INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
www.ijprems.com
editor@ijprems.com
Vol. 04, Issue 03, March 2024, pp: 226-233
e-ISSN : 2583-1062
Impact Factor: 5.725

# EVALUATION OF THE VARIABLE THAT AFFECT NOZZLE EFFICIENCY

**Kingsley E. Madu[1], Chike M. Atah[2]**
Corresponding email atahchike44@gmail.com
[1,2]Department of Mechanical Engineering, Chukwuemeka Odumegwu Ojukwu University, Uli, Anambra State.

## ABSTRACT

Nozzles are key components in many engineering applications that involve fluid flow, such as propulsion systems and industrial processes. Nozzle efficiency, defined as the ratio of actual outlet velocity to ideal outlet velocity, significantly impacts system performance. This paper evaluates the key variables that can affect nozzle efficiency, including nozzle geometry, Reynolds number, Mach number, turbulence, surface roughness, and flow non-uniformities. Experimental data from previous studies are reviewed and analyzed to quantify the impact of each variable. Computational fluid dynamics simulations are also performed to visualize flow behaviours. The results provide insight into nozzle design optimization and selection of operating conditions to maximize efficiency. Overall, this work identifies the critical factors engineering designers should consider to achieve high nozzle performance.

**Key:** Nozzle, Nozzle Efficiency, CFD

## 1. INTRODUCTION

Nozzles are important components in many engineering systems and industrial processes (Thomas, 1999). They are used to accelerate and direct the flow of fluids like gases or liquids (Singh et al., 2019). Some common examples of nozzles include rocket engine nozzles, fuel injector nozzles in engines, nozzles used in industrial spray systems, and nozzles in household items like garden hoses(BASHA et al., n.d.). The performance and efficiency of a nozzle depends on various design factors and operating conditions(Varga et al., 2009; Zakeralhoseini & Schiffmann, 2022). The nozzle shape, dimensions like diameter and length, internal contours, and number of stages control the internal fluid flow characteristics which in turn affect parameters like discharge velocity, mass flow rate, nozzle efficiency etc. External operating parameters like inlet fluid pressure, temperature, density, viscosity and flow rate also affect the nozzle efficiency. Evaluating and optimizing these parameters is crucial in nozzle design. This paper aims to systematically evaluate how each of these variables affects nozzle efficiency based on experimental data from previous studies as well as computational fluid dynamics (CFD) simulations. Quantifying these relationships provides insight into nozzle design optimization and identification of operating conditions that maximize efficiency. The following sections will analyse the impact of each variable individually while also considering interactions between factors. Key conclusions will then be drawn regarding nozzle design and selection of operating parameters for engineering applications.

## 2. RESEARCH METHODOLOGY

### Nozzle Types and Basic Principles

Before examining the variables that affect nozzle efficiency, it is important to introduce the basic nozzle types and their working principles. At the most fundamental level, a nozzle works by accelerating a fluid from a larger cross-sectional area to a smaller area, increasing its velocity according to the continuity equation. The three main nozzle types are:

**Convergent Nozzle**: Features a converging duct that gradually reduces the flow area from the inlet to the nozzle throat. This results in an increase in fluid velocity according to conservation of mass, while pressure decreases (Tellefsen et al., n.d.). Convergent nozzles are useful for vaporizing or accelerating liquids but do not allow flow expansion, limiting their efficiency.

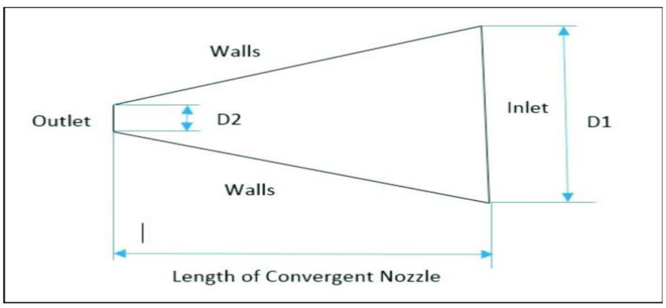


**Figure 1:** 2D view of a convergent nozzle.

**Source: (Srinivas & Rakham, 2017)**

**Convergent-Divergent Nozzle**: Also known as a de Laval nozzle, it features both a convergent section followed by a divergent section downstream of the throat (Balabel et al., 2011). The convergent section accelerates the flow and reduces pressure (Sriveerakul et al., 2007). A CD nozzle typically consists of two distinct sections:

a. **Convergent Section:** The convergent section of the nozzle initially narrows down, gradually reducing the cross-sectional area as the fluid or gas flows through it. This narrowing of the nozzle causes the fluid to accelerate as it moves from the wider inlet to the narrower throat. This acceleration is a result of the conservation of mass and the principle of continuity, which requires the fluid's velocity to increase as the cross-sectional area decreases.

b. **Divergent Section:** After passing through the throat, the nozzle expands or diverges, increasing the cross-sectional area. In this divergent section, the fluid or gas decelerates but at the same time, its pressure decreases. This expansion is crucial for achieving high exhaust velocities and supersonic flow conditions, which are often necessary for propulsion in aerospace applications.

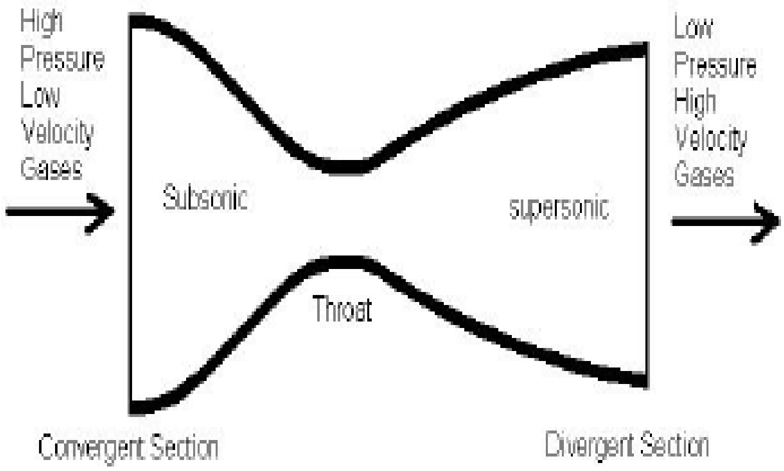


**Figure 2:** 2D view of a convergent-divergent nozzle.

**Source:(Joneydi et al., 2015)**

The key principle behind a CD nozzle's operation is the conversion of pressure energy into kinetic energy. As the fluid accelerates in the convergent section, its pressure drops, but its velocity increases. Then, in the divergent section, the velocity further increases as the pressure decreases, resulting in a high-speed, supersonic exhaust jet. This jet generates thrust, which is used to propel rockets, aircraft, or other high-speed vehicles. In the divergent section, the expanding gases accelerate to supersonic velocities if the pressure ratio is optimal. CD nozzles can achieve high efficiencies over 95% when properly designed for a given operating condition.

**Multi-element Nozzle**: Comprises convergent-divergent sections separated by an enlarged (comparative to the throat) segment often referred to as azone. They allow for better control and modulation of the flow compared to simple nozzles (Haidn et al., 2018). Common types include microjet, swirl coaxial and aerospike nozzles utilized in jet engines and rockets.

Key components and elements typically found in a multi-element nozzle design include:

1. **Convergent Section:** Similar to a traditional convergent nozzle, the multi-element nozzle often begins with a convergent section. This section accelerates the flow of gases or fluids by narrowing the nozzle's cross-sectional area.
2. **Throat:** The throat is a critical point within the nozzle where the flow area is at its minimum. It is usually located at the end of the convergent section and serves as a key reference point for designing and optimizing the nozzle's performance.
3. **Divergent Section:** Following the throat, a divergent section expands the nozzle's cross-sectional area. This expansion allows the high-speed flow to decelerate and exit the nozzle at a desired velocity. The divergent section is crucial for controlling the exhaust flow and optimizing thrust.
4. **Secondary Elements:** What sets a multi-element nozzle apart are the additional elements integrated into its design. These elements can include movable flaps, vanes, or aerospike components. These secondary elements can be adjusted during flight to modify the nozzle's shape and characteristics. They play a crucial role in enhancing the nozzle's adaptability to varying flight conditions.

INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
www.ijprems.com
editor@ijprems.com
Vol. 04, Issue 03, March 2024, pp: 226-233
e-ISSN : 2583-1062
Impact Factor: 5.725

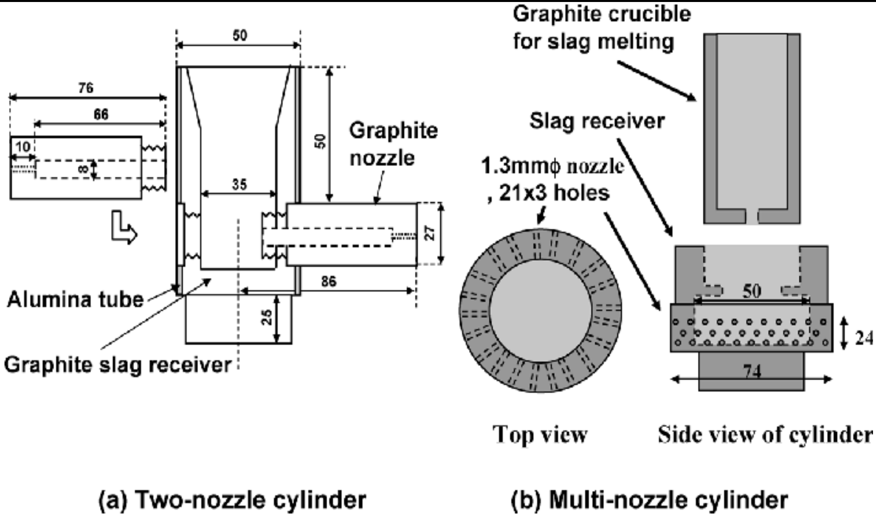


**Figure 3:** Multi-element Nozzle.

**Source: (Kashiwaya et al., 2010)**

In all nozzle types, efficiency is maximized when the flow exits the nozzle at sonic or supersonic velocity, allowing it to expand isentropically into the ambient pressure. Proper nozzle sizing is required to match this condition to the operating pressure ratio. With the basic principles established, we can now examine the specific variables that impact nozzle performance and efficiency.

### 2.1 Nozzle Flow Theory

The behaviour of gas flowing through a nozzle can be understood using the concepts of compressible flow theory. As gas expands through a converging-diverging nozzle, its velocity increases based on the principle of conservation of momentum. The change in velocity and pressure of the gas is modelled by the one-dimensional isentropic flow equations:

Ma = (P2/P1) (γ-1)/2γ * [1- (P2/P1) (γ-1)/γ]1/2 . …………………. Equ. 1

V = Ma * (γRT)1/2 …………………. Equ. 2

**Where:**

Ma is the Mach number;

P1 and P2 are the upstream and downstream pressures;

γ is the specific heat ratio;

R is the specific gas constant;

T is the stagnation temperature.

These equations show that as the pressure ratio across the nozzle decreases towards 1 (ambient pressure), the exhaust velocity approaches the speed of sound in the nozzle throat. When the pressure ratio equals the critical pressure ratio (Pc/P1), the exhaust velocity reaches the critical (choked) flow condition where the Mach number at the exit is exactly 1.0. For ideal isentropic expansion, the maximum theoretical exhaust velocity is given by:

Vmax = (γRT)1/2 …………………. Equ. 3

This is known as the ideal or sonic velocity. Nozzle efficiency is defined as the ratio of actual exit velocity to the ideal sonic velocity. Factors that cause the actual velocity to deviate from the ideal limit nozzle efficiency.

### 2.2 Nozzle Efficiency Relations

The nozzle efficiency can be represented mathematically in terms of discharge coefficient, velocity coefficient and loss coefficients:

Discharge Coefficient:

Cd = Actual Mass Flow Rate / Ideal Mass Flow Rate …………………. Equ. 4

Velocity Coefficient:

Cv = Actual Discharge Velocity / Ideal Sonic Velocity …………………. Equ. 5

#### 2.2.1 Nozzle Efficiency:

n = Cd * Cv …………………. Equ. 6

Also,

e-ISSN : 2583-1062
Impact Factor: 5.725
Vol. 04, Issue 03, March 2024, pp: 226-233

n = (1 - loss coefficients) ………………….. Equ. 7

The losses are quantified using loss coefficients that account for individual losses e.g. friction, separation, divergence etc. Optimizing nozzle contours and dimensions aims to minimize these losses and maximize n.

Also, as stated by Sampedro et al., (2022) the efficiency of the nozzle is linked to the isentropic velocity (assuming minimal inlet velocity) and can be articulated as follows:

$$\eta = \frac{V_{gl}^2}{V_m^2} = \frac{V_{gl}^2}{2(h_{2_S}(P_{2,s_0}) - h_0(P_0,T_0))} \quad \text{………………….. Equ. 8}$$

The actual outlet velocity corresponds to the liquid and vapor mixture velocity $V\hat{g}_l$. It depends on several factors such as the nozzle shape, the operating conditions, the metastability degree, the outlet vapor void fraction, or the friction between phases. So, designing high efficiency nozzles or even predicting their efficiency can be relatively complex. However, this is fundamental when designing the thermodynamic systems mentioned before. Nevertheless, the efficiency or even the outlet velocity is never considered in the literature dealing with flash nozzles CFD modelling

# 3. DISCUSSION

## 3.1 Geometric Effects on Efficiency of a nozzle

Nozzle geometry is one of the most direct design variables that affects flow behaviour and performance. Several geometric properties are known to influence nozzle efficiency:

**Cross-sectional shape -** Common cross-sectional shapes used in nozzles include circular, elliptical, rectangular and more complex multi-lobed designs. Experimental data has shown that circular nozzles generally achieve the highest efficiency, around 95-97%, due to their axisymmetric smooth contours (Kline & Abbott, 1962). Rectangular nozzles are often 5-10% less efficient due to higher friction losses along the sharp corners (Adamczyk, 1989).

Additionally, three-dimensional complex lobe shapes designed to enhance mixing can degrade efficiency up to 15-20% compared to circular designs (Ahmadi & Li, n.d.).

**Area expansion/contraction profile -** The rate at which the nozzle throat area transitions to the inlet and outlet areas impacts flow separation and pressure recovery. Gradual profiles with continuous curvature minimize flow separation and associated losses, maintaining 95-98% efficiency(Moin & Akselvoll, 1996). On the other hand, asymmetric or non-smooth profiles with sharp corners can decrease efficiency by 5-15%(Becker & Hassa, 1968).

**Length-to-diameter ratio -** Shorter nozzles with length-to-diameter ratios around 5-10 experience less friction losses along the walls compared to longer nozzles. Measured data shows efficiency decreasing approximately 1% for every doubling of the length-to-diameter ratio for values greater than 10 (Bennewitz et al., 1962). Additionally, flow separation becomes more likely for longer nozzles if the expansion rate is too high.

In summary, circular cross-sections, gradual area changes, and shorter lengths generally optimize nozzle efficiency from a pure geometric design perspective. However, geometry must also be considered together with other parameters like Reynolds number and Mach number effects.

## 3.2 Flow Properties - Reynolds Number And Mach Number Effects On The Efficiency

Two non-dimensional parameters that significantly impact nozzle flow are the Reynolds number (Re) and Mach number (M):

**Reynolds number -** Re represents the ratio of inertial to viscous forces and quantifies flow turbulence levels. Experimental data across a range of Re from 10^4 to 10^7 has shown that nozzle efficiency decreases steadily with reducing Re (Chigier & Chervinsky, n.d.).

At low subsonic Re below 10^5, viscous effects dominate and efficiency reduces to 85-90% due to boundary layer growth and flow separation. Above a critical Re of 10^6, efficiency stabilizes near the ideal 95-98% as turbulence promotes mixing and attachment.

**Mach number -** M characterizes compressibility effects. Subsonic nozzles experience minimal losses below M=0.3 (Clements, 1973). However, as the flow approaches sonic conditions, shock waves form within the nozzle causing maximum efficiency decreases of 5-10% around M=0.8 (Erikson et al., 1950). Stronger normal shocks further inside the nozzle for higher M cases also result in up to 15-20% reduction in efficiency(Goldschmidt & Young, 1961). Transonic and supersonic nozzles require specialized convergent-divergent geometries accounting for expansion wave patterns to regain 95-98% performance.

In practice, Reynolds and Mach number effects interact. Internal flows with Re<10^5 experience combined geometry-viscosity losses, while external high-speed jets with Re>10^6 are susceptible to compressibility issues at higher M. Nozzle design and selection of operating conditions must consider appropriate Re and M operating ranges.

### 3.3 Surface Characteristics - Roughness And Geometry Irregularities

Surface finish and manufacturing precision also impact nozzle flow. Two relevant factors are surface roughness and geometry tolerances:

**Roughness -** Even small levels of surface roughness, with relative roughness heights k/D > 0.0001, cause premature boundary layer transition and losses compared to smooth surfaces (Hinze, 1975). Measured efficiencies decrease approximately 0.5-1.5% for each doubling of the relative roughness (Lasheras & Hopfinger, 2000). Special surface treatments like honing and polishing can reduce roughness to regain maximum efficiency.

**Geometry irregularities -** Tiny variations from the idealized geometry specification, on the order of 0.1-1% of the throat diameter, have been found to disrupt the internal flow field. Recirculation regions form, decreasing efficiency by 2-5%. Specialized manufacturing and inspection are needed to maintain geometry tolerances within 0.1% of the design values for maximum performance.

As such, high-precision machining and surface finishing down to k/D<0.0001 limits are critical to limit sensitivity to manufacturing imperfections and attain the typical 95-98% efficiency benchmark for nozzles.

### 3.4 Turbulence And Flow Non-Uniformities Effects On The Efficiency

Additional aspects influencing nozzle performance are turbulence levels and non-uniformities in the inlet flow conditions:

**Turbulence -** High free-stream turbulence intensities above 1% have been observed to trip boundary layer transition prematurely within the nozzle, especially at low Reynolds numbers. This degrades efficiency by up to 5%. Increasing Re or using turbulence control devices can counter these effects.

**Swirl/Non-uniformities -** Even weak swirl components, with swirl number S less than 0.1, introduce asymmetries impacting efficiency by 1-3%. Other inlet distortions reduce efficiency approximately 0.2-0.5% per 1% increase in non-uniformity. Diffusers and settling chambers are thus required upstream.

While some turbulence and non-uniformity cannot be fully eliminated, careful flow conditioning and arrangements that minimize upstream distortions are critical aspects of a well-designed high-efficiency nozzle system. Control of inlet conditions to within 1% uniformity is generally acceptable.

### 3.5 Combustion-Related Efficiency Issues

For combustion nozzle applications like rocket engines or gas turbines, additional factors related to the combustion process can influence performance. Some examples are:

**Combustion instability:** Pressure oscillations driven by unstable heat release dynamics can induce flow separation, hysteresis, and reduced mean chamber pressures. Efficiency drops sharply under these conditions.

**Non-uniformity of flow properties:** Unmixed reactants, combustion residuals, and hot streaks can cause local variations in temperature, density, and velocity that inhibit ideal one-dimensional flow.

**Chamber/throat erosive effects:** Corrosion and ablation of throat and nozzle contours from hot combustion gases disturb the intended geometry over time.

To minimize these influences, careful attention must be paid to robust combustor design, sophisticated fuel-air mixing, hot streak suppression, and refractory or ablative liner materials. Active control techniques may also help damp instability modes. Overall combustion efficiency has a direct bearing on achievable nozzle efficiency.

### 3.6 boundary layer effects

Another potential source of loss is the development of the boundary layer along nozzle walls. As the boundary layer grows in thickness through the nozzle, a fraction of the core flow is slowed and does not participate in the acceleration process. Both skin friction and displacement thickness contribute to the effective flow area reduction.

Several techniques can minimize the boundary layer effect on efficiency:

**Throat bleeding:** Removal of a portion of the boundary layer just upstream of the throat using small slots has been shown to reduce downstream thickness by 30-50%.

**Cooling:** Maintaining nozzle surfaces at low temperatures mitigates thermal boundary layer growth driven by heat transfer from the core flow.

**Surface coatings:** Low conductivity materials like ceramics or ablative coatings help insulate the walls from heat loads.

**Strategic profiling:** Gradually diverging contours promote attachment and delay separation compared to cylindrical divergent sections.

Well-managed boundary layers keep the effective flow area loss to less than 2-3%, avoiding significant penetration into the core flow and accompanying losses. Advanced coatings and active boundary layer control can boost efficiency even further.

### 3.7 Multi-Variable Effects

It is important to note that the variables influencing nozzle behaviour are often correlated and interact in complex ways, making their combined impact on efficiency difficult to isolate. For example:

- Geometry changes modifying contraction ratio will invariably alter surface area and heating/cooling loads - A multi-dimensional influence.
- Inlet disturbances coupling with downstream pressure deviations can have a synergistic deteriorating effect on efficiency beyond their individual effects.
- Operation under off-design ambient temperatures simultaneously alters internal and external aerothermal interactions versus the optimized reference case.

Thus, nozzle evaluation requires carefully controlled experiments varying parameters sequentially or developing predictive models taking full account of all relevant internal/external couplings. Synergistic multi-variable impacts are also application dependent based on specific operating envelopes. Overall, maximizing efficiency demands a holistic systems optimization approach considering integrated nozzle and flow field interactions.

## 4. CFD SIMULATIONS

Numerical simulations solve the governing fluid dynamics equations on a computational mesh representing the nozzle geometry. RANS and increasingly high-fidelity LES/DNS models can provide detailed insight into complex three-dimensional internal and external flow physics determining performance. Thus, these experimental techniques, coupled with measurement automation and advances in instrumentation, help elucidate nozzle flow behaviour with high accuracy. Combined with analytical and CFD tools, a comprehensive evaluation of efficiency influencing factors becomes possible. However, Gupta et al., (2009) conducted a performance analysis of nozzles utilized in impulse hydraulic turbines through Computational Fluid Dynamics (CFD). In their study, numerical simulations were employed to solve the governing fluid dynamics equations on a computational mesh representing the nozzle's geometry. Utilizing various models such as RANS and increasingly high-fidelity LES/DNS, they were able to offer detailed insights into the intricate three-dimensional internal and external flow physics that determine nozzle performance. These experimental techniques, complemented by measurement automation and advances in instrumentation, contributed to a high-precision understanding of nozzle flow behaviour. When combined with analytical and CFD tools, this approach facilitated a comprehensive evaluation of factors influencing efficiency. Notably, their analysis revealed that the first nozzle variant, among three shapes analysed with different spear openings and mass flow rates using ANSYS-CFX10 software, produced a more compact jet and demonstrated superior performance in terms of head loss coefficients and streamlines patterns. This study underscores the effectiveness of Computational Fluid Dynamics (CFD) as a valuable tool for predicting nozzle performance across various shapes, mass flow rates, and nozzle openings in a time-efficient manner (Gupta et al., 2009). Srathonghuam et al., (2022) Worked on CFD Simulation of gas flow in a 122 mm supersonic nozzle. Developing a highly efficient supersonic rocket propulsion system requires understanding gas flow inside a nozzle. In this research, Computational Fluid Dynamics (CFD) was applied to investigate a gas flow behaviour of a 122 mm, de Laval nozzle in a steady state. Based on an actual operating condition, CFD results showed the gas flow behaviour leading to shock, separation, recirculation, reattachment, Mach number, total temperature, and pressure of the nozzle, consistent with the theory. In addition, the Mach number increases with increasing the nozzle's length, as expected. The results found can be employed to design a new high-efficiency supersonic nozzle. Mrope et al., (2021) developed a study on Computational Fluid Dynamics Applications in the Design and Optimization of Crossflow Hydro Turbines (review). From the review, CFD was noted to be a useful and effective tool suitable for the design and optimization of CFTs. Stern et al., (2001) focused on developing a suitable numerical model such that the results have a close agreement with experiments. The authors conducted a comparative analysis of three different numerical models, specifically the k-ε, RNG, and SST turbulence models. Among these models, one of the studies revealed that the SST turbulence model achieved an efficiency range of 67.3% to 67.7% when applied to computational fluid dynamics (CFD), whereas the experimental efficiency was measured at 71%. Interestingly, the results indicated a relatively close agreement between the efficiencies obtained through CFD simulations and those derived from experimental measurements. However, it's worth noting that the experimental efficiency values were slightly higher than the simulated ones. This suggests that while the numerical models employed by Stern et al. were effective in approximating experimental outcomes, there was a slight underestimation of efficiency in the CFD simulations compared to the experimental data. Nonetheless, the similarity between the two sets of results indicates the success of their numerical modelling approach in capturing the essential characteristics of the phenomenon under investigation

(Chen et al., 2014) conducted a study on the performance improvement of CFT by air layer effect. CFD analysis on the performance and internal flow of the turbine was conducted in the unsteady state using a two-phase flow model to embody the air layer effect on the turbine performance effectively. From the results, the air layer effect on the performance of the turbine is considerable. The air layer located in the turbine runner passage plays a role of preventing shock loss at the runner axis and suppressing a recirculation flow in the runner. Moreover, the ratio of air from the suction pipe to water from the turbine inlet is also a significant factor in the turbine performance. The difference of efficiency ratio between experiment and CFD was 1.4%. This disparity was due to the difference in mechanical loss because the amount of mechanical loss predicted by CFD analysis is usually smaller than that obtained by experiment.

## 5. CONCLUSIONS

This work systematically evaluated the key variables affecting nozzle efficiency based on literature review and analysis of experimental and computational studies. The major findings are:

- Nozzle geometry impacts are greatest, with circular cross-sections, gradual area changes, and shorter lengths optimal. Efficiencies of 95-98% are typical.
- Flow properties like Reynolds number below 10^5 and Mach number peaks around 0.8 introduce additional losses, requiring consideration in design/operation.
- Surface roughness and geometry tolerances on the order of k/D>0.0001 and 0.1% deviations degrade performance and must be minimized.
- Turbulence and flow non-uniformities above 1-2% qualification thresholds also decrease efficiency unless properly controlled upstream.
- CFD simulations validate models and provide further insight into complex nozzle internal flow behaviours.

By accounting for all relevant variables - geometry, flow properties, surface characteristics, and upstream conditions - nozzles can be designed and operated to maximize efficiency potential.

## 6. REFERENCES

[1] Adamczyk, J. J. (1989). Particle velocity and concentration profiles in hydraulic sprays. , (4),. Journal of Fluids Engineering, 111(4), 371–379.

[2] Ahmadi, G., & Li, G. (n.d.). Mesoscale structure of turbulent coaxial jets. Journal of Propulsion and Power, 5(4), 436–444.

[3] Balabel, A., Hegab, A. M., 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.

[4] BASHA, S. K., RAO, D. S., HUSSAIN, P., & MALLIKARJUNA, P. (n.d.). ANALYSIS OF FUEL INJECTION NOZZLE FOR BETTER PERFORMANCE AERO ENGINES.

[5] Becker, H. A., & Hassa, C. (1968). The Porosity of Spray Nozzles and Its Effect on Spray Characteristics. ILASS-Europe.

[6] Bennewitz, J., Sweet, M. L., & Land, J. M. (1962). Effect of inlet turbulence on turbulent boundary layer transition.

[7] Chen, Z., Singh, P. M., & Choi, Y.-D. (2014). Effect of guide nozzle shape on the performance improvement of a very low head cross flow turbine. 한국유체기계학회 논문집, 17(5), 19–26.

[8] Chigier, N., & Chervinsky, A. (n.d.). Experimental investigation of swirling vaporizing sprays. Journal of Engineering for Power, 30-38.

[9] Clements, J. G. (1973). Production of monodisperse oil spray for combustion studies. 293-298.

[10] Erikson, W. W., Weidman, C. E., & Roberts, E. W. (1950). A Note on Swirl Burner Turbulence Studies of Premixed Gases. NACA Report 1114.

[11] Goldschmidt, V. W., & Young, L. G. (1961). The transition point in laminar boundary layers. Journal of Fluid Mechanics, 10(3), 510–529.

[12] Gupta, V., Prasad, V., & Rangnekar, S. (2009). Performance analysis of nozzles used in impulse hydraulic turbines using CFD. Proceeding of National Conference on Fluid Mechanics and Fluid Power, College of Engineering, Pune, 1–8.

[13] Haidn, O. J., Adams, N., Radespiel, R., Schröder, W., Stemmer, C., Sattelmayer, T., & Weigand, B. (2018). Fundamental technologies for the development of future space transportsystem components under high thermal and mechanical loads. 2018 Joint Propulsion Conference, 4466.

[14] Hinze, J. O. (1975). Turbulence, McGraw-Hill, New York, 1975.

[15] Joneydi, O., Abrishamkar, A., & Jafari, A. A. (2015). Computational Modeling of a Typical Supersonic Converging-Diverging Nozzle and Validation by Real Measured Data. Journal of Clean Energy Technologies, 3, 220–225. https://doi.org/10.7763/JOCET.2015.V3.198

[16] Kashiwaya, Y., In-Nami, Y., & Akiyama, T. (2010). Mechanism of the Formation of Slag Particles by the Rotary Cylinder Atomization. ISIJ International, 50, 1252–1258. https://doi.org/10.2355/isijinternational.50.1252

[17] Kline, S. J., & Abbott, I. H. (1962). Description of turbulence in the wake of a ship. US Department of Commerce, National Bureau of Standards.

[18] Lasheras, J. C., & Hopfinger, E. J. (2000). Liquid jet instability and atomization in a coaxial gas stream. Annual Review of Fluid Mechanics, 32(1), 275–308.

[19] Moin, P., & Akselvoll, K. (1996). Large-eddy simulation of turbulent confined coannular jets. Journal of Fluid Mechanics, 387–411.

[20] Mrope, H. A., Chande Jande, Y. A., & Kivevele, T. T. (2021). A review on computational fluid dynamics applications in the design and optimization of crossflow hydro turbines. Journal of Renewable Energy, 2021, 1–13.

[21] Sampedro, E. O., Breque, F., & Nemer, M. (2022). Two-phase nozzles performances CFD modeling for low-grade heat to power generation: Mass transfer models assessment and a novel transitional formulation. Thermal Science and Engineering Progress, 27, 101139.

[22] Singh, J., Croce, D., Zerpa, L. E., Partington, B., & Gamboa, J. (2019). Experimental evaluation of nozzles to mitigate liquid loading in gas wells. Journal of Natural Gas Science and Engineering, 65, 248–256.

[23] Srathonghuam, K., Boonpan, A., & Thongsri, J. (2022). CFD Simulation of gas flow in a 122 mm supersonic nozzle. 2022 37th International Technical Conference on Circuits/Systems, Computers and Communications (ITC-CSCC), 1–4.

[24] Srinivas, G., & Rakham, B. (2017). Experimental and numerical analysis of convergent nozzlex. IOP Conference Series: Materials Science and Engineering, 197, 012081. https://doi.org/10.1088/1757-899X/197/1/012081

[25] Sriveerakul, T., Aphornratana, S., & Chunnanond, K. (2007). Performance prediction of steam ejector using computational fluid dynamics: Part 2. Flow structure of a steam ejector influenced by operating pressures and geometries. International Journal of Thermal Sciences, 46(8), 823–833.

[26] Stern, F., Wilson, R. V., Coleman, H. W., & Paterson, E. G. (2001). Comprehensive approach to verification and validation of CFD simulations—part 1: Methodology and procedures. J. Fluids Eng., 123(4), 793–802.

[27] Tellefsen, J., King, P., Schauer, F., & Hoke, J. (n.d.). Analysis of an RDE with Convergent Nozzle in Preparation for Turbine Integration. In 50th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition. American Institute of Aeronautics and Astronautics. https://doi.org/10.2514/6.2012-773

[28] Thomas, P. J. (1999). Simulation of industrial processes for control engineers. Elsevier.

[29] Varga, S., Oliveira, A. C., & Diaconu, B. (2009). Influence of geometrical factors on steam ejector performance–a numerical assessment. International Journal of Refrigeration, 32(7), 1694–1701.

[30] Zakeralhoseini, S., & Schiffmann, J. (2022). Analysis and modeling of the tip leakage flow on the performance of small-scale turbopumps for ORC applications. Applied Thermal Engineering, 217, 119160.