

Optimization of Aircraft Fuel Dump Rate Towards the Mitigation of Post-Impact Fire

Abdulbaqi Jinadu^{1,a*}, Olalekan Adebayo Olayemi^{1,2,b}, Joshua Daniel^{1,c},
Oluwatomiwa John Odenibi^{1,d}, Volodymyr Koloskov^{3,e}, Dmytro Tiniakov^{4,f}

¹Department of Aeronautics and Astronautics, Faculty of Engineering and Technology,
Kwara State University, Malete, Kwara State, Nigeria.

²School of Engineering, Cranfield University, Cranfield, United Kingdom.

³Department of Applied Mechanics and Environmental Protection Technologies, Faculty of
Technogenic and Ecological Safety, National University of Civil Defence of Ukraine,
Kharkiv, Ukraine.

⁴Nanjing University of Aeronautics and Astronautics, Nanjing, China.

^aabdulbaqi.jinadu@kwasu.edu.ng, ^bolalekan.olayemi@kwasu.edu.ng,
^c008danieljoshua@gmail.com, ^dodenibioluwatomiwa@gmail.com, ^ekoloskov@nuczu.edu.ua,
^ftiniakov_d@nuaa.edu.cn

Keywords: Fuel Jettisoning, Flight Safety, Aircraft Performance, Fuel Dumping Rate, Gas Ejector, Venturi Effect, Emergency Procedure.

Abstract. This study seeks to improve the utilization of compressed air towards a faster fuel jettisoning, to increase the survival rate of passengers in the event of an accident or aborted takeoffs by augmenting the already existing means of dumping fuel with no considerable increase in overall weight. The aircraft fuel dump sub-system was isolated, this process was achieved with the aid of the venturi effect. The engine compressor marks the start of the aircraft fuel dump sub-system while an exterior nozzle for displacing the fuel marks its end. This system achieved jettisoning through bled-off air from the compressor, passing through a converging-diverging nozzle (primary supersonic nozzle), thereby creating a vacuum in the mixing chamber. A jet which provides a direct connection between the fuel tank and the mixing chamber sucks fuel from the tank, where bypassed air from the compressor expels the sucked air in fine particles. After running the simulation, the mass flow rate was computed. The compressed air inlet has a mass flow rate of 58.5193(Kg/S), the kerosene inlet 1.2385(Kg/S) while the outlet has a relative value of -59.6541(Kg/S).

Nomenclature

A, Area
m, Mass flow rate
M, Mach number
p, Pressure
Re, Reynolds number
T, Temperature

Subscripts

0 Stagnation condition
1 Primary nozzle section
2 Throat section
3 Mixing chamber section
4 Diffuser section
p Primary stream
s Secondary stream
t Nozzle throat

Introduction

A landing is termed overweight when an aircraft lands beyond its maximum structural landing weight. There are two major aircraft weight limits viz the maximum structural landing weight and the maximum takeoff weight. Often, the maximum takeoff weight is always higher than the landing weight. Taking off at a heavy and landing light is normal as fuel onboard is consumed in flight.

Unforeseen circumstances may require a pilot to land overweight, others abort takeoff after going full throttle on the runway. This could arise from an urgent medical emergency or a mechanical failure

that could exacerbate in the long run. Such instances would require an emergency landing. In all the instances, the airplane is landing overweight. If an aircraft lands more than its maximum permissible weight, it might suffer damage, structurally [1] which is also likely in the case of an aborted takeoff overshooting the runway. At the very least, an overweight landing would require a thorough inspection for damage. The pilot must then decide whether to reduce the weight before landing or land overweight [1]. This is achieved either by loitering to burn off fuel or by jettisoning fuel.

The Federal Air Regulations (FAA) puts the requirement of a fuel jettison system at 10 minutes for general aviation and 15 minutes for transport category aircraft.

Over the years, the alarming number of casualties of post-crash fires made it apparent that a means to reduce fuel levels in tanks in cases of emergency is paramount. Delaying landing with a malfunctioning system or engine failure to reduce weight or jettison fuel may expose the airplane to additional system deterioration, [1]. A comprehensive study shows the fuel dumped by the Airforce (worldwide) averages 600 metric tons (well over 1 million pounds) per month. and this quantity of fuel amounts to 0.07 percent of the total Air Force jet fuel consumed at an annual cost of \$1.1 million [2]. Commercial aircraft also jettison fuel, but complete records are not kept. Maintenance reports provided to the Federal Aviation Administration (FAA) by commercial airlines show 458 incidents of fuel discarded over the five years ending March 1980 (FAA). These records are for incidents associated with aircraft maintenance alone.

Reported fuel dumping events covered by the FAA as shown in Fig. 1 below occur at an estimated rate of 0.00000588% of flights per year. Between fiscal years 2015 and 2019, there was an average of more than 25.5 million flights over the U.S. per year. In that same period, there was an annual average of 15 reported fuel dumps [3].

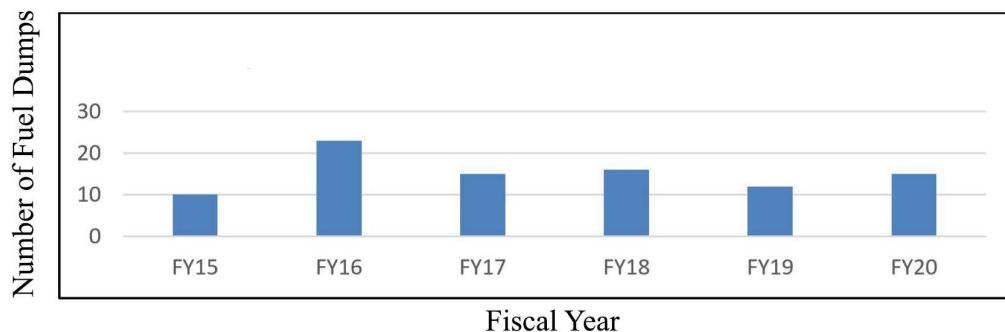


Fig. 1: Reported Fuel dumps per Fiscal Year [3]

The dumping of excessive fuel before landing is necessary because of safety concerns. This is an important aerodynamic issue requiring resolution by combining wind tunnel tests, computational fluid dynamics (CFD) simulations, and flight tests. Since jettisoned fuel is parallel to the external air stream, it has less tendency to impinge on the aircraft surface than fuel jettisoned normal to the air stream [4].

In 1936, J.H. Hammond, Jr., invented an aircraft dumping system [5]. The invention, which is shown below in Fig. 2 describes a means for dumping gasoline from the fuel tank of the airplane by the application of high pressure to the fuel tanks and at the same time, a means is provided for opening the dump valve and for shutting off the supply of fuel to the engine.

Alvin [6] designed a fuel dump device for airplanes as shown below in Fig. 2. This dump device is contained in a suitable casing attached to the underbelly of the aircraft, and included a section of coiled fuel tubing which will be discharged by spring pressure from within the casing, and the further provision of shock absorbing means to take up the shock when the tubing has reached its extended position, to which a folded parachute will be attached, whereby, when a pilot desires, the doors at the rear end of the casing may be opened and the parachute and tubing pushed from the casing into the slipstream of the airplane which will strike against the parachute thus quickly drawing the tube from the casing to its extended position.

In 2010, Morgia, et al [7] designed a fuel jettison system that accords the pilot a direct manual control to fuel jettisoning to as low as 20 minutes usable fuel remaining level, other than the 60 minutes usable fuel remaining level for the automatic.

What all these inventions share is their inability to contest with the pumps currently used for jettisoning. Instances, where runways were overshoot after an aborted takeoff, are also a downside for the existing technology.

The current study aims at augmenting the already existing means of dumping fuel with no considerable increase in aircraft's overall weight and increase how efficiently compressed air is being used as a resource made available by the jet engine.

Methodology

This research involves the modeling, simulation, and analysis of a system using finite element modeling (FEM) software. The system's inlet gets its primary flow from bypassed compressed air from the jet engine compressor, which then induces a secondary flow from the fuel tank for jettisoning. This system, when used together with the already existing means of fuel jettison, would increase the overall jettisoning rate.

1. A 2D model will be designed using SOLIDWORKS, while analysis and simulation will be done using ANSYS Fluent.
2. The system will be simulated with two-phase flow (air and kerosene).
3. The system's behavior will be further investigated by comparing it with ejectors.

Physical Model Description

Fig. 2 shows a 2-D physical model for the ejector employed for the present study while table 1 entails the dimensions for the ejector employed for the present study. The solution of ejectors in 3-D is not economical as it requires a lot of time and requires an expensive computational resource. Nevertheless, this ejector can be simulated as 2D-axisymmetric as it is close to the asymmetrical model (except for the inlet of primary and secondary flow). Thus, this assumption shall significantly reduce the time and resources to simulate it. However, past CFD studies demonstrate that overall simulation of 2D does not affect the accuracy [8].

The most important section in the ejector is the area around the mixing chamber between the primary and secondary inlets. The area densely meshes at the mixing area and diffuser.

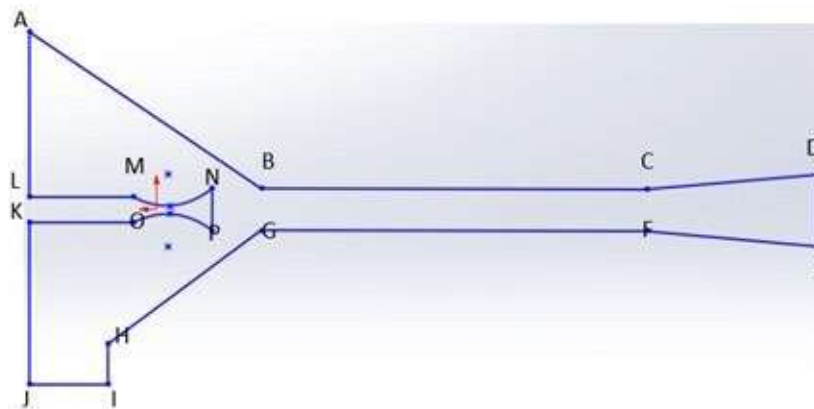


Fig. 2: Physical Model of the ejector

Table 1: Ejector dimensions (mm) (current study)

AB	AL	KL	JI	IG	HG	GF	FE	DE	NP	CF
629	408	70	186	442	450	1034	420	189	90	70

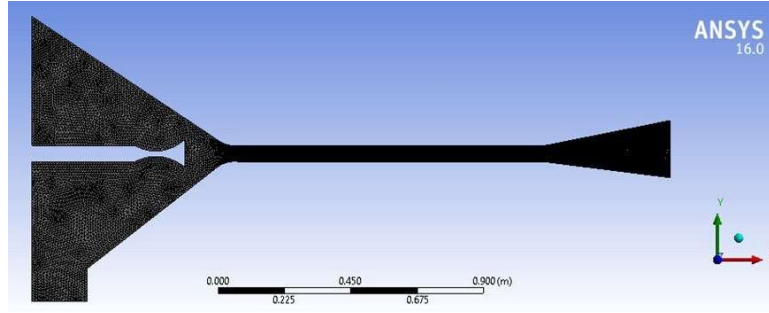


Fig. 3: Mesh profile of the fuel jettisoning system under study

Table 2: Mesh statistics of the fuel jettisoning system

Nodes	33813
Element	65280

Fig. 3 above shows the mesh profile of the fuel jettisoning system, while Table 2 details the number of nodes and elements in the meshed profile of the fuel jettisoning system.

Supersonic Primary Nozzle

Fig. 4 shows the converging-diverging supersonic primary nozzle of the ejector used in the fuel jettisoning system. The nozzle's inlet throat and exit sections are represented by 0, t and 1, respectively. Eq. 1 and Eq. 4 were used to calculate the maximum mass flow rate and the area ratio of nozzle exit to nozzle throat. The downstream pressure of the system is normally not operated at the unique design conditions of the nozzle geometry. Irreversible over- or under-expansion, i.e., shock waves, will occur outside the nozzle. Therefore, a coefficient η_n , relating to the isentropic efficiency of the compressible flow in the diverging expansion part of the nozzle, is utilized.

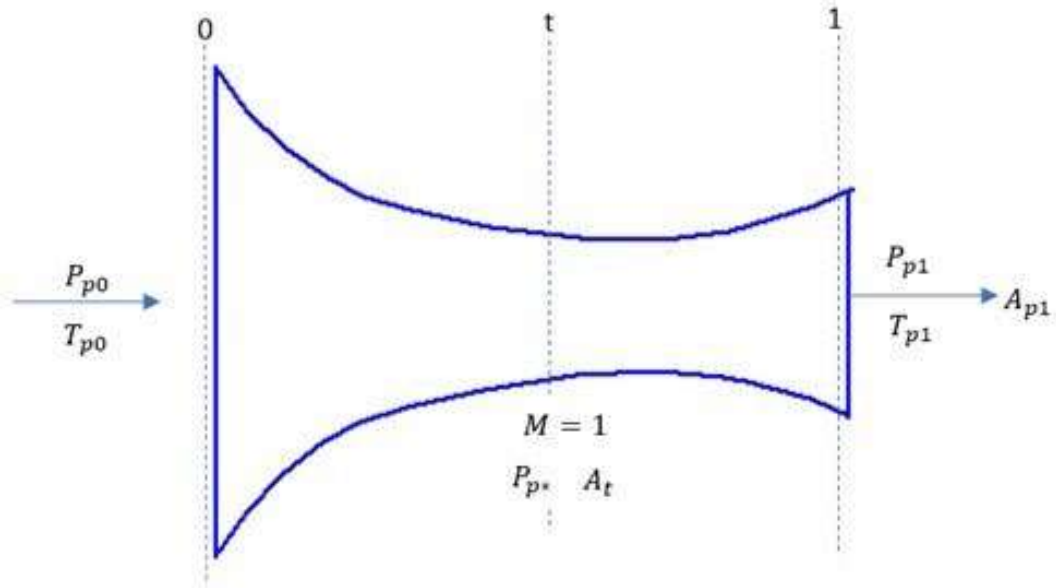


Fig. 4: Supersonic primary nozzle of the ejector

Governing Equations

For the converging part through the nozzle throat, the isentropic assumption is adopted. The mass flow rate of the primary stream is given by Eq. 1.

$$m_p = \frac{A_t P_0}{T_0} \sqrt{\frac{\gamma_p}{R_p} \left(\frac{2}{\gamma_p + 1} \right)^{\frac{\gamma_p + 1}{\gamma_p - 1}}} \quad (1)$$

The isentropic efficiency of diverging expansion part of the nozzle is defined as

$$\eta_n = \frac{h_{p0} - h_{p1}}{h_{p0} - h_{p1s}} \quad (2)$$

The primary nozzle pressure ratio is as expressed in Eq. 3.

$$\frac{P_{p1}}{P_{p0}} = \left[1 - \frac{1}{\eta_n} + \frac{1}{\eta_n \left(1 + \frac{\gamma_p - 1}{2} M_{p1}^2 \right)} \right]^{\frac{\gamma_p}{\gamma_p - 1}} = f_1(\gamma_p, M, \eta_n) \quad (3)$$

Eq. 4 expresses the primary nozzle area ratio of exit to throat.

$$\frac{A_{p1}}{A_t} = \frac{P_{p*}}{P_{p1}} \frac{1}{M_{p1}} \left(\frac{T_{p1}}{T_{p*}} \right)^{\frac{1}{2}} \quad (4)$$

Constant pressure mixing model

Keenan and Neumann [9] introduced the continuous pressure mixing approach's fundamental ideas. It was assumed that the mixing of the primary and secondary streams occurs in a chamber with a uniform, constant pressure. The constant-pressure flow model of a typical ejector is shown in Fig. 5 below. The mixing chamber is between sections 1 and 2, within which the pressure is assumed to be uniform. If the velocity of the fully mixed flow is supersonic ($M_{m2} > 1$), a normal shock wave is assumed to occur in the constant-area chamber between sections 2 and 3. The static pressure of the mixed flow leaving Section 3 at uniform subsonic velocity is increased in the diffuser.

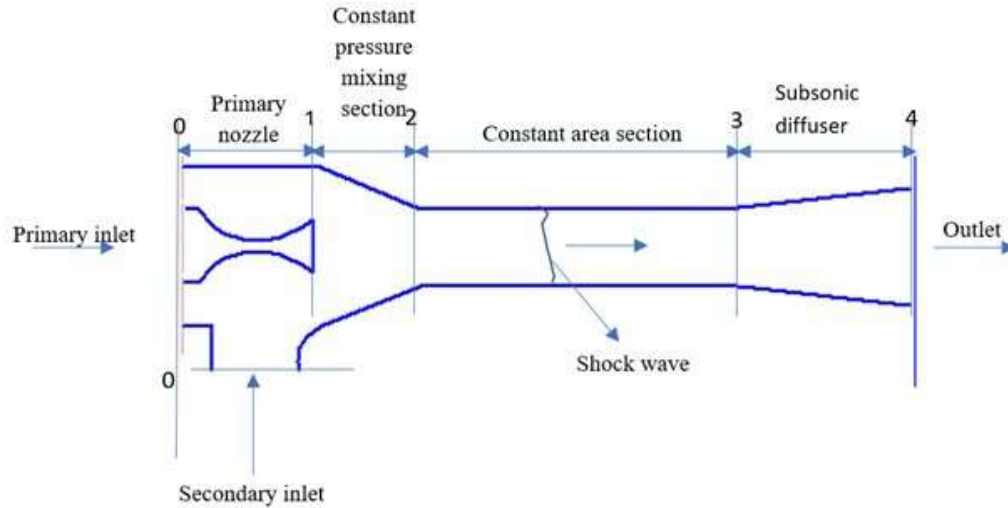


Fig. 5: Constant-pressure flow model of ejector

The entrainment ratio (ER) which is denoted by ω , is the mass flow ratio of secondary to primary streams at the inlet of the mixing chamber as given by Eq. 5.

$$\omega = \frac{m_{s1}}{m_{p1}} = \frac{P_{s1}}{P_{p1}} \frac{A_{s1}}{A_{p1}} \left(\frac{T_{p0}}{T_{s0}} \right)^{\frac{1}{2}} \left(\frac{R_p}{R_s} \right)^{\frac{1}{2}} \frac{f_2(\gamma_s, M_{s1})}{f_2(\gamma_p, M_{p1})} \quad (5)$$

Subsonic diffuser

The diffuser pressure ratio is expressed by Eq. 6.

$$\frac{P_{40}}{P_{m3}} = \left[1 + \eta_d \frac{\gamma_m - 1}{2} M_{m3}^2 \right]^{\frac{\gamma_m}{\gamma_m - 1}} \quad (6)$$

While the overall compression ratio through the gas ejector can be found from the following expression:

$$\frac{P_{40}}{P_{s0}} = \frac{P_{40}}{P_{m3}} \frac{P_{m3}}{P_{p1}} \frac{P_{p1}}{P_{p0}} \frac{P_{p0}}{P_{s0}} \quad (7)$$

Supersonic primary nozzle simulation

ANSYS is the software used for analysis. This analysis is made concerning both inlet and outlet temperature and pressure.

1. Primary nozzle surface

The surface to sketch option under the concept produces a two-dimensional surface that is used for additional investigation. The primary supersonic nozzle is seen in a two-dimensional sketch in Fig. 6 below.

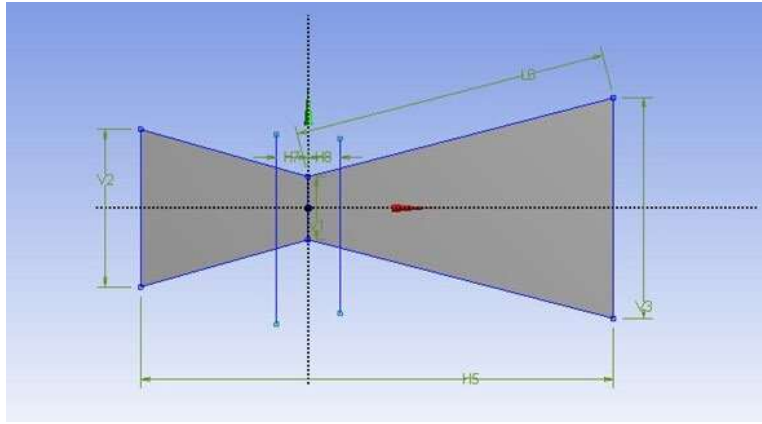


Fig. 6: Primary nozzle sketch of a Convergent Divergent Nozzle (Current Study)

2. Meshing and setup

This is the disintegration of the geometric body into a set of finite elements.

Mesh generation: A generated mapped face mesh approximates a geometric structure. Fig. 7 shows the meshed profile of the supersonic primary nozzle, while Table 3 shows the imposed boundary conditions before simulation.

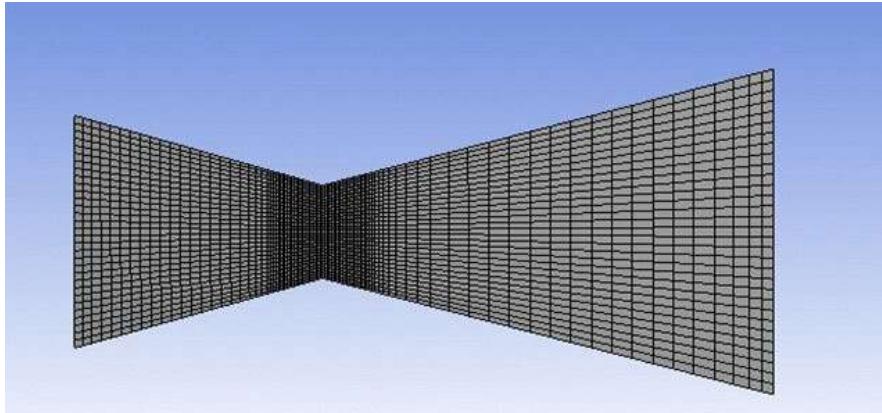


Fig. 7: Mesh of a Convergent Divergent Nozzle (Current Study)

Table 3: Boundary Conditions used for the Convergent Divergent Nozzle (Current Study)

Input	Value
Viscosity	1.7894e-05 (Kg/ms)
Thermal conductivity	0.02619 (W/mK)
Specific heat capacity	1006.43 (J/kg.K)
Mean molecular mass	28.966 (g/mol)
Inlet temperature	300 (K)
Inlet pressure	150000 (Pa)
Solver type	Density-based
2D space	Planar
Time	Steady

3. Simulations

For controlling direction and speed, the convergent-divergent nozzle finds application in aerospace and rocketry; the carburetor also uses the convergent-divergent architecture. The convergent and divergent nozzle analysis is carried out in ANSYS to compute flow properties like density, pressure, velocity, and temperature.

Solution Technique

The existing aircraft fuel jettisoning system independently jettisons fuel from multiple fuel tanks to maintain equal aircraft balance [7].

Computational fluid dynamics (CFD) modeling involves physical laws in the form of partial differential equations called Navier-Stokes equations. The standard k-epsilon model in ANSYS Fluent was used. The standard k- ϵ model is a model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ϵ) [10]. In this model, it is assumed that the flow is turbulent and other effects like molecular viscosity are negligible. Therefore, it is only valid for turbulent flows [11].

The 2D model in Fig. 2 was designed in SOLIDWORKS with the dimensions in Table 1 and then imported into ANSYS for preprocessing. The working fluid for both the primary and secondary inlets was defined as air for the primary and kerosene for the secondary inlet, respectively. The appropriate boundary conditions were then imposed on the respective boundaries, as shown in Table 4, and the computational domain was then discretized into distinct finite element meshes, as seen in Table 2. The mesh of the mixing chamber and diffuser were generated with an extremely fine grid size option for a better result, and before meshing, the double precision options were selected. The desired energy was selected, and assumptions like whether the flow is viscous or inviscid, compressible or non-compressible, were made.

The fuel jettisoning system will consequently be divided into two separate, independent systems, one for each wing, so that lateral stability can be maintained by jettisoning fuel from the "heavy" wing if it is necessary to do so. Normally, if an unbalanced fuel load exists, fuel will be used from the "heavy" wing by supplying fuel to engines on the opposite wing. This working system is applicable to a typical gas ejector, as seen in Fig. 8 below.

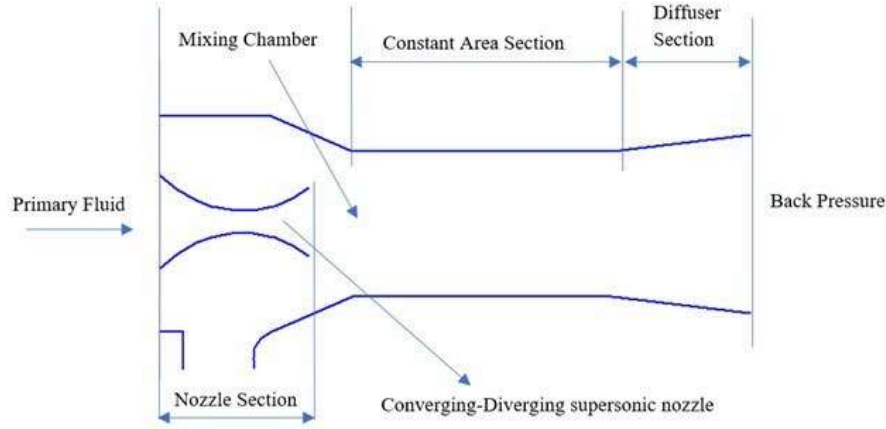


Fig. 8: A typical gas ejector (ESDU A9242)

A (k-epsilon) model of turbulent flow

The K-epsilon ($k - \epsilon$), a two-equation model of turbulence, is employed mostly in computational fluid dynamics (CFD) to simulate average flow properties for turbulent flow conditions. Expressed as transport-type equations depicting turbulent kinetic energy and the dissipation rate, i.e., the two-equation model of turbulence.

In comparison with previous models, this one hinges on the mechanisms that affect the turbulent kinetic energy. The underlying assumption of this model is that the turbulent viscosity is isotropic, i.e., the ratio between Reynolds' stress and the average rate of deformations is the same in all directions.

Turbulent kinetic energy transport equation (k).

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right] + 2\mu_t E_{ij} E_{ij} - \rho \epsilon \quad (8)$$

Turbulence dissipation transport equation (ϵ).

$$\frac{\partial(\rho \epsilon)}{\partial t} + \frac{\partial(\rho \epsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\frac{\mu_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} 2\mu_t E_{ij} E_{ij} - C_{2\epsilon} \rho \frac{\epsilon^2}{k} \quad (9)$$

Where.

u_i represents the velocity component in the corresponding direction.

E_{ij} represents the component of the rate of deformation.

μ_t represents eddy viscosity.

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \quad (10)$$

The equation also consists of calibrated constants:

$$C_\mu = 0.09 \quad \sigma_k = 1.00 \quad \sigma_\epsilon = 1.30 \quad C_{1\epsilon} = 1.44 \quad C_{2\epsilon} = 1.92$$

Mass flow rate

The mass flow rate is the mass of a substance that is passed through any cross-sectional area per unit of time. The primary parameters upon which the mass flow depends are P, T, and A [11].

$$m = \frac{PA}{(RT_0)^{\frac{1}{2}}} f_{2(\gamma, M)} \quad (11)$$

Static pressure within ejector

Figs. 9(a) and 10(a) show the static pressure distribution contour and XY-plot, respectively, along the simulated ejector wall based on the ideal gas primary inlet and liquid kerosene secondary inlet, compared with the static pressure contour and plot in Figs. 9(b) and 10(b) of a gas ejector from [13]. The comparison is based on the boundary conditions presented in Table 4 for the primary and secondary streams. For this simulation, the primary nozzle is converging and diverging, thereby making the flow transonic at the throat and supersonic at the diverging section. This configuration was chosen to achieve a high-pressure drop at the mixing chamber and a higher-pressure recovery at the diffuser. Both pressure contours show that the static pressure profiles from CFD simulations are in reasonable agreement with available experimental data and are like those presented in the literature [1]. The simulated pressure profile driven by compressed air at the primary inlet sucking kerosene-liquid from the secondary inlet displays significant structure; two distinct zones of pressure rise are observed. The first distinct pressure rise occurs in the mixing chamber and is attributable to the higher static pressure of the compressed air primary stream leaving the nozzle. The second distinct pressure rise occurs near the outlet of the constant area zone and brings the static pressure close to the diffuser exit static pressure.

The results from the simulations presented in Figs. 9 (a and b) and 10 (a and b) are representative of the level of agreement achieved in the other operating conditions where corresponding experimental data were available for comparison. Despite the average relative error, the difference is not particularly significant. It is concluded that the Fig. 10s model is about as capable as the Fig. 9 model in simulating the static pressure profile.

The working fluids for the present study are air and aviation kerosene, while those of the journal are R141b refrigerant and air. Aviation kerosene has a lower vapor pressure compared to refrigerants. This must have contributed to the difference in behavior in both studies, as well as the difference in density.

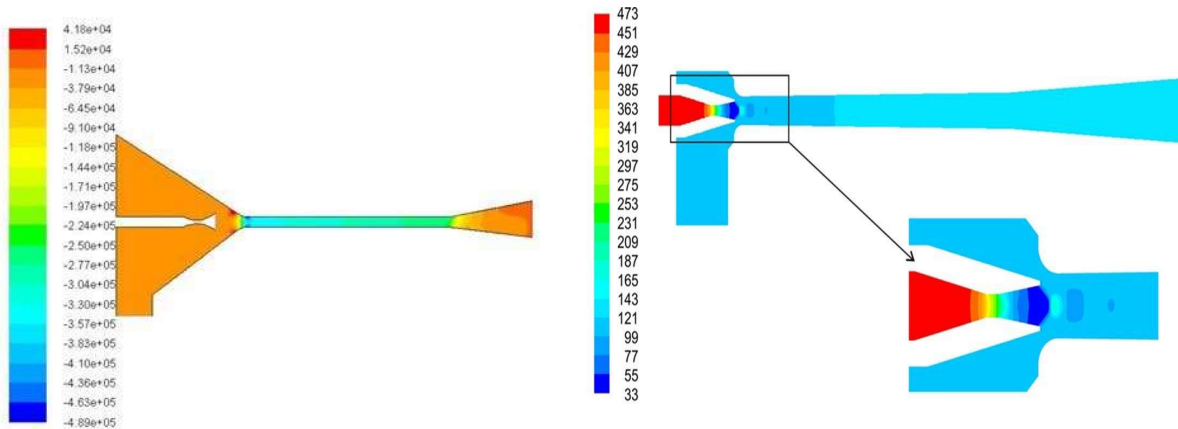


Fig. 9: Static pressure contour within the ejector: (a) current study; (b) J. Smolka et al. [13]

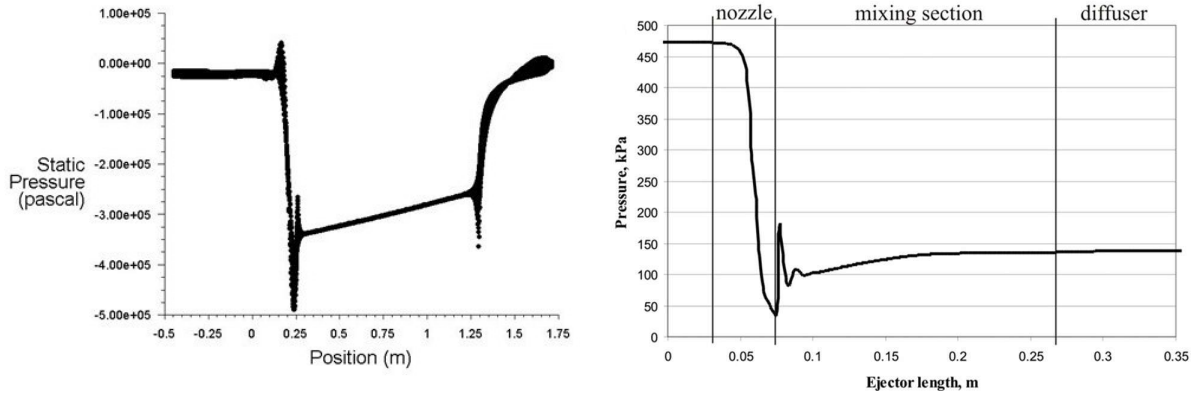


Fig. 10: Static pressure plot within the ejector: (a) current study; (b) J. Smolka et al. [13]

Table 4: Stream pressure and temperature values applied at the boundaries in the computational simulation

Stream	Temperature(K)	Pressure (pas)
Primary stream	300	150000
Secondary stream	273	-

Velocity within Ejector

In comparing the velocity contours in Figs. 11(a) and (b), it can be seen that the flow has three regimes. The least velocity is experienced at the secondary inlet, owing to the pressure gradient around that area, the maximum velocity is experienced at the constant area mixing section, while the diffuser section is in the middle. Both contours agree with the experimental data in the literature [1].

The velocity XY plots, Figs. 11(a) and 12(b), illustrate the trend in velocity reduction through the nozzle length. High velocity is experienced at the exit of the primary nozzle, a decline is experienced in the mixing section; and the farthest reduction is experienced at the nozzle exit. Fig. 9b's velocity drop in the diffuser section is higher than that in Fig. 10b. This is due to the fact that the faster flow in Fig. 9b encounters a wider diffuser compared to Fig. 10b.

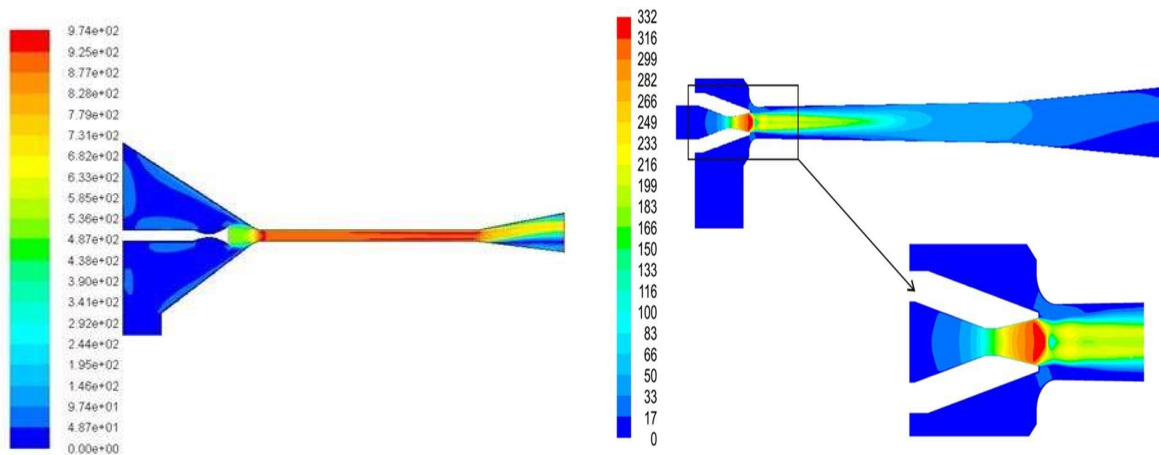


Fig. 11: Velocity contour within Ejector: (a) current study; (b) J. Smolka et al. [13]

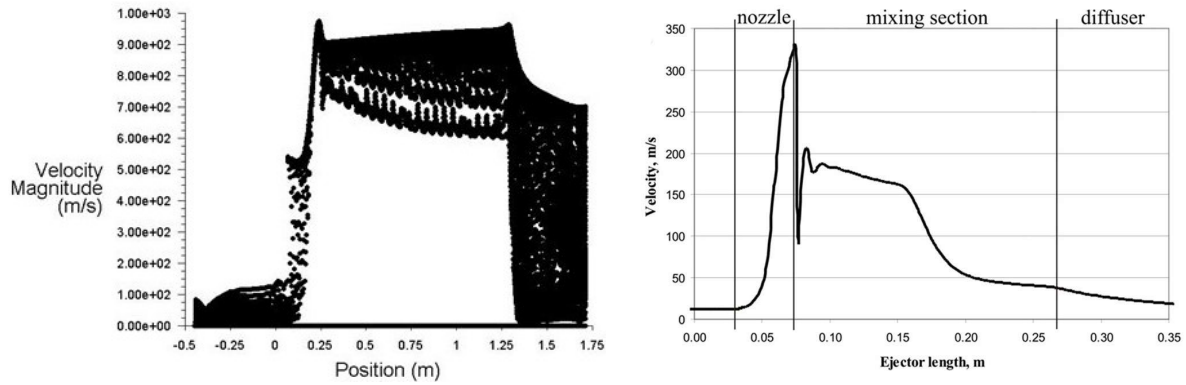


Fig. 12: Velocity plot within Ejector: (a) current study; (b) J. Smolka et al. [13]

Results and Discussion

Temperature within primary nozzle

Fig. 13 below shows the supersonic primary nozzle temperature contour simulation result. The blue color at the end of the spectrum shows the minimum temperature of $1.16\text{E}+002\text{K}$ while the red color at the upper end shows the maximum temperature of $2.98\text{E}+002\text{K}$ at the outlet and inlet of the nozzle, respectively, as seen.

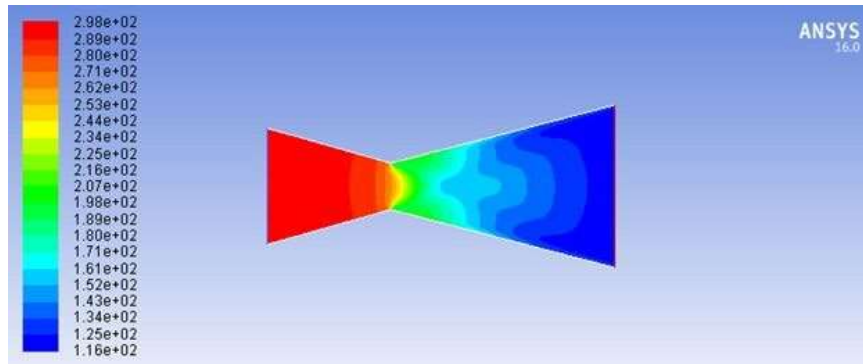


Fig. 13: Primary nozzle temperature contour (current study)

Pressure contour within primary nozzle

Fig. 14 below shows the supersonic primary nozzle pressure contour simulation result. Higher pressure is experienced at the outlet compared to the pressure at the inlet since the divergent end behaves as a form of pressure recovery. The inlet pressure is $2.924\text{E}+005$ pa while that of the outlet is $-8.71\text{E}+004$ pa. While the inlet pressure of $9.5\text{E}+006$ pa and an outlet pressure of $4.8\text{E}+006$ pa.

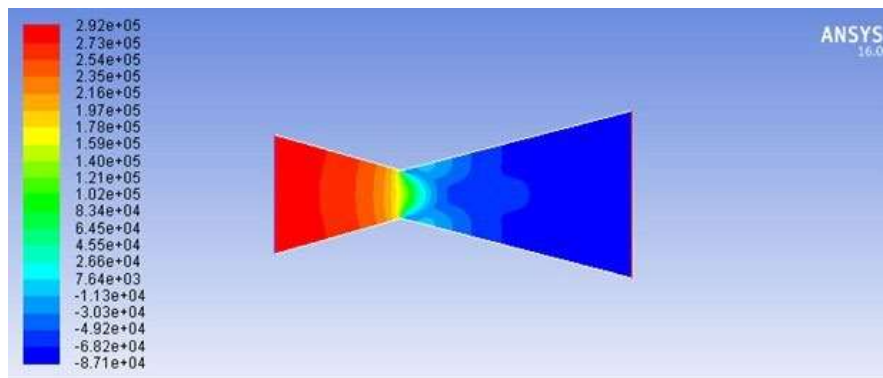


Fig. 14: Primary nozzle pressure contour (current study)

Velocity contour within primary nozzle

Fig. 15 below shows the supersonic primary nozzle velocity contour simulation result. As expected, the velocity of the fluid in the convergent-divergent nozzle's inlet is 0 m/s. The velocity is transonic at the throat. At the divergent section, fluid exiting the throat skyrockets to 6.09E+02 m/s, while the paper reviewed has an inlet velocity of 0 m/s and an outlet velocity of 1.25E+02 m/s. This means the current study ejects faster.

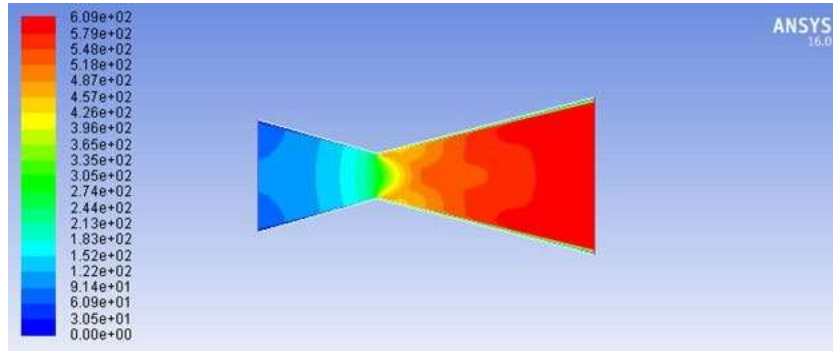


Fig. 15: Primary nozzle velocity contour (current study)

Density contour within primary nozzle

Fig. 16 below shows the supersonic primary nozzle density contour simulation result. A gradual change in density is experienced by the fluid in the nozzle from the inlet down to the outlet. As seen, the density at the inlet is about 4.61E+00 kg/m³, while it is 4.09E-00 kg/m³ at the outlet of the nozzle.

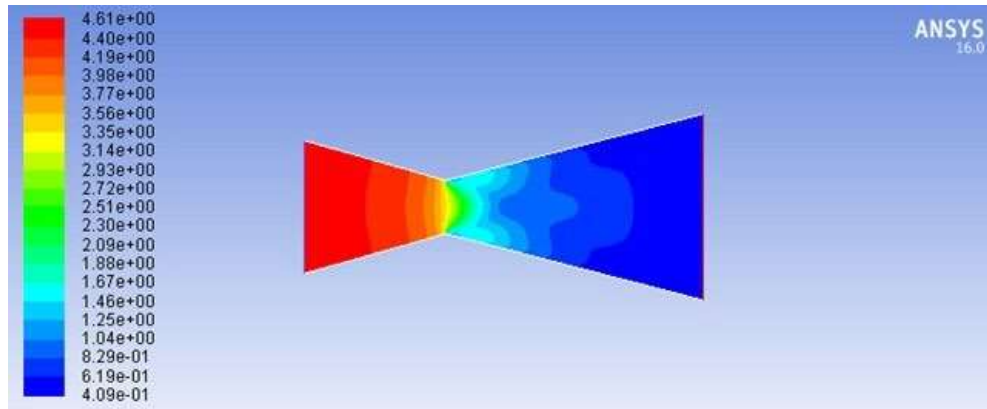


Fig. 16: Primary nozzle density contour (current study)

After running the simulation, the mass flow rate was computed and detailed in Table 5 below. According to the table above, the compressed air inlet has a mass flow rate of 58.5193(kg/s), the kerosene inlet has a mass flow rate of 1.2385(kg/s), and the outlet has a mass flow rate of -59.6541(kg/s).

Table 5: Mass flow rates for compressed air, kerosene inlet and outlet

Cool kerosene inlet	1.2385(Kg/S)
Outlet	-59.6541(Kg/S)
Compressed air inlet	58.5193(Kg/S)

The outlet has a higher mass flow rate compared to the primary nozzle compressed air inlet. Kerosene mixed with compressed air will be jettisoned in an atomized form through the outlet.

Conclusions

Computational Fluid Dynamics (CFD) has been successfully used to help reduce the likelihood of post-impact fire in the event of an air crash by designing and simulating a fuel jettisoning system that expels fuel at the rate of 59.6541(kg/s) when compared to the Boeing 777's fuel jettisoning rate of 41.667(kg/s).

Other advantages noticed from the current study are numerous. Unlike the pumps used in fuel dumping, the current study has no moving parts, making it easier to adopt and thus more dependable. Also, owing to the lack of moving parts and simplicity of the design, the system is less likely to encounter a breakdown, and even when it does, it will be easier and less expensive to repair. Its lightweight design is impressive when compared to the existing fuel system. For a system proposed to be used in conjunction with the already existing fuel jettisoning system, its contribution to the aircraft's maximum takeoff weight is negligible. The current study is economical, giving it the ease of use of an attachment to the already existing fuel jettisoning system.

Data Availability Statement

The authors affirm that the data that supports the results of the current investigation is present in this article.

Disclosure of competing interests

The authors have no conflicting interests to disclose.

References

- [1] Information on <https://www.boeing.com>. Overweight landing? Fuel Jettison? What to consider [Accessed 27 March 2023]
- [2] R. E. Good and H. J. Clewell, Drop formation and evaporation of JP-4 fuel jettisoned from aircraft, *Journal of Aircraft*, vol. 17, no. 7, (1980), 450–456.
- [3] Information on <https://www.faa.gov>. Annual occurrences of Fuel Dumps in the United State, Fiscal Year 2015-2020. [Accessed 23 March 2023]
- [4] T. Tai and S. Woods, Simulation and analysis of V-22 aircraft fuel dumping, 42nd AIAA Aerospace Sciences Meeting and Exhibit, (2004).
- [5] J. H. Hammond, Jr, U.S. Patent 2,038,998. Gas tank dischargers for airplanes. (1932)
- [6] J. F. Alvin, U.S. Patent 2,138,970A. Gasoline dump devices for airplanes. (1938)
- [7] J. J. Morgia and A. P. Cutting U.S. Patent 20,080,173,763A1. Fuel jettison system (2007)
- [8] B. Elhub, S. Mat, K. Sopian, A. M. Elbreki, M. H. Ruslan, and A. A. Ammar, Performance evaluation and parametric studies on variable nozzle ejector using r134a, *Case Studies in Thermal Engineering*, vol. 12, (2018) 258–270.
- [9] J. H. Keenan, E. P. Neumann, and F. Lustwerk, An investigation of ejector design by analysis and experiment, *Journal of Applied Mechanics*, vol. 17, no. 3, (1950) 299–309.
- [10] M. W. Khalid and M. Ahsan, Computational fluid dynamics analysis of compressible flow through a converging-diverging nozzle using the K- ϵ turbulence model, *Engineering, Technology & Applied Science Research*, vol. 10, no. 1, (2020) 5180–5185.
- [11] M. Ahsan, Numerical Analysis of friction factor for a fully developed turbulent flow using K- ϵ turbulence model with enhanced wall treatment, *Beni-Suef University Journal of Basic and Applied Sciences*, vol. 3, no. 4, (2014) 269–277.

- [12] J. Anderson, Fundamentals of Aerodynamics, sixth ed., McGraw Hill, New York, 2017.
- [13] J. Smolka, Z. Bulinski, A. Fic, A. J. Nowak, K. Banasiak, and A. Hafner, A computational model of a transcritical R744 ejector based on a homogeneous real fluid approach, Applied Mathematical Modelling, vol. 37, no. 3, (2013) 1208–1224.
- [14] H. J. Clewell, Ground contamination by fuel jettisoned from aircraft in flight, Journal of Aircraft, vol. 20, no. 4, (1983) 382–384.
- [15] R. E. Good and H. J. Clewell, “Drop formation and evaporation of JP-4 fuel jettisoned from aircraft,” Journal of Aircraft, vol. 17, no. 7, (1980) 450–456.
- [16] H. K. Versteeg, W. Malalasekera, An introduction to Computational Fluid Dynamics, second ed., Pearson, London, 2007.