ence **67**: 199-208.

Fritz BK, Hoffmann WC, Bagley WE, Kruger GR, Czaczyk Z and Henry RS 2014. Measuring droplet size of agricultural spray nozzles-measuring distance and air speed effect. *Atomization and Sprays* **24**(9): 747-760.

Hazim M and Reshaq A 2017. Performance of knapsack sprayer: Effect of technological parameters on nanoparticles spray distribution. *International Journal of Engineering Trends and Technology* **46**(4): 199-207.

Hitesh B, Khura TK, Mani I, Kushwaha HL, Lande SD and Sarkar SK 2019. Development of liquid urea ammonium nitrate fertilizer foliar application system for enhanced nutrient use efficiency. *Indian Journal of Agricultural Sciences* **89**(1): 22-27.

Hong WS, Zhao L and Zhu H 2017. CFD simulation of airflow inside tree canopies discharged from air-assisted sprayers. *Article in Computers and Electronics in Agriculture* July 2017 DOI: 10.1016/j.compag.2017.07.011.

Ishak HHM, Ismail F, CheMat S, Abdullah ZM, Abdul Aziz MS and Idroas YM 2019. Numerical analysis of nozzle flow and spray characteristics from different nozzles using diesel and biofuel blends. *Energies* **12**: 281; DOI:10.3390/en12020281.

Pushpendra K and Nitish RB 2018. Status and use of pesticides in forage crops in India. *Journal of Pesticide Sciences* **43**(4): 225-232.

Received 22 August, 2023; Accepted 11 February, 2024