Owen Tolve
University of New Hampshire, Durham, NH, USA

Abstract

The key function of a nozzle is to provide thrust in order to propel the object. Through research done by NASA and other industries using nozzles, by expanding the flow at a proper rate, the thrust can be optimized. However, when using smaller nozzle designs, the boundary layer can have a negative effect on the flow as it uses most of the available space. Additionally, it was found that the angle of the nozzle expansion affected the flow of the fluid, as an angle of 15° allowed for less skin friction [1]. Finally, it is important to regulate the development of the flow as it exits the nozzle. As a fully developed flow means the velocity gradient is uniform, then a fully developed flow will reduce the overall thrust due to a lower velocity[1]. In this case it is important to try and reduce the development of the flow to have a less uniform velocity gradient with higher velocities. In order to optimize a nozzle for use on a drone in a low-pressure system, a simplified 3D model was built based off Fig. 1. The model was performed on the expansion portion shown in Fig. 2 using initial condition calculations. The results of the study suggest that the development of the boundary layer caused the velocity gradient to remain constant throughout the tests.

Nomenclature

F_T	Force generated from thrust
$\dot{m}$	Mass flow rate
V_e	Fluid exit velocity of nozzle
P_e	Pressure at nozzle exit
P_0	Pressure outside nozzle (atmospheric)
A_e	Nozzle exit cross-sectional area
CO_2	Carbon Dioxide
ID	Inner diameter
Q	Volumetric flow
R_e	Exit radius
R_t	Throat radius
A	Expansion angle

1 Introduction

The author is part of the University of New Hampshire's Quad-X Swarm teams and working on developing a hexacopter drone. The project encompasses movement generated from propellers, with additional movement through use of thrust generated from compressed CO_2. This will allow for translational movement in the longitudinal and latitudinal directions as a first-step process in simulating the dynamics of a satellite.

As the fluid subsystem of the drone contains a limited supply of CO_2, both the mass flow rate and exit pressure will change, causing a non-linear flow. Using a Lee Company high speed in-line solenoid valve for each nozzle, the mass flow rate at various pressures can be calculated allowing for a map of the thrust over time after optimization was complete. The nozzle design was tested using expansion angles of 2° to 10° in increments of 2°, with an exit diameter of 1.5 mm allowing for an extensive mapping of data. After meshing the different designs, they were all run in Open FOAM to get results.

The testing provided relevant data, while slightly inaccurate due to two assumptions, the first of which being incompressible flow. The assumption allows for simpler simulations as compressible flow

[1] Thrust Equation: $F_T = \dot{m} V_e + (P_e - P_0) A_e$

requires extensive processing power and this assumption then assumes the fluid is hydraulic rather than pneumatic. The second assumption was using a square pipe mesh rather than a circular. This assumption allows for simpler meshing, while increasing the frictional loss due to a higher surface to wetted area [2].

2 Experimental Setup

2.1 SolidWorks

In order to perform the analysis, the nozzle was first designed in SolidWorks, shown in Fig. 1. Using this model, the initial conditions of the flow entering the expansion portion in Fig. 2 were calculated to reduce the number of cells, in turn reducing the runtime per test. Using the conservation of mass[2], the initial conditions led to an initial velocity of 6.58 m/s at the inlet of the expansion. This model was used in every test in order to calculate the change in length of the expansion due to the change in angle.

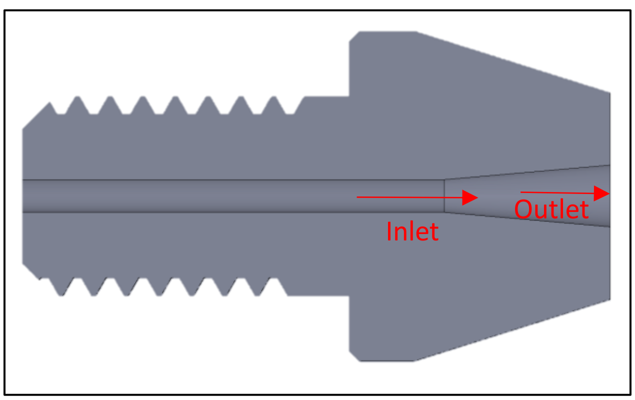

FIGURE 1. CROSS-SECTION OF NOZZLE WITH LEAD FROM SOLENOID VALVE. INLET AND OUTLET ARE SHOWN TO INDICATE THE DIRECTION OF THE FLOW. NOTE EXPANSION ANGLE IS 5°.

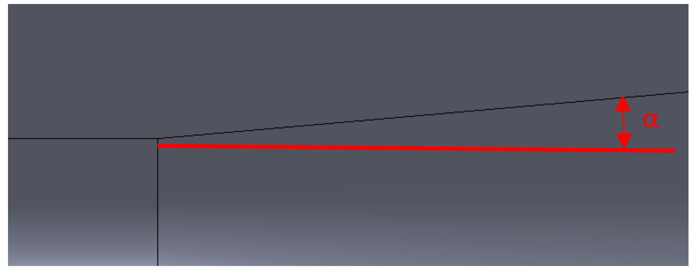

FIGURE 2. DETAILED VIEW OF THE NOZZLE'S EXPANSION SHOWING THE EXPANSION ANGLE

2.2 OpenFoam

Next, a blockMesh file was created in OpenFoam in order to define the region of the flow. The mesh used a single block of 8 points defining 100 cells in the streamwise direction and 10 in both the height and width totaling 100,000 cells. This led to a quick runtime, with data being recorded in 5 second periods and being written to view every 500 seconds.

Before writing any additional files, the effect of a sudden expansion on laminar flow was studied. This led to the finding that even though the flow was calculated to be fully developed and laminar before entering the expansion, the change in geometry can cause a sudden increase in turbulence [3]. This led to the use of simpleFoam as a solver as it is useful for studying incompressible flow. Due to the increase of turbulence, the solver used a k-ω, turbulence profile. The dynamic viscosity of CO_2 was then defined using the constant/transportProperties with a value of $7.67\text{x}10^{-6}$. Finally, the inlet velocity was defined in the 0/U directory finishing the setup for the simulation.

[2] Q = A * V

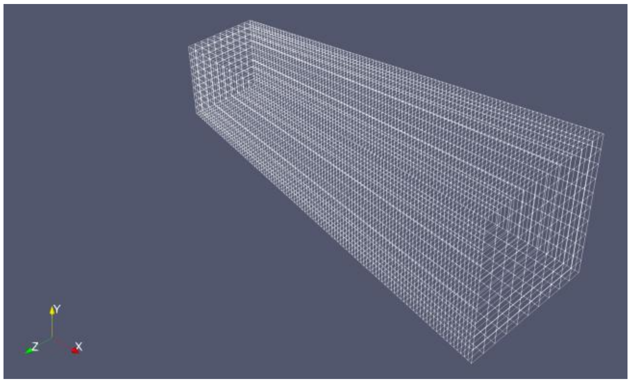

FIGURE 3. ISOMETRIC VIEW OF MESH IN PARAVIEW. NOTE EXPANISION ANGLE IS 5°.

3 Results and Discussion

All the data from the tests suggested that the overall velocity of the fluid is much higher than originally calculated using Bernoulli's Equation. The tests converged in a range of 2230-4210 iterations with an error of $1x10^{-6}$. When looking at Fig. 5 through Fig. 10, the velocity gradients at the exit are identical. This means that the expansion of the nozzle had an effect on the boundary layer development, causing the flow to have very little development. As the geometry is square rather than circular, there is added surface friction, meaning that the gradient should be slightly more expanded, with a smaller boundary layer [2].

Expansion Angle	# of Iterations
2°	2230
4°	2305
5°	2845
6°	3065
8°	3400
10°	4210

FIGURE 4. TABLE DEFINING THE EXPANSION ANGLE VERSUS THE NUMBER OF ITERATIONS.

2°

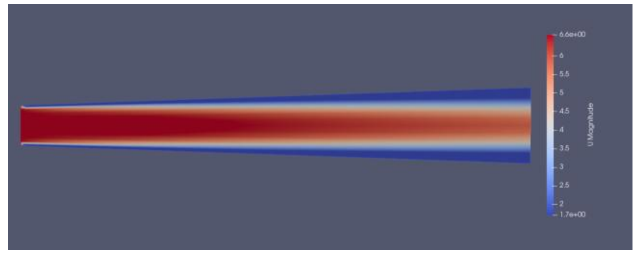

FIGURE 5. CROSS-SECTION OF THE FLOW THROUGH THE NOZZLE WITH AN EXPANSION ANGLE OF 2°.

4°

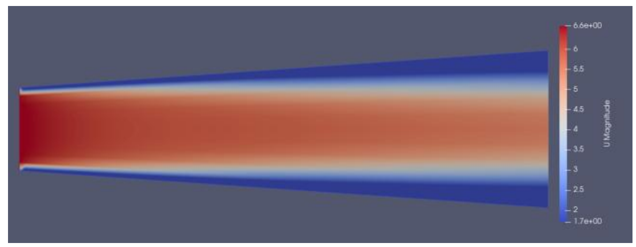

FIGURE 6. CROSS-SECTION OF THE FLOW THROUGH THE NOZZLE WITH AN EXPANSION NOZZLE OF 4°.

5°

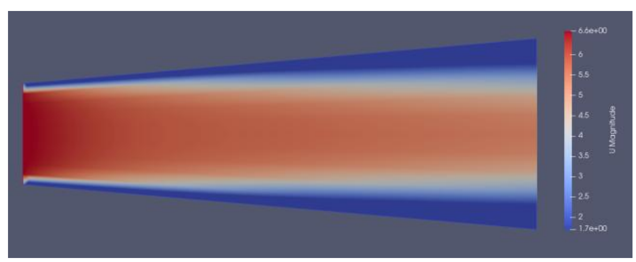

FIGURE 7. CROSS-SECTION OF THE FLOW THROUGH THE NOZZLE WITH AN EXPANSION NOZZLE OF 5°.

6°

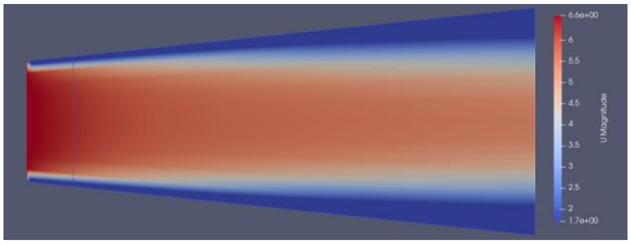

FIGURE 8. CROSS-SECTION OF THE FLOW THROUGH THE NOZZLE WITH AN EXPANSION NOZZLE OF 6°.

8°

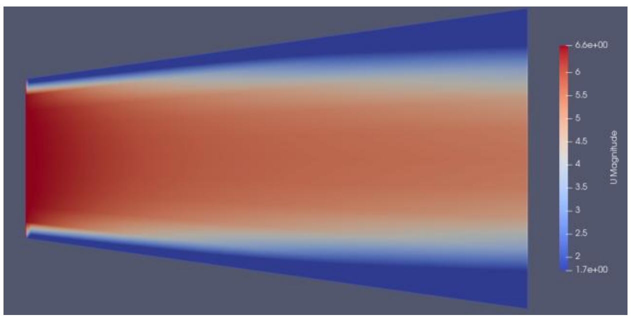

FIGURE 9. CROSS-SECTION OF THE FLOW THROUGH THE NOZZLE WITH AN EXPANSION ANGLE OF 8°.

10°

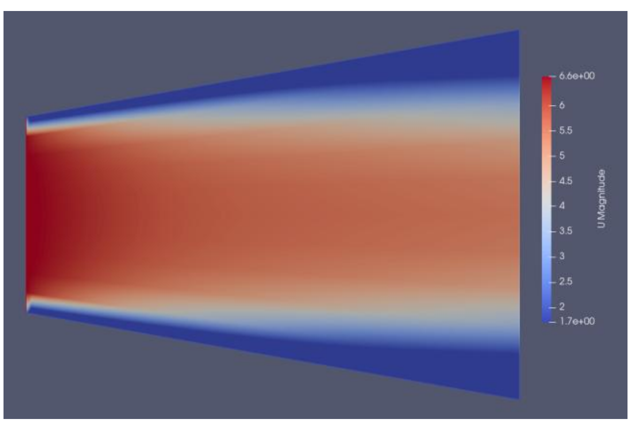

FIGURE 10. CROSS-SECTION OF THE FLOW THROUGH THE NOZZLE WITH AN EXPANSION ANGLE OF 10°.

Future Considerations

To provide the author with more accurate data, a simulation will be run in the future using compressible fluid on a device capable of handling the simulation. Additionally, a proper mesh will be used to simulate the flow through a round pipe providing the correct boundary layer and surface friction results. Results will also be tested using a 3D printed nozzle and measuring the thrust output.

Conclusions

The results from this study show that the expansion angle should be 5° to reduce the development of the flow. Since the velocity gradients are all identical at the exit, the most important feature would then be its ability to configure into the geometry of the compression fittings being used. Testing with this geometry will move forward using a circular pipe geometry as well as real-world testing. As previously mentioned, due to the assumed geometry of a square pipe, there is more friction generated than if the geometry were circular. This would lead to an even higher exit velocity at the exit providing additional thrust. Additionally, as the fluid was assumed to be incompressible, the testing fluid would have different densities at the inlet and exit, causing the velocity gradient to likely expand.

References

[1] Jiang, T., Huang, Z., Li, J., Zhou, Y., and Xiong, C., 2022, “Effect of Nozzle Geometry on the Flow Dynamics and Resistance Inside and Outside the Cone-Straight Nozzle,” ACS Publications [Online]. Available: https://pubs.acs.org/doi/epdf/10.1021/acsomega.1c07050?ref=article_openPDF&articleRef=control.

[2] Young, C., “Square Pipe vs Round Pipe Flow: Comparing Performance and Efficiency,” Engineer Excel [Online]. Available: https://engineerexcel.com/square-pipe-vs-round-pipe-flow/.

[3] Wong, M.K. et al (2015) *Numerical Study of Turbulent Flow in Pipe with Sudden Expansion.* Available: https://akademiabaru.com/doc/ARFMTSV6_N1_P34_48.pdf