Project 055 Noise Generation and Propagation from Advanced Combustors

Georgia Institute of Technology (GT)
Raytheon Technologies Research Center (RTRC)

Project Lead Investigator

Timothy Lieuwen
Professor
Aerospace Engineering
Georgia Institute of Technology
270 Ferst Drive, G3363 (M/C 0150)
Atlanta, GA 30332-0150
Phone: 404-894-3041
E-mail: tim.lieuwen@aerospace.gatech.edu

University Participants

Georgia Institute of Technology

- PIs:
    - Timothy Lieuwen, Professor
    - Suresh Menon, Professor
    - Adam Steinberg, Professor
    - Vishal Acharya, Senior Research Engineer
    - Benjamin Emerson, Senior Research Engineer
    - David Wu, Research Engineer
    - Samuel Grauer, Post-Doctoral Researcher
- FAA Award Number: 13-C-AJFE-GIT-058
- Period of Performance: February 5, 2020 to February 4, 2021
- Tasks:
    1. **Facility Development at GT:** This task addresses the design of experiments that will be performed at GT. The task involves coordination between the teams for developing and defining the aerodynamic design of a Rich-Quench-Lean, quick quench, lean burn (RQL) combustor to study. The task is led by Professor Lieuwen, Professor Steinberg, and Dr. Emerson.
    2. **Simulations of GT experiment:** This task consists of the simulation of the GT experiment focusing on the pre-combustion flow dynamics, flame dynamics, and post-combustion dynamics of pressure and entropy disturbances. This task is led by Prof. Menon.
    3. **Reduced Order Modeling:** This task consists of creating a reduced order modeling framework for the unsteady response of the flame as well as the generation of entropy disturbances due to unsteady heat release. This task is led by Dr. Acharya and Prof. Lieuwen.

Raytheon Technologies Research Corporation

- PIs:
    - Jeffrey Mendoza, Technical Fellow Acoustics
    - Duane McCormick, Principal Research Engineer
    - Julian Winkler, Staff Research Engineer
    - Peter Cocks, Project Leader, P&W Program Office
    - Lance Smith, Principal Research Engineer
- FAA Award Number: 13-C-AJFE-GIT-058 (sub-award through GT)
- Period of Performance: February 5, 2020 to February 4, 2021

- Tasks:
    4. **Facility Development at RTRC:** This task addresses the design of experiments that will be performed at RTRC. The task involves coordination between the teams for developing and defining the aerodynamic design of an RQL combustor to study. This task is led by Jeffrey Mendoza, Lance Smith, and Duane McCormick.
    5. **Spray Modeling:** This task consists of performing direct numerical simulation (DNS) of the air blast atomizer in the high shear swirler used for both the GT and RTRC experiment. The main goal of this task is to provide inputs for spray modeling used in the GT simulation task. This task is led by Jeff Mendoza and Xiaoyi Li.
    6. **Swirler Impedance Modeling:** This task focuses on modeling the acoustic impedance of the high shear swirler in order to provide boundary conditions for the GT simulations as well as for the direct noise modeling task. This task is led by Jeffrey Mendoza and Duane McCormick.
    7. **Post-Combustion Modeling:** This task consists of both a post-processing and simulation effort. First, data mining the post-combustion simulation data from the simulation of the GT experiment is used to understand the dynamics of entropy fluctuations and their transport. Next, simulations are used to model the propagation of noise in the post-combustion architecture of the engine. The simulations are split across the different sections: nozzle, turbine, and far-field. This task is led by Jeffrey Mendoza and Julian Winkler.

Project Funding Level

FAA funding: $1,500,000 split equally between Georgia Institute of Technology and sub-awardee Raytheon Technologies Research Corporation.
Cost-share: $1,500,000 total, split equally between Georgia Institute of Technology and Raytheon Technologies Research Corporation.
Total funding: $3,000,000.

Investigation Team

Tim Lieuwen (Georgia Institute of Technology): Principal Investigator. Professor Lieuwen is the lead PI overseeing all tasks. Specifically, he leads the GT experiments and design in Task 1 and 2 along with Professor Steinberg. In addition, he also co-leads the modeling tasks in Task 1 for pre-combustion, flame response, and post-combustion along with Dr. Acharya.
Adam Steinberg (Georgia Institute of Technology): Co-Principal Investigator. Professor Steinberg manages the design of experiment diagnostics and the measurements.
Suresh Menon (Georgia Institute of Technology): Co-Principal Investigator. Professor Menon is the manager of the simulation tasks for simulations of the GT experiment.
Vishal Acharya (Georgia Institute of Technology): Co-Principal Investigator. Dr. Acharya co-manages all modeling tasks for the pre-combustion, combustion, and post-combustion physics along with Professor Lieuwen. In addition, as administrative coordinator, he is responsible for the general project management such as project deliverables and group meetings along with interfacing with the FAA project manager.
Benjamin Emerson (Georgia Institute of Technology): Co-Principal Investigator. Dr. Emerson is responsible for designing and maintaining experimental facilities, as well as experimental operations and management and safety of graduate students.
David Wu (Georgia Institute of Technology): Co-Principal Investigator. Mr. Wu is responsible for designing and maintaining experimental facilities, as well as experimental operations and management and safety of graduate students.
Samuel Grauer (Georgia Institute of Technology): Co-Principal Investigator. Dr. Grauer is a post-doctoral researcher and reports to Professor Steinberg. He is responsible for designing the post-combustion diagnostic capabilities in the GT experiment.
Orlando Ugarte-Almeyda (Georgia Institute of Technology): Post-Doctoral Researcher. Dr. Ugarte-Almeyda reports to Professor Menon and works on the simulation of the GT experiment.
Lane Dillon (Georgia Institute of Technology): Graduate Student. Mr. Dillon works on the design of the experiment at GT.
Parth Patki (Georgia Institute of Technology): Graduate Student. Mr. Patki works on the hydrodynamics modeling sub-task (pre-combustion disturbances).
Tony John (Georgia Institute of Technology): Graduate Student. Mr. John works on the entropy modeling sub-task (post-combustion disturbances).
Jeffrey Mendoza (Raytheon Technologies Research Center): Co-Principal Investigator. Dr. Mendoza is the team leader for the RTRC team and oversees their contributions to the project. He leads the sub-tasks related to modeling, measurements, and simulation for post-combustion disturbances, nozzle interactions, turbine interactions, and far-field sound propagation.

Lance Smith (Raytheon Technologies Research Center): Co-Principal Investigator. He is responsible for the design and measurements of the RTRC experiment. He works closely with the GT team to ensure similarities between both experiment setups.
Duane McCormick (Raytheon Technologies Research Center): Co-Principal Investigator. He is responsible for the design and measurements from the RTRC experiment as well as finite element calculations that are part of the design process.
Jordan Snyder (Raytheon Technologies Research Center): He is responsible for design, measurements, and data processing using tunable diode laser absorption spectroscopy (TDLAS) and chemiluminescence in the RTRC combustor rig.
Julian Winkler (Raytheon Technologies Research Center): Co-Principal Investigator. He is responsible for the simulation tasks at RTRC and focuses on the post-combustion disturbances, nozzle interactions, turbine interactions, and far-field sound propagation.
Jin Lee (Raytheon Technologies Research Center): He is responsible for the entropy wave transport modeling which models the transfer function for the entropy disturbances at the flame leading to pressure disturbances generated at the nozzle.
Xiaoyi Li (Raytheon Technologies Research Center): He is responsible for the pre-combustion DNS simulations of the spray dynamics. This task generates the spray information required for input to the GT simulation task led by Professor Menon.
Kenji Homma (Raytheon Technologies Research Center): He is responsible for the far-field sound propagation simulations.
Aaron Reimann (Raytheon Technologies Research Center): He is responsible for reduced order modeling and high-fidelity modeling of the propagation of direct and indirect noise sources through the turbine nozzle and supports the far-field sound propagation simulations.

Project Overview

The objective of this project is to develop and validate physics-based design tools that are able to predict noise production mechanisms, the relative significance of the noise production mechanisms, and ultimately reduce the noise output from future engines. The motivation for this project stems from the recent advances and future advances in aircraft engine technology. High-bypass engine technology has significantly reduced the traditionally dominant engine noise sources; namely, fan and jet exhaust noise. The noise generated in the combustor has become a dominant source of engine noise for future advanced aircraft designs. In addition, as combustors evolve to increase efficiency and reduce pollutant emissions, methods of predicting and mitigating combustion noise have severely lagged; legacy methods are insufficient for predicting noise from next-generation combustors. This motivates the objective of this project which is a critical need to develop physics-based design tools. The resultant understanding of noise generation and propagation, along with the validated noise prediction tools, will enable more rapid and cost-effective design of low noise engines for future aircraft.

The project objectives will be achieved through a program of cooperative experiments, high-fidelity simulations, and physics-based reduced order modeling. The physical processes involved are tightly coupled and directly determine the project tasks as shown in Figure 1.

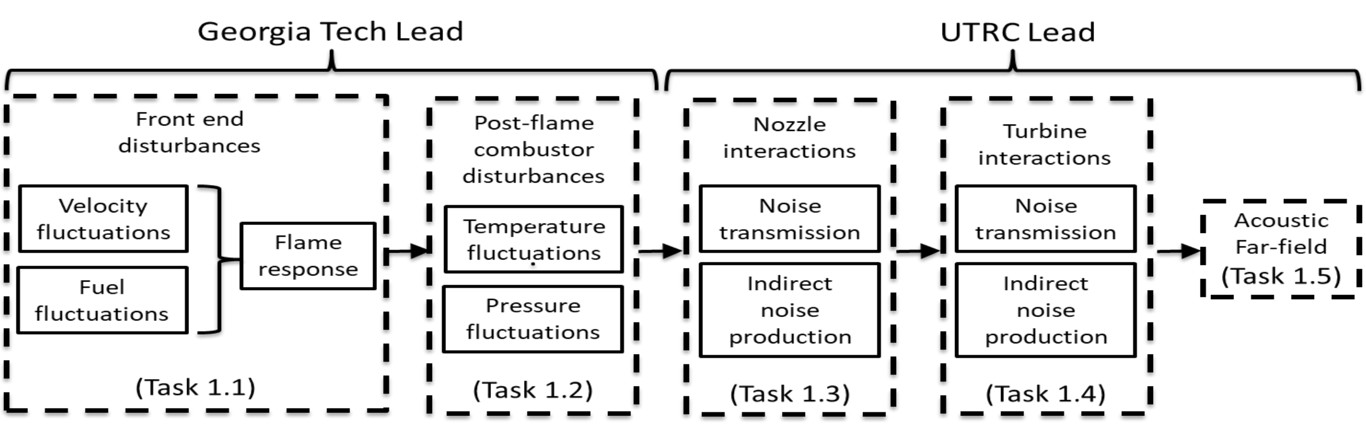


Figure 1. Physical processes and project tasks for noise generation.

The physics of noise generation begin with the source disturbances upstream of the combustion zone that involve unsteady dynamics in the flow and incoming fuel (spray). This is followed by the response of the combustion zone (flame) to these upstream disturbances. The fluctuations in the unsteady heat release, lead to both the generation of pressure fluctuations as well as entropy fluctuations. These fluctuations propagate further downstream in the combustor and interact with the nozzle and turbine and eventually lead to far-field sound generation. With the complex interplay of unsteady physics in the different parts of the engine, developing reduced order models is a challenge.

An important goal of this project is to generate high-quality reference data from both measurements and validated high-fidelity simulations. This includes measurements of the flow, spray, and flame unsteadiness in the head-end of the combustor. Followed by this, the secondary combustion zone is characterized. The generation of entropy and pressure disturbances are then characterized through measurements of the temperature fluctuations and pressure fluctuations. This is followed by measurements of the reflection and transmission of noise through the turbine and nozzle section and finally measurement of sound in the far-field. The measurements are accompanied by large eddy simulations (LES) and finite element simulations that are validated against the measurements. Collectively, this data is generated across a range of operating parameters and serves as the source database for the modeling task.

The main goal of this project is the development of a robust design tool that can predict noise at operating points where prior measurements/data are unavailable. To achieve this goal, there are two major tasks involved. First, reduced order models and frameworks must be developed for different aspects of the engine architecture: flow/spray models, flame response models, entropy generation models, entropy propagation models, nozzle interaction models, turbine interaction models, and far-field noise generation models. The reduced order modeling for each of these involves simplifications and assumptions that are validated against the source database. This validation study and iterative improvement of model predictions serves as the second task to achieve this goal.

In this report, we summarize the effort by both teams from February 2020 (start of project) until September 2020. The effort primarily includes the development of the facilities at GT and RTRC. In addition, frameworks for the reduced order modeling and simulations have been setup and are being executed using available data and publications for validation.

Task 1 – Facility Development at GT

Georgia Institute of Technology

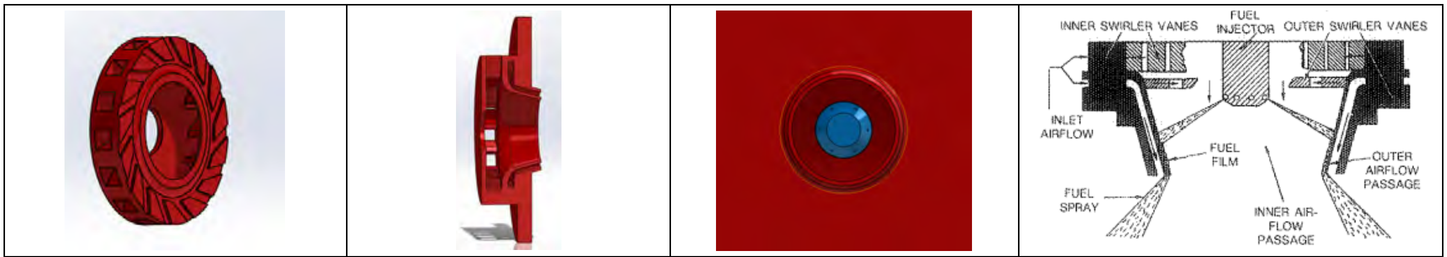
Objective

The objective of this task is to design an RQL combustor that closely mirrors the design by RTRC in order to measure different physics in the combustor architecture alone. The goal of the design is to allow for a multitude of optical diagnostics and other diagnostics capabilities that collectively measure the unsteady flow, spray, flame, temperature, and pressure fluctuations. The GT experiment will focus on the lower operating conditions to facilitate detailed optical measurements.

Research Approach

The goal of this task at GT is to leverage an existing combustor test rig facility that was developed in partnership with Pratt & Whitney (PW). This task modified the facility to utilize the existing plumbing, instrumentation, data acquisition systems, pressure vessel, and structural steel. With these modifications, the new experiment setup resulted in: (a) a generalized hardware whose measurements can be shared in the public domain, (b) optical access and instrumentation access to measure flow, spray, flame, pressure, and temperature dynamics, and (c) replication of the general physics and operational characteristics of a modern RQL combustor. The modifications resulted in a new liner, fuel/air injection system, and exhaust system.

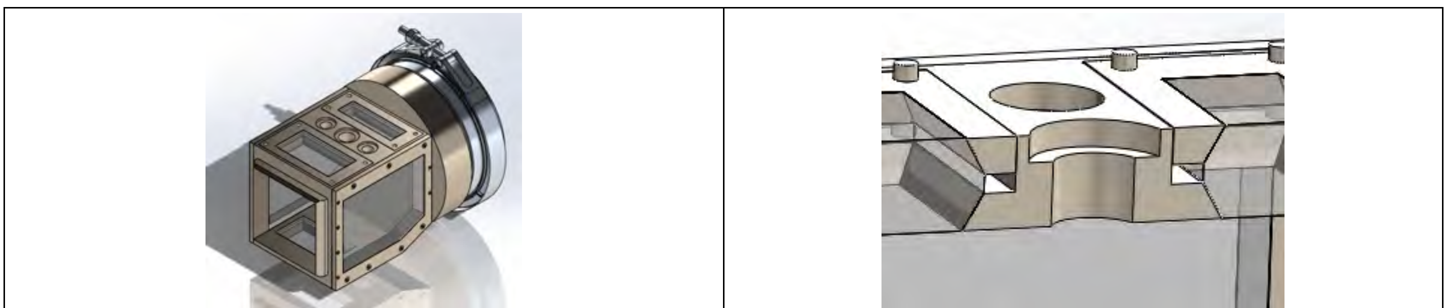
Aircraft engines use swirling inflow to aid both spray atomization as well as flame stabilization. In the project, we have used a swirler from an earlier FAA program in the current rig. This swirler is a high shear dual radial swirler as shown in Figure 2(a) and 2(b). The air blast atomizer is mounted in the center-body and is as shown in Figure 2(c). The air blast sprays are distributed evenly around the circular center-body. The collective flow path and spray are shown in Figure 2(d).



(a)	(b)	(c)	(d)

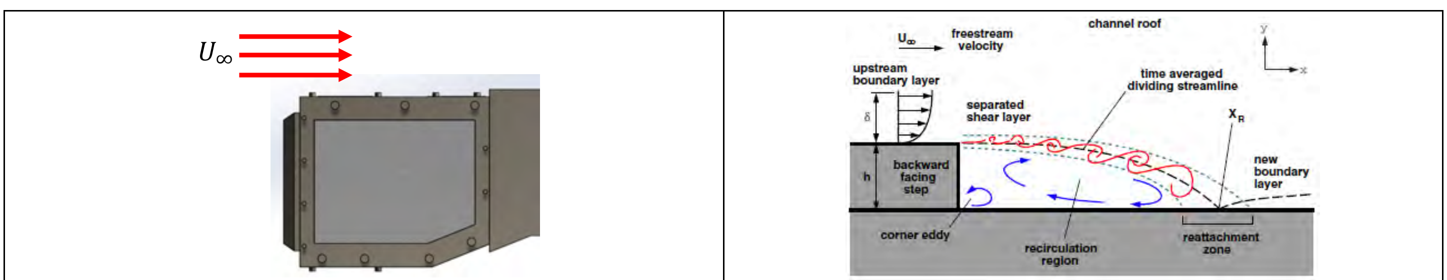
Figure 2. Inflow swirler showing: (a) one of the radial swirling inflow vanes, (b) side-view of the swirler cut-out without the center-body atomizer, (c) top view of center-body air blast atomizer (blue), and (d) side view of swirler showing swirler vanes, fuel injector, and spray.

The finalized liner design is as shown in Figure 3(a). It consists of five windows: two in the front and back, one in the bottom, and two in the top on either side of the quench hole section. Collectively, these optical access windows allow for highly detailed measurement of the flow, spray, and flame heat release. There are three quench holes in the top and two in the bottom. The quench holes are designed to have a “stepped” design as shown in Figure 3(b). The flow coming into the quench hole comes in the transverse direction as shown in Figure 4(a). The stepped design takes advantage of the recirculation flow created as shown in Figure 4(b). This creates a “turbulator” effect that induces components to the flow other than just the transverse component. This design is also adopted since it most closely represents the engine design and the rig designed/used by RTRC.



(a)	(b)

Figure 3. Combustor section showing: (a) liner with windows and frames, (b) “stepped” quench hole design.



(a)	(b)

Figure 4. Physics of “stepped” quench hole design showing: (a) transverse flow over the liner coming to the quench, and (b) flow physics at a backward facing step.

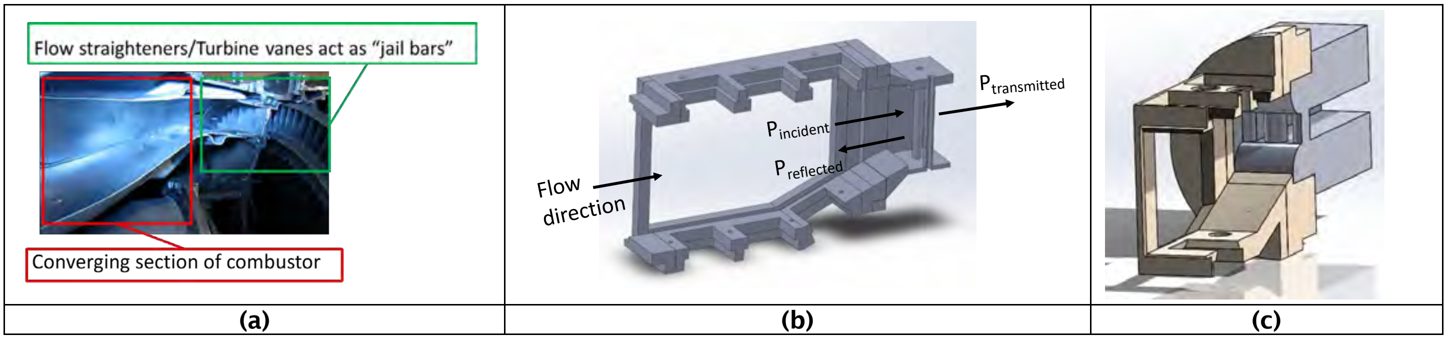


Figure 5. (a) Cross-section from a real engine showing converging section and array of turbine vanes, (b) cross-section from GT design showing converging section followed by, (c) "jail bars" to mimic turbine vanes.

The post-combustion zone architecture of the combustor is designed to closely resemble that seen in the real engine. As seen in Figure 5(a), the real engine architecture involves a converging section post-combustion that leads to the blade/vane array in the turbine section. This is replicated in the GT design. The repeating array of turbine blades is replicated through a jail bar type design. These designs in the GT rig are shown in Figure 5(b) and 5(c).

Finally, an important diagnostic feature of this rig is the measurement of post-combustion disturbances such as temperature and pressure fluctuations. Multiple pressure taps are installed, and the available pressure probes can be mounted at any of the taps and varied across different experiment runs. For temperature fluctuations, the TDLAS method is used. For this purpose, a TDLAS ring geometry is installed in the convergent section between the combustion zone and the jail bar exhaust as shown in Figure 6(a). In this ring, multiple lasers and absorption probes will be installed (see Figure 6(b)). Collectively, these measurement probes and lasers will allow for a detailed measurement of the temperature field that can then be used to understand the fluctuations in entropy that are a source of indirect noise.

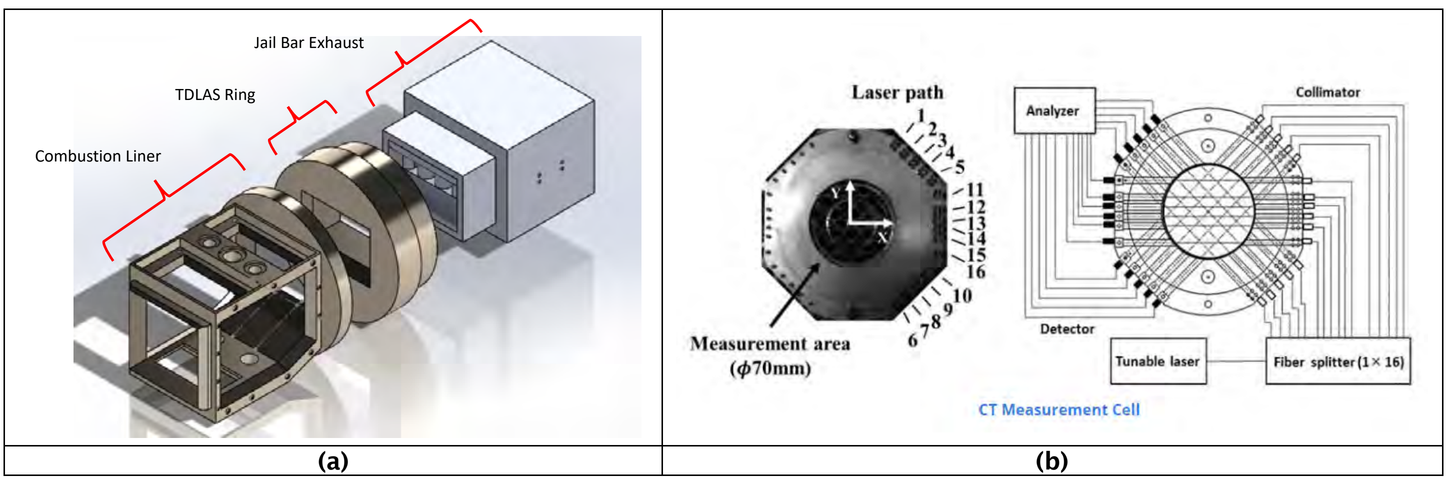


Figure 6. (a) Exploded view showing combustion liner, TDLAS, and jail bar exhaust system in the GT rig, (b) TDLAS probe array in the ring.

Milestones

- GT experiment configuration finalized.
- Operating conditions for the experiments have been finalized.

Major Accomplishments

The experiment rig has been designed to include a swirler relevant to aircraft engine operation and matching the experiment setup used by RTRC. The combustor section is designed as a RQL type borrowing from an existing lab-scale configuration.

In order to mimic the combustor exhaust to capture the effect of the turbine stage, jail bars were used to create periodic blockages that were each choked. The resultant configuration is suitable for detailed measurements of combustion noise. The range of operating conditions have been finalized. This has a major impact on this program as the second year shall focus on obtaining detailed measurements of unsteady flow, spray, flame, pressure, and temperature fluctuations. Collectively, these datasets will serve as validation data for the simulation as well as the reduced order model.

Publications
None

Outreach Efforts
None

Awards
None

Student Involvement
Lane Dillon is the graduate student taking the lead in the design of the GT experiment.

Plans for Next Period
In the second year of this project, the finalized GT rig design will be machined and installed in the pressure vessel as shown in Figure 7. The first step is to shakedown the experiment followed by which initial measurements will be taken across the board, covering flow, spray, flame, pressure, and temperature dynamics. The initial array of operating conditions and the corresponding measurements will serve as validation data for high-fidelity simulations. In addition to this, the measurements from the GT experiment campaign will be compared against those from the RTRC experiment campaign at similar operating conditions.

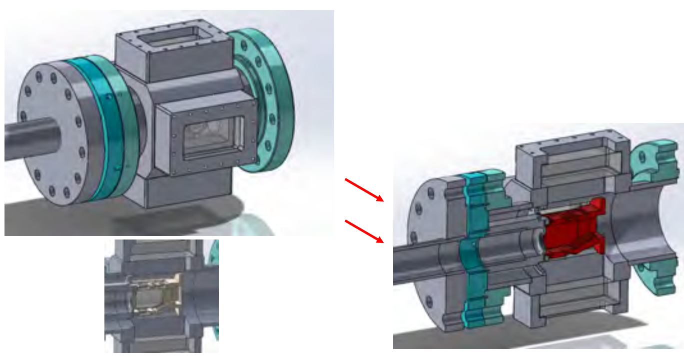


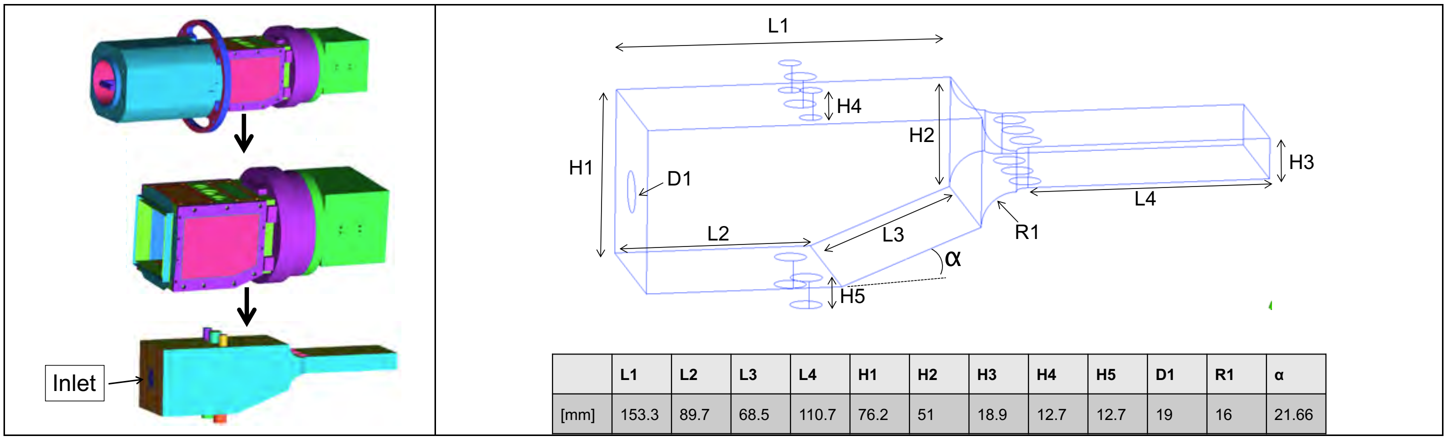
Figure 7. Placement of combustion liner inside the pressure vessel at the GT combustion lab.

Task 2 – Simulations of GT Experiment
Georgia Institute of Technology

Objectives
The objective of this task in the first year is to setup the facility geometry and flow conditions to begin the LES studies. The initial LES will involve cold-flow simulations without the swirler with appropriate boundary conditions. The spray modeling will take inputs from the RTRC Spray DNS task and a key objective of this task is to appropriately communicate the DNS data to the LES solver at GT.

Research Approach
The GT simulation effort uses the well-established compressible LES solver LESLIE to simulate the combustor configuration from the inflow swirler and spray to the choked downstream nozzle. The solver is well-suited to capture the required acoustic-vortex-heat release interactions. It is noted that the full combustor assembly has to be incorporated in order to produce

realistic acoustic disturbances and its coupling with the boundary conditions. The nozzle is needed to allow for the acoustic reflection/transmission by the throat and also to determine what disturbances are transmitted to the turbine and beyond. By modeling the entire combustor geometry, we can naturally couple the inflow with the spray and downstream choked throat's acoustic boundary. However, in this reporting period, the effort focused on a reduced geometry that incorporates a swirling inflow without the swirler but considers the jail bar exhaust as shown in Figure 8.



	L1	L2	L3	L4	H1	H2	H3	H4	H5	D1	R1	α
[mm]	153.3	89.7	68.5	110.7	76.2	51	18.9	12.7	12.7	19	16	21.66

Figure 8. Initial computational domain used for the GT experiment simulation. Left: reduction from experiment CAD model to computational CAD model. Right: dimensions of computational domain.

The initial meshing effort on this geometry resulted in a hex-only grid with 5.6 million cells and 709 blocks. The cell sizes vary between 0.25 to 0.65 mm and are calculated depending on the flow velocity and length scales of the domain. The initial cold flow simulations are performed for an inflow with air incoming at 100 m/s, 300K, and a swirl number of 1. The dilution holes are set to inflow air at 10 m/s, 300K. Since the code is compressible, a subsonic non-reflective outlet boundary condition is used. An example snapshot from this simulation is shown in Figure 9. The simulation effort is on-going and further statistical analysis will be done at the different probe points (L1-L6) shown.

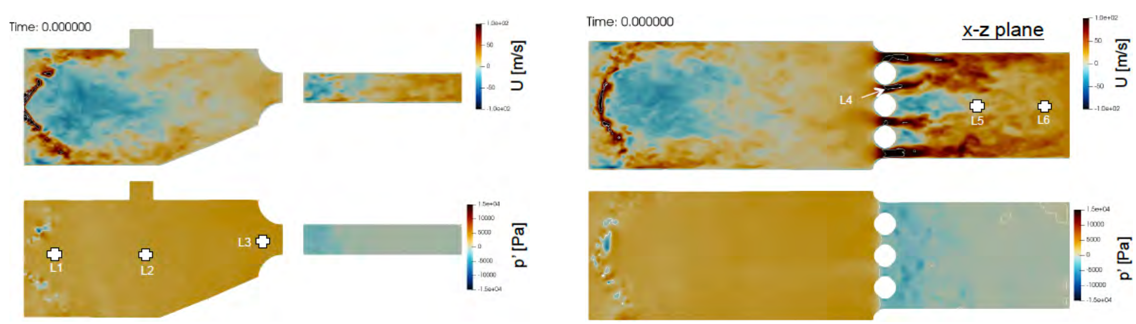


Figure 9. Snapshots of the velocity (top) and pressure fluctuations (bottom) in a cut-plane passing vertically through the center of the swirler (left) and in a cut-plane passing horizontally through the center of the exhaust section (right).

Milestone
Initial LES of cold flow in the computational domain of the GT experiment.

Major Accomplishments
The finalized GT experiment was reduced to an equivalent computational domain. This domain excludes the swirler and was meshed for an initial cold flow LES. The cold flow results are being analyzed for their statistics to understand the noise content and dominant features. In addition to this, the spray DNS by RTRC is generating data that is then being converted to an appropriate input form that can be read by the GT LESLIE code in order to perform both cold flow spray simulations as well as reacting flow simulations.

Publications
None

Outreach Efforts
None

Awards
None

Student Involvement
None

Plans for Next Period
In the second year, the simulation efforts will focus on including an accurate representation of the swirling inflow along with a swirler impedance boundary condition measured by RTRC. In addition to this, the swirler geometry shall also be included as an option for future simulations, although will be considered secondary due to its computationally expensive nature. The spray modeling needed for reacting flow simulations will come from the spray DNS provided by RTRC. The cold flow and reacting flow simulations shall also be validated against measurements from the GT experiment. Finally, additional post-processing tools will be developed in order to generate the required information from the simulation for the RTRC post-combustion post-processing task geared towards entropy wave transport modeling.

Task 3 – Reduced Order Modeling
Georgia Institute of Technology

Objectives
The overarching objective of this task is to create quick-action, reduced order models that can accurately predict different aspects of the noise generation mechanism that then collectively feed into a design tool for noise prediction. The specific objective of the GT reduced order modeling task focuses on the head-end physics in the architecture, namely flow and spray dynamics, flame dynamics, and generation of entropy disturbances by the flame. The spray/flow dynamics feed into the flame dynamics that cause direct combustion noise. The flame dynamics also result in entropy disturbances which then lead to indirect combustion noise at the nozzle. The flame response modeling and the model for the generation of entropy disturbances are provided as inputs to the post-combustion models that will be developed by RTRC. Depending on the prediction results from the RTRC models, these “head-end” models will be iteratively refined.

Research Approach
In this reporting period, the reduced order modeling activities at GT focused on developing the flame response framework in order to model the unsteady heat release disturbances. The generated unsteady heat release disturbances then lead to entropy disturbances at the flame.

Flame Response Modeling
The flame response modeling for a spray flame requires the extension of prior gaseous diffusion flame models to include the effect of spray droplets. The configuration used for this framework is as shown in Figure 10 with fuel droplets injected in a center duct and air injected in the outer ducts. The fuel flows in the inner duct $0 < r < R_F$ and the air/oxidizer flows in the outer ducts $R_F < r < R$. The fuel exits the duct and enters the combustion zone as a mix of fuel gas and a spray of liquid fuel droplets, which after evaporation and diffusive mixing result in the spray diffusion flame being modeled.

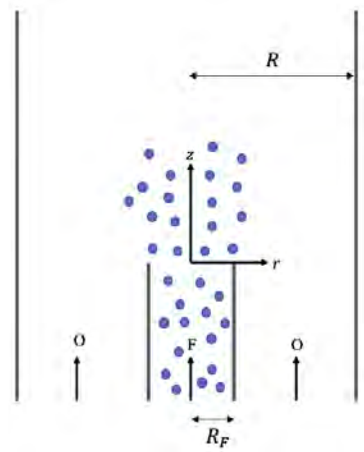


Figure 10. Schematic of the ducted spray flame configuration. Fuel droplets are injected in the center duct (shown in blue) and oxidizer gas is injected in the outer ducts.

In this work, the atomization physics for the liquid state is not considered. We assume the fuel to be injected as droplets far upstream that are then convected downstream in the fuel duct. These droplets are assumed to follow the gas flow in the combustion zone. A sectional approach is used to model the spray physics. In this approach, the continuous droplet-number distribution is divided into distinct size sections from which averaged sectional conservation equations are obtained. This simplifies the representation of an otherwise infinite size distribution of droplets. Note that while the droplets discretely exist, the sectional approach provides for a continuum representation of the droplets through their number density, which results in a continuum-based partial differential equations (PDE) for the droplet mass fraction. Under these assumptions, a governing equation for the droplet mass fraction can be derived. Following the Schvab-Zeldovich formulation, the gaseous phase is converted to a single gaseous mixture fraction ($\mathcal{Z}$), and along similar lines, the droplet phase is converted to a droplet mixture fraction ($\mathcal{Z}_d$). These mixture fractions are one-way coupled through vaporization of the droplet generating fuel gas. In non-dimensional form, the governing equations become:

$$\frac{\partial \mathcal{Z}_d}{\partial t} + u_z \frac{\partial \mathcal{Z}_d}{\partial z} + u_r \frac{\partial \mathcal{Z}_d}{\partial r} = \frac{1}{Pe_d}\left[\frac{\partial^2 \mathcal{Z}_d}{\partial r^2} + \frac{\partial^2 \mathcal{Z}_d}{\partial z^2}\right] - \Gamma_V \mathcal{Z}_d \quad (1)$$

$$\frac{\partial \mathcal{Z}}{\partial t} + u_z \frac{\partial \mathcal{Z}}{\partial z} + u_r \frac{\partial \mathcal{Z}}{\partial r} = \frac{1}{Pe_g}\left[\frac{\partial^2 \mathcal{Z}}{\partial r^2} + \frac{\partial^2 \mathcal{Z}}{\partial z^2}\right] + \Gamma_V \mathcal{Z} \quad (2)$$

Here, $Pe_d = u_0 R/\mathcal{D}_d$ is the droplet Peclet number and $Pe_g = u_0 R/\mathcal{D}_g$ is the gaseous Peclet number. The Peclet number is the ratio of the diffusion timescale to the axial convection timescale. A large Peclet number denotes diffusion dominant transport. The Damkohler number for vaporization is denoted by Γ_V and denotes the ratio of the axial convection timescale to the vaporization timescale. Large values of this parameter indicate that droplets tend to evaporate completely to fuel gas before they are transported sufficiently downstream. Notice that this parameter controls the one-way coupling from the droplet phase to gaseous phase and thus introduces the effect of the spray parameters on the gaseous mixture fraction and hence the local unsteady heat release. This local unsteady heat release is important for both the direct combustion noise modeling as well as for generation of entropy disturbances. The generation of noise by both mechanisms is a RTRC task and thus requires key inputs from this modeling task.

Generation of Entropy Disturbances

The governing equation for entropy dynamics is given by:

$$\rho T \frac{Ds}{Dt} = -\nabla \cdot \vec{q} + \underline{\tau} : (\nabla \vec{u}) - \rho \sum_{i=1}^{N} \mathcal{D}_i \nabla Y_i \cdot \vec{F}_i - \sum_{i=1}^{N} \frac{\mu_i}{MW_i}\left[\dot{w}_i + \nabla \cdot (\rho \mathcal{D}_i \nabla Y_i)\right] \quad (3)$$

As seen from this equation, entropy generation can be attributed to molecular transport (diffusion and conduction) and chemical reactions (4th term). At the flame, molecular transport processes are negligible when compared to the chemical term. Retaining only the chemical term and expanding the chemical potential results in:

$$\rho T\frac{Ds}{Dt} = -\sum_{i=1}^{N}\tilde{\mu}_i\dot{w}_i = \dot{q} - \sum_{i=1}^{N}\dot{w}_i\int_{T_o}^{T}c_{p,i}dT + \sum_{i=1}^{N}Ts_i\dot{w}_i \qquad (4)$$

Here, $\dot{q}$ is the chemical heat release term. Several prior research efforts on entropy dynamics during combustion have assumed that only the heat release term contributes to the generation of entropy disturbances at the flame and that the other terms are negligible, without evidence. A first focus of this task has been to investigate the relative contributions of the heat release term when compared to the full chemical term. To test the dominance of heat release rate over the remaining terms, a 1-D Cantera simulation was conducted for methane-air combustion. In the simulation, the ratio of the magnitude of the heat release rate to the sum of heat release rate and remaining terms was computed. The mathematical equivalent of this ratio, α, is defined below. It can be noted that if α is in close proximity to unity, $\dot{q}$ can be considered the dominant term and the remaining terms can be neglected.

$$\alpha = \dot{q}/-\sum_{i=1}^{N}\tilde{\mu}_i\dot{w}_i \qquad (5)$$

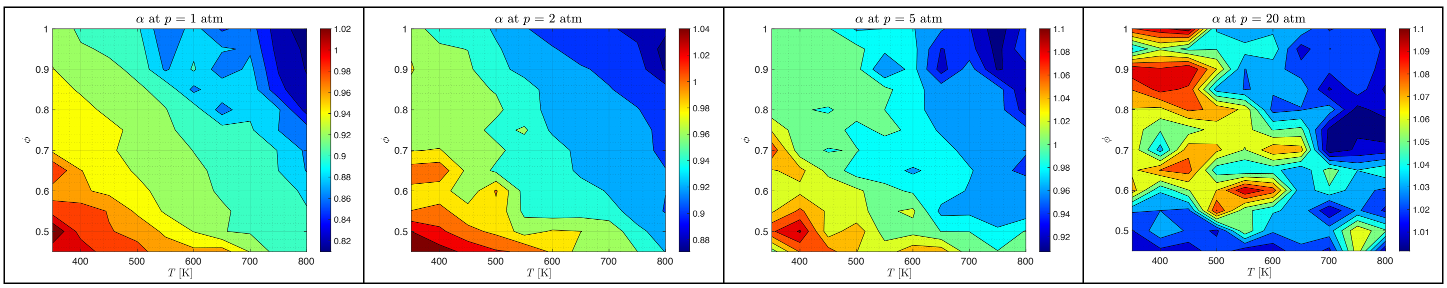


Figure 11. Variation in α for methane-air combustion simulation for a range of equivalence ratio and pre-heating temperatures at different fixed pressures.

Figure 11 shows the variation of α at different operating pressures, for a range of preheating (T ranging between 350K and 800K) and equivalence ratios (ϕ ranging between 0.45 and 1). At lower pressures, the ratio is furthest from 1 and can go as low as 0.82, indicating that the assumption breaks down. In contrast, at the highest pressure, the ratio indicates that the heat release captures 90% of the source term. This indicates that the entropy dynamics at the flame can more or less be determined by the equation:

$$\rho T\frac{Ds}{Dt} = \dot{q} \qquad (6)$$

Decomposing the entropy into its base state (subscript 0) and fluctuating component (subscript 1), the governing equation for the dynamics of the entropy fluctuations is given by:

$$\frac{Ds_1}{Dt} = \frac{\dot{q}_1}{\rho_0 T_0} \qquad (7)$$

The unsteady heat release rate disturbance term is the source of the entropy disturbance generation at the flame. Although not shown here, a second term is present but can be shown to be negligible for multi-dimensional low Mach number flows. For a 2D flame in an axial only mean flow, the solution is given by:

$$s_1(x,y,t) = \frac{1}{\rho_0 T_0 u_0}\int_{-\infty}^{x}\dot{q}_1\left(\eta, y, t - \frac{x-\eta}{u_0}\right)d\eta \qquad (8)$$

The flame response modeling work then feeds the heat release model for the above equation. The resultant entropy fluctuations generated by the above model are then provided as inputs to the RTRC entropy wave modeling sub-task.

Milestones

- Initial framework for the response of a spray flame.
- Initial framework for the generation of entropy disturbances by unsteady heat release.

Major Accomplishments

The response of a premixed flame and gaseous diffusion flame to imposed disturbances has been significantly addressed in the literature. However, the framework developed in this task is the first step towards the response of a spray flame to imposed disturbances. Specifically, the formulation brings in the parameters of the spray and explicitly shows how they affect the diffusion flame and hence the overall heat release.

Prior research on entropy dynamics has assumed that the heat release was the only dominant source for the generation of entropy disturbances at the flame. However, this assumption was never validated. The chemical kinetics analysis showed that the heat release term covers between 80–100% of the source term, thus validating the assumption that the heat release is the sole contributor to the generation of entropy disturbances at the flame. In addition, the model for the generation of entropy disturbances shows how heat release disturbances are converted to entropy disturbances at the flame.

Publications

Accepted submission to the 2021 AIAA SciTech virtual conference for the Flame Response Modeling task.

Outreach Efforts

None

Awards

None

Student Involvement

- Graduate student Parth Patki has been involved in understanding the entropy budget of the entropy dynamics equation to determine the dominant source terms for entropy disturbances.
- Graduate student Tony John has been involved in modeling the generation of entropy disturbances due to a heat release source term.

Plans for Next Period

In the next year, the reduced order modeling task will expand to include hydrodynamics stability analysis to model the pre-combustion flow disturbances. The velocity model generated from this analysis feeds directly into the flame response model. In addition to this, the spray measurements and spray DNS will be used to generate model parameters for the spray droplets used in the flame response model. In addition to this, the flame response model will be further improved to relax assumptions made in the current model. Furthermore, the results from the models will be validated against the new measurement and simulation data and iteratively improved.

The model for the entropy generation at the flame will be used with the validated flame response model to generate the source entropy disturbances which are then plugged into the entropy wave transport sub-task by RTRC. The predictions from RTRC's model at the nozzle will be validated against measurements of the temperature fluctuations. This will then iteratively feedback to improvements in both the GT entropy source model and the RTRC entropy wave transport model.

Task 4 – Facility Development at RTRC

Raytheon Technologies Research Center

Objective

The objective of this task is to design an RQL combustor that closely mirrors the design by GT, with the specific goal being to focus on the higher operating points not possible for the GT rig. Collectively, the GT and RTRC rig capabilities will encompass a broad range of operating conditions thus resulting in a robust data set to train the design tools.

Research Approach

The approach to designing the RTRC rig largely mirrors the effort at GT through close discussions and iterations of the design process. The key features of the finalized RTRC rig are discussed here.

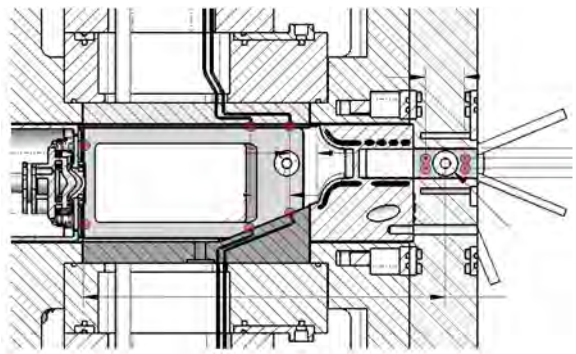


Figure 12. Cut-away of the RTRC experiment showing side view with the inflow in the left, combustion zone, convergent section, followed by bell mouth feature before jail bars. Pressure tap locations shown in red.

Figure 12 shows the cut-section of the RTRC experiment. The high shear swirler is inserted in the left. The optical access windows are present only on one side (unlike the GT rig where it is present on all sides). This RTRC liner design is incorporated into the GT rig discussed earlier with more optical windows being added for the GT experiment. The proposed pressure probe locations to measure pressure fluctuations is as shown in red. The intention of the pairs of probe locations upstream and downstream of the jail bars are for decomposing the acoustics into downstream and upstream propagating waves. This wave decomposition may help quantify the indirect noise magnitude.

Milestones

- RTRC experiment configuration finalized.
- Operating conditions for the experiments have been finalized.

Major Accomplishments

The experiment rig has been designed to include a swirler relevant to aircraft engine operation and has been designed in coordination with GT. The GT rig mirrors that of the RTRC rig except for the diagnostic capabilities. The combustor section is designed as a RQL type borrowing from an existing lab-scale configuration. In order to mimic the combustor exhaust to capture the effect of the turbine stage, jail bars were used to create periodic blockages that were each choked. The resultant configuration is suitable for detailed measurements of combustion noise. The RTRC rig shall focus on the higher end of the operating condition space.

Publications

None

Outreach Efforts

None

Awards
None

Student Involvement
None

Plans for Next Period
In the second year of this project, the finalized RTRC rig design will be used to take initial measurements of the combustion heat release (deduced from chemiluminescence measurements), pressure fluctuations at various points, and temperature fluctuations. In conjunction with the GT experiment, the data will be used for validating GT simulations performed at higher conditions.

Task 5 – Spray Modeling
Raytheon Technologies Research Center

Objectives
The objective of this task is to perform direct numerical simulations (DNS) of the proposed high shear air blast atomizing swirler in order to capture the detailed atomization process and obtain the statistics of the spray. The output from these simulations will feed into the spray modeling settings used for the GT simulations.

Research Approach
Accurate prediction of liquid fuel atomization is crucial for LES combustor noise prediction. RTRC has leveraged its internal DNS capability for spray atomization to predict droplet statistics near the injector exit. Over the last decade, RTRC has developed and extensively validated a state-of-the-art DNS capability in the form of the HiMIST code, which stands for High-fidelity Multiphase Injection Simulation Tool. This code has been applied to a wide variety of problems ranging from impinging jet atomization to liquid jet atomization in crossflow, as well as the first-ever full aero-engine swirling-flow injector atomization at ambient and high temperature-pressure conditions. Achieving these results involves some of the most advanced numerical methods, including the coupled level set and volume of fluid (CLSVOF) method for interface transport, the ghost fluid sharp-interface approach, adaptive mesh refinement (AMR), and the embedded boundary approach for flexible solid geometry handling. Realistic thermodynamic and transport properties are obtained using the National Institute of Standards and Technology's (NIST) Reference Fluid Thermodynamic and Transport Properties Database (REFPROP) in combination with established empirical correlations. The code is also massively parallelized in high performance computing (HPC) systems and can scale up to 10,000 cores.

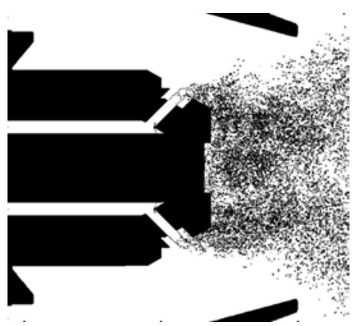


Figure 13. Snapshot of droplets and atomization from baseline production simulation. Spray is seen to remain in the core with minimal wall filming.

In this reporting period, a baseline condition was identified. This baseline production run used a 100-130 million grid on 1500–2000 HPC cores. The simulation completed roughly 2–3 flow through times and the flow-field reached a stationary state. A large region of Eulerian mesh refinement was applied in the startup simulation to ensure stability. The refinement region was manually adjusted to ensure both a stable and efficient production run. The computational cost was further reduced using grid coarsening with Lagrangian droplet transformation. These baseline conditions showed that a majority of the droplets remained in the core of the inner swirling flow (see Figure 13), indicating that the filming process could be neglected when considering spray models for a larger combustor LES in the GT simulation task.

Milestones
Established a baseline simulation to generate spray information from HiMIST.

Major Accomplishments
The HiMIST code has been used for an initial baseline simulation case in order to interface its results with the GT simulation code (LESLIE).

Publications
None

Outreach Efforts
None

Awards
None

Student Involvement
None

Plans for Next Period
In the next year, the efforts will focus on validating the simulations for conditions relevant to the GT rig. This will be done using the spray measurements from the GT experiment. In addition to this, the GT LESLIE code's spray input parameters/input file format will be used for post-processing the results from HiMIST in order to directly generate spray modeling input files for the GT simulation.

Task 6 – Swirler Impedance Modeling
Raytheon Technologies Research Center

Objectives
The major objective of this study is to characterize the acoustic impedance of the chosen high shear swirler at a range of operating conditions. This is achieved through a combination of impedance tube measurements and finite element simulations. The measurements and simulations collectively result in a validated data set of the swirler acoustic impedance. This data helps reduce the GT simulation cost by eliminating the swirler geometry from the computational domain and instead replacing it with the impedance and swirling inflow boundary condition. Additionally, the impedance is needed for the numerical Green's function approach used to characterize the direct combustion noise from unsteady heat release.

Research Approach
The swirler geometry of the high shear swirler involves several small passages of air flow that along with the grid requirements can result in computationally expensive simulations. An alternative to modeling the effect of the swirler is to use an acoustic impedance boundary condition along with a swirling inflow. In addition, the numerical Green's function approach that is part of RTRC's direct combustion noise sub-task would require the heat release modeling from GT's sub-task and the swirler impedance for noise computations. The RTRC flowing impedance tube is a well-trodden and validated approach at RTRC to characterize the acoustic impedance of several geometries. The acoustic damping of the swirler pressure drop is modeled by locally linearizing flow resistance. An important drawback of these methods is that the effect of the fuel jets/spray is not captured.

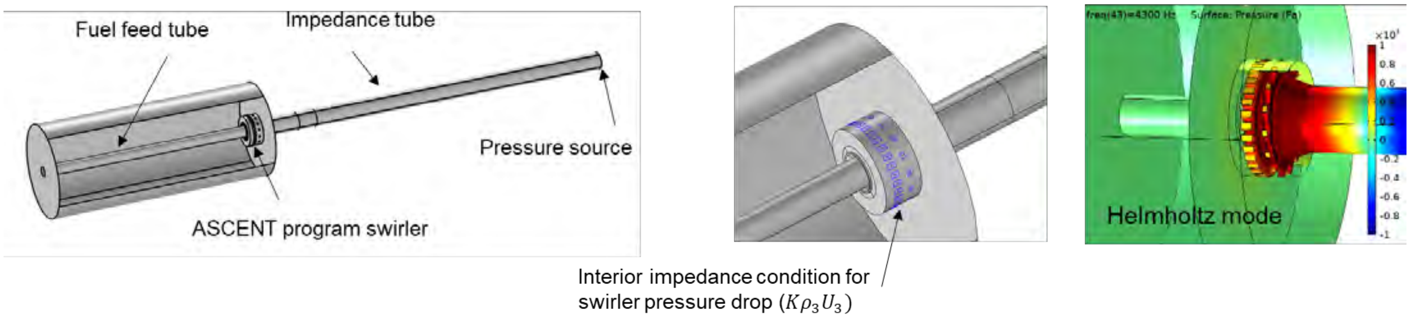


Figure 14. Geometry used for COMSOL simulations with boundary conditions to characterize swirler impedance and example mode visualization. Flow is from bottom left to top right.

In this reporting period, finite element calculations in COMSOL are used for modeling the swirler impedance as shown in Figure 14. The COMSOL simulations were performed at GT approach conditions of 118 psia, 752F. The swirler impedance is calculated just downstream of the swirler at the entrance to the impedance tube. In addition to the above geometry, the GT inlet, consisting of a perforated plate upstream of the swirler, is used.

Milestone

Established a COMSOL simulation and post-processing framework to numerically characterize swirler impedance.

Major Accomplishments

The high shear swirler has been used in a numerical impedance tube in COMSOL and its complex impedance was obtained for both the RTRC impedance tube plenum case as well as the GT combustion rig inlet case (includes perforated plate upstream of swirler). Once validated, the COMSOL finite element framework will continue to serve as a quick use tool to generate swirler impedance values at new operating conditions where impedance tube measurements are not available. This is helpful for reducing the computational cost of the corresponding GT simulations where this swirler boundary condition is required.

Publications

None

Outreach Efforts

None

Awards

None

Student Involvement

None

Plans for Next Period

Validate the finite element results using the impedance tube when hardware is available.

Task 7 – Post-Combustion Modeling

Raytheon Technologies Research Center

Objectives

The goal of this task is to develop transfer functions from the combustion zone to the nozzle, nozzle to turbine, and turbine to far-field. This involves modeling the physics for:

a) Entropy wave transport post-combustion, since unsteady heat release rate disturbances at the flame generates entropy disturbances that are then transported through the post-combustion zone.
b) Direct noise modeling using a numerical Green's function approach with the heat release model.
c) Nozzle interactions for dynamics of pressure disturbances through the nozzle. Specifically, the effect of the jail bar configuration used in both GT and RTRC rigs is investigated.
d) Turbine interactions for dynamics of pressure disturbances through the turbine.
e) Far-field sound propagation.

Research Approach

The post-combustion zone physics involves the effects of the combustion unsteady heat release rate disturbances and the post-combustion geometry on the eventual noise generation outside the engine. This involves:

a) The direct generation of combustion noise due to the heat release and the interaction of these pressure disturbances with the rest of the engine geometry which and lead to far-field noise.
b) The entropy disturbances generated by the flame interacting with geometric changes at the nozzle and causing pressure disturbances that then interact with the rest of the engine geometry and lead to far-field noise.

Entropy Wave Transport Modeling

This task focuses on how the entropy disturbances at the upstream flame are transported through the combustor to the nozzle. The modeling in this task is performed through data mining from a simulation that generates and transports the entropy disturbances as shown in Figure 15. First, the rich combustion zone in the head-end labeled "A" generates entropy disturbances through unsteady heat release rate disturbances as the source. These disturbances are also manifested as temperature fluctuations. These disturbances are convected and diffused through the downstream region and also undergo turbulent dispersion. At the secondary lean combustion zone "B", further disturbances are generated and collectively these disturbances are then transported downstream to the nozzle. At the nozzle, these disturbances are converted back into pressure disturbances depending on their magnitude at the nozzle. Thus, the important goal of this task is to measure the magnitude of the entropy disturbances at the nozzle and hence understand the importance of this indirect sound generation mechanism.

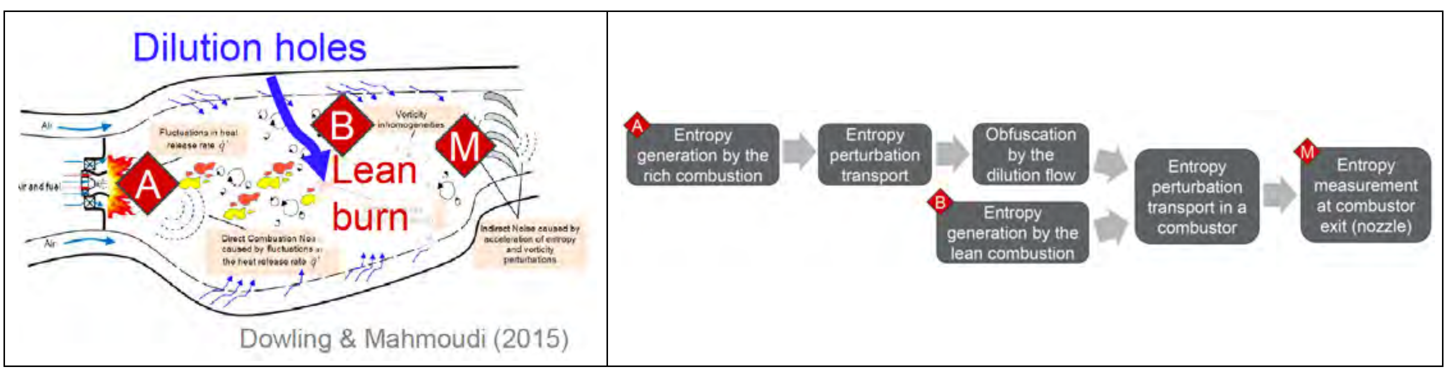


Figure 15. Mechanism of entropy wave generation and transport in combustor. Left: aeroengine combustor showing different regions with generation and transport. Right: flow chart for modeling and understanding entropy wave transport through simulation.

In order to accurately quantify the changes in magnitude of the entropy disturbances, the propagation of entropy disturbance is simulated with additional entropy fluctuations in the upstream controlled by user input. In this simulation, a fluctuation in temperature is introduced at "A" and then its magnitude is calculated at "M". Additionally, the effect of the lean burn at the dilution holes is evaluated by introducing a temperature fluctuation at "B" and calculating the magnitude at "M". In both cases, a transfer function is evaluated between "A" and "M" and between "B" and "M". The imposed disturbances in temperature are at an amplitude of 5% of the mean and the frequencies are varied. The analysis workflow is presented in Figure 16.

The results from the initial simulations showed that the transfer function between "A" and "M" is very small. The advection of the flow from "A" is much weaker than the turbulent diffusion by the lateral flow from "B". The transfer function between "B" and "M" is greater. However, note that canonical RQL combustors can have different results. An important limitation from the current analysis is that the simulation time is shorter. Furthermore, additional probe locations must be used to continually track the changes in amplitude between the sources ("A" or "B") and "M".

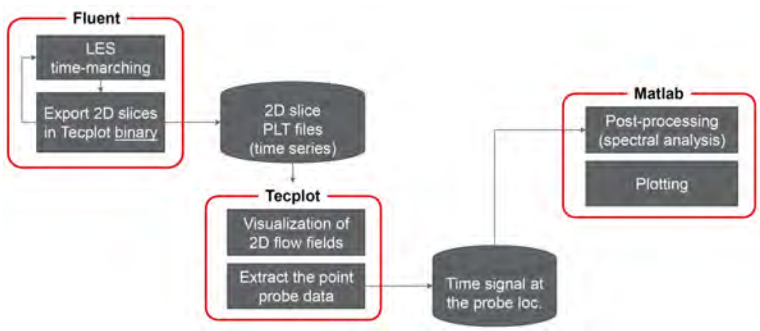


Figure 16. Analysis workflow from using Ansys Fluent for simulation to Tecplot for post-processing and visualization to Matlab for final transfer function calculation.

Direct Noise Modeling
The direct noise modeling is performed using a numerical Green's function approach that uses an appropriate Green's function in conjunction with either a measured or simulated/modeling unsteady heat release rate disturbance field, in order to calculate the pressure disturbance at a particular location in the combustor. While this was not performed in the current reporting period, a prior workflow established by RTRC under a NASA program has been reviewed and will be leveraged for this work.

Nozzle Interactions Modeling
The goal of this task is to use high-fidelity simulations to support the rig design of the jail bar configuration used by both GT and RTRC. This will help screen the jail bar nozzle design concept for potentially undesirable aerodynamics and acoustic behavior. The results also help with down selecting the final jail bar configuration and for placement of pressure sensors in the rigs. The jail bar configuration explored is as shown earlier in Figure 12(a). The equivalent computational domain is as shown in Figure 17 with a downstream extension to allow for numerical dampening of outgoing waves. A Lattice-Boltzmann compressible transonic scale-resolving flow simulation was performed. The grid consists of a total of 5 million voxels and the grid resolution used 40 points per diameter of the jail bar in the region of the jail bar as shown in Figure 17.

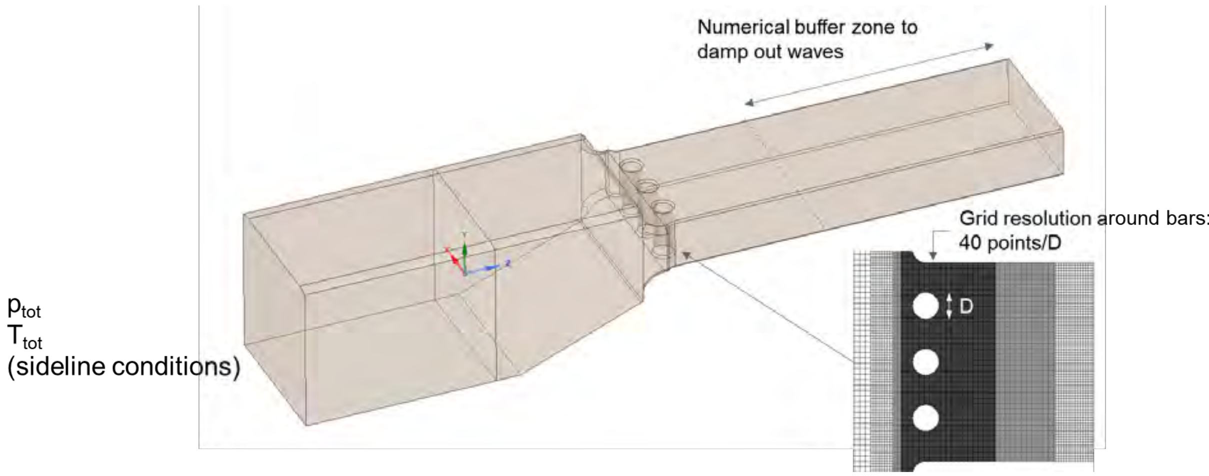


Figure 17. Computational domain and jail bar grid details for the Lattice-Boltzmann methods (LBM) simulations of the nozzle interactions.

The simulations were performed for different back pressure ratios ($p_{s,outlet}/p_{s,inlet}$) in order to assess the reflection and transmission of sound through the jail bars. A snapshot of the results from the different back pressure cases is shown in Figure 18. For the choked case (which corresponds to the highest back-pressure ratio), the presence of the vortex shedding affects shock formation, leading to oscillations in the shock location. A dynamic coupling between cylinder wake vortex shedding and shock oscillation was observed that leads to the flow going in and out of choke. This flow condition may therefore be called “nominally” choked. As the ratio is decreased, shock oscillation is strongly reduced and an asymmetric shock pattern between bars is observed. For the lowest ratio, the shocks are stable and so are the wakes behind the cylinders. For the highest back pressure case (nominally choked), it can be clearly seen that there is upstream noise propagation.

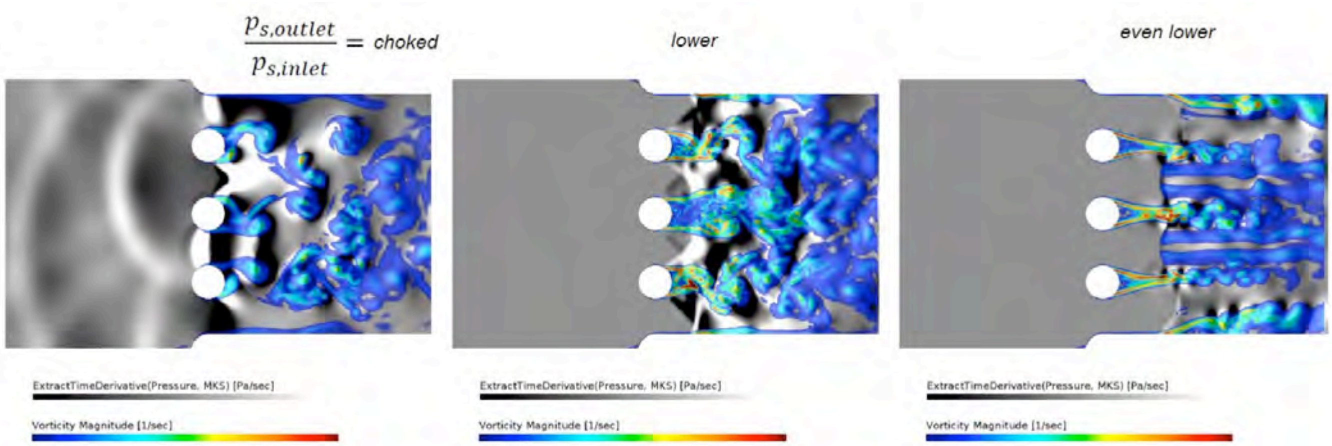


Figure 18. Snapshot of pressure field (grayscale) and vorticity (colors) for the three different cases simulated starting with a choked case until a lower back-pressure ratio.

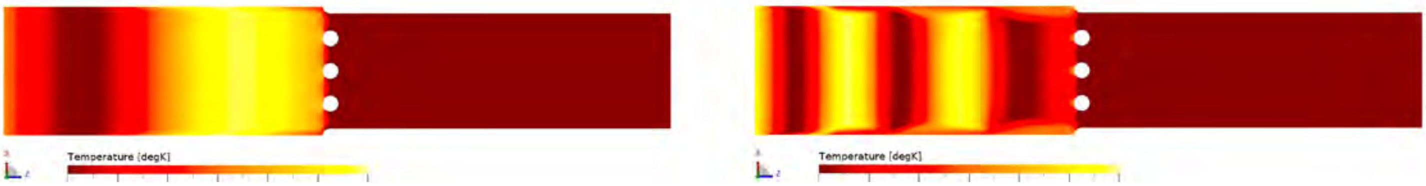
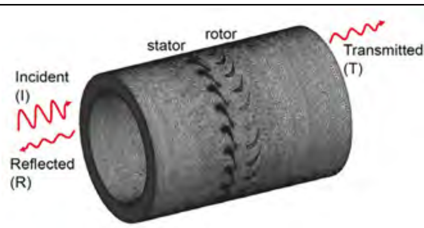


Figure 19. Snapshot of temperature field for two different frequencies. Left: lower frequency. Right: higher frequency.

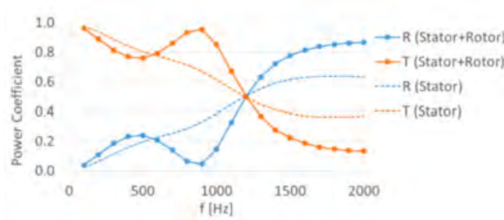
In addition to the above analysis, the effect of indirect noise due to the jail bars was also investigated. For this analysis, a similar strategy as the entropy wave transport task was adopted. The inlet was forced with temperature fluctuations that are of a convective nature. This was analyzed for two different frequencies as shown in Figure 19. The higher frequency case shows multiple waves due to the shorter wavelength. As the temperature fluctuations pass through the area change introduced by both the bell mouth and jail bars, this acceleration leads to sound generation that is both transmitted and reflected. Using an upstream and downstream probe, the pressure signal was analyzed, and it was seen that in the region downstream of the jail bars there was broadband increase in noise levels, partially due to the complex flow field resulting from the cylinder wake flow and the shock-induced flow separation near the combustor rig walls. Additionally, at the forcing frequency, the noise level was seen to be slightly higher in the downstream region, due to the noise generation from the acceleration of the temperature disturbance passing through the flow contraction.

Turbine Interaction Modeling

This task focuses on simulations of a high-pressure public domain turbine rig (Polytechnic University of Milan and the German Aerospace Center (DLR) to understand both direct and indirect noise propagation through a representative high-pressure turbine stage. The first set of simulations focused on ideal wave propagation through the turbine where there are no loss mechanisms to understand reflection and transmission of sound. For this study, the domain and mesh are as shown in Figure 20(a). The wave equation is solved using FEM Actran. This simulation provides a reference solution which can be used to verify the LBM setup and prediction results and to cross-compare with the experimental data. Plane wave acoustic duct modes are injected at the inlet of the domain and get either reflected or transmitted upon reaching the turbine stage. Both reflected and transmitted waves pass freely through the inlet and outlet of the domain via non-reflecting boundary conditions. Simulations were performed for the stator only and for the stator-rotor configuration. The transmitted (T) and reflected (R) sound power coefficients are shown in Figure 20(b). For the present turbine geometry, the stator alone provides increasing wave reflection with frequency. The combination of stator and rotor creates a dip near 900 Hz where almost no sound is reflected back. The cross-over frequency where reflection and transmission are on par is around 1200 Hz.

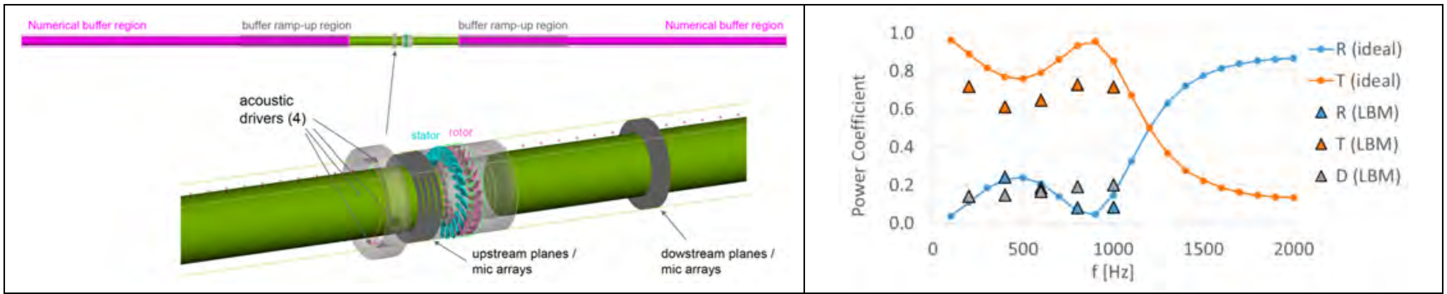


(a)



(b)

Figure 20. (a) Ideal wave simulation mesh. (b) Predicted sound power coefficients as a function of frequency.



(a) (b)

Figure 21. (a) Domain used for real wave simulations showing extended buffer regions to control wave damping. (b) Comparison of power coefficients between real and ideal wave simulations.

The next simulation focused on the real wave propagation through the turbine and used the setup shown in Figure 21(a). The simulations were performed using LBM in PowerFLOW. The comparison between the ideal wave and real wave simulations is shown in Figure 21(b). There is a close agreement for the reflection coefficients (R). The trend for the transmission (T) is the same, but some of the sound power is lost due to dissipation (D). The pumping of acoustic waves through the blade rows introduces viscous losses on the airfoil surfaces, with particularly high values at the nozzle guide vane throat (i.e., the minimum open area provided by the stator row). Upon scattering at the sharp vane and blade trailing edges and the rotor blade tip gap, vorticity waves are produced that add to the sound damping effect as shown in Figure 22 for the 1000 Hz case.

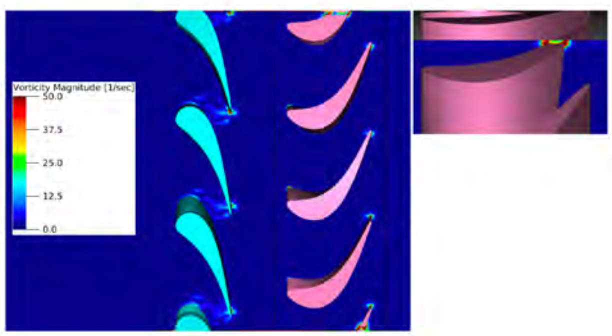


Figure 22. Induced vorticity generation in the LBM simulations of the acoustically forced stationary turbine (no flow). The forcing frequency is 1000 Hz.

The final simulation focused on the acoustic forcing for a high-subsonic flow operating condition for which the rotor speed is set to 7000 rpm. Horseshoe vortices are produced around the stator blade row, and the very large eddy simulation turbulence model quickly switches to scale-resolving mode downstream in the wake mixing region of the stator. The wave propagation behavior with acoustic forcing by the virtual duct speakers in presence of this unsteady high-subsonic flow is shown in Figure 23. As shown in Figure 23(a), the wave reflection behavior seems to be less sensitive to frequency compared to the no-flow case, with a reflection coefficient between 0.2 and 0.4. The transmitted noise is also much lower compared to the no-flow case, leaving larger values for the unaccounted term D, which is partly due to increased dissipation as the acoustic waves pass through the accelerated flow. The experimental data from DLR is plotted in Figure 23(b). The experimentally obtained reflection coefficient hovers around a similar value as in the LBM simulations and also shows little sensitivity to frequency. The transmission coefficient is also very low for all frequencies. Combined with the rather low reflection coefficient, this also leaves large values for the unaccounted part D, and the role of cut-off modes, mode scattering, and physical dissipation needs to be looked at in more detail to understand the different behavior.

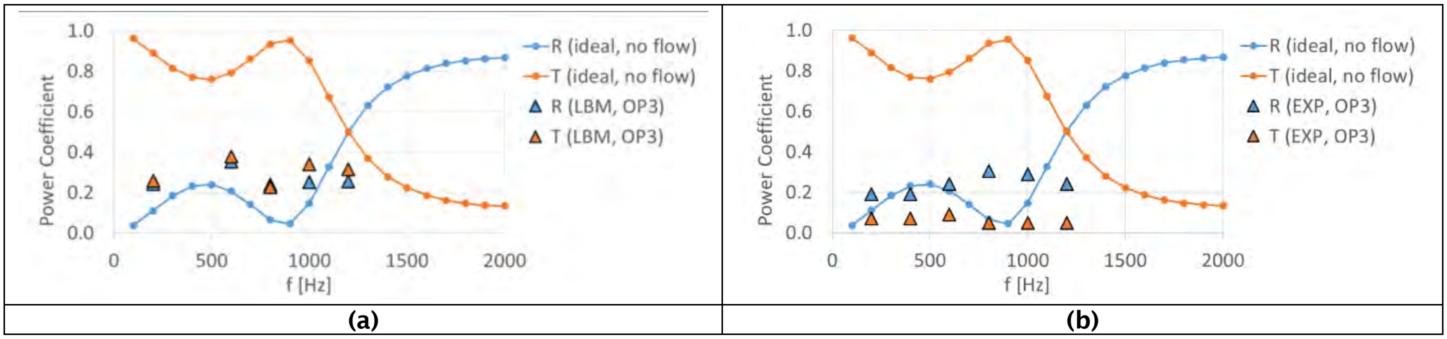


Figure 23. Comparison between (a) LBM with flow and ideal without flow, (b) experimental data with flow and ideal without flow.

Far-field Noise Modeling

This task uses the acoustic mode identified at the turbine exit as input and simulates the different physics involved in far-field noise propagation. This includes refraction due to sheared flow and temperature gradients before sound is perceived in the far-field.

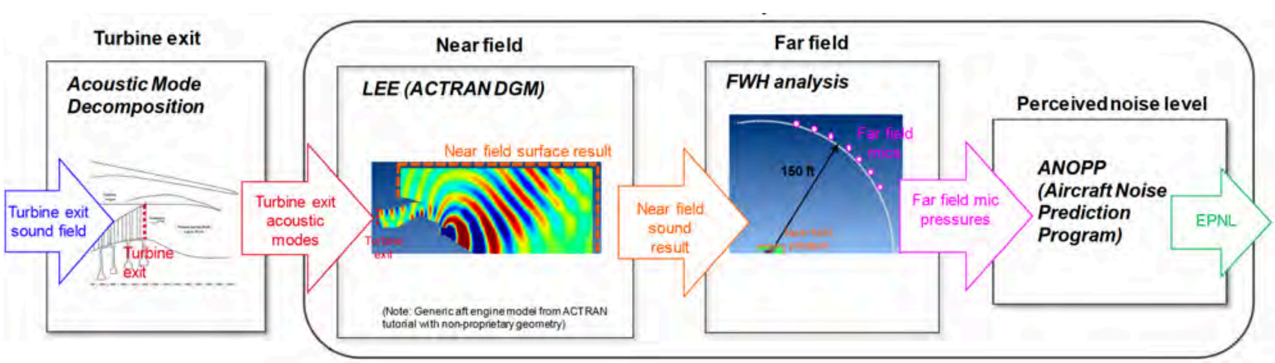


Figure 24. Simulation approach for turbine exit to far field pressure and perceived noise level.

The simulation approach used is as shown in Figure 24. First, the acoustic modes from the turbine exit are injected into the near-field simulation which uses a linearized Euler equation (LEE) approach in Actran DGM. From the near-field simulation output, the input to the far-field analysis is provided and calculated using the Ffowcs-Williams-Hawking equations. The far-field is set at 150 ft and from these far field mics, the pressure is used to measure the effective perceived noise level (EPNL) using the Aircraft Noise Prediction Program (ANOPP).

Milestones

- Entropy wave transport: Established a workflow from simulation to data processing to transfer function calculation.
- Nozzle interactions: Established an LBM simulation framework to simulate noise generation due to jail bars.
- Turbine interactions: Established a validated workflow to explore reflection/transmission of sound through a stator-rotor stage.
- Far-field sound generation: Established a multi-framework simulation workflow to go from turbine data to far-field noise.

Major Accomplishments

For the entropy wave transport modeling effort, a robust workflow has been established that performs the simulations in Fluent, post-processes the data in Tecplot, followed by which the final transfer functions are calculated using Matlab. This workflow can be used when simulation data specific to the GT rig is made available.

For the nozzle interaction task, the Lattice-Boltzmann simulation framework was used for the jail bar design to understand the effect of this geometry on noise propagation. An important result from this study was the identification of the back-pressure ratio at which there is a steady shock at the jail bars. This is important since it identifies operating conditions at which reflection from the nozzle section is minimized. It has important impacts on the transmitted noise through the nozzle which eventually affects the noise through the turbine and in the far-field.

For the turbine interaction task, a public domain turbine model with experiment data was simulated in the LBM framework and the resulted matched well with the measurements. This validates the simulation method that will be used to create a reduced order model for reflection/transmission of sound through the turbine stage.

For the far-field noise modeling task, a multi-framework simulation workflow was established and was successfully used with test data. When a validated turbine model is available, it can directly be used in this workflow to generate far-field sound data. Along with measurements, this will enable the creation of the final piece in the design tool which is to predict the perceived noise level in the far-field.

Publications

Extended abstract submitted to AIAA Aviation 2021 Conference for work in the Turbine Interaction Modeling task.

Outreach Efforts

None

Awards

None

Student Involvement

None

Plans for Next Period

For the entropy wave modeling task, the main effort next year will be establishing a workflow pipeline that can take the raw simulation data from the GT LESLIE code and convert that to a form for post-processing in the exiting workflow. A new workflow will need to be established to analyze the LES data and generate transfer functions. In addition, a reduced order model for the transport physics will be developed, which will take the heat release model as input to generate source disturbances and then a transfer function for the disturbances at the nozzle.

For the nozzle interactions task, the main effort next year will be to perform simulations on the finalized rig with accurate inflow and exit boundary conditions and generate data that can then be used to create a reduced order model for the transmission and reflection of pressure waves before and after the nozzle.

For the turbine interaction task, there are several efforts that will be addressed in the coming year. In particular, the role of the downstream struts in the experiments in mode scattering and contribution to the "lost" sound power, captured by the D contribution will be studied numerically. More details and insight into the sound dissipation mechanisms will be explored. The analysis will be expanded towards transonic flow conditions, for which initial computations have already been performed. In addition, to address the indirect noise generation (entropy conversion to acoustics) at the turbine stage, entropy wave injection near the stator leading edge will be studied with the objective to perform validations with the available published experimental data. The injection ports and forcing are well-defined and the implementation into the current simulation setup is straightforward. The indirect noise source may be an important mechanism of future combustors and we expect to capture salient trends and observations with this initial study.

For the far-field noise propagation task, further simulations in the multi-simulation framework will be performed to build a database of core noise directivity as a function of frequency and source modes from turbine exhaust at multiple flight conditions. The results will be reduced into far field transfer functions mapping core source modes to far field pressures, which will be used for predicting total far field pressures once specific source modes are determined from upstream core propagation simulations.