

The Design and Optimisation of a Micro-Jet Engine Combustion Chamber

Bradley Nolan

Mechanical & Aerospace Engineering, The University of Manchester, The United Kingdom.

Abstract

Modern micro-jet engines remain limited by anisotropic temperature distributions and over-reliance on legacy analytical design methods, which fail to capture relevant viscous effects and impose bold geometric approximations, restricting efficiency, durability, and integration with UAV propulsion systems. Despite the potential of computational fluid dynamics (CFD), with the capability to resolve these phenomena with far greater resolution, it is underutilised in current design practice, rarely coupled with optimisation techniques. To address these shortcomings, we must focus on the annular combustor and its effectiveness upon mixing coolant and reactant fluids. With a goal to i) implement a tangential annulus flow pattern design ruling and ii) optimise fluid interaction, resulting in improved combustor performance and durability.

Combustor geometry was generated from a combination of Lefebvre-based preliminary analysis and atomiser design principles, using outlet conditions of diffuser design studies as boundary conditions. A three-dimensional COMSOL Multiphysics® cold run CFD model was validated against analytical problems and existing academic work, complemented by a mesh convergence study (1.4 million cells) concerning the convergence of energy weighted average vorticity in the primary zone. The simulations revealed limitations in traditional analytical predictions, as well as quantifying improvements from implementing the TAP principle. Concerning hot run CFD, heat transfer was added to the simulation with a source in the primary zone. Further simulation of the atomiser validated the analytical approach due to similarities with experimental correlation. However, considering cost and machinability, it is concluded that commercially available technology can achieve the same performance.

Multi-fidelity mesh optimisation of airflow characteristics resulted in corrected oxygen supply in the primary zone, hence, reasonable grounds to assume the equivalence ratio shall be actuated upon running the combustor. Furthermore, improvements in pattern factor via Nelder-Mead optimisation - altering dilution zone and secondary zone positions - and cohesion between the TAP and Conventional Design, reached 88.2% with negligible pressure and efficiency losses and an optimised value of 0.05778. Beyond the specific findings, this report suggests an innovative design principle and provides a practical pathway to further research by using a multi-dimensional optimisation algorithm, experimental validation and integration of chemical reaction simulation, for more efficient and maintainable micro-jet engine combustors.

Keywords & Phrases: TAP – Tangential annulus flow principle, a method in which the bleed holes are plotted along the annulus in the direction of the swirling velocity vector from the compressor stage.

Primary zone – Where most or often all combustion takes place in annular combustors.

Secondary zone – Follows the primary zone, begins turbulent mixing and cooling of the hot reacted flow, after combusting remaining fuel (if any).

Dilution zone – Completes turbulent mixing and cooling of the flow prior to the turbine inlet plane.

pattern factor – A measure of temperature uniformity in the turbine inlet plane, a lower value represents a uniform distribution.

Equivalence Ratio – The actual fuel-air ratio divided by the stoichiometric fuel-air ratio.

Periodic Conditioning – Using a face of a simulation geometry as the inlet condition for another face, allowing for small scale models to represent the results of a large-scale model.

Vaporiser Tubes – In annular combustors, fuel is atomised into hot vaporiser tubes, where the fuel is evaporated allowing for more efficient, mixed combustion.

Introduction

Beginning where most great engineering stories do, during World War II, in March of 1941 a pulse jet was mounted upon a radio-controlled airframe, birthing the McDonnell TD2D Katydid, or perhaps more relevantly, the first jet-powered unmanned aerial vehicle. This breakthrough in aeronautics, began a multi-decade push of operational envelope and efficiency, leading to the development of micro-jet engines such as the BMW 8025 and Lyulka TS-31M (1950s). Furthermore, input of Arthur H. Lefebvre in 1983 formalised the annular combustor and its governing design principles. The aviation industry reached a new apex, which when coupled with the speculation of computationally aided design, made the future seem promising. This is not the case. As desire to miniaturise grew, issues with thermal distribution, vibration and fatigue became more apparent in combustors, companies that were successful became reluctant to publish detailed design methodologies, and those who failed disappeared. This leads to modern day design, with reliance on legacy analytics, sparse contributions to improvement and limited trace in the open literature; forming the plateau of efficiency, required maintenance and the ever-rising price the consumer market is subjected to. Despite the availability of large scale CFD, chemical reaction simulation and optimisation algorithms, these tools are rarely integrated into combustor design stages. Reviews of annular combustor design [1] confirm that analytical methods remain the industry standard despite demonstrated CFD capability, suggesting a gap between academic research and engineering practice.

This report aims to design and optimise a micro-jet engine combustor to improve temperature uniformity at the turbine inlet, increasing maintainability and endurance, imposed via CFD modelling and a clear optimisation workflow. The work outputs a new tangential-annulus flow design principle enabling enhanced cooling of the reacted fluid, an optimised combustor geometry and the framework for informed design to take place in further works. Impacting the cost of operations for UAV vehicles of this scale on a civilian level, but also the standard design procedure taken by combustion engineers.

Analytical Combustor Design

Liner Geometry & Fluid Flow Rates

This section follows principles denoted by Lefebvre [10] and implemented by Santoso et al. [1].

Inlet combustor parameters are defined as: mass flow rate of air = 0.7 kg/s, inlet pressure = 4.4 bar, inlet temperature = 523.15 K and inlet radii = 10.578 cm (inner) 11.578 cm (outer) attained from compressor stage design and CFD simulation [2]. Assuming steady-state, axisymmetric flow with negligible inlet heat loss, a turbine inlet temperature of 1100 K [6] and pressure loss of 5% [3], analysis may begin with computation of a reference area and complementing corrective parameter 'K', yielding a liner radius of 9.748 cm.

$$A_{ref} = \sqrt{\frac{R}{2} \left(\frac{\dot{m}_{air} \sqrt{T_{in}}}{P_{in}} \right)^2 \left(\frac{P_{in}}{q_{ref}} \right)}$$

$$K = 1 - \left(\frac{(1 - \dot{m}_{air})^2 - \gamma}{\frac{\Delta P}{q_{ref}} - \gamma r^2} \right)^{\frac{1}{3}} \quad A_{liner} = A_{ref} K$$

Fuel mass flow rate is calculated from an energy balance between power required to heat the air mass and the power released from the combustion of a mass of kerosene; this approximation neglects the heating of fuel products, however, as this represents a small fraction of the total fluid, is deemed valid. Assuming a combustion efficiency of 95% and specific heat capacity at average combustor temperature [4], it is determined 0.0114 kg/s of fuel releasing 0.49101 MW is required.

$$\dot{m}_{fuel} = \frac{\dot{m}_{air} C_{p,hot} (T_{out} - T_{in})}{LHV_{kerosene} \zeta}$$

Airflow is partitioned across successive zones, governed by combustion and cooling requirements. To determine first, the primary zone airflow, we must set an equivalence ratio of 0.9 (as the optimisation of pattern factor shall negate the otherwise anisotropic temperature distribution introduced at the turbine) and stoichiometric ratio of 15.6.

$$\dot{m}_{pz} = 15.6 \cdot \eta \cdot \dot{m}_{fuel}$$

Secondary zone air mass is split into parts i) to combust the remaining fuel, and ii) to begin cooling mixing of the air – determined as a chosen fraction of the inlet air mass, in this case 30% [10]

$$\dot{m}_{Sz,combustion} = 15.6 \cdot (1 - \eta) \cdot \dot{m}_{fuel}$$

$$\dot{m}_{Sz,mixing} = (0.3 \cdot \dot{m}_{in}) - \dot{m}_{Sz,combustion}$$

Calculation of zone lengths is based on empirical relations (Primary and Secondary) and correlation (Dilution).

$$L_{pz} = L_{Sz} = \frac{2}{3} D_{Liner}$$

$$L_{Dz} = D_{Liner} \left(3.83 - 11.83 \frac{\dot{m}_{pz}}{\dot{m}_{In}} + 13.4 \frac{\dot{m}_{pz}^2}{\dot{m}_{In}} \right)$$

Now, assuming a drag coefficient of 0.65 [7], a pressure drop of 3% [10] and a number of bleed holes in each zone informed by Lefebvre's work, geometry can be finalised using the equations below.

$$D_{Zone} = 2 \sqrt{\frac{\dot{m}_{Zone} \sqrt{2 \rho_{In} \Delta P_{loss}}}{\pi C_d n_{Bleed\ Holes}}}$$

Table 1: Combustor geometry specification

Zone	Primary	Secondary	Dilution
MFR [kg/s]	0.1603	0.21	0.3297
Hole Diameter [mm]	4.7	6	7.6
No. of holes	50	40	40

Imposition of the Tangential Annulus Flow Principle (TAP)

Conventional combustion chambers align bleed holes axially, neglecting the swirl component of the inlet air. It is proposed that the bleed holes should be plotted in line with the velocity vector. By aligning the holes with the swirl induced flow, resulting geometry shall theoretically propagate the same airflow into the Primary zone. Whilst downstream the inlet for each bleed hole will be admitting distorted air from a previous inlet, allowing more turbulent flow to enter the inner combustor, aiding mixing. The magnitude of swirling and the resulting velocity vector is attained from CFD study of the diffuser stage by McNeill [2], resulting in a swirl number of 1.4. Radially velocity can be computed as 10.5055 m/s via the equation below.

$$V_{radial} = 2 \cdot V_{in} \cdot S \cdot \frac{r}{R}$$

This derived velocity distribution allows computation of streamlines along the annulus; via projection of these onto the liner surface, bleed hole coordinates are determined in alignment with the local velocity vector – defining the TAP geometry. Note that for the initial simulations of this report, both designs (conventional axial and TAP) shall be used to conclude an optimal design to proceed with for hot run optimisation.

Atomiser analytical design

First, a swirl pressure atomiser is chosen for this system, due to its lack of moving parts and relatively simple design, with no compromise of performance. This atomiser works by transferring a pressure driven fluids kinetic energy, partially into swirling vortex energy; the resulting rotation causes

centrifugal force on the particles, spreading the fluid at a given injection angle “ 2θ ” upon release into a domain. Assuming a target spray angle of 180 degrees – corresponding to the upper possible limit – allows for the maximum spread design, as it is determined from experimental correlation [7] that analytics vastly overestimate the spray angle. By imposing an injection pressure difference of 4 bar, we allow for the required vorticity effects to form – informed by similar scale systems. In cohesion with the geometric properties of Ghaffar [9], the Hagen-Poiseuille equation [5] can be used below to find the diameter of the inlet pipe as 0.60583mm.

$$D_p = \left(\frac{0.1 \cdot \dot{m}_{fuel}}{\pi \rho_k \Delta P} \cdot 128 \mu_k \right)^{0.25}$$

It is now pertinent to validate the laminar flow assumption by instilling an upper limit of 2300 to Reynolds number and working backwards to calculate the maximum laminar flow velocity – 8.952 m/s; yielding a much lower flow rate than required. This directly enforces the requirement of CFD validation, potentially prompting corrective geometry changes in further design. Via relation of the diameters of the outer (D_o) and choked (D_s) sections through two equations [8] we can iteratively solve for a solution, generating the geometry in described in Table 2.

$$K = \frac{A_p}{D_o D_s} \quad 2\theta = 6K^{-0.15} \left(\frac{\Delta P D_o^2 \rho_L}{\mu_L^2} \right)^{0.11}$$

Where K is the atomiser geometrical constant.

Table 2: Atomiser geometry specification

Dimension	D_o	D_s	D_p
Size [mm]	1.6	4	0.60583

Prior to simulation, it is necessary to acknowledge and mitigate the inaccuracies introduced by the absence of vaporiser tubes present in the geometry. Omitting this feature allows the flow to propagate without physical constraints imposed in operation. Therefore, a conceptual representation must be incorporated. For both the TAP and conventional designs, the tube is oriented from the turbine plane, between the planes of Primary Zone air injection. This shall be done at a height above the neutral axis which coincides with the largest airmass.

Simulation Setup

Computational Fluid Dynamics allows us to validate the analytics and identify any discrepancies resulting from the reduced order one-dimensional formulation. These simulations will provide spatially resolved velocity, pressure and temperature fields, enabling the assessment of performance against analytical targets.

Solver Selection & Model Details

The model is configured in COMSOL Multiphysics®, using the Reynolds Averaged Navier Stokes equations with the Shear Stress Transport turbulence model. This physics model solves the Reynolds averaged continuity and momentum equations, closed with the Boussinesq hypothesis and two additional transport equations for turbulent kinetic energy (k) and specific dissipation rate ω . The SST formulation blends the $k - \omega$ model near walls with the $k - \epsilon$ resolution in the free stream for improved boundary layer separation prediction and overall resolution. Note that higher fidelity solvers are available, including Large Eddy Simulation, Smoothed Particle Hydrodynamics and particulate solutions, however, due to the computationally and time constrained nature of this work, the RANS-SST model is chosen

– blending accuracy and efficiency. Aiming to mitigate computational cost, simulation is conducted of a one tenth slice of the engine in isolation; achieved using Periodic conditioning with interpolating functions allowing nonidentical mesh compatibility.

Replication Validation Work

By replication of an annular combustor study [1], we introduce confidence to the model; successful replication demonstrates the solver configuration, meshing strategy and turbulence model can capture the dominant physics. Using data available in the reference paper and pixel counting for unknown dimensions, the study geometry can be recreated. Noting some potential deviation in parts, however, concluding that the essential geometric features to replicate flow behaviour are preserved.

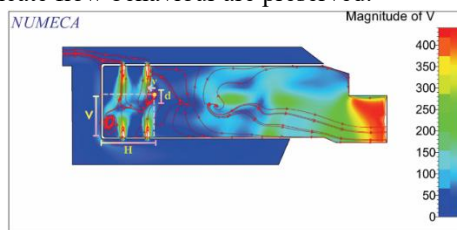


Figure 1. Results attempted to replicate [1]

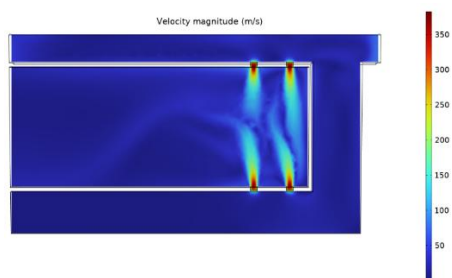


Figure 2. Replication Study Results

The model reproduces the expected jet nozzle at bleed holes, with a 4.5% discrepancy in Primary zone peak velocity. Reasons for this may be that the study here used a cell count of 10.8 million, 3.2 million less than is later discovered as the required cell count for a converged solution. In addition, large differences between the analytics and model are found, perhaps discrediting the NUMECA solver or methods used in the simulation setup – enforced by the absence of validation work. From this it can be determined that the solver demonstrates credible accuracy. Further confidence shall be obtained by using simulations. The closeness to the results of analytical design stages will highlight deviation if it arises.

Mesh Convergence Study

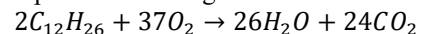
The meshing algorithm used is the physics-informed tetrahedral mesher in COMSOL, utilising a conventional convergence study approach in which samples are taken across a domain, initially from 600,000 to 1.1 million cells. Evaluating energy weighted averaged vorticity in the centre of the primary zone generates data representative of density, velocity and vorticity, which can be used to determine field convergence.

$$\text{Avg. Vorticity [1/s]} = \frac{\int (0.5\rho(u^2 + v^2 + w^2)\sqrt{\omega_x^2 + \omega_y^2 + \omega_z^2}) dA}{\int 0.5\rho(u^2 + v^2 + w^2) dA}$$

The initial sample space disclosed non asymptotic convergence, that was not yet completed, therefore a new domain varying 1.1 to 1.5 million was simulated, concluding a convergence at 1.4 million cells to 1.012% error. Whilst higher cell counts would represent a more accurate solution, they exceed the capacity of the computational tools available.

Hot Run Simulation

Utilising the reaction below, with surrogate Kerosene fuel - Dodecane - we can implement a heat flux representative of its completion at the centre of the primary zone. By implementing the heat flux ourselves into the engine, the species formed by the fuel and their thermodynamic implications on the downstream flow are neglected, but the model can be solved efficiently. Due to the small fraction that the product species represent in the model, the results obtained through this method remain credible. Furthermore, improvements in pattern factor from this work will be synonymous to actual improvements, even in the absence of explicit species modelling.



This method is representative of operational conditions, with small change during transient operation (startup & shutdown) where the fuel may not completely vaporise, as this study concerns

operational endurance conditions, this is deemed valid. Similarly, operation of the engine is possible with non-vapour fuel in startup configurations, however, penalties in efficiency will be prominent. Accordingly, a power input into the Primary zone is applied, equal to the power released from one tenth of the fuel reaction (as the model is for a one tenth slice), calculated with the equation below, concluding 49101 Watts.

$$P_{released} = \left(\frac{1}{10} \dot{m}_{fuel} LHV_{Kerosene} \right)$$

Implementing this flux with a heat domain and coherent periodic conditioning in the Heat Transfer in Fluids physics interface, a base turbine inlet temperature of 1011.8 K is obtained – within the expected temperature region. A reason for discrepancy between this and the analytical target of 1100 K being that some power is trapped behind the recirculating zone of the Primary Zone, this is due to the method of imposing the heat flux and could be rectified with reaction modelling; leading to the preposition of future works.

Atomiser CFD Simulation

Using a time dependent Level Set Mixed Flow Physics, we can distinguish between two fluids (air and kerosene) and govern their interaction; as prior, a RANS-SST solver shall be used. In attempts to reduce the runtime of this model whilst attaining results that can indicate suitability, slip wall conditioning is implemented; an advantage of this is that we are able to directly verify the analytics used, which do not account for wall viscosity. However, to gauge operational behaviour experimental correlation must be adopted. A mesh convergence study is conducted in the same method as the combustors', resulting in 200,000 cells being required for 0.954% convergence.

Atomiser Experimental Correlations

Firstly, the Giffen and Massey [7] correlation is highlighted, noting that it consistently over predicts the spray angle, even for much smaller atomisers which it was designed to be used for.

$$\tan \theta = 0.169 v_L^{-0.131}$$

Furthermore, the simplified semi-empirical correlation proposed by Sun [7] can be used also; noting its dependence on three experimentally determined fitting parameters (Q, M & N), which can only be attempted to solve for from CFD data – where the spray angle 2θ is equal to the square root below.

$$\sqrt{1 + \left(1 - 10.7348 Q \frac{\rho^{0.1} \mu^{0.2} L_o^{0.8}}{\Delta p^{0.1}} \left(1 + M \frac{r_s - r_o}{L_s \tan \alpha} \right)^2 \left(\frac{W_{\delta\theta}}{U_{\delta x}} \right)^2 \right)} \exp \left(N \left(\mu \sqrt{\frac{\rho}{\Delta p}} \right)^{0.25} r_o^{-0.75} L_o \right) - 1$$

As a secondary approach, the correlation above can be used with our design parameters, and existing

atomisers of similar scales experimental parameters (Q, M & N) to attain a reliable solution, to be used in conjunction with other correlation to determine the success of the design.

Optimiser Development

Focusing on the axial position of the secondary and dilution zone, improvements in pattern factor can be attained; possibly resulting in a lower average turbine inlet temperature, meaning that more thrust could be attained for the same engine size. Firstly, geometric boundaries must be defined to avoid overlap of the zones, respectively as (68.4157, 76.5978) & (97.2025, 137.782) – the coordinate system is described below as millimetres from the origin axis.

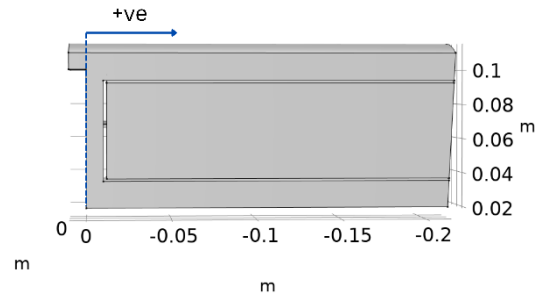


Figure 3. Coordinate reference

A random sample of design points is generated via Latin Hypercube Sampling, a function built into the pyDOE2 library in Python; for this study an initial sample size of 24 was chosen due to the expected linear slope of results. From this, each design is simulated and its pattern factor calculated and the data is fed to a Gaussian Process Regressor – which fits a surrogate model to be used by an L-BFGS-B optimiser to determine the minima. This prompts the recursive simulation of the minima until a convergence criterion is reached. Attempting to avoid a false minimum the script is validated with test data containing a known minimum, in conjunction with this uncertainty data is analysed at each stage to determine sensibility of results.

Results

Cold Flow Simulation of the Conventional and TAP Design

It is important to examine the primary zone in isolation for both designs and evaluating the optimal via comparing recirculation characteristics. The designs yield an energy weighted averaged vorticity of 4195.85 1/s and 3783.56 1/s for the conventional and TAP respectively; from this it is inferred that either a less turbulent or less recirculating flow is propagated into the TAP design primary zone. Via further visual examination of circulation characteristics in the plot

below, it is concluded that the conventional design is superior in recirculating flow for a stable flame.

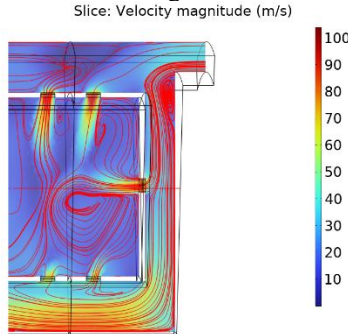


Figure 4. Primary Zone Velocity Profile for the Conventional Design

It is important to note that a less turbulent flow in the primary zone is ideal to some extent, however, in this case turbulence overpowers the necessary mechanisms in the TAP design. Proceeding solely with the TAP design would result in a higher equivalence ratio than desired in the combustor and potentially catastrophic failure of the engine. As a result, a design that combines the primary zone principles of Lefebvre and the TAP for the secondary and dilution zones is reasoned as superior, denoted hereby as the Nolan annulus pattern design. For further optimisation work concerning the mixing of the combustor, the TAP design shall be proceeded with, as the improvements will be theoretically synonymous for both the TAP and Nolan annulus pattern design.

Deviation in Primary Zone Mass Flow Rate

The simulation results yield lower zone mass flow rates than the analytical predictions.

Table 3: Mass Flow Discrepancy

MFR [kg/s]	Primary	Secondary	Dilution
Analytics	0.1603	0.21000	0.32970
Conventional	0.14669	0.20461	0.33414
TAP	0.14799	0.29660	0.3365

Examining the mass flow in the Primary zone for the Conventional design, we find the error to be - 8.49%; indicating the reaction shall be starved, causing a significantly larger portion of combustion to occur in the secondary zone and reducing the cooling effect of added fluid supplied in the dilution zone. To rectify this, a range of primary zone bleed hole diameters are sampled to determine the correct diameter to deliver 0.1603 kg/s of air – achieved using a multi-fidelity meshing approach.

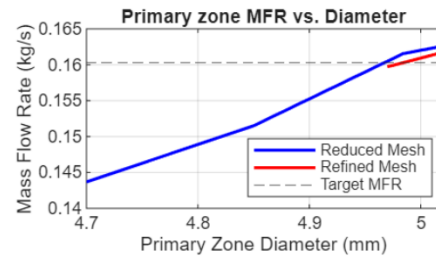


Figure 5. Primary Zone Mass Flow Rate Correction

Refinement here was completed until convergence lied within a 0.506% bound of the target mass flow rate, representative of half the worst case numerical dependent error outlined in the mesh convergence study. By imposing half the error of the convergence here, the compound amplification of final error in Primary zone mass flow rate is minimised whilst attaining reasonable computational strain. Combining the two independent errors using a quadrature results in $\pm 1.131\%$ confidence. Implications of this are directly that the equivalence ratio may be over or under-actuated by this amount due to the overfed or starved reaction. Furthermore, how this contributes to pattern factor and combustion efficiency can be assessed with further chemical reaction simulation, however for further error analysis this will be treated as a direct independent error to pattern factor optimisation.

Hot Flow Simulation

The initial design simulation result in the turbine inlet temperature distribution to the left of Figure 9, yielding a less than adequate pattern factor of 0.49. Furthermore, a turbine inlet average temperature of 1049 K is determined, indicating the design is operable in practice; however, such an anisotropic temperature distribution suggests elevated thermal loading on the turbine stage, thereby reduction of component lifespan and maintainability.

Optimisation of the TAP Design

By running the 24 Latin Hypercube samples the heatmap in Figure 6 is generated.

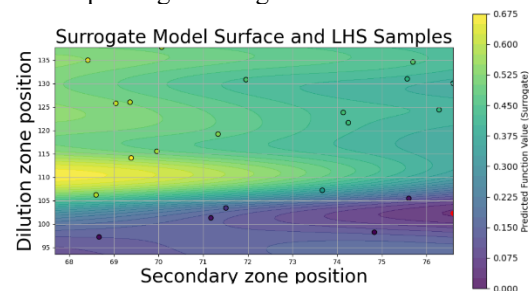


Figure 6. Optimisation Heatmap (Sampling Stage)

From the initial 24, 21 simulations ran as expected, translating to an 87.5% success rate – large enough to warrant higher order optimisation, with built in

error handling functions. Note the initial suggested minima (red point) which lies on the upper boundary of Secondary zone position, this is attributed to there not being enough data points locally to close the surrogate contours and shall be rectified upon following the workflow. From the sampling stage a pattern factor of 0.136 was found, showing that simple exploration of a sample space, can be extremely beneficial to development and durability of combustors. A further 26 runs yielded a pattern factor of 0.05778, or an 88.2% reduction. Considering the validity of this data, the previously highlighted mesh convergence error bound of 1.012% is adopted, resulting in a confidence interval of ± 0.0005847 . Despite this error, the optimisation is considered massively beneficial to the design workflow and shall likely result in more significant improvements when subjecting higher order dimension optimisation.

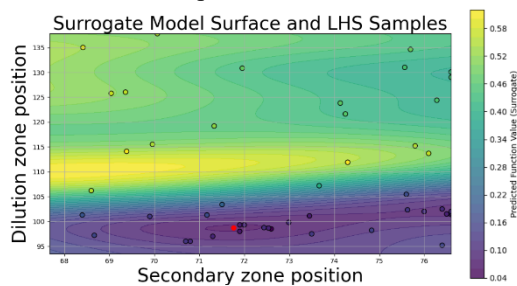


Figure 7. Optimisation Heatmap (Final Stage)

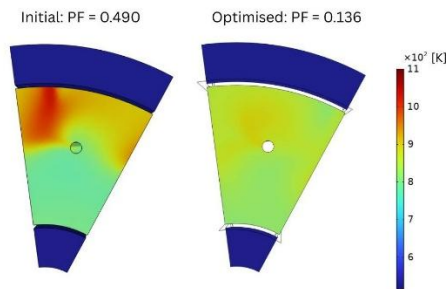


Figure 8. Comparison of turbine inlet temperature distribution

Regarding relationships observed, the injection of dilution zone air as upstream as possible in this case is beneficial. Regarding secondary zone, the same trend is not found, instead we see an optimal at the midpoint of our bounds; explained by considering the turbulent, recirculating flow inside the chamber close to the secondary zone, causing understanding of fluidic interaction to be invalid & fronts of circulation to dominate the mixing of the flow. Using the optimal design point: Secondary zone- 71.76 mm & Dilution zone- 98.69 mm, the Nolan Annulus Pattern geometry can be built, and verification of this optimisations' application can be undertaken.

Assessment of the Nolan Annulus Pattern Design

A pattern factor of 0.182 is computed, complemented by a maximum turbine inlet temperature of 923.16 K. The pattern factor evaluated here, is larger than the optimised TAP design, however, this penalty is considered acceptable for the improved recirculation characteristics discussed prior, which can attain a stable, safe, flame in operation. Building on the reduced turbine inlet temperature, it is now theoretically possible to burn a higher portion of fuel in this engine – reaching an optimal thrust to weight ratio. This can be quantified by evaluating the temperature difference between simulation and the final design as 176.84 K and using an energy balance to compute that an additional 0.0035 kg/s may be burnt.

Atomiser CFD Simulation

After successful simulation the spray angle is extracted using high contrast filters and pixel counting methods, as this is not a direct output from COMSOL, determining 77.32° . It is concluded that this value is sufficient to design an operable vaporiser tube, however, as discussed, the expected discrepancy due to simulation simplifications applied requires the use of experimental correlation.

Table 4: Atomiser Experimental Correlation Results

	Rizk	CFD	Giffen & Massey	Sun & Tian	Lei Sun
Spray Angle	77.59	77.32	39.73	44.89	44.1

Correlation consistently predicts a much lower spray angle, within the expected range, however due to the range of these values prompts for pending experimental validation.

Conclusions

As a result of work conducted in this project, a novel design principle - the "Nolan Annulus Pattern Design" has been discovered as a leading tool, aiding the modern design of annular combustors. Reasoned by the cohesion of two previously examined designs' advantages: the conventional with improved recirculation characteristics in the Primary zone, and the TAP- with optimised mixing in the Secondary and Dilution zone, yielding this optimal design. Beyond specific deliverables, workflows suggested here carry broad applicability across engineering disciplines, and their adoption at early design stages is strongly suggested. Furthermore, future validation work shall take place prior to investigating the results of higher order and multi-target optimisation of the annular combustor.

References

- [1] R. R. Santoso, “Design of mini turbojet engine combustion chamber liner,” *Indonesian Journal of Aerospace*, vol. 21, 2023.
- [2] O. Mcneill, “Development of a surrogate-assisted optimisation workflow for radial compressor design using fully-automated CFD processes,” *Unpublished thesis - The University of Manchester*, 2026.
- [3] U. Honegger, “Gas turbine combustion modelling for a parametric emissions monitoring system,” *Kansas State University*, 2009.
- [4] NASA, “Propert tables and charts (si units),” *Wayne State University*, 2025.
- [5] J. X. J. Zhang, “Microfluids and micro total analytical systems,” *Molecular sensors and nanodevices*, 2014.
- [6] H. I. H. Saravanamuttoo and G. C. Rogers, *Gas Turbine Theory*, Pearson, 2009.
- [7] L. Sun and D. Tian, “Available simplified semi-empirical correlation to predict hollow spray cone angle for practical pressure swirl atomisers with viscous liquid,” *Triennial International Conference on Liquid Atomisation and Spray Systems*, 2021.
- [8] N. Dombrowski and D. Hasson, “The flow characteristics of swirl (centrifugal) spray pressure nozzles with low viscosity liquids,” *AIChE Journal*, vol. 14, no. 4, 1969.
- [9] Z. A. Ghaffar, “Characteristics of spray angle and discharge coefficient of pressure-swirl atomiser,” *Advanced research in fluid mechanics and thermal sciences*, 2026.
- [10] D. R. Ballal and A. H. Lefebvre, *Gas Turbine Combustion: Alternative Fuels and Emissions*, Third Edition, CRC Press (Taylor & Francis Group), 2010.

Further information

This paper presents a condensed account of a broader research project, for any further enquiries or a copy of the complete report please contact bradley.nolan@student.manchester.ac.uk.

Acknowledgements

I would like to express my sincere gratitude to Professor Yasser Mahmoudi Larimi for his guidance throughout this project. I am also grateful to Dr. Seohee Jang, who consistently met with me over the past seven months and offered valuable insight that would otherwise have been overlooked. As well as this I would like to thank Oliver Mcneill for his invaluable input to the wider engine design process.

Finally, I wish to thank my partner Keira for her support and encouragement through happy - and some difficult, times during the writing of this report.