Master Van Huy Khuat, 2026, 14:4
ISSN (Online): 2348-4098
ISSN (Print): 2395-4752

International Journal of Science, Engineering and Technology
An Open Access Journal

Study of Changes in Gas Flow Parameters in a Tube with Gradually Reducing Cross-Section Using Ansys Fluent Application

[1]Master Van Huy Khuat, [2]Master Le Thanh Nguyen, [3]Engineer Van Viet Vo
Air Force Officer's College, Khanh Hoa, Vietnam

Abstract- This paper presents the results of a study investigating changes in airflow parameters such as pressure and velocity as the flow moves through a converging nozzle, using the Ansys Fluent software platform. Converging nozzles are widely used in the jet engines of subsonic aircraft. The study analyzes the behavior of airflow parameters within this type of nozzle using Ansys Fluent and compares the findings with theoretical and experimental results. This approach enables the determination of specific parameters under varying conditions and identifies the underlying causes of these parameter changes.

Keywords— Jet engine; Nozzle; Subsonic; Ansys Fluent

I. INTRODUCTION

Among the various types of jet engine components, the nozzle converts the thermal energy of the airflow into kinetic energy and expels the gas at high speed to generate thrust. A subsonic nozzle is characterized by a converging cross-section. It increases subsonic airflow velocity by transforming the flow's static pressure and thermal energy into kinetic energy. This velocity increase occurs because the converging cross-section ($A2 < A1$) compresses and accelerates the airflow in accordance with the continuity equation for compressible flow in the subsonic regime ($Ma < 1$). The reduction in static pressure is explained by the law of conservation of energy: as total energy remains constant, an increase in kinetic energy results in a corresponding decrease in the airflow's static pressure and temperature. In this type of nozzle, the maximum velocity at the exit of the converging section (the throat) is limited to the speed of sound ($Ma = 1$); this phenomenon is known as "choked flow."

II. METHODOLOGY

The choice of computational method for a specific problem depends on the required accuracy, the method's simplicity and consistency, economic considerations, available resources, and controllability. The resulting outcomes must also be verified.

We selected Ansys Fluent software to analyze various converging nozzles with different cross-sectional profiles. This powerful software facilitates the creation of diverse models and delivers rapid computational results; it also allows for easy parameter adjustment during the analysis while maintaining the necessary level of accuracy.

In practice, converging nozzles are widely used in the jet engines of subsonic aircraft.

To study parameter variations during flow through a converging nozzle, we selected a three-section model: an initial section with a constant cross-section, a second section with a converging cross-section, and a final section with a constant cross-section.

Smooth curves are used for transitions between these regions to avoid abrupt parameter changes. The dimensional parameters can be easily modified to conduct the analysis.

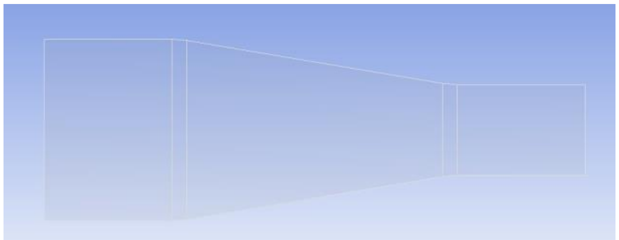


Figure 1: Pipeline diagram for the survey

After defining the geometric model of the test pipe, we utilized ANSYS Fluent to create models featuring various contraction angles. We then generated a mesh within the region of interest, applied boundary conditions, and analyzed the airflow dynamics inside the pipe. For the airflow simulation, we selected an automated structured quadrilateral meshing approach; we defined the mesh size across the pipe's cross-sections using a fixed number of divisions with uniform spacing to ensure a consistent mesh count. Additionally, we set the spacing along the pipe walls to be proportional to the length, ensuring uniform mesh element sizes. The resulting mesh was notably uniform and stable (Figure 3).

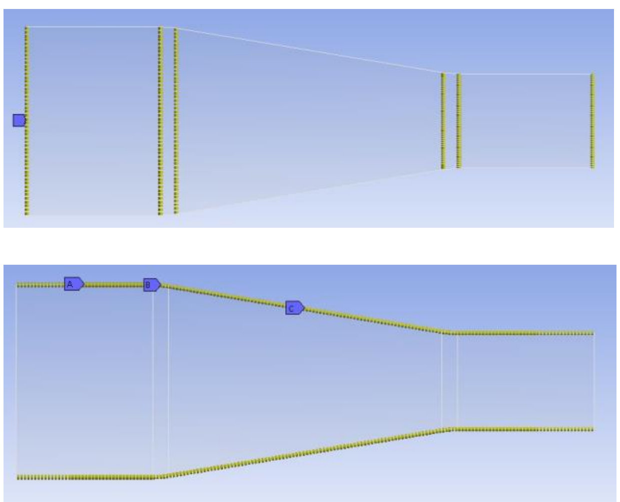


Figure 2: Determining dimensions during the meshing process.

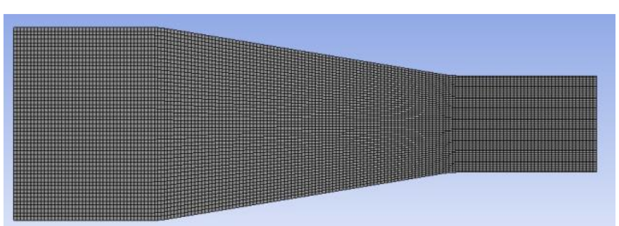


Figure 3: Survey grid results

III. RESULTS AND DISCUSSION

After constructing the computational model, we conducted an analysis using it. To perform this analysis, we specified the inlet airflow velocity and set boundary conditions appropriate for studying airflow within a converging duct. We then varied the duct's outlet dimensions to examine how velocity changes in response to different degrees of contraction.

The results are presented as images and graphs illustrating the variations in pressure and airflow velocity along the length of the test duct for the different dimensions.

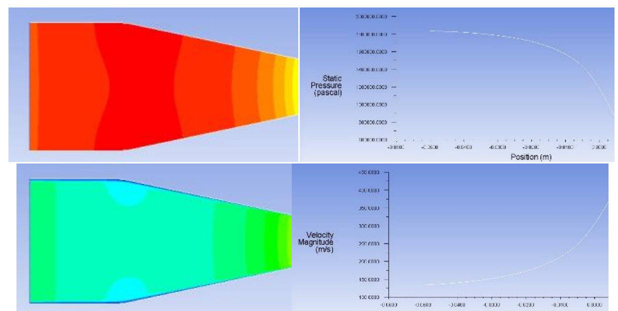


Figure 4: Images and graphs showing changes in pressure and airflow velocity for a duct with an inlet width of 40 mm and an outlet width of 15 mm.

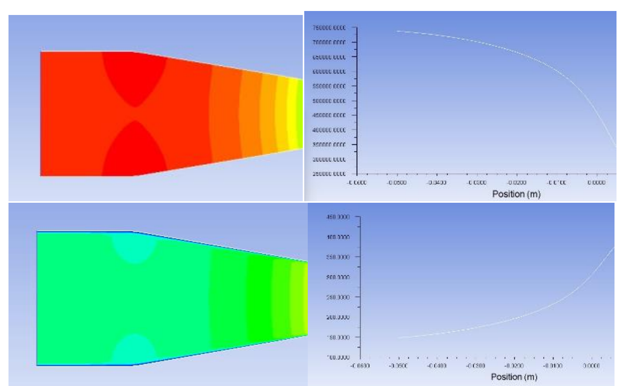
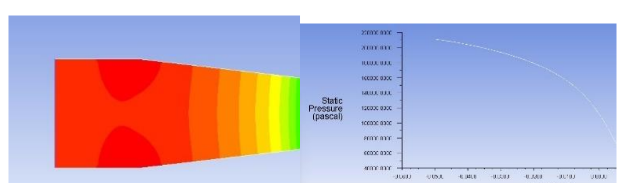


Figure 5: Images and graphs showing changes in pressure and airflow velocity for a duct with an inlet width of 40 mm and an outlet width of 20 mm.



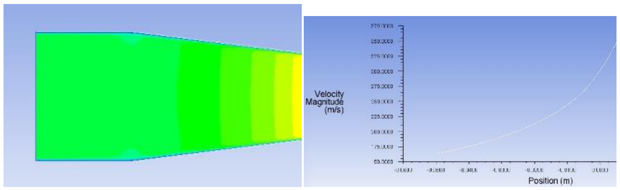


Figure 6: Images and graphs showing changes in pressure and airflow velocity for a duct with an inlet width of 40 mm and an outlet width of 25 mm.

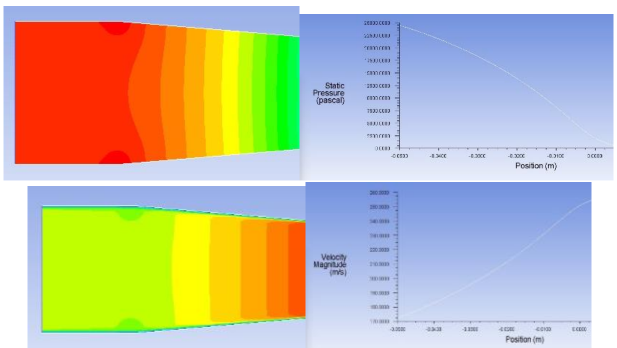


Figure 7: Images and graphs showing changes in pressure and airflow velocity for a duct with an inlet width of 40 mm and an outlet width of 30 mm.

Based on the calculation results shown in the images and graphs, it can be observed that:

- When the gas flow moves at subsonic velocities through a converging duct, the flow velocity increases along the length of the duct, while the static pressure decreases;
- Regarding changes in duct dimensions, the greater the degree of constriction, the more rapidly the parameters change (provided that flow choking has not yet occurred);
- When flow choking occurs (pressure drops to a critical value while velocity reaches the speed of sound), the gas flow rate at the outlet ceases to increase, forcing the parameters in the upstream section of the pipe to change more slowly.

Significance: By examining the changes in gas flow parameters within a converging pipe, we observe the following:

- The obtained results are entirely consistent with theoretical principles;
- The analysis determines specific values for gas flow velocity and pressure at various locations along the pipe;
- The results provide visualizations of the gas flow dynamics, as well as changes in velocity and pressure within the area of study; this offers an intuitive view of the flow process, facilitating observation and a clear understanding of the underlying physical phenomena;
- The findings can be utilized in the academic training of students;
- This analysis method can be applied to further research on gas flow parameter variations in different types of piping.

IV. CONCLUSION

Based on the computational results obtained from the model, we have determined the variations in airflow parameters as the flow moves through a converging duct. The analysis reveals that when using a converging nozzle, the airflow velocity increases as it travels through the duct; however, once the velocity reaches a maximum value equal to the speed of sound at the exit, it ceases to increase further. This acceleration of the airflow at the nozzle ensures the generation of the necessary thrust for the engine.

The study also facilitates an analysis of the airflow dynamics, helping to identify the underlying causes of these parameter changes. Visual data obtained during the study can be utilized in instruction to provide students with a clear, visual understanding of the physical phenomena occurring as the airflow passes through the nozzle.

REFERENCES

1. L. N. Thuy, Gas thermodynamics, Air Force Academy, 2015 (in Vietnamese);
2. T. V. Bay, Textbook on Engine Theory, Air Force Academy, 2005 (in Vietnamese);