

UNIVERSITY TURBINE SYSTEMS RESEARCH (UTSR) 2023 GAS TURBINE INDUSTRIAL FELLOWSHIP PROGRAM

Prepared by:

Isabel Moore
Bachelor of Science in Mechanical Engineering
Texas A&M University

**FINAL REPORT
UTSR Summer 2023 Fellowship**

Prepared for:

Southwest Research Institute

October 11, 2023



SOUTHWEST RESEARCH INSTITUTE®
6220 Culebra Road
San Antonio, Texas 78238

UNIVERSITY TURBINE SYSTEMS RESEARCH (UTSR) 2023 GAS TURBINE INDUSTRIAL FELLOWSHIP PROGRAM

Prepared by:
Isabel Moore
Bachelor of Science in Mechanical Engineering
Texas A&M University

FINAL REPORT UTSR Summer 2023 Fellowship

Prepared for:
Southwest Research Institute

October 11, 2023

Approved:



Griffin C. Beck
Manager
Propulsion & Energy Machinery
Machinery Department

ABSTRACT

As part of the Great Horned Owl (GHO) program, Southwest Research Institute (SwRI) has developed a small, lightweight gas-turbine generator to provide power for electric or hybrid electric Unmanned Aerial Vehicles (UAV). The development program focused on advancing the state of the art in UAV power systems by targeting key metrics in weight, fuel efficiency, and noise generation [1]. This innovative fuel-to-electricity component, designed, built, and tested at the SwRI facility in San Antonio, TX, underwent multiple major design iterations and influenced the evolution of the GHO machine's design to its current state.

A combustor test rig was designed to facilitate the component testing of the combustor. Previous testing in 2021 revealed that the current design could not meet the performance goals of low-pressure drop and flame stabilization. To address these issues, this research investigated the combustor design and conditions through physical measurements, computational fluid dynamics (CFD) modeling and analysis, and additional experimental cold flow testing. The primary objective of this research was to characterize the combustor's pressure drop and use this information to propose design improvements for the annular gas turbine combustor.

By iteratively modifying geometric and flow variables in the CFD model, an updated model was created for the analysis. Rerunning simulations allowed observation of total pressure loss and airflow distribution. The results agree very well with previous experimental findings in 2021. Having validated the model with the inlet conditions of the combustor, a study was conducted with an updated geometry to examine the effects of the flow pattern in terms of the pressure and velocity profiles. The predicted cold flow pressure drop was much lower than the previous experimental results, and a redesign of the flow path is likely necessary.

TABLE OF CONTENTS

<u>Section</u>	<u>Page</u>
1. Introduction	4
2. Great Horned Owl (GHO) Combustor	5
2.1 Background	5
2.2 Combustor Design and Testing Rig	5
2.2.1 Combustor Features	6
2.2.2 Flow Pattern.....	7
2.2.3 Approach.....	8
2.2.4 Boundary Conditions.....	10
2.3 Results	11
2.3.1 Original Experimental Data	11
2.3.2 CFD Model Preliminary Results	12
2.3.3 Geometry Discrepancies	12
2.4 Updated Model Results	13
2.4.1 Updated Cold Flow Results.....	16
2.5 Conclusions and Future Work.....	17
3. Acknowledgements	18
4. References.....	19

NOMENCLATURE

dP	differential pressure, psi
\dot{m}	mass flow rate, kg/s
P	total pressure, psi
p	static pressure, psi
q	dynamic pressure, psi
T	total temperature, °F
u	velocity, in/s
\dot{V}	volumetric flow rate, m ³ /s
ρ	density, kg/m ³ or lbm/ft ³
ω	speed, rpm

SUBSCRIPTS

<i>comp</i>	compressor
<i>in</i>	inlet
<i>out</i>	outlet
<i>ref</i>	reference
<i>turb</i>	turbine

1. INTRODUCTION

As part of the University Turbine Systems Research (UTSR) Program, the Gas Turbine Industrial Fellowship (GTIF) Program offers the opportunity for qualified university research students to perform cutting-edge research for the next generation of industrial gas turbines to enhance U.S. energy security and provide a cleaner environment. This fellowship is managed by the U.S. Department of Energy Office of Fossil Energy Advanced Turbine Program, which strives to improve U.S. gas turbine technology.

Among the select group of participants, I was honored to be chosen as one of seven undergraduate and graduate students for the GTIF Program. The ten-week research fellowship, hosted by Southwest Research Institute (SwRI) in San Antonio, Texas, allowed me to delve into cutting-edge research, engineering, and design projects related to industrial gas turbines, including analysis for the GHO.

2. GREAT HORNED OWL (GHO) COMBUSTOR

2.1 BACKGROUND

In 2012, SwRI was initially contracted by the Air Force Research Laboratory to assist the Intelligence Advanced Research Projects Agency (IARPA) in its efforts to develop a UAV. The IARPA program involved developing two systems for this UAV: a fuel-to-electricity generator and an electricity-to-thrust machine. SwRI's development effort focused on a lightweight gas turbine-driven generator in response to the fuel-to-electricity program requirement. The generator developed for the Great Horned Owl (GHO) program is a single-shaft gas turbine generator system. The GHO program aims to develop a quiet, lightweight, and high-power density UAV system meeting the IARPA requirements.

Fabrication and acquisition of all the components and assembly of the Great Horned Owl machine were completed, and testing began in November 2013 [2]. The program basis originated with a patented SwRI gas-turbine configuration and went through five major design iterations to come to the final configuration before the National Aeronautics and Space Administration (NASA) Glenn Research Center (GRC) gained control of the GHO program.

As part of the overall machine development, SwRI built the combustor test rig to characterize the combustor and fuel system. Moving forward, the project goals involve the test rig currently being used to characterize the combustor pressure drop, light-off characteristics, and combustion dynamics; however, it serves as an ideal development bed for small gas turbine combustion testing. SwRI plans to use the test rig for sensor development and combustor component testing.

2.2 COMBUSTOR DESIGN AND TESTING RIG

The benchtop combustor test rig was developed to conduct cold-flow pressure drop testing, fuel system calibrations, light-off testing, and operational stability tests independent of the turbine rotating components. The test rig uses a fixture to house the combustor liner in a pressure vessel machined from 6-in schedule 40 pipe [1], as shown in Figures 1 and 2.



Figure 1 Combustor Test Rig

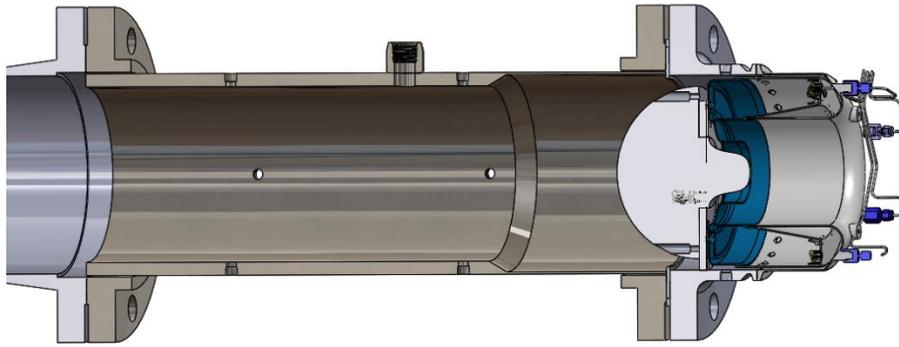
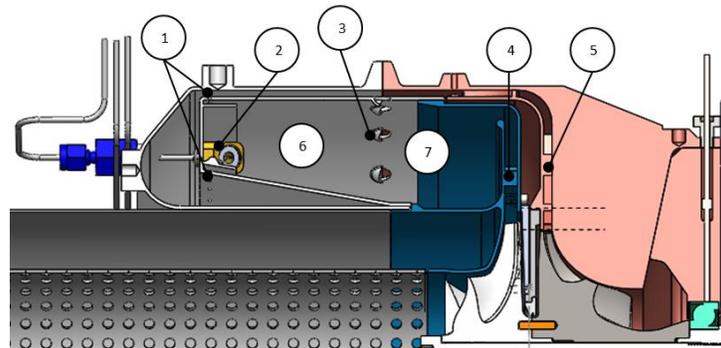


Figure 2 Combustor Containment Spool

2.2.1 COMBUSTOR FEATURES

The combustor design process required several different features to meet program goals for combustor weight, efficiency, and pressure drop. The final design selected was a conventional annular combustor with a liner. The liner contains the combustion process and introduces the various airflows into the combustion zone. Because this combustor is very small, the orientation of the fuel nozzles was adjusted to maximize the spray area volume while minimizing the number of fuel nozzles. This design incorporates six air-blast fuel injectors and dilution air jets with a residence time of 8 ms [1]. The modeled geometry of the combustor is shown in Figure 3. The combustor is shown oriented horizontally with the aft or injector end of the combustor on the left, and the combustor exit pointed toward the bottom of the page labeled as turbine inlet. The individual combustor parts are designed to friction slip fit together for easy assembly and disassembly during testing. This modular design allows for easy customization and design iteration. SwRI disassembled and reassembled the combustor several times during testing, with downtimes of only a few hours [1].



Key:

1. Cooling/Primary Air Jets
2. Airblast Injector (Air+Fuel)
3. Dilution Air Jet w/ Lip
4. Turbine Inlet Guide Vanes
5. Compressor Discharge Guide Vanes
6. Primary Combustion Zone
7. Dilution Zone

Figure 3 Combustor Design Features

For measurement through the combustor, Figure 4 outlines the instrumentation locations for the thermocouples, pressure transducers, and fuel injectors.

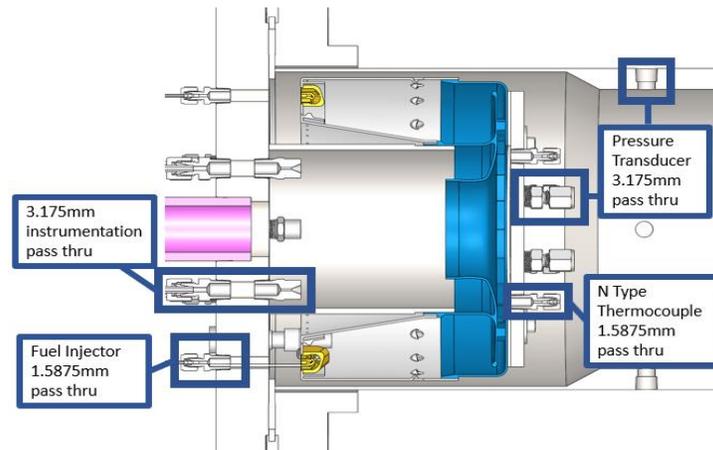


Figure 4 Instrumentation Locations

2.2.2 FLOW PATTERN

Control of the airflow rate distribution within the combustor chamber is critical to the performance of the combustor, including the ability to complete combustion, stable operation, and effective temperature control. The primary function of a combustion chamber is to burn the fuel/air mixture, adding heat energy to the air. Efficient combustion requires proper mixing of fuel and air, followed by controlled burning of the mixture. Additionally, the combustion chamber must cool down the hot combustion products to a temperature that can be withstood by the turbine inlet guide vanes under operating conditions. To achieve this, orifices are created to induce the flow and allow for the overall cooling to the liner. Figure 5 outlines the intended flow directions from the orifices: cooling/primary air jets, air blast injectors, and dilution air jets.

Generally, the flow through the liner holes does not depend only on their size and the corresponding pressure drop but also on the flow conditions in the vicinity of the hole, which strongly influence the flow distributions amongst the various zones of the chamber [3]. The air entering the combustion chamber is divided into three main streams: primary, secondary, and dilution air, as shown in Figure 6.

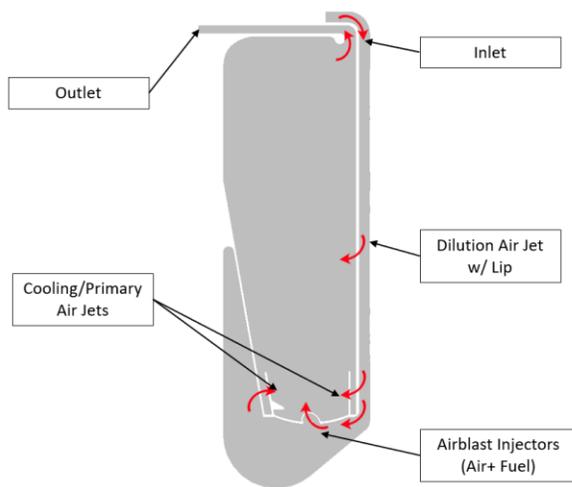


Figure 5 Airflow Pattern through Orifices

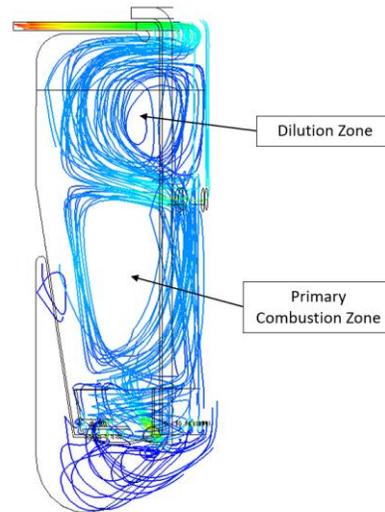


Figure 6 Velocity Streamline

The primary air is directed inside the liner, where it mixes with fuel and undergoes combustion. Secondary and dilution air is directed between the outer casing and enters the liner through specific holes. The dilution air cools the combustion gases to an appropriate temperature. The induced swirl pattern shown in Figure 6 is intended to help maintain stable combustion by increasing the residence time of the air in the combustor. The swirl pattern also recirculates the hot products of combustion to achieve stable flame. Thus, in order for each zone to perform its function perfectly, the airflow rate should be properly distributed, and the proper number, size, and location of the orifices should be analyzed.

The operation of a combustor, more specifically a reverse-flow annular combustor like the GHO, must satisfy a wide range of conditions and requirements, including high combustion efficiency, reliable and stable ignition, and low-pressure loss. These factors govern whether light-off can be achieved and how efficiently the combustor can operate. During testing in June 2021, failure to light off three times demonstrated the need for more refined control of the fuel system and a better understanding of the combustor airflow prior to operating. Thus, cold flow predictions were evaluated and analyzed to reflect the pressure drop and the flow patterns.

Low-pressure loss is a key performance target for gas turbine combustors. A low-pressure drop combustor is desirable to maximize the gas turbine's efficiency. When total pressure is lost due to mixing and flow constrictions in the combustion chamber, less energy is available to work on the turbine blades and lower overall power. The pressure loss depends on many parameters, especially the chamber design and working gas flow rate.

2.2.3 APPROACH

Computational and experimental approaches were used. For simulations, first Ansys CFX and then later Ansys Fluent were employed to simulate the cold flow in the combustor. The Ansys CFX model was adapted from earlier simulations performed in 2015. When difficulties duplicating the experimental results occurred, the project team switched to Ansys Fluent when the geometry was updated. In both cases, the inlet was set to a fixed mass flow rate, and the outlet was assigned to a fixed pressure, either atmospheric or elevated, depending on the test conditions. Pressure-based solvers with compressible ideal gas were used. The k- ϵ turbulence model with $y^+ \sim 50$ across the domain was deemed appropriate. Second-order discretizations were employed.

The benchtop combustor test rig was developed for various purposes, including cold-flow pressure drop testing, fuel system calibrations, light-off testing, and operational stability tests, all independent of the gas turbine's rotating components. The physical machine features a specialized plate with precision-cut channels for placing combustor exit thermocouples and pressure ports, as depicted in Figure 7.

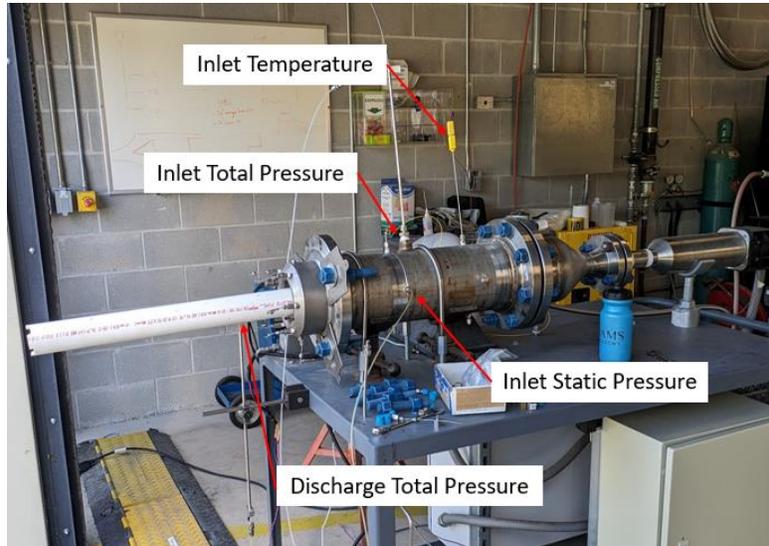


Figure 7 2023 Cold Flow Testing Rig

This arrangement accommodates various N-type thermocouples positioned at the combustor's outlet, complemented by pressure transducers at the same location, as illustrated in Figure 8 and Table 1 for reference. Additionally, 3.175mm instrumentation pass-throughs are conveniently available at the pressure vessel's outlet, offering versatility for various applications. These include the potential integration of an exhaust quench system, utilizing tap water from the testing facility, or facilitating instrumentation testing.

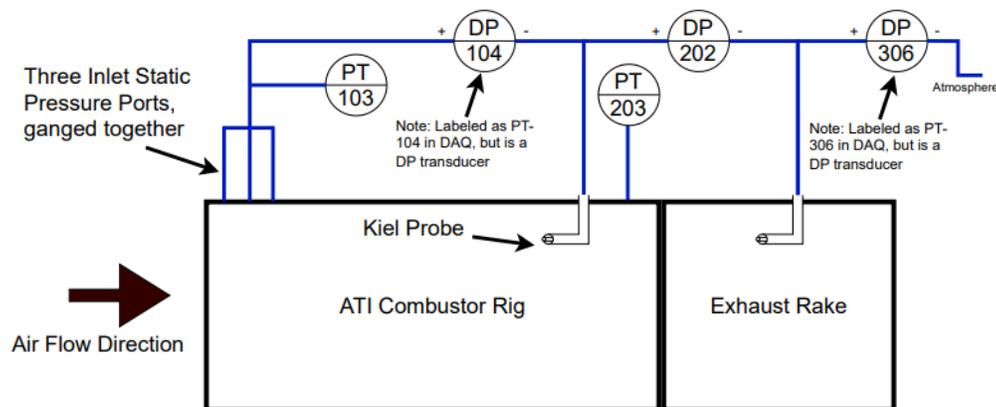


Figure 8 Updated P&ID for 2023 Cold Flow Testing

Table 1 Figure 8 Reference

Measurement	Title
PT-103	Inlet air static pressure [psig]
DP-104	Inlet air total pressure [psid]
PT-203	Combustor static pressure [psig]
DP-202	Combustor pressure delta [psid]
DP-306	Exhaust total pressure [psig]

2.2.4 BOUNDARY CONDITIONS

An analysis of the pressure loss throughout the combustor required establishing defined boundary conditions, with particular attention to the inlet and outlet of the combustor. Properly setting these parameters prior to the investigation was crucial, especially for the CFD modeling and analysis carried out using ANSYS. The governing parameters of pressure drop led to the evaluation of these boundaries, including inlet air mass flow rate and reference outlet pressure, which was set to an atmospheric pressure of 14.7 psi. These determinants played a significant role in understanding and quantifying the pressure loss throughout the combustor.

To calculate the inlet air mass flow rate, the Fan Law Calculation was utilized, which establishes a direct proportionality between volumetric flow and speed:

$$\dot{m}_1 = \dot{m}_2 \frac{\omega_1}{\omega_2}$$

Here \dot{m} and ω are the respective mass-flow rate and speed of the turbine and compressor flows.

After commissioning of the combustor test rig, the initial test campaign focused on combustor cold flow pressure drop testing. During a cold flow combustor test, the combustor inlet air mass flow was set to match a simulated combustor loading at a given percent turbine load. Table 2 presents the system conditions replicated by the test rig ordered by the turbine operational speeds.

Table 2 Simulated Turbine Operational Points

Speed [rpm]	Mass Flow [kg/s]
25000	0.0280
50000	0.0559
70000	0.0783
100000	0.1119
118000	0.1320

Continuing with the investigation into pressure loss throughout the combustor, specific areas of interest were chosen, as depicted in Figure 9.

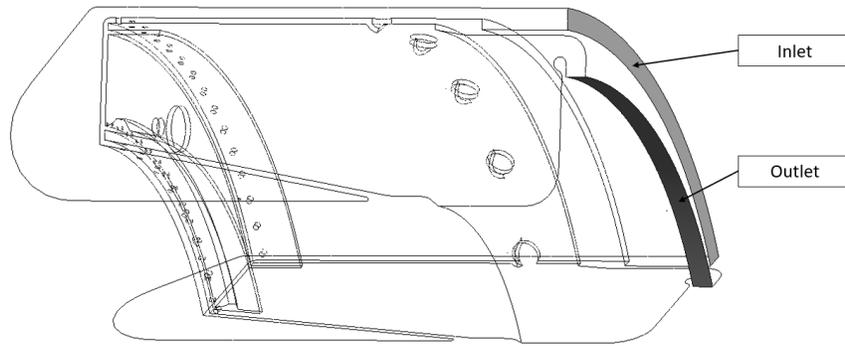


Figure 9 Combustor Inlet and Outlet Areas

The pressure change within the combustor was analyzed using one-dimensional compressible flow equations, including the overall pressure drop equation:

$$\frac{dP}{P} = \frac{P_{in} - P_{out}}{P_{in}}$$

The dP/P represents the ratio of the total pressure drop across the combustor to the inlet total pressure, which ranges between 4-8% for a typical combustor [3]. However, the pressure loss factor denotes the flow resistance introduced into the airstream between the compressor outlet and the turbine inlet and is expressed as:

$$\frac{dP}{q} = \frac{P_{in} - P_{out}}{q_{ref}}$$

The pressure loss factor represents the ratio of the total pressure drop across the combustor to the reference dynamic pressure. Unlike the dP/P or overall pressure loss, which depends on operating conditions, the pressure loss factor is a fixed property of the combustion chamber, as it characterizes the velocity and density of the flow [3]. Changes in velocity are related to area changes and density variations (heat input). Thus, retaining dynamic pressure data collection is of prime importance within the combustor analysis.

2.3 RESULTS

The outcomes of the investigation were explored, commencing with the 2021 testing phase, which includes cold flow tests and unsuccessful light-off attempts. Subsequently, the preliminary results from CFD modeling, followed by an examination of the geometric discrepancies discovered during the analysis, to determine the root causes of the observed discrepancies and provide valuable insights into the factors influencing the performance of the system.

2.3.1 ORIGINAL EXPERIMENTAL DATA

As previously referenced, testing in 2021 encompassed unsuccessful light-off attempts. Additionally, during this time, there were several cold flow tests conducted. Illustrated in Figure 10, these cold flow tests served as the benchmark for the subsequent CFD analysis to be used for a comparative assessment and help identify areas for potential CFD model enhancement.

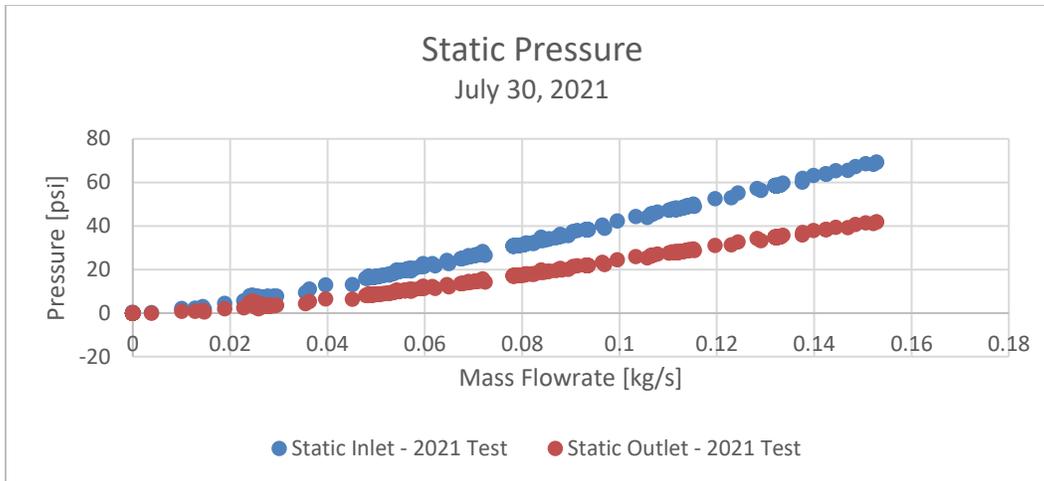


Figure 10 Static Pressure of 2021 Cold Flow Tests

2.3.2 CFD MODEL PRELIMINARY RESULTS

The initial CFD model (Ver1) was assumed to represent the combustor accurately. However, the CFD model was run and did not support this assumption. The simulations were run using the input mass flow parameters from Table 2, and the outlet pressure was made relative to atmospheric pressure. The results revealed a lack of correlation in static pressure and $dP/P\%$ between the experimental data from 2021 with the Ver1 CFD model, as shown in Figure 11. This doubling of pressure losses was unexpected. A failure of the CFD approach to validate pressure losses indicated a problem with the model setup.

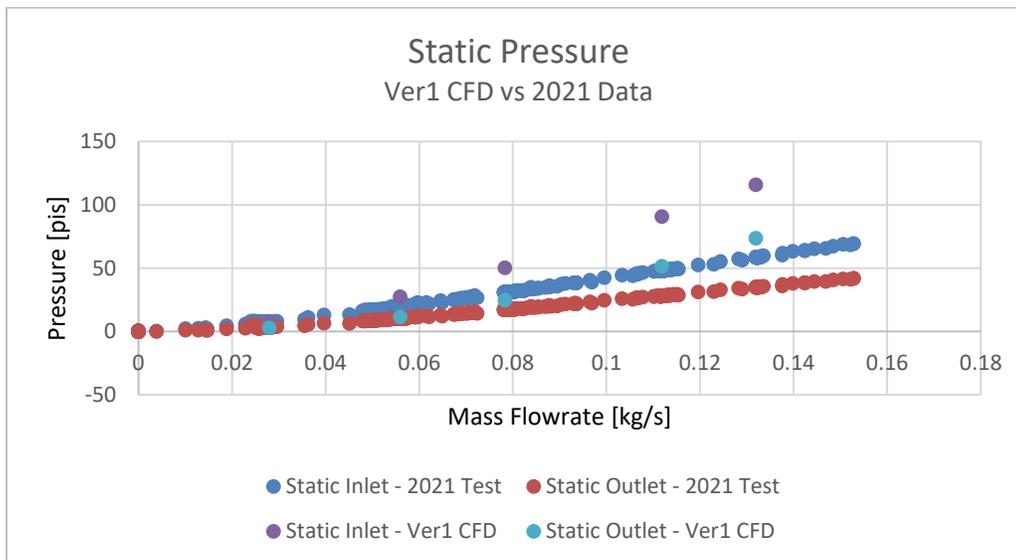


Figure 11 Static Pressure between 2021 Cold Flow and Ver 1 CFD

2.3.3 GEOMETRY DISCREPANCIES

To address the discrepancies between the CFD and experimental data, an in-depth geometric analysis was performed. Pin gauges were used to precisely measure the orifice diameters within the liner, including cooling/primary air jets, air blast injectors, and dilution air jets, represented in Figure 12. The dimensions of these orifices were compared between the actual combustor liner

and the Ver1 CFD model. The pin gauge measurements of the liner orifices showed significant discrepancies between the original Ver1 CFD size in ANSYS and the actual size, as shown in Table 3.

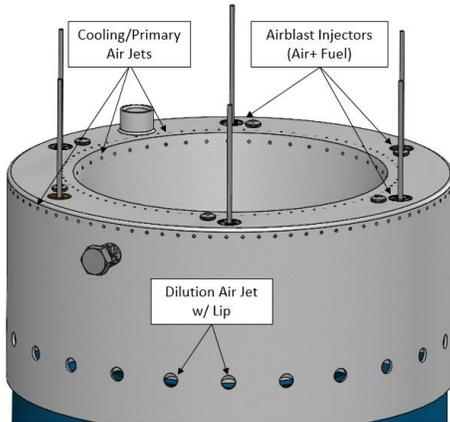


Figure 12 Orifices Locations

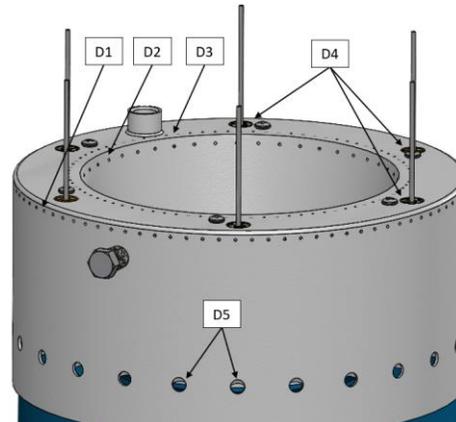


Figure 13 Table 3 Reference

Table 3: Pen Gauge Measurements of Liner Orifices

Location	D1	D2	D3	D4	D5
Original ANSYS Size [in]	0.0415	0.0415	0.0260	45 deg, 0.16 x 0.14	0.1970
Average Actual Size [in]	0.0328	0.0292	0.0280	0.2860	0.1933

The orifice diameters' discrepancies suggested a causal link to the failure to light off the combustor, as the actual hole sizes were much smaller than intended. These results proved crucial to the investigation of why the previous testing in 2021 had multiple unsuccessful light-off attempts, as decreasing the hole sizes would result in a higher-pressure loss. This liner was designed during the early days of 3D capabilities through direct metal laser sintering (DMLS), which would explain the inconsistencies within the intended design dimensions.

2.4 UPDATED MODEL RESULTS

In addition to the findings mentioned above, a critical observation was made regarding the physical dimensions and the design of the combustor liner, which deviated from the Ver1 CFD model. It is important to note that the Ver1 CFD model was one of the early design iterations of the GHO, which differs from what was manufactured and tested. To address the geometric discrepancies, a new geometry (Ver2) was created based on a recovered SolidWorks model. The Ver2 CFD model featured larger passageways, smaller chamber size in relation to the passageways, and modified fuel nozzle angles, providing a more accurate representation of the combustor, as depicted in Figure 14.

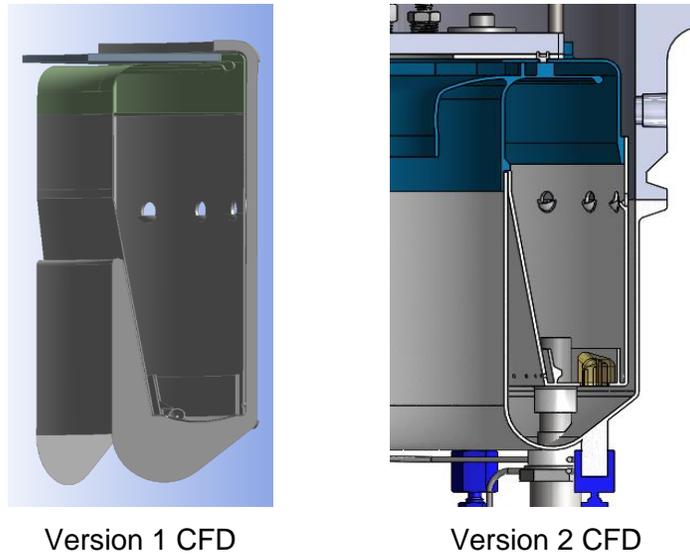


Figure 14 CFD Cross Section between Ver1 and Ver2

As a result of this in-depth investigation into pressure drop characterization, it was evident that the discrepancies between the experimental data and the Ver1 model were primarily influenced by Ver1's excessively small outlet, as well as geometric variations in the combustor liner's orifices. Following the same approach as Ver1, Ver2 was rerun using the input mass flow parameters from Table 2, and the outlet pressure was made relative to atmospheric pressure, as outlined in Table 4.

Table 4: Inlet Conditions for Version 2 CFD

Outlet Pressure [psi]	Inlet Air Mass Flow [kg/s]	Inlet Temperature [K]
14.696	0.02797	300
14.696	0.05593	300
14.696	0.07830	300
14.696	0.11186	300
14.696	0.13200	300

By introducing the refined CFD geometry, the accuracy of the combustor's representation was significantly improved, leading to more reliable results and valuable insights for future design and analysis, as shown in Figure 15.

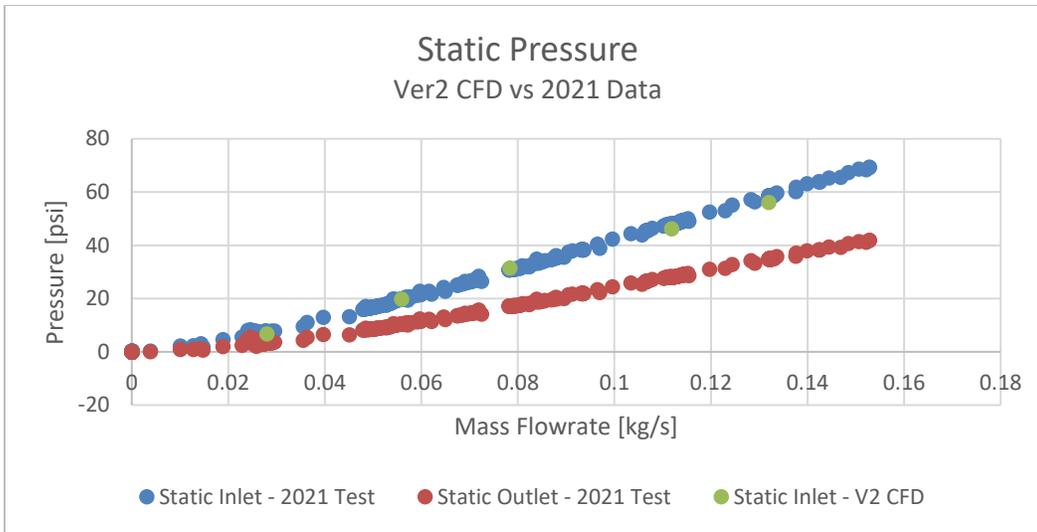


Figure 15 Static Pressure between 2021 Cold Flow and Ver2 CFD

The results in Figure 15 revealed the input static pressure matched the experimental data, which shows the potential use for this model to make predictions. However, the output static pressure showed to be consistently zero. Reasons for this discrepancy potentially had to do with not taking into consideration inlet guide veins, relation to atmospheric as part of boundary conditions, and unknown location for the outlet on previous data. While the model was unable to produce comparable results for the outlet pressure, the inlet pressure results are deemed to be a great sign and can predict the combustor and its behavioral response within given conditions. Figure 16 reveals the simulated velocity streamline of the combustor at an inlet mass flow rate of 0.02797 kg/s.

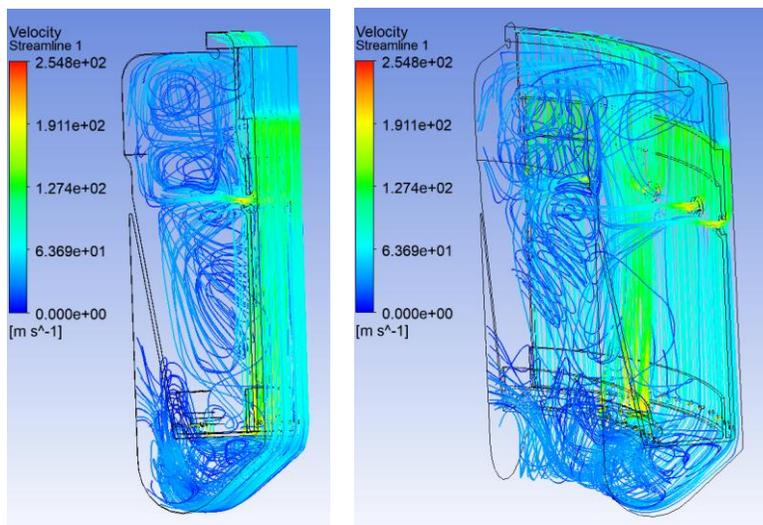


Figure 16 Velocity Streamline of Ver2 CFD Model

Ver2 CFD predicts deep jetting from undersized cooling holes and impingement on the far liner wall. This phenomenon occurs when the cooling holes in the liner of the combustor are not adequately sized to provide sufficient cooling air to the surrounding hot gases [4]. Consequently, the cooling air exits the holes at a high velocity, resulting in an impinging on the far liner wall with significant force. This impingement can result in pressure losses in the vicinity of the impingement zone. This pressure loss further contributes to an overall increase in pressure drop across the

combustor. Variations in pressure drop along the combustor may cause uneven cooling, temperature imbalances, and hotspots in different sections of the combustor, affecting its performance, efficiency, and structural integrity.

2.4.1 UPDATED COLD FLOW RESULTS

To investigate the impact of deep jetting, impingement, and pressure drop further, additional cold flow testing was conducted. This testing aimed to validate the findings from the CFD simulations and previous testing and gain more comprehensive insights into the complex flow behavior within the combustor. Unlike the previous testing conducted in 2021, which utilized static pressure probes, this round of testing employed a more appropriate method for pressure measurement. The previous approach lacked the capability to account for dynamic pressure, which is an essential factor in accurate pressure assessment across the combustor.

To address this limitation, the current testing strategy integrated both static and differential pressure probes. This combination enabled the calculation of total pressure, thus encompassing dynamic pressure as per Bernoulli's equation:

$$P - p = \frac{1}{2} \rho u^2$$

By incorporating the velocity and density of the flow, this approach provided a more comprehensive overview of the flow behavior. The calculated dP/P values obtained from the DAQ are outlined in Figure 17.

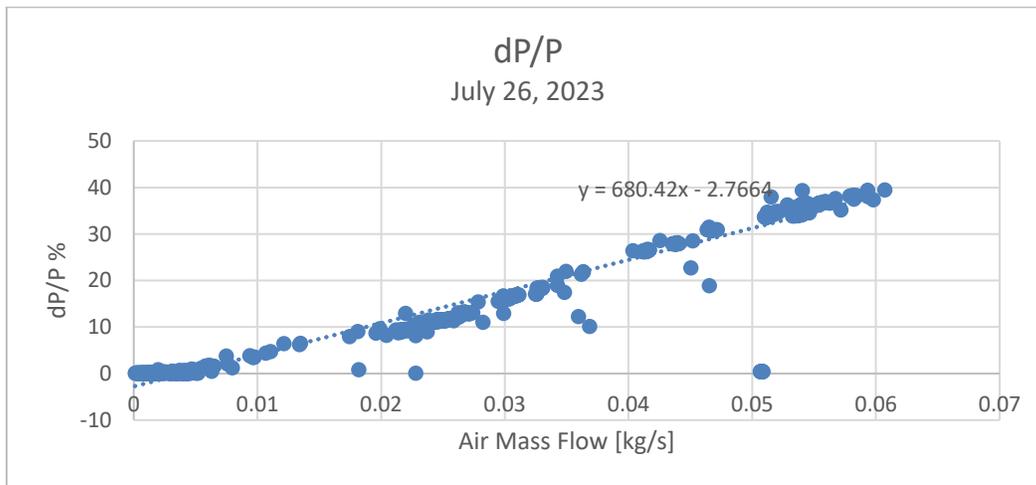


Figure 17 dP/P of 2023 Cold Flow Test

The testing results confirmed the preliminary findings from the earlier 2021 testing, indicating that the pressure loss remained excessively high. These outcomes corroborated several potential causes of pressure loss, including the CFD findings highlighting impingement on the far liner walls due to the undersized holes and broader issues in the combustor design affecting flow circulation.

The combination of insights gleaned from both the CFD simulations, and the additional cold flow testing presented a comprehensive depiction of pressure drop patterns and flow characteristics within the combustor. These shed light on the reasons behind the combustor's inability to support light-off conditions.

2.5 CONCLUSIONS AND FUTURE WORK

Previous testing revealed that the current design could not meet the performance goals of low-pressure drop and flame stabilization. To address these issues, this study investigated the combustor design and conditions through physical measurements, CFD modeling and analysis, and additional experimental cold flow testing. The primary objective of this research was to characterize the combustor's pressure drop and use this information to propose design improvements for the GHO combustor. Several key factors influencing pressure drop and overall combustion system performance were enlightened by the findings.

Initially, the measured cold flow pressure drops were observed to be higher than predicted in the previous testing. This discrepancy prompted a thorough examination of the combustor's design and flow dynamics. The CAD model's outlet channel was much smaller than the experimentally tested combustor, leading to higher predicted pressure drops in the CFD than observed experimentally. It was also discovered that the measured liner hole sizes were significantly smaller than initially intended, attributable to manufacturing tolerances associated with the Direct Metal Laser Sintering (DMLS) process. This discrepancy may help to explain the high-pressure losses observed experimentally.

To improve the accuracy of the CFD predictions, the boundary conditions and geometry were updated to match the tested combustor. These updates enabled the new CFD model (Ver2) to closely match pressure drop predictions to the previously obtained test data, validating the reliability and effectiveness of the simulation approach. Additional test data confirmed CFD and experimental agreement.

Furthermore, the CFD simulations highlighted critical issues within the combustion system. The deep jetting phenomenon resulting from improperly sized cooling holes and impingement on the far liner wall emerged as a significant concern. These findings underscore the necessity for a comprehensive liner redesign, one that adequately accounts for cooling hole dimensions, flame holding, and total pressure losses.

In conclusion, this research paper has made significant strides in understanding the intricate interplay between pressure drop and cold flow dynamics in the GHO combustor system using CFD simulations and experiments. The comprehensive approach, incorporating both experimental data and advanced simulations, has paved the way for future advancements in the combustion system design. By addressing the identified issues and implementing the proposed liner redesign, future work may aim to enhance combustion efficiency and ensure sustained performance and reliability. SwRI plans to use the GHO machine for instrumentation development, as a test bed for new technologies such as ceramic or additive manufactured parts, and as a component in a hardware-in-the-loop system.

3. ACKNOWLEDGEMENTS

I would like to thank SwRI and the UTSR fellowship program for granting me the opportunity to participate in the GTIF program, funding, and support. Thank you to the SwRI intern program for providing the opportunity to learn about the variety of work performed at the Institute throughout the summer.

I would like to extend a special thank you to my coworkers in the Division 18 Machinery Department, to my mentors and supervisors Griffin Beck, Seth Cunningham, and Brian Connelly for their guidance, support, and many great discussions, to staff and administration, Laura Garcia, and Dorothea Martinez for helping me navigate a new workspace, and to my fellow interns for an awesome work environment.

4. REFERENCES

- [1] Baye-Wallace, L., Cunningham, C. S., Musgrove, G. O., "Liquid Fuel Combustor Test Rig for 7 kW Gas Turbine Generator," in *ASME Turbo Expo 2021 Turbomachinery Technical Conference and Exposition*, Virtual, Online, 2021.
- [2] Bryner, E. PhD., Ransom, D. PE., Bishop, J., Coogan, S., Musgrove, G., "Design of a Small-Scale Gas Turbine for a Hybrid Propulsion System," in *ASME Turbo Expo 2015: Turbine Technical Conference and Exposition*, Montreal, Canada, 2015.
- [3] Lefebvre, A. H., Ballal, D. R., *Gas Turbine Combustion Alternative Fuels and Emissions*, Boca Raton, Florida: Taylor & Francis Group, 2010.
- [4] Tacina, R. R., Grobman, J., *Analysis of Total-Pressure Loss and Airflow Distribution for Annular Gas Turbine Combustors*, Cleveland, Ohio: NASA Lewis Research Center, 1969.