Combined Computational and Theoretical Analysis of a Pulsejet Engine

Principal Investigator: Garrett Lee
Research Mentor: Dr. Yanir Maidenberg

ENGINEERING PHYSICS PROPULSION LAB

Abstract

The purpose of this project is to create a predictive model to define the performance of a Pulsejet engine. We will do this by creating a computational fluid dynamics model (CFD) and generating unified equations to define the relationships between the chemical kinetics of combustion, wave generation and propagation, and exhaust flow characteristics. Our goal for this project is to unify data collected from a CFD analysis of the engine that would otherwise be too difficult to account for theoretically, in combination with theoretical equations created to define the performance parameters of the engine.

Background

The Pulsejet is an unsteady propulsive device, with reed valves being the only moving parts in the engine. The Pulsejet sustains its combustion by utilizing the acoustics generated by the combustion, in combination with residual exhaust gas recirculation, to incite a new combustion cycle. There are several theories to describe the operation of combustion of a Pulsejet, the most popular of which is the acoustic theory. The acoustic theory states that once an ignition has occurred, the pressure rises in the combustion chamber, closing the reed valves, and forcing the exhaust gases out of the tailpipe resulting in thrust production. As the exhaust gases at the exit expand to atmosphere pressure, their momentum generates expansion waves that propagate through the tailpipe and into the combustion chamber. This expansion wave causes the pressure in the combustion chamber to drop below ambient, resulting in the reed valves opening to allow a new mixture to enter.

Methods

- Theoretical equations are being generated to define the performance parameters of the engine. These equations will account for the chemical kinetics of the combustion, the exhaust propagation, and the resultant wave generation.
- ANSYS Fluent is being used to run simulations of a CFD analysis of the engine. This CFD model will be used to gather data and expected performance, from the combustion and exhaust process', to be used in combination with theoretical equations to generate our predictive model.
- A Brauner Pulsejet will be constructed along with a test bed that allows accurate data to be recorded. This data will be used to improve the input conditions of the CFD model and gather a greater understanding of the operating conditions of the engine.

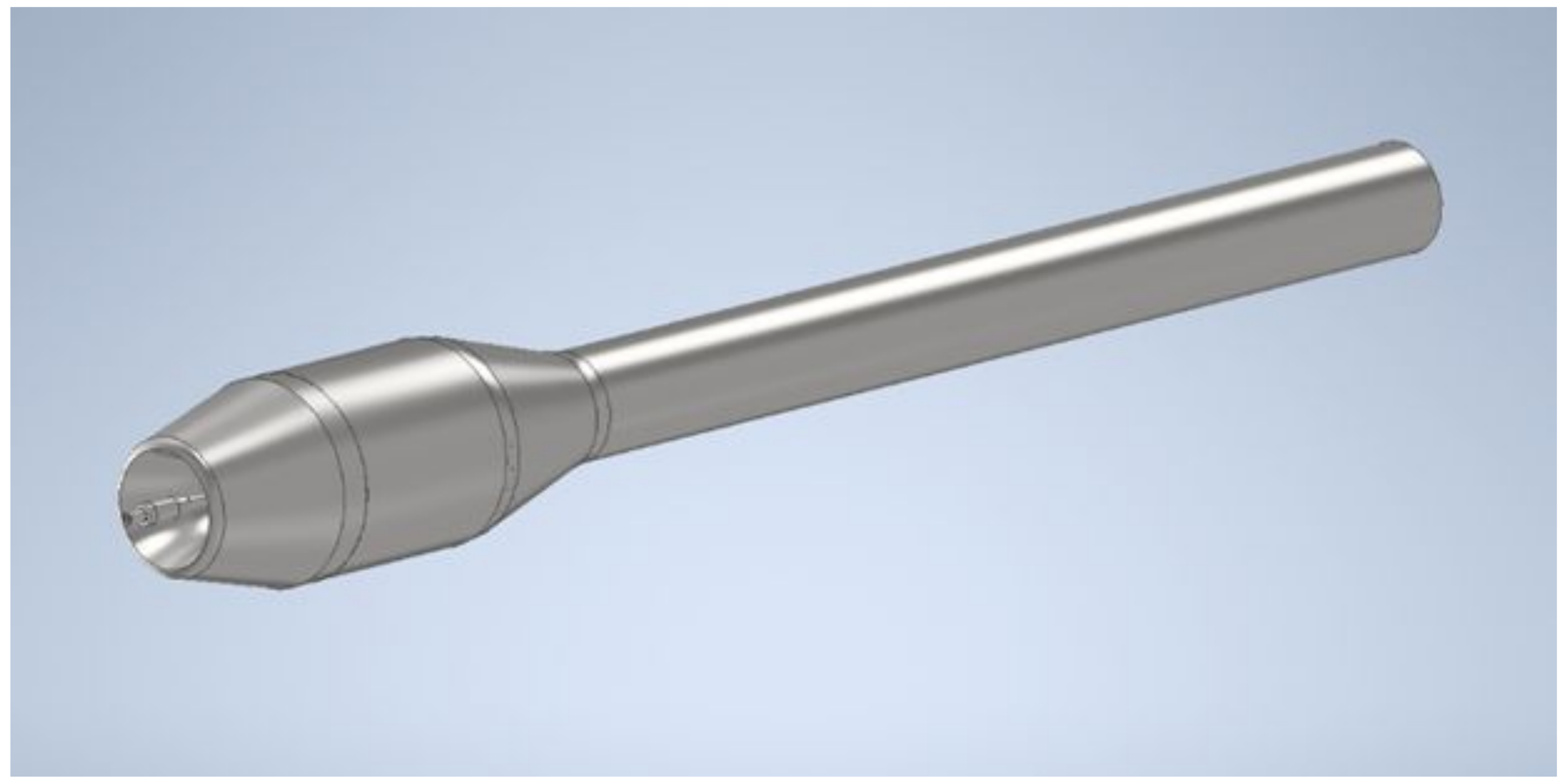

Figure 1: 3D model of Brauner Pulsejet

Current Work

- Theoretical equations are being developed to define the operation of the engine. These equations are currently focusing on the post-combustion exhaust gas and their recirculation throughout the engine,
- Proper ANSYS Fluent set up conditions are being investigated to properly analyze the engine conditions.
- An UDF is being created to alter the boundary conditions of the ANSYS Fluent simulation.

Future Work

- Continue to develop the theoretical model to define the performance parameters of the engine.
- Model combustion and exhaust cycle in ANSYS Fluent to gather performance data of the engine.
- Conduct hot fire tests of a Brauner Pulsejet engine and collect data to be used for the improvement and validation of the CFD and theoretical model.

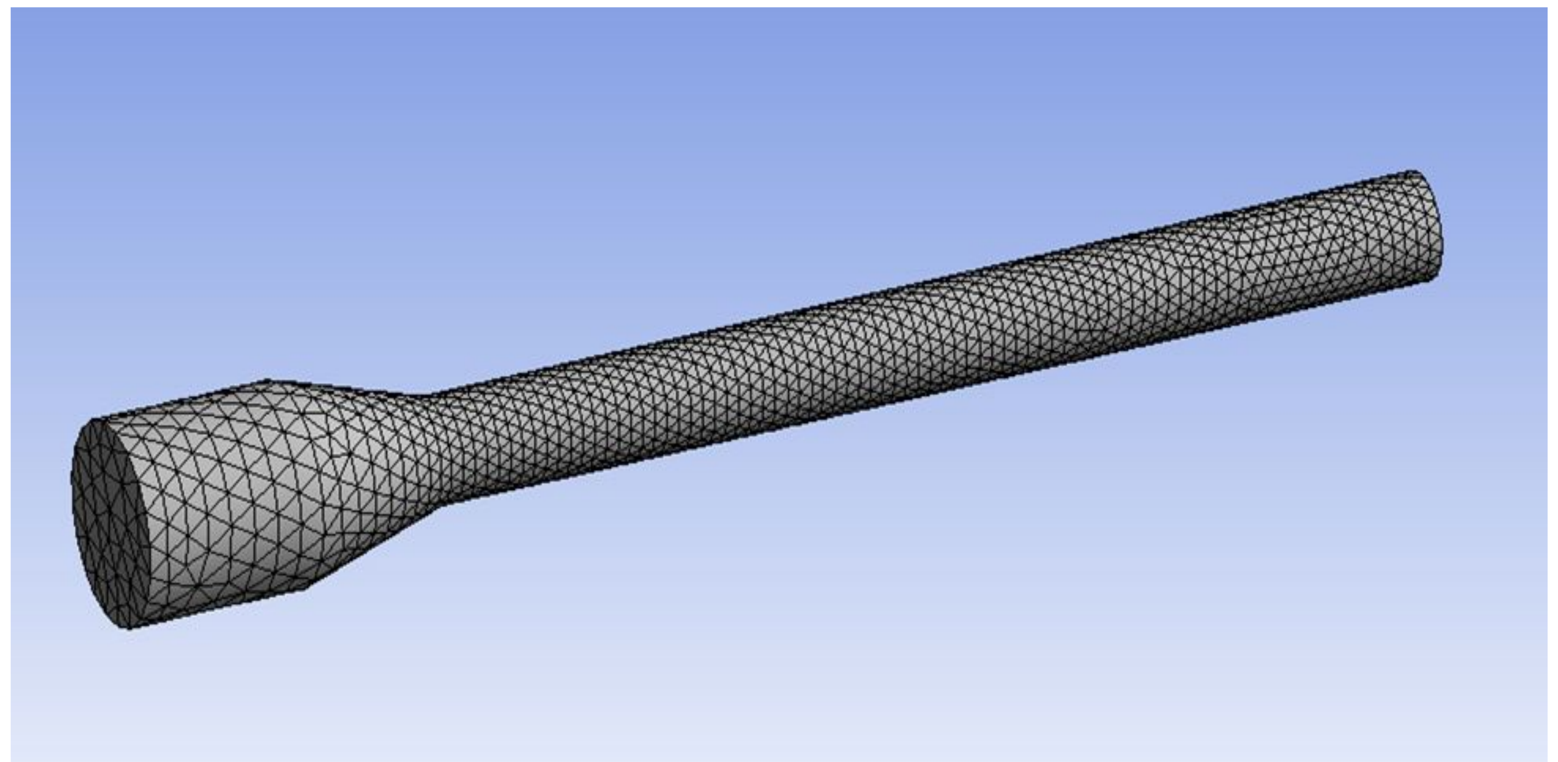

Figure 2: Mesh of 3d model in ANSYS Fluent