Analysis of Avgas Fuel Spraying Schemes Using the ANSYS Application Approach

Dimas Endrawan Putra, Nasrul Ilminnafik*, M Fahrur Rozy Hentihu, Muh. Nurkoyim Kustanto, Danang Yudistiro, Skriptyan N.H. Syuhri

Graduate Faculty of Mechanical Engineering, Jember State University 68121, Jember, Indonesia
**nasrul.teknik@unej.ac.id*

Article history:
Received: 14 December 2023 / Received in revised form: 17 March 2024 / Accepted: 9 April 2024
Available online 23 May 2024

ABSTRACT

Avgas fuel consumption on Cessna trainer aircraft is very high. However, there has been little research regarding spray impacts in Cessna aircraft engines. The phenomenon of avgas spray colliding with the cylinder wall may occur during fuel injection, resulting in a changed spray radius and height, which will affect the mixing of fuel and air. In several aspects, this affects engine performance and exhaust emissions on Cessna aircraft. This research aims to determine and study the phenomenon of spray impact on avgas-fueled aircraft engines. The fuel spray in the study occurred in the combustion chamber using pressure from a fuel hand pump whose pressure was supplied from a compressor with a pressure adjusted to the original pressure on the aircraft, namely 2 Bar (30 Psi). The experiments in this research used a high-speed camera system to study the phenomenon of avgas spray on walls to get better spray distribution. The results of this research were processed using the CFD application. The result of this research is that the greater the pressure, the more concentrated the resulting jet will be on the jet wall so that the atomization of the fuel jet will be dispersed. When the burst occurs, a change of 3.80e+00 occurs compared to other burst pressures.

Keywords: *Avgas, ANSYS, CFD, impigment, spray*

I. Introduction

Avgas is one of the fuels used in Cessna training aircraft. The characteristics of an avgas-fueled aircraft engine require a mixture of fuel and air and require a thrust force to operate [1]-[3]. In recent years, the use of avgas fuel itself has been very high in high-wing light aircraft engines, namely the Cessna [3]-[5]. However, until now, there has yet to be any research regarding the spray collision in this aircraft engine. The phenomenon of avgas spray colliding with the cylinder wall may occur during fuel injection, resulting in a changed spray radius and height, which will affect fuel mixing and air, engine performance, and exhaust emissions will also be affected [4]-[6]. Engine performance will be maximum if the mixture of fuel and air matches the engine's needs[7]-[9].

Therefore, it is important to know and study the phenomenon of spray impact on avgas-fueled aircraft engines[7], [10], [11]. The characteristics of avgas-fueled sprays, based primarily on fuel density, injection pressure, and impact angle, serve the purpose of understanding how these factors influence spray behavior and engine performance [12], [13], [14]. Several previous studies have revealed a relationship between fuel density and spray characteristics [3], [15], [16]. For example, the average peak velocity of the spray tends to be positively correlated with the velocity of the spray center [12], [17]. Research has shown that the higher the injection pressure, the higher the spray speed and spray

DOI: 10.17977/um016v8i12024p071

penetration [18], [19]. This can later have an impact on more efficient combustion and better engine performance [20]. The angle of impact is also an important factor in spray characteristics [21]. Research has shown that lower impact angles tend to result in faster spray penetration. The optimal impact angle can provide better fuel distribution and higher combustion efficiency. Apart from these factors, research has also compared the spray characteristics of avgas fuel and other fuels. This research shows that avgas fuel has a wider spray angle and shorter penetration time than its mixture at the same injection pressure.

The research considers the influence of all aspects on spray characteristics and, through these experiments, provides important insights into optimizing machine performance. The method used in this research is experimental methods with measured nozzle pressure. The image on the spray nozzle can be analyzed by computational fluid simulation (CFD)[13], [19], [22].

II. Material and Methods

The research considers the influence of all aspects on spray characteristics and provides an important understanding of engine performance optimization through these experiments. The research methods started with experimental jets with a specified pressure and were jetted on the combustion chamber wall.

1. Experimental Tools and Procedures

This data collection used an experimental system by visualizing a constant volume jet of fuel. The experimental equipment used is seen in Figure 1.

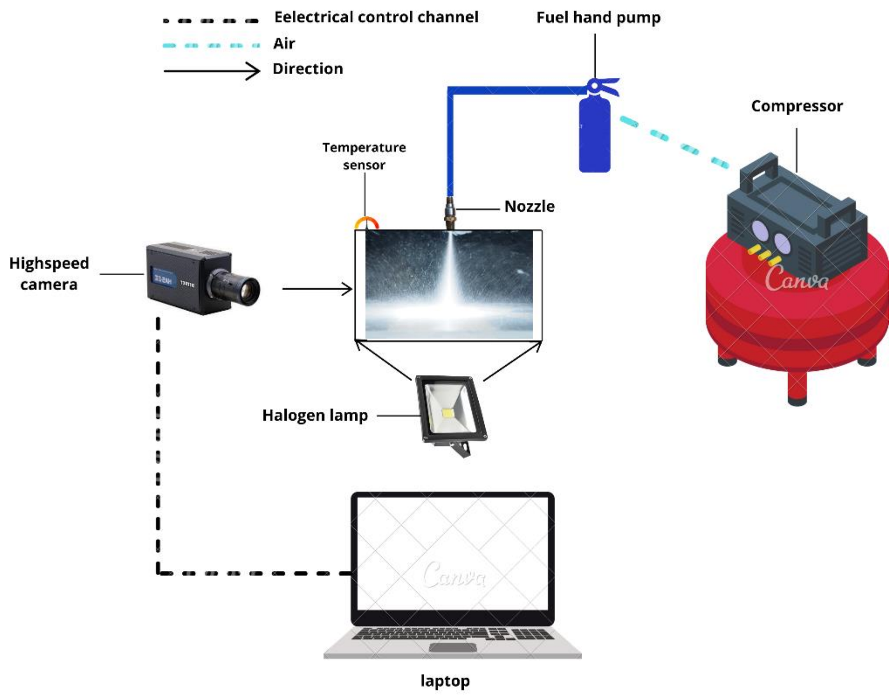

Fig. 1. The flow of data collection

A burst of fuel that occurs indoors. This combustion uses pressure from the fuel hand pump. The pressure was supplied from the compressor with pressure adjusted to the original pressure on the aircraft, namely 2 Bar (30 Psi); the spray could later be seen with the help of a halogen lamp placed at the bottom at a certain angle to get the best spray results desired, installation of a highspeed camera to capture. The jet that came out until it hit the wall was clamped on laptops. The temperature in the combustion chamber was sterilized with the room temperature sensor set to 26^{o}C to be stable at the time of data collection; this data was carried out in an air-conditioned room where the temperature was stable and set at 26^{o}C.

Data was collected when the fuel hand pump was pressed. It ejects fuel through the nozzle, and a burst occurs; at that time, the camera records the movement. The burst burst ends are repeated until you get satisfactory results. Later, data will be sent to a computer for storage, allowing a complete recording of the spray process. With experimental conditions using the system high-speed cameras to study phenomena of avgas spray on the wall, to get better spray distribution, this experiment uses a single-hole nozzle. The special conditions for these parameters are shown in Table 1.

Table 1. Experimental conditions

Injection condition	Parameter	Environmental condition	Parameter
Fuel	AVGAS	Ambient temperature (^{o}C)	26
Spray (mass/mg)	1	Injection angle to the wall (^{o})	90
Injection pressure (Bar)	2	Nozzle to wall distance (cm)	7.21
Injection duration (m/s)	1	-	-
Nozzle type	Single hole	-	-
Nozzle hole diameter	2	-	-

2. ANSYS CFD Simulation Procedure

One of the data support using CFD is the ANSYS application to see how the movement of the impacted fluid, basically in the experimental taking of the spray, there will be an impact by describing the development of the spray after contact with the wall, namely the spray height (H) and spray radius (R) as shown in Figure 2.

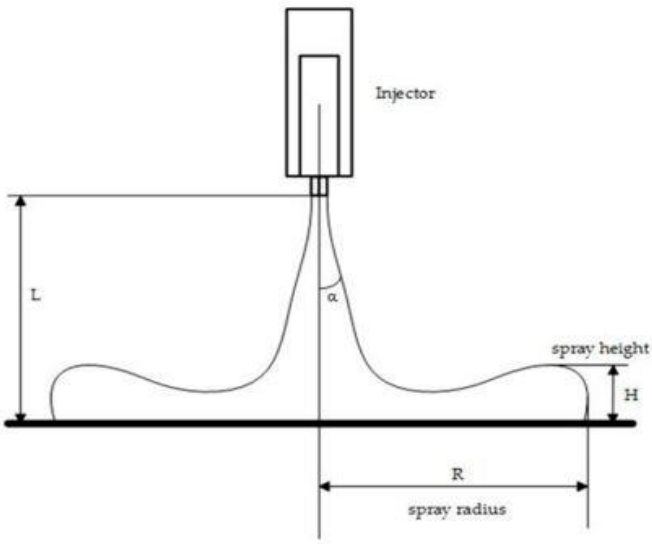

Fig. 2. Calculation of impact bursts

Figure 2 shows a diagram of the vertical wall spray parameters. The spray radius R is the jet penetration distance spray along the direction of the wall after contact with a wall; wall spray height H is the penetration distance perpendicular to the wall; L is the distance from the injector to the wall surface; parameters such as diameter diffusion and high volume can be used for reflects the volume of fuel on the walls and volume air[27], [28]. The calculation of penetration of the spray passing through the wall [29], [30] follows Eq. (1).

$$S = B * (tW - t0) * (Pamb/P0) * (L / cos(q)) * tan(j) \quad \text{..........(1)}$$

Where:
- S : spray penetration distance (including free spray and collision with the wall).
- B : wall collision constant.
- t0 : start time of injection.
- tW : time when the spray hits the wall.
- Pamb : environmental back pressure.
- L : distance from nozzle to wall.
- q : angle of inclination of the wall to the horizontal plane.
- j : angle between the entire oil jet and the spray wall.

This empirical formula is suitable for evaluating the penetration of a single spray. By knowing the formula for the experimental spray, the spray can also be implemented in CFD by looking at its behavior and movement. ANSYS software has an automatic mesh feature that uses real-time meshing techniques and only needs to enter the size and type of base material. The meshing in question is a design image of a size with the material we want, as shown in Figure 3.

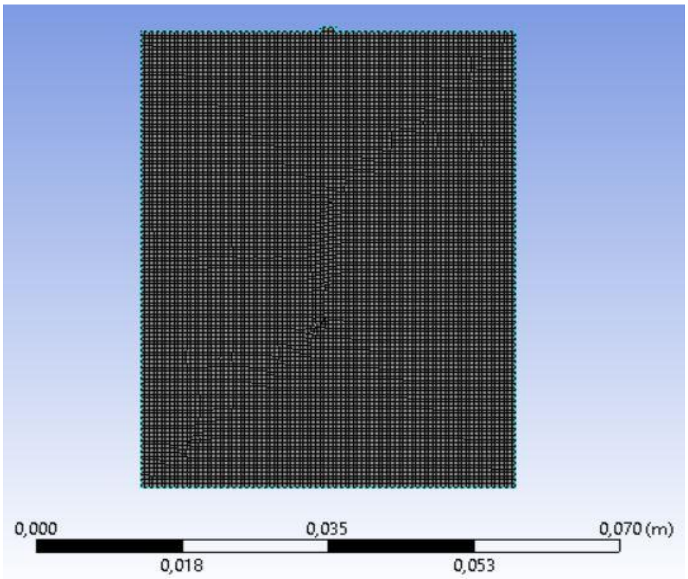

Fig. 3. ANSYS meshing

The meshing is generated directly when we input the size and is calculated immediately. This experimental simulation is of a combustion chamber with a constant volume jet. The model is a combustion chamber with a length of 4.5 cm and a width of 6.15 cm, and a nozzle with a size of 0.2 cm is drawn in the center with a height of 0.05 cm. To achieve calculation accuracy and speed simultaneously, the meshing size was reduced to 0.0005 mm using CFD ANSYS simulation software to simulate the Avgas spray process in the combustion chamber with constant pressure. Then, the original pressure of 2 Bar is varied to 3 and 4 Bar; the initial conditions of fluent basic fluid and Eulerian Multiphase parameters added with viscous k-omega are shown in Table 2.

Table 2. Material solution setup

Multiphase Eulerian	Phase 2 implicit
Viscous model	k-omega SST Mixture
Phases	Air- primary avgas- secondary
	mixture, pressure outlet, gauge 200000
Boundry condition	Velocity inlet, gauge 200000
	Avgas, velocity inlet, magnitude 2 m/s Avgas, backflow 1
Calculation	200

3. Numerical Simulation (ANSYS Simulation)

By using CFD applications to help see in more detail how the burst is produced and its growth coupled with the temperature of the burst and the results of the burst, the CFD application ANSYS is used, using the simulation setup and mesh obtained.

4. Simulation Setup

At this setting stage, there are several steps, namely geometry, mesh, setup, and solution. In the geometry section, the nozzle size is made, and the nozzle distance and the spout space are set here with a predetermined size, namely:

Table 3. Geometry size

Parameter	Value
Length	4.50 cm
Wide	6.15 cm
Nozzle height	0.05 cm
Total area	6.20 cm
Diameter of the nozzle	0.20 cm

By determining the size of the blast room, the sketch can be made, and then a line from the sketch to apply the room is frozen so that it becomes a form of combustion room with a nozzle that is ready to be simulated.

5. Mesh Convergence

The meshing used uses an element size of 0.0005 m so as to get a tighter and smoother size, as shown in Figure 4. It is added with the inlet section for the input of the burst and the wall or wall for part of the blast chamber, as shown in Figure 5.

Vol. 8, No. 1, July 2024, pp. 71-81

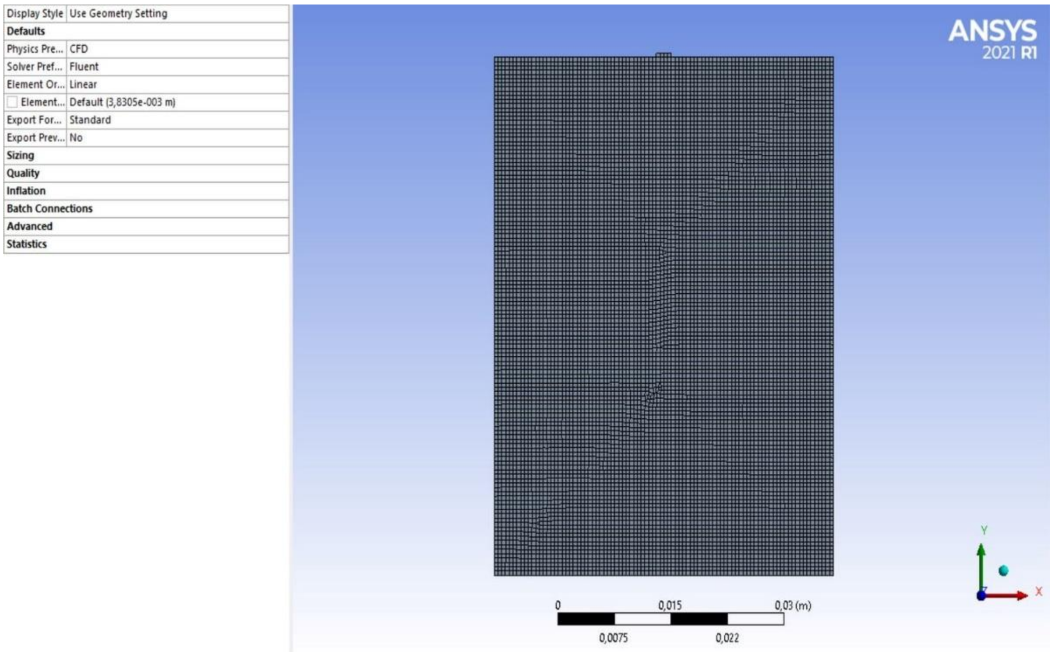

Fig. 4 .Meshing

6. Setup and solution

The parameter used at Setup is shown in Table 4.

Table 4. Parameter for Setup

Set-Up	Double precision
Solver Proces	1
Multiphase	Eulerian
Formulation	Implicit
Phase	2
Viscous	SST-k-omega, Mixture
Material	Air
	N-Octane Liquid (C8H18)

Material Setup set on Boundary Conditions is centered on the inlet and set on the mixture and N-octane parts. Determination of velocity magnitude was obtained during the experiment where the pressure at 2 Bar was 2.79 m/s, 3 Bar was 2.87 m/s, 4 Bar was 3.02 m/s. at the time of determining the nominal initial gauge 2 Bar was changed to pascal [Pa] so that 2 Bar = 200000 Pa, 3 Bar = 300000 Pa, and 4 Bar = 400000 Pa. After setting the Boundary conditions, continue with setting Initialization and Patch to determine which part will be focused on the mesh field so that the final part will be Run Calculation with 200 iterations and reporting interval 1, and the iteration results are obtained from the 3 pressures.

7. Experimental Setup

The experimental setup uses avgas fuel with adjusted pressures of 2 Bar, 3 Bar, and 4 Bar with a spout distance of 7.21 cm and a wall width of 4.5 cm, and then it is sprayed at room temperature. The size application is shown in Figure 5.

Fig. 5. Avgas R, H and α burst results.

III. Results and Discussions

1. Experimental Results

Obtained burst results with a criterion pressure, the pressure criteria that are varied, namely 2, 3, and 4 Bar.

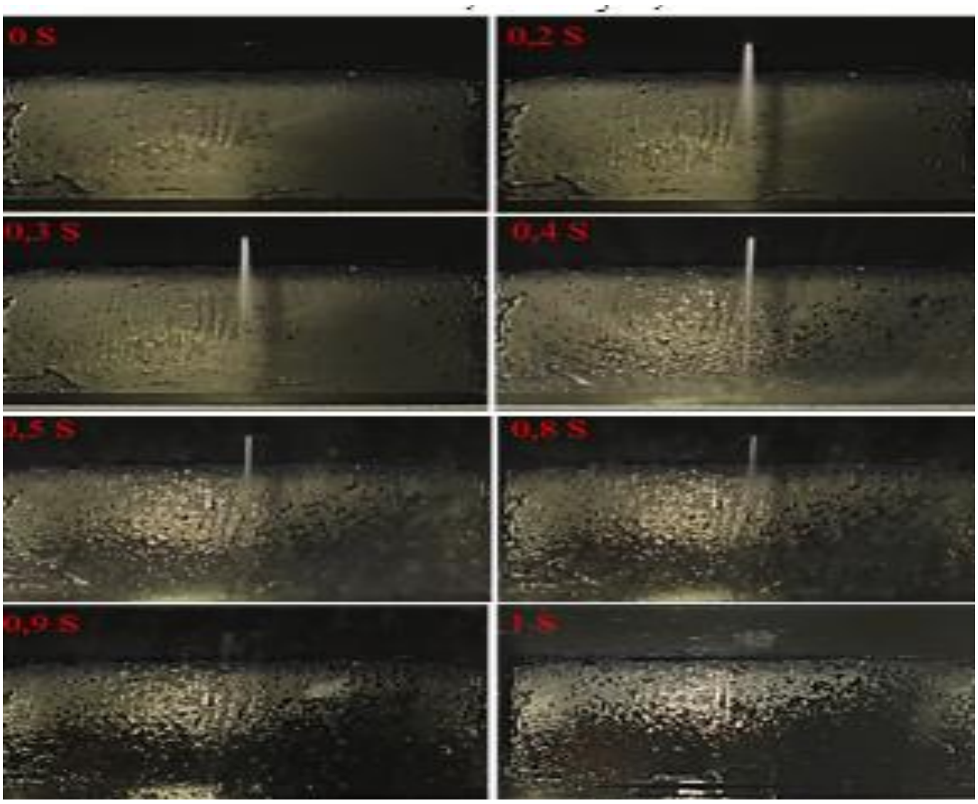

Fig. 6. Journey of the burst

Figure 6 shows photographs of the shape of the avgas fuel spray as it develops over time. The spray that comes out and looks slightly distorted before hitting the wall and the front of the body shows a more pronounced diffusion phenomenon. A "mist" appears at the edge of the sprayed head [19]. At 0.2s, after the onset of injection, a clear oil beam can already be seen. Avgas oil beam After five measurements at each pressure, it was found that the cone angle of the avgas fuel jet did not change significantly. The results agree with the conclusion that the cone angle change is small at high injection pressures. At this point, the phenomenon of avgas spray hitting the wall has occurred. The spray radius increases obviously at the spraying stage after impact. The author found that the change of injection pressure on the spray cone angle is mainly affected by the injection pressure [20] and the influx of air around the spray bundle. The injection pressure and environment are not changed in this test, so in the whole process of spraying the injector can generate maximum data. Observation shows that there is an obvious thinning oil mist area around the outer wall of the avgas mist, and the thin avgas mist area becomes gradually larger as time passes.

3. ANSYS Results

Based on the iteration results, 3 different pressures are obtained where based on the results of the burst obtained at 2 Bar pressure the heat generated due to collisions is 3.52e+00, 3 Bar pressure, the heat generated by collisions is 3.62e+00, and 4 Bar pressure produces heat of 3.80e + 00, where of the three collisions the 4 Bar pressure collision is the largest heat generated by avgas fuel, as shown in Figure 7.

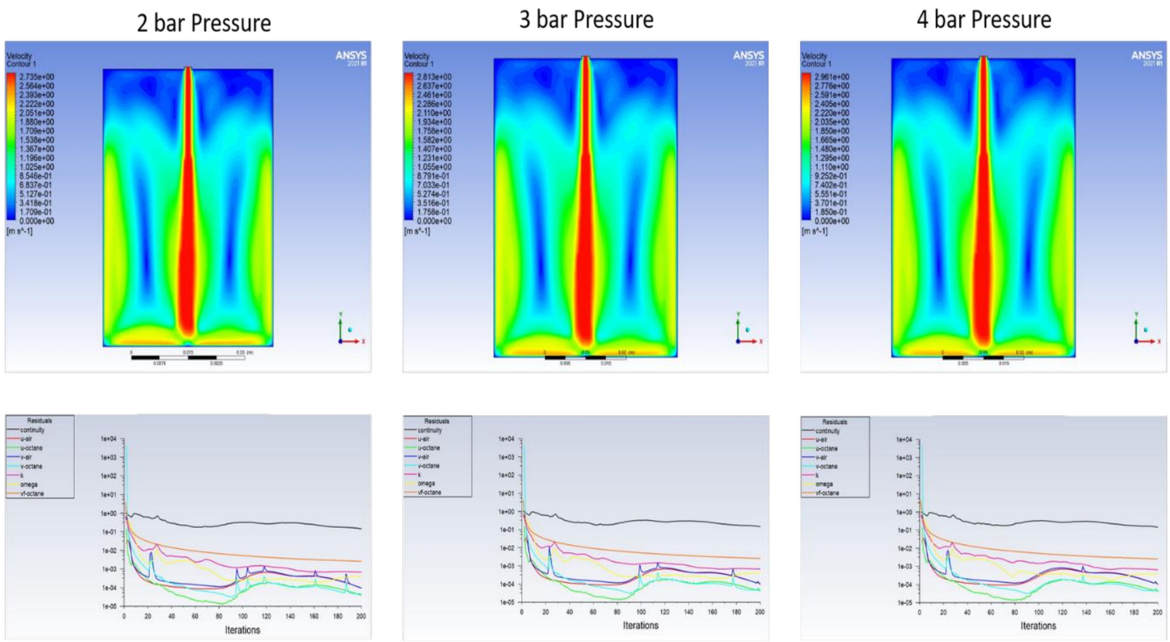

Fig. 7. Iterations results for each pressure

4. Numerical Simulation Results and Discussion

Based on experimental and ANSYS results, the pressure produces a different burst pattern from the air pressure variation entered in the burst as in the experimental burst of 2 Bar pressure. The avgas burst is triangular and visible spray results, but the atomization of the burst is split before being impacted by the wall, and the heat disappears when the burst process has not been impacted by the wall, whereas at 3 Bar and 4 Bar jets. The impact process makes the atomization split due to high pressure so that the heat carried from the fuel jet is not lost on the way to the impact, while in the ANSYS experiment. It was found that the highest heat was generated by a pressure of 4 Bar, whereas in Figure 8 last and bottom image, the highest heat is 3.80e+00 at the time after being impacted by the wall at 0.2 seconds. This is compared to other experimental pressures where at 2 Bar pressure 3.51e+00 at 0.4 seconds and at 3 Bar pressure 3.62e+00 at 0.3 seconds.

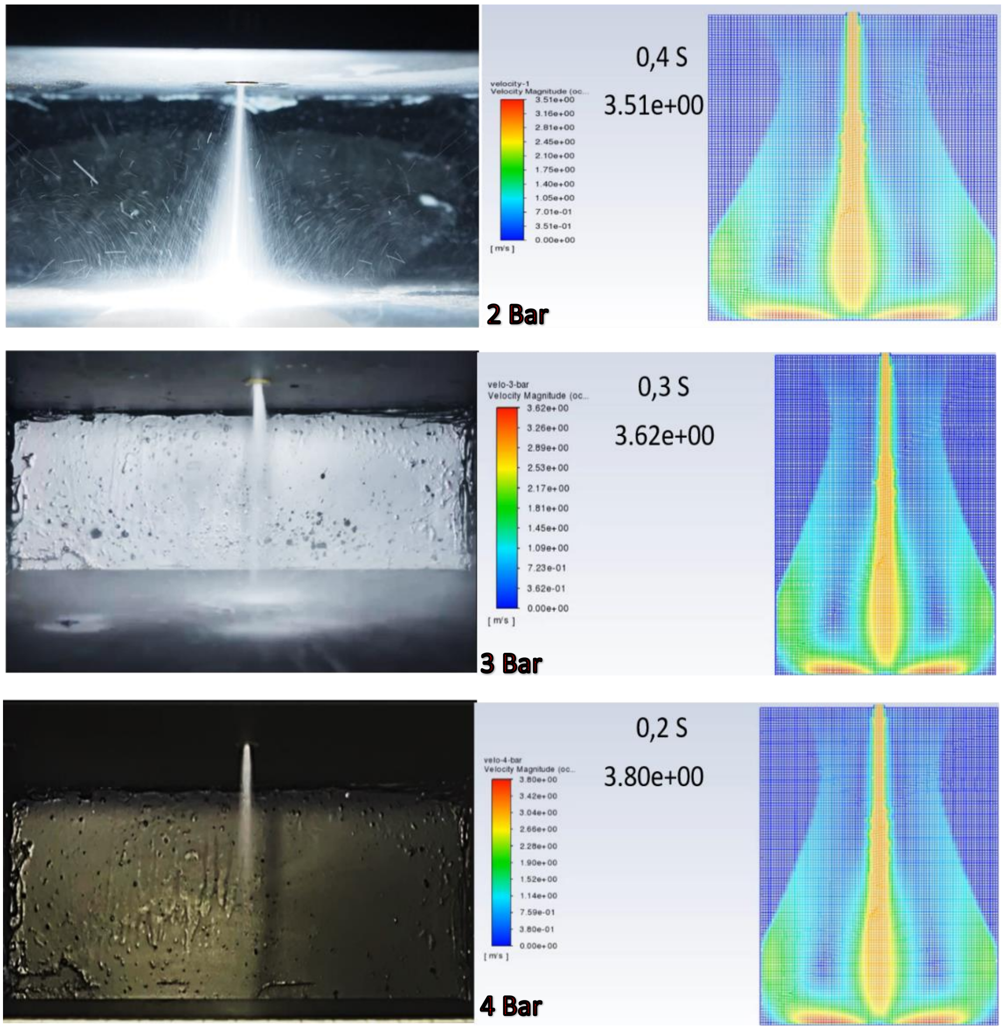

Fig. 8. Burst difference

The occurrence of the collision in Figure 8 with the ANSYS comparison can be seen that the greater the pressure on the nozzle, the more concentrated the resulting spray, so that the results of the collision are also well atomized as in Figure 8. The bottom picture with a pressure of 4 bar gets the largest heat pressure resulting from the collision, namely 3.80 e+00.

IV. Conclusions

From the experimental results of data collection and ANSYS results, we can take aspects that we cannot see, such as the highest heat output, impact time, and also the distribution of impacts resulting from each additional pressure, so that the results above can later be implemented on aircraft nozzles which vary the pressure. which is higher to support the maneuverability of the Cessna 172S aircraft itself and also without ignoring other factors.

Acknowledgment

The authors would like to express their sincere gratitude to all parties who have contributed to the preparation of this journal. Special thanks to Jember State University and Akademi Penerbang Indonesia Banyuwangi for supporting this study.

References

[1] R. Mahmud, T. Kurisu, K. Nishida, Y. Ogata, J. Kanzaki, and T. Tadokoro, “Experimental study on flat-wall impinging spray flame and its heat flux on wall under diesel engine–like condition: First report—effect of impingement distance,” *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, vol. 233, no. 8, pp. 2187–2202, Jul. 2019, doi: 10.1177/0954407018778153.

[2] K. Yamagishi, Y. Onuma, S. Ohara, K. Hasegawa, *et al.*, “Computations and Experiments of Single-Point Autoignition Gasoline Engine with Colliding Pulsed Supermulti-Jets, Single Piston and Rotary Valve,” in *SAE Technical Papers*, SAE International, 2016. doi: 10.4271/2016-01-2334.

[3] Q. Zhang, J. Xia, J. Wang, Z. He, Y. Qian, and X. Lu, “Experimental investigation on spray and combustion characteristics of dual-fuel collision of biodiesel and n-butanol,” *Fuel*, vol. 340, May 2023, doi: 10.1016/j.fuel.2023.127613.

[4] S. Wu, S. Yang, M. Wooldridge, and M. Xu, “Experimental study of the spray collapse process of multi-hole gasoline fuel injection at flash boiling conditions,” *Fuel*, vol. 242, pp. 109–123, Apr. 2019, doi: 10.1016/j.fuel.2019.01.027.

[5] B. K. Fritz, W. C. Hoffmann, W. E. Bagley, G. R. Kruger, Z. Czaczyk, and & R. S. Henry, “Measuring droplet size of agricultural spray nozzles-measurement distance and airspeed effects,” 2014.

[6] R. Jumadi, A. Khalid, N. Jaat, I. S. Abdullah, *et al.*, “Analysis of spray characteristics and high ambient pressure in gasoline direct injection using computational fluid dynamics,” *CFD Letters*, vol. 12, no. 5, pp. 36–51, 2020, doi: 10.37934/cfdl.12.5.3651.

[7] M. H. M. Zin, “Computational study of fuel spray structure,” Bachelor Thesis, Faculty of Mechanical Engineering, Universiti Malaysia Pahang, Malaysia 2013.

[8] B. Tekgül, H. Kahila, O. Kaario, and V. Vuorinen, “Large-eddy simulation of dual-fuel spray ignition at different ambient temperatures,” *Combust Flame*, vol. 215, pp. 51–65, May 2020, doi: 10.1016/j.combustflame.2020.01.017.

[9] Hadi Prayitno, Dimas Endrawan Putra, M. Khoirul Anam, Wisnu Kuncoro, and M. Nur Cahyo Hidayat, “Effect of Cessna 172S Aircraft Engine Vibration on the ground on Aviator Academy Pilots,” *Journal of Science Technology (JoSTec)*, vol. 5, no. 1, pp. 1–6, Jul. 2023, doi: 10.55299/jostec.v5i1.337.

[10] M. Muzammil, “Study of spray breakup and mixture in a gasoline direct injection engine by using simulation,” Bachelor Thesis, Faculty of Mechanical Engineering, University Malaysia Pahang,” 2013.

[11] Y. Duan, D. Han, P. Li, C. Wang, H. Lin, and Z. Huang, “Experimental study on injection and macroscopic spray characteristics of ethyl oleate, jet fuel and their blend on a diesel engine common rail system,” *Atomization and Sprays*, vol. 25, Jan. 2015, doi: 10.1615/AtomizSpr.2015011145.

[12] W. Mathews, C. Lee, and J. Peters, “Experimental investigations of spray/wall impingement,” *Atomization and Sprays*, vol. 13, pp. 223–242, Mar. 2003, doi: 10.1615/AtomizSpr.v13.i23.40.

[13] A. Hussain and K.S. Rao, “Design analysis and optimization of internal combustion engine piston using CAE tool ansys,” 2017. *Anveshana's International Journal of Research In Engineering and Applied Sciences*, vol. 2, no. 11, pp. 22-29.

[14] X. Liang, H. Zhang, G. Shu, Y. Wang, *et al.*, “Experimental investigation on effect of wall roughness and lubricant film on the adhered fuel film of N-butanol-diesel blends after spray impingement,” *Energies (Basel)*, vol. 11, no. 6, Jun. 2018, doi: 10.3390/en11061576.

[15] A. F. Alhikami and W. C. Wang, “Experimental study of the spray ignition characteristics of hydro-processed renewable jet and petroleum jet fuels in a constant volume combustion chamber,” *Fuel*, vol. 283, Jan. 2021, doi: 10.1016/j.fuel.2020.119286.

[16] D. Sun, W. Cai, C. Li, and J. Lu, “Experimental study on atomization characteristics of high-energy-density fuels using a fuel slinger,” *Energy*, vol. 234, Nov. 2021, doi: 10.1016/j.energy.2021.121222.

[17] H. Luo, K. Nishida, S. Uchitomi, Y. Ogata, W. Zhang, and T. Fujikawa, “Effect of temperature on fuel adhesion under spray-wall impingement condition,” *Fuel*, vol. 234, pp. 56–65, Dec. 2018, doi: 10.1016/j.fuel.2018.07.021.

[18] S. J. M. Algayyim, A. P. Wandel, T. Yusaf, and I. Hamawand, “The impact of n-butanol and iso-butanol as components of butanol-acetone (BA) mixture-diesel blend on spray, combustion characteristics, engine performance and emission in direct injection diesel engine,” *Energy*, vol. 140, pp. 1074–1086, 2017, doi: 10.1016/j.energy.2017.09.044.

[19] Y. Liu, Q. Xiang, Z. Li, S. Yao, X. Liang, and F. Wang, “Experiment and simulation investigation on the characteristics of diesel spray impingement based on droplet impact phenomenon,” *Applied Sciences (Switzerland)*, vol. 8, no. 3, Mar. 2018, doi: 10.3390/app8030384.

[20] N. Ilminnafik, P. Dimas Endrawan, H. M Fahrur Rozy, K. Muh. Nurkoyim, and Y. Danang, “Karakteristik Semburan Bahan Bakar Aviation Gasoline dengan Nozzle Pesawat Cessna 172S,” *Journal of Mechanical Engineering*, vol. 1, no. 1, pp. 9–22, Jan. 2024, doi: 10.47134/jme.v1i1.2182.

[21] F. J. Salvador, J. J. Lopez, J. De la Morena, and M. Crialesi-Esposito, “Experimental investigation of the effect of orifices inclination angle in multihole diesel injector nozzles. Part 1 – Hydraulic performance,” *Fuel*, vol. 213, pp. 207–214, Feb. 2018, doi: 10.1016/j.fuel.2017.04.019.

[22] I. Malgarinos, N. Nikolopoulos, and M. Gavaises, “Numerical investigation of heavy fuel droplet-particle collisions in the injection zone of a Fluid Catalytic Cracking reactor, part II: 3D simulations,” *Fuel Processing Technology*, vol. 156, pp. 43–53, Feb. 2017, doi: 10.1016/j.fuproc.2016.09.012.