41st AIAA/ASME/SAE/ASEE Joint Propulsion Conference
Tucson, AZ, July 10-13, 2005

AIAA 2005-4530

Status of the Combustion Devices Injector Technology Program at the NASA MSFC

Gregg Jones[*], Christopher Protz[†], Huu Trinh[‡], Kevin Tucker[§], Tomas Nesman[**]
NASA Marshall Space Flight Center, Huntsville, AL, 35812

James Hulka[††]
Jacobs Sverdrup, MSFC Group, Huntsville, AL, 35812

To support the NASA Space Exploration Mission, an in-house program called Combustion Devices Injector Technology (CDIT) is being conducted at the NASA Marshall Space Flight Center (MSFC) for the fiscal year 2005. CDIT is focused on developing combustor technology and analysis tools to improve reliability and durability of upper-stage and in-space liquid propellant rocket engines. The three areas of focus include injector/chamber thermal compatibility, ignition, and combustion stability. In the compatibility and ignition areas, small-scale single- and multi-element hardware experiments will be conducted to demonstrate advanced technological concepts as well as to provide experimental data for validation of computational analysis tools. In addition, advanced analysis tools will be developed to eventually include 3-dimensional and multi-element effects and improve capability and validity to analyze heat transfer and ignition in large, multi-element injectors.

The task on thermal compatibility and heat transfer is to reduce local peak heat flux due to element or injector design. It is applicable to all systems, but is especially relevant to in-space engines and upper stage engines with expander cycles. Selected small-scale injectors are being hot-fire tested at The Pennsylvania State University in a highly-instrumented, 1-inch internal diameter heat sink combustion chamber with liquid oxygen and gaseous hydrogen propellants. Combustor wall local heat flux is calculated from an array of Medtherm coaxial thermocouples. These experimental data are being compared to model results from the Finite-Difference Navier-Stokes (FDNS) CFD code currently in use at the NASA MSFC. An unstructured code called STREAM using the Loci parallel processing platform is currently being developed and will be validated in the future with these test data.

The task on ignition is to model time-dependent propellant flows in the injector and combustion chamber before ignition is generated, and is applicable to all non-hypergolic systems. It is especially critical for restartable upper-stage and in-space engines where ignition is one of the critical factors in engine reliability. Mixing of two gaseous flow streams at ambient pressure is being measured at Purdue University using a laser-based scheme with seeding in one of the propellant flows. These results are being compared to a time-accurate, spatially-resolved transient CFD code developed at Purdue.

Finally, combustion stability model development is focused in two areas. An injection-coupled response model with acoustic as well as lumped-parameter components is being developed in-house to provide more capability to analyze injector elements and designs where acoustic features are present. A non-linear energy-based stability model is being developed at the University of Tennessee Space Institute to enhance prediction capability.

---

[*] Combustion Specialist, M/S ER32, NASA MSFC, AL, 35812.
[†] Aerospace Engineer, M/S ER32, NASA MSFC, AL, 35812; Member, AIAA.
[‡] Aerospace Engineer, M/S ER32, NASA MSFC, AL, 35812.
[§] Aerospace Engineer, M/S ER43, NASA MSFC, AL, 35812.
[**] Team Lead, Unsteady Fluid Dynamics, M/S ER42, NASA MSFC, AL, 35812; Senior Member, AIAA.
[††] Engineering Specialist, MSFC Group, M/S ER32, NASA MSFC, AL, 35812; Senior Member, AIAA.

I. Introduction

The NASA Marshall Space Flight Center (MSFC) has been conducting a program in fiscal year 2005 focused on improving the reliability and durability of combustion devices for the NASA Space Exploration Mission. The Combustion Devices Injector Technology (CDIT) program will improve the technology and the capability to analyze three critical requirement areas in thrust chamber design: 1) injector and chamber thermal compatibility and heat transfer, 2) ignition, and 3) combustion stability. These three areas are the dominating factors that define combustor reliability, and – significantly – engine reliability. Currently, design analysis capability in each area is largely one-dimensional and empirical. The use of advanced analysis techniques such as combustion computational fluid dynamics (CFD) in each area is limited and not widespread. Unfortunately, failures are local, not global, so use of current one-dimensional and empirical models means that developing new designs (or better understanding of current designs) requires extensive full scale testing. The objective of CDIT is to increase the analytical fidelity of each requirement area to include three-dimensional and multi-element effects, and to include the effects of real fluid properties. Thus, more detailed information about reliability-critical factors can be made available earlier in the engine development process.

For heat transfer and ignition, combustion CFD computer models are being developed and validated with specific experiments to analyze the three-dimensional, real fluid, and multi-element effects. For combustion stability, combustion CFD models are not yet tractable. Other means are being used to increase fidelity from one-dimensional and empirical limitations, including modeling more of the acoustic injector and feed system parts, and developing a non-linear predictive technique utilizing an energy equation based methodology.

For the following fiscal years, CDIT is proposing to build upon the work conducted in fiscal year 2005. The development of local heat transfer analysis capability is proposed to expand from the current LO_2/H_2 single element injectors to LO_2/H_2 and LO_2/CH_4 multi-element injectors in carefully planned steps consistent with the CFD development roadmap pioneered by the NASA MSFC. The capability to analyze local time-dependent ignition processes, which is immature, is proposed to proceed from gaseous simulants to cryogenic simulants to real propellants. Ignition is often cited as the number one factor defining engine reliability for restartable and in-space engines. The heat transfer and ignition work will support development of all types of new engines, including LO_2/CH_4 engines, as well as evaluation of the *real* reliability and durability of existing engines (which would otherwise require thousands of tests to effectively demonstrate). The current focus in combustion instability modeling on injection-coupled response and non-linear gas dynamics can significantly increase the analysis fidelity, especially in LO_2/CH_4 engines where combustion instability is an increased risk.

II. Injector/Chamber Thermal Compatibility and Heat Transfer

Injector/chamber thermal compatibility is one of the critical design requirements for any rocket combustor. Real effects of local overheating are seldom included in the design process, but are usually factored in by including empirical margins of safety on the thermal and structural analyses. Although it is usually never determined how much these combustors may be over- or under-designed, sometimes other surprising influences are discovered late in the development program. During Space Shuttle Main Engine (SSME) development, for example, blanching of the main combustion chamber wall – an injector effect – severely limited the initial predicted life of the chamber and increased the reusability operating costs due to unanticipated maintenance. The focus on injector/chamber thermal compatibility in CDIT is to improve the fidelity of heat transfer analysis capability by validating a CFD-based analysis methodology with highly-resolved small-scale experiments.

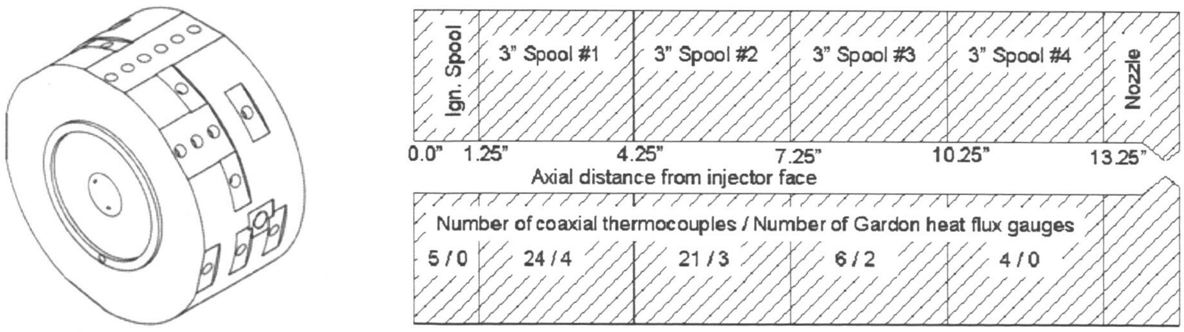
A. Overall Plan

The CDIT thermal compatibility/heat transfer task for fiscal year 2005 has two parts, an experimental program and an analytical tool development program. The experimental program is being managed by the NASA MSFC with hot fire testing to be conducted in the Cryogenic Combustion Laboratory (CCL) at The Pennsylvania State University. This program will gather local heat flux data on both conventional and promising advanced wall compatibility injector elements, using liquid oxygen and gaseous hydrogen (LO_2/gH_2) propellants. The conventional elements were selected to provide varying levels of complexity to the experimental data sets to be generated for combustion CFD code validation. The advanced elements were selected to demonstrate the technology required to improve local heat flux to less than 10% of the mean heat flux. Both single element and small multi-element hardware was fabricated and tested. A copper heat-sink combustion chamber that is highly instrumented with Medtherm coaxial thermocouples and Gardon heat flux gauges was designed and fabricated to resolve local heat flux. The injectors and chamber are discussed in more detail below.

The analytical tool development effort is also led by the NASA MSFC. Combustion CFD prediction tools are being developed to model the injector and combusting flows and to predict the injector face and combustion chamber wall heat flux environment. The NASA MSFC Finite Difference Navier-Stokes (FDNS) code with the real gas model was exercised in the injection element design phase to provide insight into proper scaling procedures and the effects of various injector features, as well as to generate pre-test predictions, using axisymmetric and 3-D geometries. These analyses will be compared to heat flux measured in the combustion chamber to validate their capability to predict single element and small multi-element heat transfer. In parallel, NASA is developing the Loci-STREAM CFD code to increase NASA's capability to run on large parallel computer networks and to add unstructured grid capabilities to the combustion CFD analysis. These codes and some of their analysis results are discussed below.

B. Combustion Chamber Design

The combustion chamber for the compatibility/heat transfer experiments is a modular, heat sink design with a water-cooled nozzle. The outer diameter of the chamber is 6 inches, and the inner diameter is 1 inch. The chamber is made up of six OFHC sections, which include one instrumented ignition spool, four 3-inch long instrumented measurement spools, and a water-cooled nozzle section. The sections mate to each other and to the injector body with a tongue and groove joint and seal and are held together with a hydraulic ram, typical of previous combustion chambers used at the CCL. An illustration of the chamber is shown in Fig. 1.



(a) (b)

Figure 1. (a) Isometric view of combustion chamber spool piece, showing typical locations for instrumentation. (b) Combustion chamber cross section, listing instrumentation in each spool piece.

Fig. 2 shows some examples of cross sectional layouts of instrumentation from these chamber spools. Generally, one side of the chamber has a higher concentration of thermocouples than the other side; for nonaxisymmetric elements, the injector will be flipped 180 degrees and re-tested so both sides of the element will be measured with high resolution.

The Medtherm coaxial thermocouple heat flux gauges consist of two Copper-Constantan thermocouples separated by a 0.250" plug, with the inner one located along the inner wall of the chamber. Fig. 3 shows a schematic of this instrument. A similar hot-fire experiment using similar instrumentation was previously conducted at the CCL using gaseous propellants.[1,2]

The chamber was designed to sustain run pressures of 1200 psia. Because the main chamber spools are uncooled, the run durations are limited by the heat flux from the hot gases to the chamber wall. Typical run durations are predicted to range from 5-10 seconds at 600 psia chamber pressure, 2-3 seconds at 900 psia, and ~ 1 second at 1200 psia.

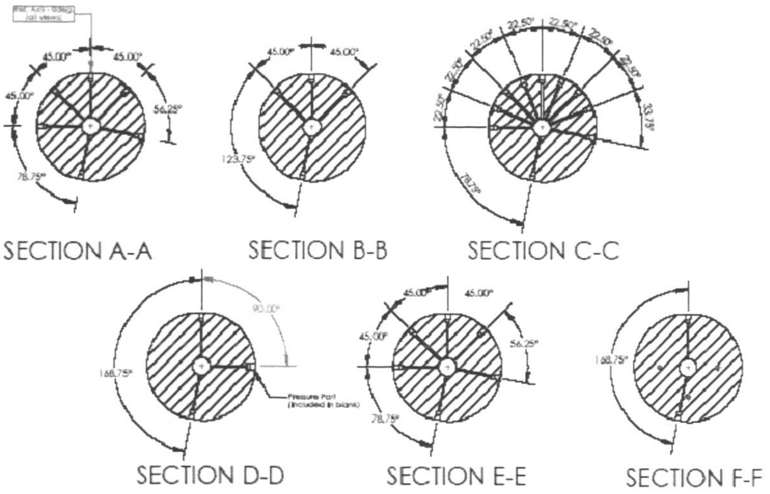


Figure 2. Examples of layouts of heat flux instrumentation in the combustion chamber spools

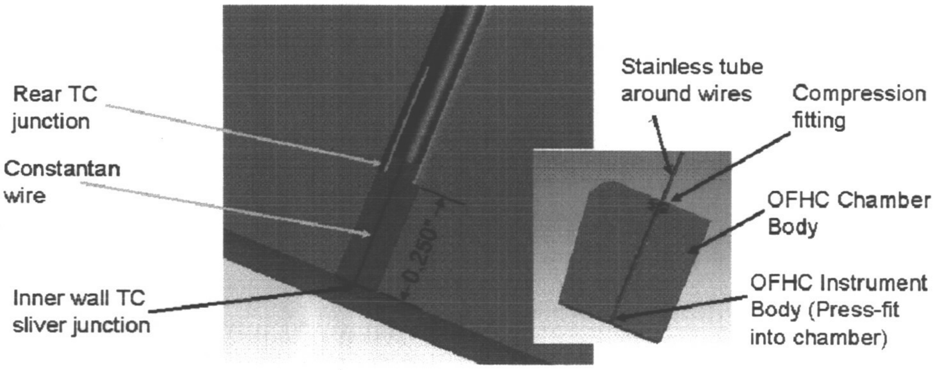


Figure 3. Schematic of Medtherm Coaxial Thermocouple

C. Injector Element Designs

Injection elements for the heat transfer rig were selected from two types – conventional historical elements with varying levels of complexity for modeling, and advanced elements specifically tailored for generating local heat flux at the chamber wall more uniform than +/-10% about the mean heat flux. The elements also were selected with regard toward applications for cryogenic upper-stage and in-space engines, for which there is some interest for Exploration missions. A summary of the elements planned for test is provided in Table I.

	1	**2**	**3**	**4**	**5**	**6**
Type	Shear Coaxial	Shear Coaxial	Swirl Coaxial	Swirl Coaxial	Shear Coaxial	Advanced Swirl Coaxial
Number of Elements	1	1	1	1	3	3
Oxidizer Post Characteristics	Recessed	Recessed, off-set	Flush	Scarfed	Recessed, off-set	Scarfed
Fuel Gap Characteristics	Concentric	Non-concentric	Concentric	Concentric	Non-concentric	Non-concentric

Table I. Injection Element Summary for Compatibility/Heat Transfer Testing stereo lithography and flowed with water in ambient conditions, where the sprays were photographed and the mass flux distributions measured in a patternator.

The oxidizer mass distribution that would reach the chamber wall unimpeded in the 3-element injector was used to simplistically determine the "perfect" element for this chamber diameter and element spacing. The model for this mass distribution analysis is illustrated in Fig. 5. Surprisingly, this mass distribution turned out to resemble most closely the swirl coax with a 45-degree scarfed oxidizer post. For this reason, this design was chosen for element #6.

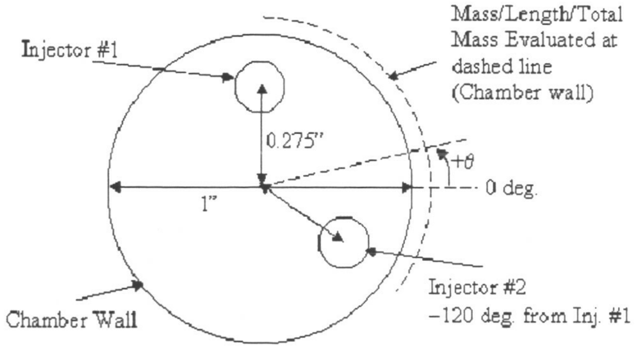


Figure 5. Analysis applied to balance oxidizer mass distribution at chamber wall for 3-element injector

However, there are multiple means to provide a uniform oxidizer mass distribution on the chamber wall with a multielement injector. The scarfed oxidizer element provides a mirror symmetrical flow. Other elements may not have the reduced oxidizer flowrate at the wall as does a scarfed element, but may still have uniform oxidizer mass flowrate on the chamber wall. One example is shown in Fig. 6, which due to summation of non-mirror symmetrical flows, as shown in the patternator measurements in Fig. 7, results in uniform oxidizer mass distribution at the wall but less mass bias than the scarfed elements. In Fig. 6, the tube cross section is circular at the inlet and gradually transforms to a D-like shape at the tube exit. It was observed that a sharp curvature on the cross section exit would lead to high mass flux concentrations, which was not desired, so more gradual curvatures were designed. An element such as shown in Fig. 6 is being considered as an additional design to bring forward to hot-fire test, or to fabricate as a backup.

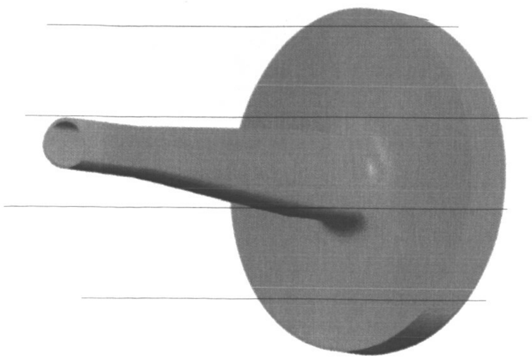


Figure 6. Advanced "D-shape" oxidizer post element

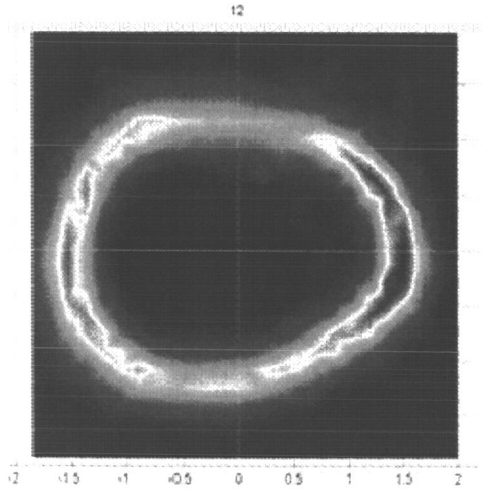


Figure 7. Water-only distribution of "D-shape" oxidizer post element

D. Test Program

The test plan was designed to provide heat transfer data at a variety of chamber pressure and mixture ratio conditions under both subcritical and supercritical oxygen conditions. Mixture ratios of 5, 6, and 6.5 will be run at various chamber pressures between 300 and 1200 psia. Mixture ratio will be changed by holding LOX flow rate constant and varying hydrogen flow rate, which will result in minor chamber pressure variations with mixture ratio around each of the test points. Test times will be limited to a few seconds to avoid overheating the heat sink chamber. Some of the skewed elements (such as the swirl coaxial element with the scarfed oxidizer post) may not reach the maximum chamber pressure due to high heat flux on one side.

Local heat flux will be calculated from the local temperatures measured by the Medtherm coaxial thermocouples in the combustion chamber with a 1-dimensional cylindrical conductive heat flux model with a lumped capacitive term for the coaxial thermocouple instrument body. This simplified calculation agreed within 1% of results from a finite element 2-dimensional ANSYS model.

E. CFD Validation

The CFD code validation effort in the heat transfer task is focused on the injector since the injector design is primarily responsible for combustor performance, compatibility, and reliability. As noted earlier, combustor failures are typically local. Historically, injector/chamber designs have been developed with one-dimensional analyses followed by extensive testing to discover and mitigate local issues resulting from the actual three-dimensional character of the complex multi-element flows. It has long been known that small changes in the injector design can have significant impact on performance and thermal environments.[6] Modern CFD codes hold the promise of quantitatively evaluating the effects of these small changes on performance and local thermal environments. However, CFD codes currently are not widely used for injector design in the rocket propulsion industry. There are several reasons for this. First, until recently, modeling the details of the physics and geometry has been difficult. Assumptions for ideal gas, simple chemistry and turbulence have been typical. Most injector solutions to date have been axisymmetric representations of single element injectors. Second, the slow solution turnaround times for even these simplified representations of the real problem have minimized their utility in the design phase time frame. Finally, there is concern about the lack of demonstrated accuracy of the solutions. There is no agreed upon validation process, and very little methodical validation work has been done. In the past few years, significant progress has been made relative to the first two issues. More realistic physical models are being used in CFD codes and the use of unstructured flow solvers has enabled the simulation of more complex geometries. Modern codes are designed to run efficiently across large numbers of increasingly fast computers. However, the continued lack of careful validation represents a significant impediment to the use of CFD codes for injector design.

A Combustion Devices Analysis Roadmap at MSFC has been developed to foster the use of CFD for design of combustion devices. Accordingly, code validation in this context is a significant part of the CDIT effort. To have any real bearing in a design setting, the CFD code must be validated to the point of qualitative agreement of relevant measures for one representative problem. This represents the minimum threshold for trend analysis, i.e., to ascertain that one design is preferred over another one. The validation conducted in CDIT reaches this level for each element considered. The relevant measure is wall heat flux and the representative problem is the single element injector. The ultimate validation goal is for the code to demonstrate quantitative agreement of relevant measures over a parametric space of the actual multi-element problem.

MSFC currently uses the FDNS code for most of the reacting flow calculations.[7-9] Other more modern codes are being developed under the Loci computing framework.[10] The FDNS code has some significant limitations since it uses a structured solver and was not initially designed for parallel execution. However, it contains significant physics, especially a fairly robust real fluids model.[11] The new codes, Loci-CHEM[12] and Loci-STREAM[13], feature solvers for generalized grids and are much more scalable in a parallel environment than FDNS. However, inclusion of all the physics required to simulate rocket injectors is in progress. Version of both codes which include real fluids models are scheduled to be available in the next several months.

Validation of CFD codes for design of combustion devices is a process. The process at MSFC began with GO_2/GH_2 propellants in a single element injector test rig where wall temperatures were measured for a coaxial injector element.[1,2] With gaseous oxygen, the complexities of multi-phase physics are avoided. Similar to the CDIT test series, measured wall temperatures were used as boundary conditions to facilitate wall heat flux calculations. The calculated heat fluxes were then compared to the experimental heat fluxes.[1,14] Both FDNS and Loci-CHEM solutions represented the data fairly well qualitatively. The Loci-CHEM solution was better in the near injector recirculation zone, while the FDNS solution better replicated the data downstream of the recirculation zone. The Loci-CHEM solution integrated the equations to the wall, while FDNS heat fluxes were calculated using wall functions. Ref. 14 provides a detailed discussion of these data and the analyses. Note that the heat flux measurements do not provide any guidance on how to improve the predictions relative to the data. This level of validation is more efficient when under-pinned by data on simpler "unit physics" problems that have detailed measurement of the flow field.

The CDIT program is testing a variety of injector elements using LO_2/gH_2 over a range of pressures from 300 psia (subcritical for O_2) to 1200 psia (supercritical for O_2). MSFC has made pre-test calculations with FDNS, using the real fluids model since LO_2 was present. Since FDNS requires structured grids, calculations were only provided for the shear coaxial single and multi-element and axisymmetric swirl injectors. As real fluid models are added to the Loci-STREAM and Loci-CHEM codes, they will be employed to model the other elements (scarfed swirl and advanced elements). A sample calculation and comparison to preliminary hot-fire test data for the concentric shear coaxial element is shown in Fig. 8. For the calculation, the wall temperature was assumed to be constant at 800 °F. The simulation matches the rise rate in the near-face region and the peak heat flux in the barrel fairly well, but underpredicts the mean level in the near-face region after the reattachment point (shown by the spike at an axial distance of approximately one inch). From the designer's standpoint, the rise rate is an important feature to obtain using CFD. Fig. 9 shows a temperature iso-surface from the FDNS simulation of the offset shear coaxial element. Note the highly three-dimensional character of the flame surface. Hot-fire test data for this element were not available for comparison at the publication date for this paper.

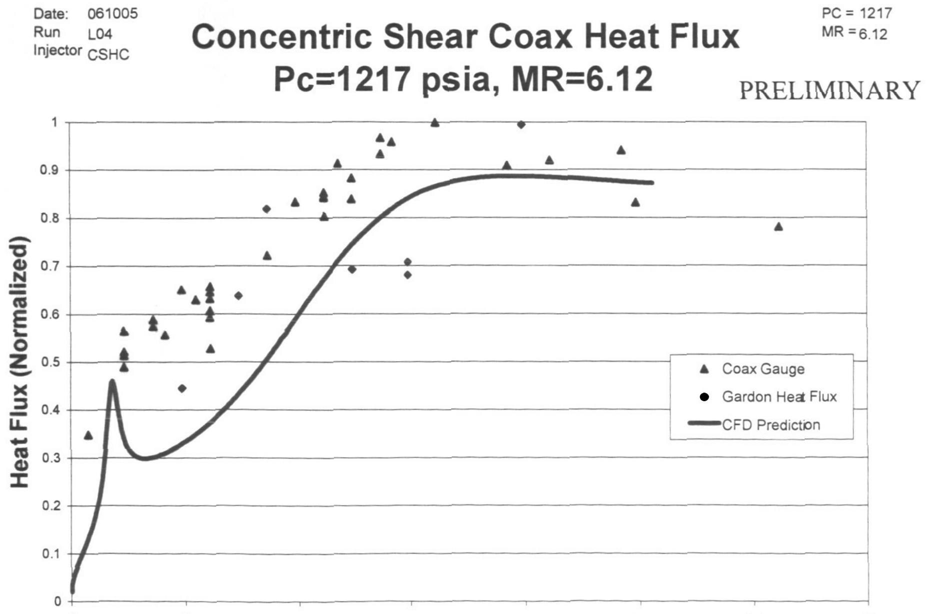


Figure 8. Preliminary FDNS heat flux comparison to measured data for the axisymmetric shear coaxial element

F. Current Results

All combustion chamber spool pieces have been fabricated, and the assembly installed in the Cryogenic Combustion Laboratory (CCL) at The Pennsylvania State University. The first three injection elements have been fabricated. Photographs of the experimental setup at the CCL are shown in Fig. 10.

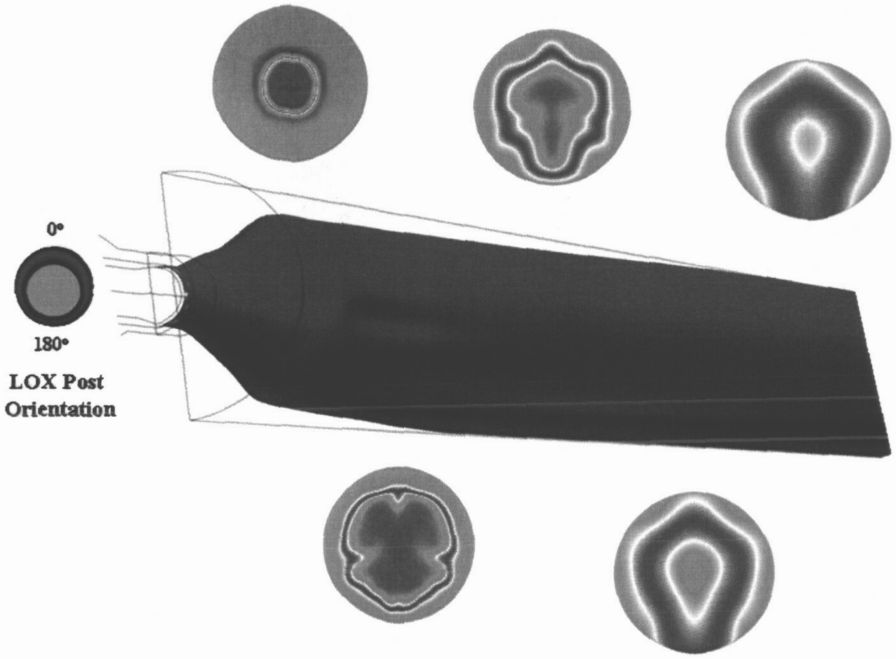


Figure 9. 5400 R iso-temperature surface from 3-D FDNS simulation of the offset shear coaxial element, and various cross-sections

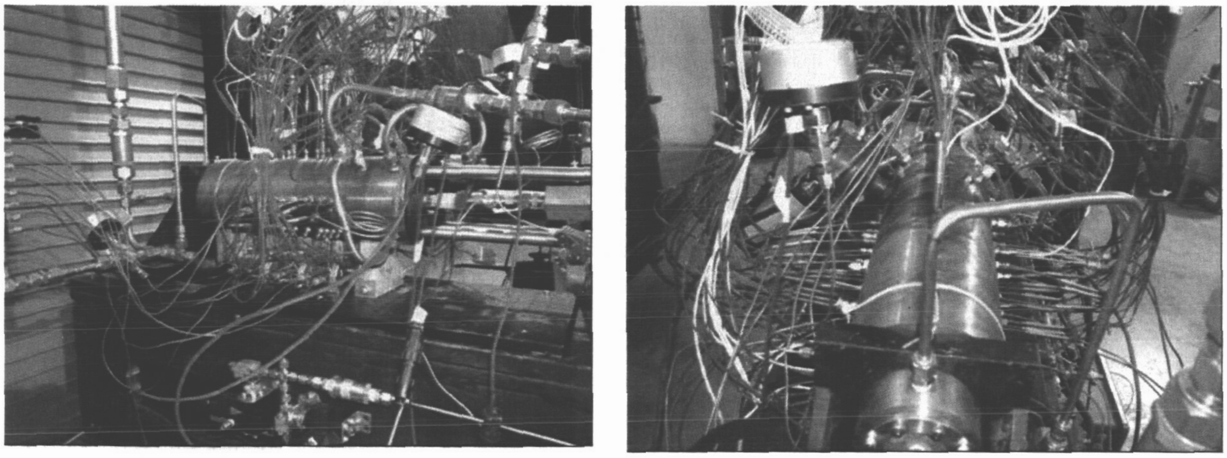
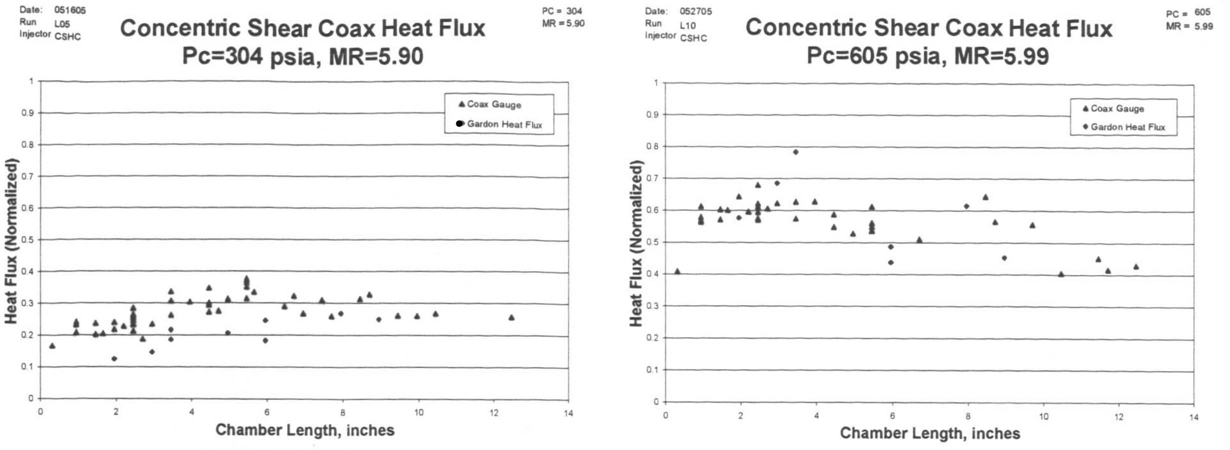


Figure 10. Photographs of the heat transfer rig installed in the CCL at The Pennsylvania State University

At the publication date for this paper, tests with injection element #1 have been completed. Preliminary heat flux measurements for a range of chamber pressures are shown in Fig. 11.

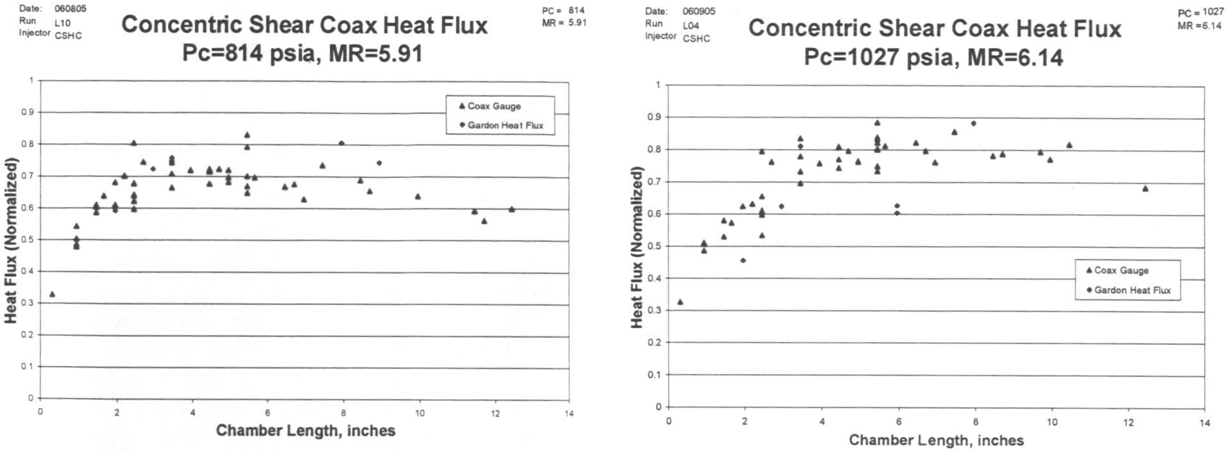
III. Ignition

For upper-stage and in-space engines that use non-hypergolic propellants and must relight, ignition may be the most important factor in the overall engine reliability. The capability to analyze local transient events such as ignition in the liquid propellant rocket combustion chamber is surprisingly immature. The focus on ignition in CDIT is to improve the fidelity of time-accurate flow analysis capability by validating a CFD-based analysis methodology with appropriate experiments.



(a) Pc=304 psia

(b) Pc=605 psia



(c) Pc=814 psia

(d) Pc=1027 psia

Figure 11. Preliminary heat flux data for concentric shear coaxial element (Element #1) at MR ~ 6. Note – heat flux for Pc=1217 psia and MR ~ 6 shown in Fig. 8.

A. Overall Plan

Predicting local mixture ratios at the time of ignition is a critical need to improve upper stage and in-space ignition system designs. While there are literature readily available to calculate flammability limits from local mixture ratios, there is little analytical methodology or empirical data available to assist in predicting the evolution of mixture ratio prior to the ignition event. The objective of the ignition task in CDIT is to provide both empirical data and a validated analytical modeling process capable of predicting local, transient mixture ratios in realistic in-space ignition scenarios.

Similar to the thermal compatibility/heat transfer task, the ignition task has two parts – an experimental program and an analytical tool development program. Both parts are directed by the NASA MSFC and both are being performed at Purdue University. The experimental program will gather space- and time-dependent mixture fraction data on typical injector/chamber geometries, which can then be used for development of advanced geometries as well as provide validation data sets for the analytical effort. The analytical prediction tool will be developed to model the pre-ignition transient from the initial opening of the fuel valve to the time the spark plug, laser, torch, or other ignition source is activated.

For cryogenic upper-stage and in-space engines, the flow conditions in the injector and chamber are two-phase oxygen mixed with gaseous fuel (hydrogen or methane) at near-vacuum conditions, with multi-element, three-dimensional geometries. To simplify this extremely complex problem, the planned activity for fiscal year 2005 will use gaseous simulants for both propellants at ambient pressures with a two-dimensional injector in a rectangular chamber. Gaseous nitrogen was selected for the oxidizer stimulant and helium was selected for the fuel (hydrogen) stimulant. The nitrogen will be seeded with 100 ppm of nitric oxide (NO) as a fluorescent tracer species. These simulants were selected because the molecular weights of these gases are similar to oxygen and hydrogen and they are non-hazardous, easily handled, and readily available.

These geometrical and propellant simplifications make the time-dependent calculation readily tractable with current computational capabilities, and provide simpler experimental requirements, but still leave enough of the overall physics intact to make substantial improvements in modeling capability. Activities for following fiscal years are planned to extend these experiments and analyses to two-phase flow, sub-atmospheric pressures, and 3-dimensional geometries, and then finally to ignite real propellants and model the flame spreading properties.

B. Experimental Hardware Design

The hardware for the ignition rig includes a two-dimensional "slab" injector and a rectangular chamber. A cross-section of this hardware is illustrated in Fig. 12.

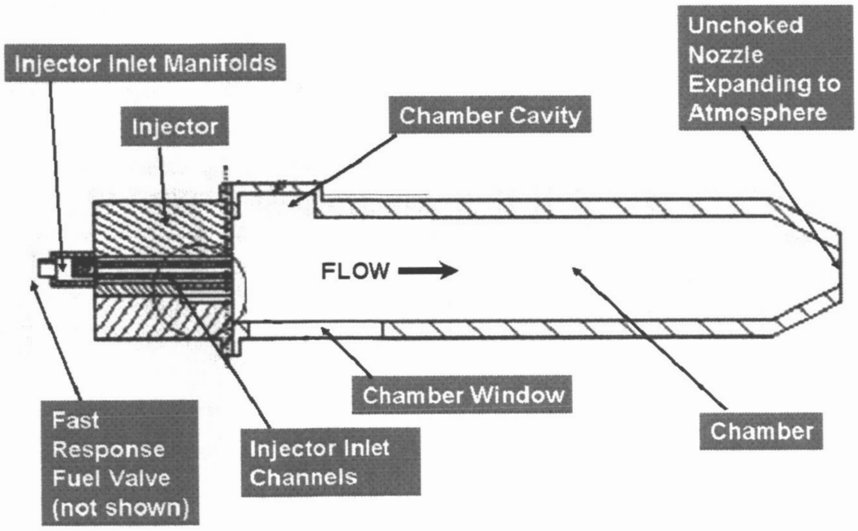


Figure 12. Illustration of hardware for the ignition experimental task

The injector is a two-dimensional "slice" of a typical coaxial injector. The injection flow areas were made equal, about 8-9% of the baseline chamber area for each stimulant, approximately simulating an injector design with warm hydrogen and an oxygen-to-hydrogen mass ratio about 6:1. Injection mass flows and mach numbers were made typical of the conditions at the introduction of propellants into the chamber prior to ignition. A first generation injector, which is being used for diagnostic setup, is shown in Fig. 13. A second generation injector has been fabricated that eliminates some of the supporting ribs in the flow slots shown in Fig. 13. A Rayleigh scattering image of a flow in ambient air is shown in Fig. 14.

The baseline rectangular chamber, shown in Fig. 15, has a width-to-height aspect ratio of 2.7:1. Following the results of the CFD calculations described in Sec. III.D, other chamber sizes are being considered for additional testing. The chamber incorporates fused silica windows on perpendicular faces which provide for laser-based imaging measurements to be conducted in the "2-D" view of the injector.

In addition to the aspect ratio, the internal chamber geometry will also be varied to simulate alternative injector/chamber ignition system configurations. The transient mixing characteristics will be measured to determine if a favorable and consistent mixture ratio can be achieved in the varying geometries.

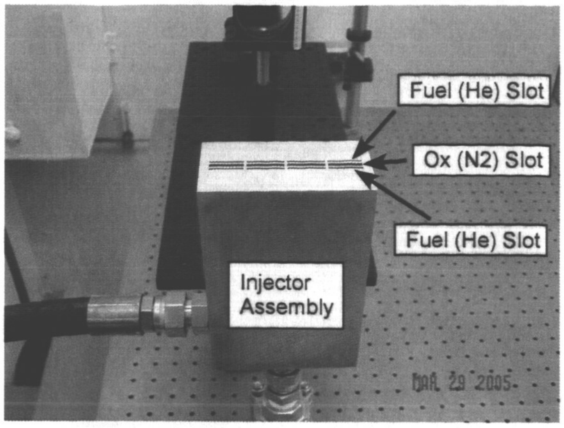


Figure 13. First-generation 2-D injector assembly.

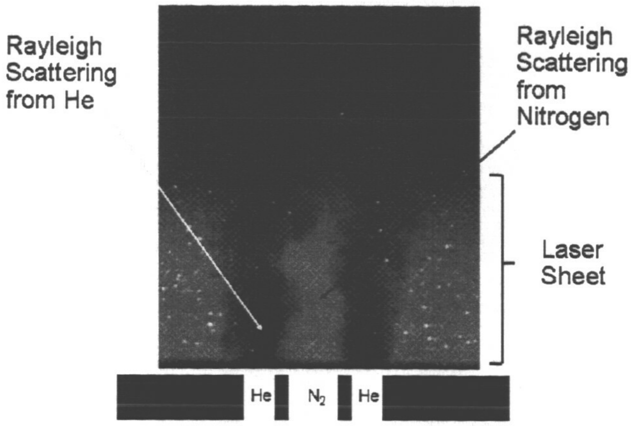
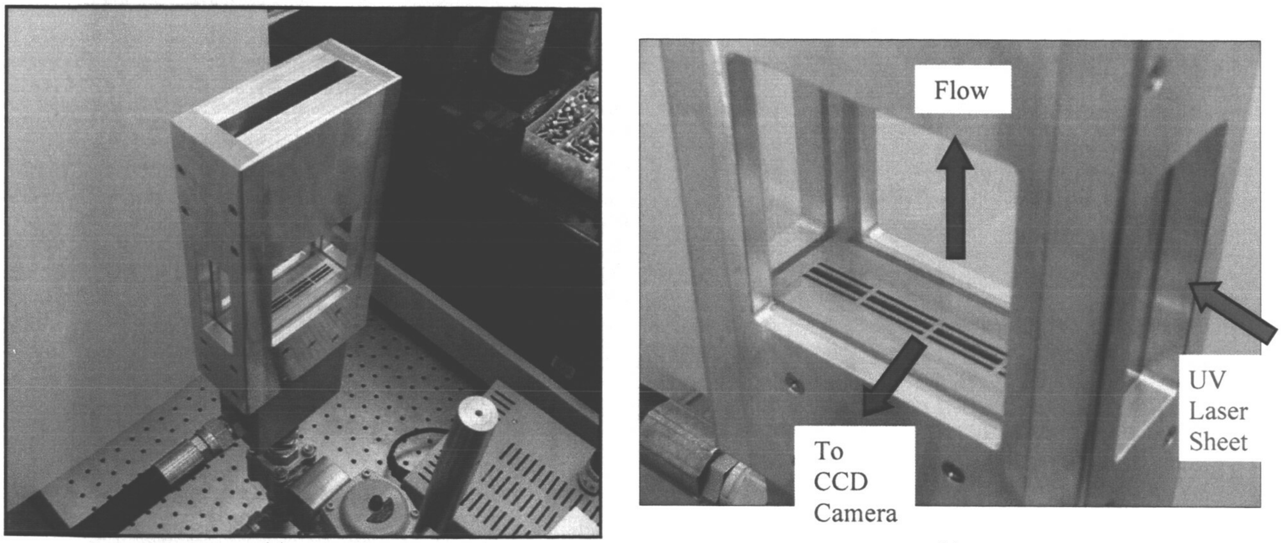


Figure 14. Rayleigh scattering images of injector flow to ambient (without chamber).



(a)

(b)

Figure 15. Rectangular combustion chamber installed for testing

C. Test Plan and Diagnostics

The optical access to the chamber allows a 226 nm laser sheet to be directed into the chamber in one plane while a CCD camera records a Planar Laser-Induced Fluorescence (PLIF) image through the opposing window, as illustrated in Figs. 15 (b) and 16. The PLIF image is made possible by seeding the nitrogen stimulant with 100 ppm of nitric oxide (NO) as a fluorescent tracer species. The use of nitric oxide as a tracer for PLIF has an extensive background.[15,16] The chamber will be at ambient conditions at the start of the experiment and will exhaust to ambient conditions throughout the experiment.

Fig. 17 illustrates the mixing experiment features. Because the nitric oxide tracer in the nitrogen will fluoresce while the helium will not fluoresce, the intensity of the fluorescence is a measure of how much helium is mixed with the nitrogen, or essentially a measure of the local mixture ratio. The measured intensity will be converted to a mixture ratio by using the intensities of pure helium and of the nitrogen seeded with NO as the baseline intensities. Regions of pure helium will result in zero signal, while regions of pure nitrogen/NO will give off the maximum signal recorded. The intensity of the mixed region will vary between these two values in relation to the relative amount of simulant propellants.

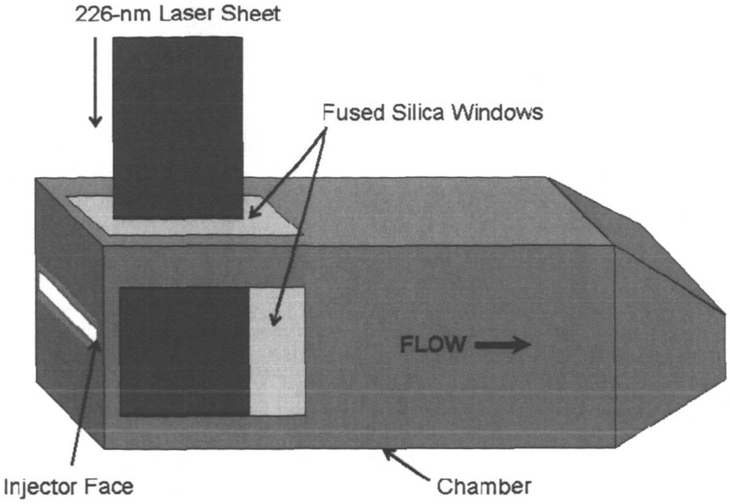


Figure 16. Illustration of transient mixing experiment.

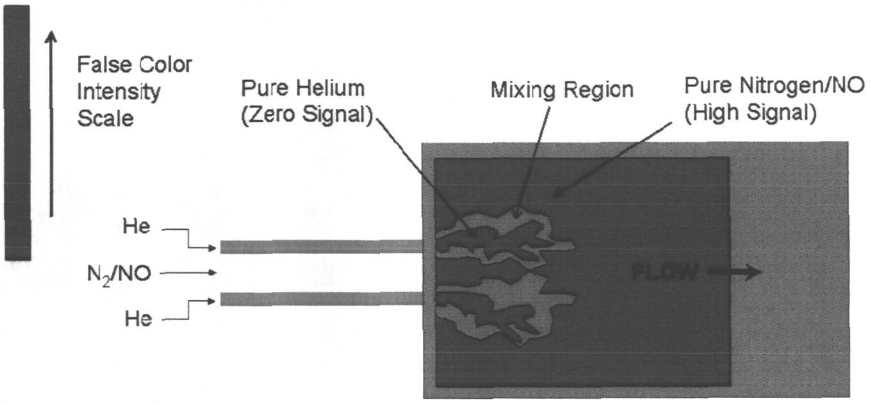


Figure 17. Representation of PLIF mixing experiment

To capture the transient mixing process, a fast-response valve in the helium line will be opened and closed in a repetitive manner. The time interval between the helium valve opening and the ND:YAG laser firing will be varied systematically, so that a series of images at different times after the valve opening can be combined. In addition, high-speed Schlieren measurements will also be taken.

D. CFD Validation

There are three objectives for CFD in the ignition subtask. The first objective is to guide the design of the experiment. Second, the CFD effort will provide an assessment of existing capability for modeling unsteady mixing phenomena in the context of rocket engine ignition. Ultimately, this assessment will serve as a point of departure for improving the accuracy and turn-around time so CFD can be used as a tool for designing ignition systems for rockets. To do this, the tool must be able to accurately predict at what time in the start transient ignitable mixtures exist as well as the spatial location of such mixtures.

The CFD code being used at Purdue University in this task is the General Equation and Mesh Solver (GEMS) code.[17] The GEMS code features second order discretization in space (flux split finite volume approximate Riemann solver) and time (fully implicit). GEMS is formulated for unstructured grids.

In terms of guiding the experimental design, the CFD effort has been used to provide insight into several issues ranging from the physical size of the experimental rig components such as chamber height and exit dimension and

N_2 and He channel lengths to rig operational issues such as global flow characteristics, required back pressure and chamber Mach number. Fig. 18 shows Mach number contours for the N_2 jet for three different combinations of N2 channel length and back pressure. Fig. 19 shows the evolving morphology of a two-propellant jet inside the chamber.

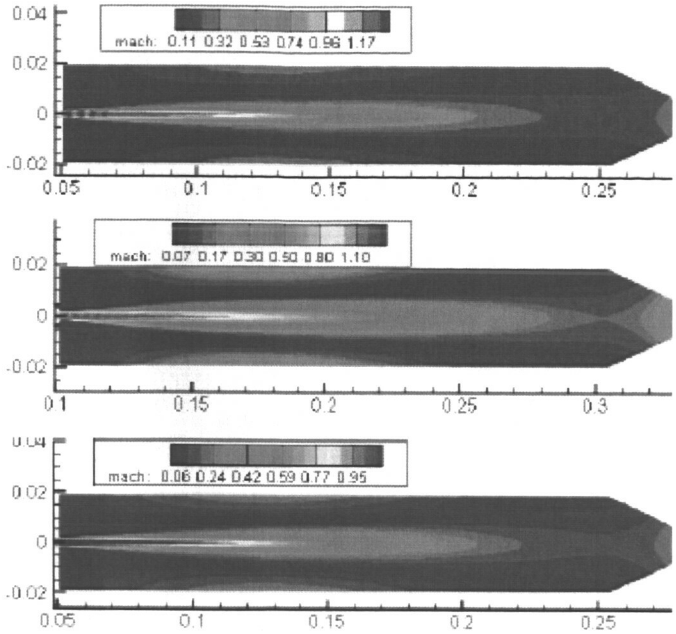


Figure 18. Mach number contours as a function of N_2 channel length and chamber back pressure.

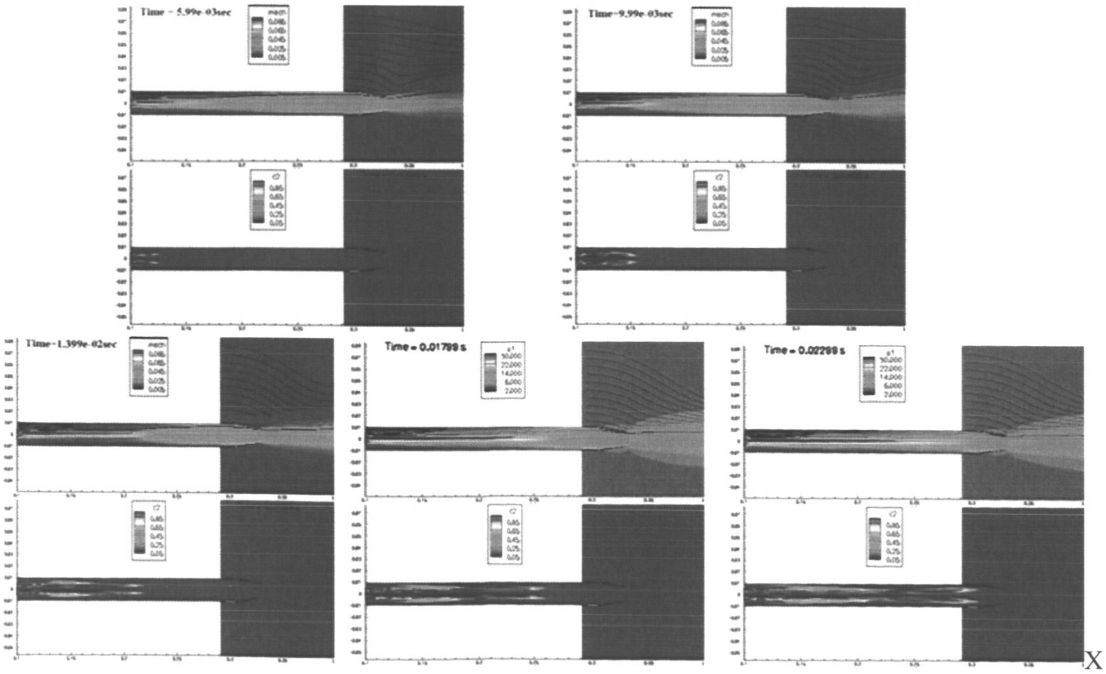


Figure 19. Evolving morphology of helium jet into a chamber filled with nitrogen.

As the experimental effort yields data, the CFD simulations will be used to quantify the level of accuracy that can be expected of CFD in terms of unsteady mixing flows with jets of dissimilar molecular weights. This effort will lead to an assessment of the utility of CFD for this type of unsteady mixing problem.

E. Current Results

Experiments are in progress. At publication time, PLIF imaging experiments were just beginning. A shadowgraph image of an injector flow with the chamber installed is shown in Fig. 20.

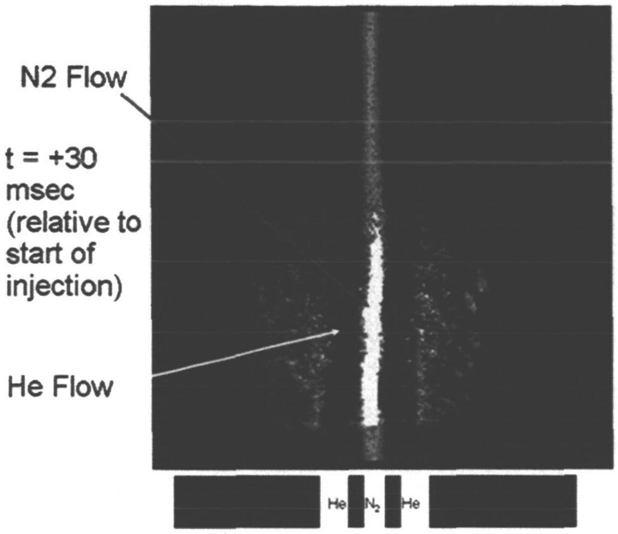


Figure 20. Shadowgraph image of simulant flow inside the chamber

IV. Combustion Stability

Some aspect of combustion instability – whether low, intermediate, or high-frequency – has occurred on every liquid propellant rocket engine development program ever attempted. The capability to analyze combustion instability or predict combustion stability margin has struggled to improve dramatically in the decades since the well-funded period of the 1950s and 1960s. A small portion of the CDIT budget has been allocated to two areas that could provide immediate payback for the future Exploration missions at NASA, as described in the following sections.

A. Injection-Coupled Response Model

The injector is a source for combustion instability – it can create, promote, or amplify combustion chamber oscillations through complementary injector element acoustics and flow instabilities. Injector element acoustics include the influences of piping, manifolds, and element feed systems. Injector-induced flow instabilities include column flow instabilities, shear layer oscillations, and vortex generation. In addition, the injection process, along with atomization, vaporization, mixing, and chemical reaction, can influence the dynamic combustion processes. An injection-coupled response occurs when the injection mass flow rate fluctuations coincide with wave motion in the combustion chamber. A worst case scenario occurs when these frequencies coincide with chamber acoustic mode frequencies.

Analysis of dynamic characteristics of liquid rocket injectors should include injector interaction with chamber and feed system, since the injector is a sensitive element with intrinsic unsteadiness that serves to modify flow oscillations or even generate flow oscillations. Over the past decades, several models have been developed to evaluate the potential for injection-coupled instability.[18-27] Early models assumed that chamber oscillations perturb the injection flow rate which in turn amplifies subsequent chamber oscillations.[9] The idea of a time lag between injection and combustion accounted for the many processes of injection and combustion. Although models of some of the processes were created, they were not a part of an overall system stability formulation.

Since those early models, some degree of complexity has been added to the analytical formulations. Furthermore, direct numerical simulation has advanced significantly with detailed description of some injection processes. Finally, experimental data and injector test data are now available from a variety of impinging, coaxial shear, and coaxial swirl elements using a variety of propellants.

In this subtask for CDIT, the injection-coupled combustion instability codes[18-27] were reviewed to determine desired features and identify possible improvements for a more comprehensive code. From this review, an analytical method that incorporates pertinent injector hydraulics was selected for further code development in the CDIT program. The effect of injector acoustics is added using a transfer matrix method to treat the various contributions as an acoustic network. The code will combine hydraulics and acoustics with sub-process models to provide an advanced injection-coupled combustion instability analysis tool. When completed, it will be made available to industry individually or as part of the ROCCID family of analysis codes.[28] The code is also planned to be integrated into the non-linear gas dynamic model funded by CDIT and described in Sec. IV.B.

The linear injection-coupled response model with acoustic as well as lumped parameter components will provide the capability to analyze injector elements and designs where acoustic features are present. Consider a model that combines feedlines, manifolds, and injection elements to get a function for injection-coupled combustion. If the model indicates a potential instability, then the injection admittance can be modified by varying the contributing geometries to reduce overall injection admittance. Another example includes propellant feed elements in the model where the code would be used to assess variations in valve pressure drop, feedline length, feed diameters, and the effects on injection admittance.

A complete dynamical system can be constructed from individual transfer functions, as shown in Fig. 21.[25,26] A closed-loop analysis can then be performed on the model subject to forced or random disturbances. Transfer functions can be used to describe all relevant sub-processes: injection, vaporization, mixing, energy release, chamber feedback, etc. A comprehensive model of injector pulsations/stability can be created by treating the injector as a combination of relevant subprocesses.

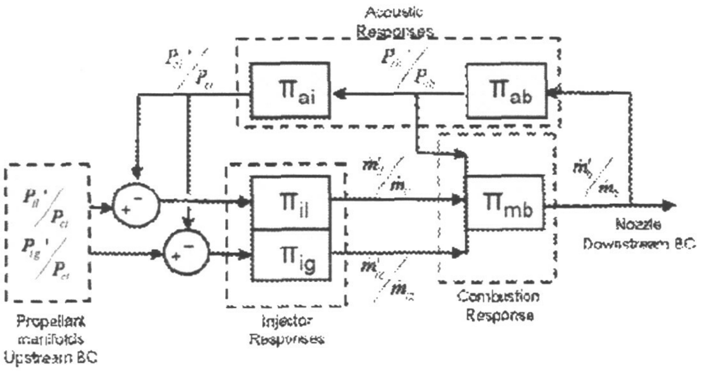


Figure 21. Response function representation of combustion instability in a liquid rocket engine.[25,26]

B. Non-Linear Gas Dynamic Model

A comprehensive modeling effort to provide improved liquid rocket engine combustion instability predictive capability and based on non-linear asymptotic perturbation techniques is being conducted at the University of Tennessee Space Institute. Features not accommodated in traditional analytical models are represented. These include steepened shock-like waveforms in place of the usual acoustic modes, limiting amplitude behavior, and interactions of waves with mean chamber properties (the "DC pressure/temperature shift" effect). Traditional simplifications are not employed – the unsteady flow model includes rotational flow effects, heat transfer, and entropy wave generation.

In order to properly represent the system behavior, it is necessary to account for nonlinear interactions as the initially low-amplitude independent acoustic wave system steepens into a shocked waveform. Fig. 22 illustrates the nonlinear effects of wave amplitude as it evolves in time. The abscissa reflects the amplitude of the composite wave system. Each of the curves represents a time-history that depends on the initial state of the system. If the oscillations grow from very low amplitude noise, an initially linear behavior ensues with each chamber acoustic mode growing independently (shown as the lower solid curve in Fig. 22). For a linearly unstable system, as the amplitude grows with time, nonlinear effects appear, and wave steepening begins; energy cascades from the lower-order to the higher-order modes. Then a finite limit amplitude is approached as illustrated. If the system is naturally or artificially disturbed by a sufficiently large pulse (shown as the upper solid curves in Fig. 22), the system may be triggered to higher limit amplitude. All of these features are captured in the nonlinear model to be developed.

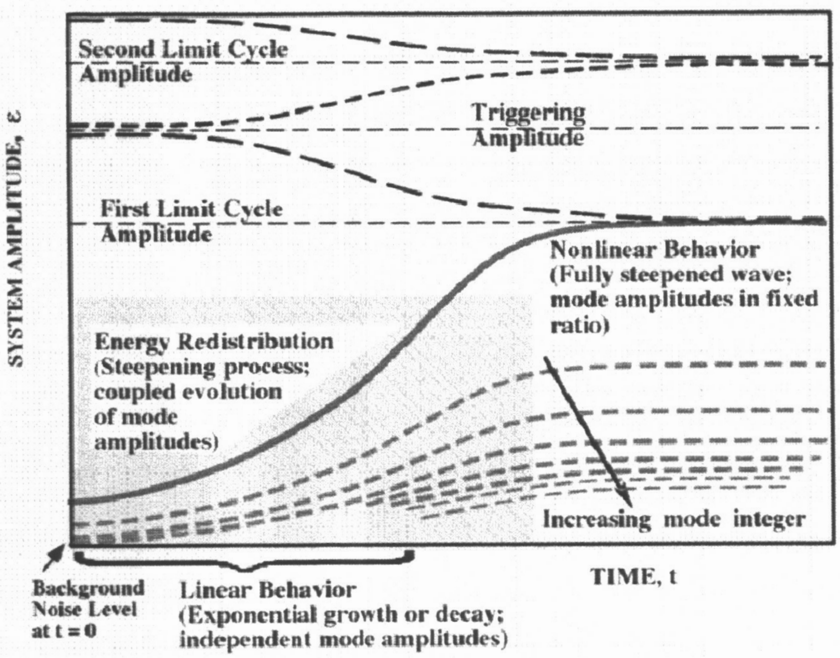
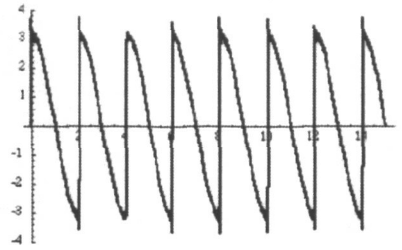


Figure 22. Nonlinear evolution of system amplitude

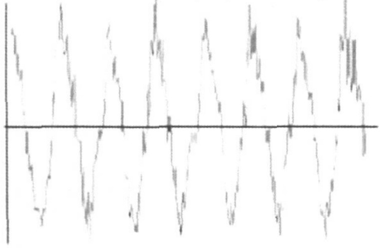
Initial model development focus is on the interaction of the chamber unsteady processes with the injector. At the injector/chamber interface there are at least three interaction mechanisms that must be included: 1) mean flow coupling with the unsteady pressure field ("mean flow driving effect"), 2) injector surface response function effect accounting for phase lag in the unsteady injector mass flux relative to the chamber pressure oscillation phase, and 3) unsteady boundary layer displacement effects. Only the second of these processes has been included in earlier models. It has been demonstrated that each of these mechanisms plays an equally important role in the overall unsteady energy gain/loss balance. Not only do the injector surface coupling mechanisms act as sources of unsteady energy to the chamber pressure fluctuations, but they also control subsequent unsteady behavior of the atomization, vaporization, mixing, and reaction processes. For example, the time-dependent combustion model under development as an integral part of the program must account for the boundary conditions at the injector face. Effects of surface heat transfer, unsteady mass and energy flux, etc., control subsequent oscillatory energy release and coupling with the chamber pressure fluctuations.

Only if all of these mechanisms are included in the system model can a reliable predictive capability be accomplished. Other secondary interactions that must be accounted for in the system unsteady energy balance include 1) coupling with large-scale vortex structures formed in highly sheared regions of the chamber flow field, 2) viscous damping effects at all inert surfaces of the combustion chamber, and 3) nozzle damping effects.

The new combustion instability model has been successfully used to analyze longitudinal mode instability observed in fuel-rich preburner experiments.[29,30] The computer algorithm correctly represented the spectrum and rate of growth oscillations in the combustion chamber. High-amplitude waves characterized the gas motion in the chamber. These were found to closely resemble traveling detonation waves rather than standing acoustic waves as treated in previous models. Fig. 23 compares the predicted and measured waveforms. All aspects of the unsteady wave motion are captured in the nonlinear combustor model, including wave reflections from the mixing ring.



a) Waveform calculated using Eq. (1) - twenty modes



b) Preburner waveform during severe oscillation

Figure 23. Comparison of Predicted and Measured Waveforms in Preburner Combustion Chamber

Experimentally determined injection response function data were utilized, and simple short-nozzle wave damping was included. Nonlinear losses brought into play by the steep wave effects accounted for the limit cycle behavior. The calculated limiting pressure fluctuation amplitude agreed reasonably well with the experimental measurement.

Current work is focused on extending the computational capabilities to include transverse oscillations. These are the most dangerous type of combustion instability confronted in liquid rocket engine development. The wave steepening has been demonstrated to play a role for these modes similar in all respects to the transition to shocked gas oscillations observed in the longitudinal mode preburner observations. It is now possible to correlate and understand historical data in a detailed way not possible using the traditional analytical methods.

The final product of this effort will be a comprehensive computer program designed for use during the design phase or during post-test diagnosis of combustion instability problems in new rocket systems. The program will enable the user to test proposed corrective procedures and to conduct detailed design parameter studies in an efficient manner. The new combustion instability analysis program is based on detailed representation of all the key physical mechanisms so that the usual guesswork can be avoided. The program is configured to utilize experimentally determined closure data (e.g., measured injector response functions) when complete analytical representations for complex parts of the system behavior are not available. The injection response model described in Sec IV.A. will also be used to supply such information.

V. Summary and Conclusions

The Combustion Devices Injector Technology (CDIT) program at the NASA MSFC was designed to provide critical analytical model development for and technology infusion into three critical areas in liquid propellant rocket engine development for NASA's Exploration Mission for the FY 2005: thermal compatibility, ignition, and combustion stability.

The thermal compatibility/heat transfer task was designed to provide a unique capability to measure *local* heat transfer and use these unique data for validation of prediction of local heat transfer by combustion CFD codes. A

1"-diameter combustion chamber highly instrumented with Medtherm coaxial thermocouples and heat flux gauges has been used to hot-fire test axisymmetric and non-axisymmetric shear coaxial and swirl coaxial injection elements, including single- and multi-element configurations. Some initial test data are presented, along with comparisons to initial modeling using the combustion CFD code FDNS at the NASA MSFC.

The ignition task was designed as the first step to develop capability to analyze transient ignition flows, here focused on the earliest pre-ignition flows prior to initiation of reacting flow. Laser-based optical imaging techniques are being used to measure the transient mixture fraction in a rectangular chamber fed with a 2-dimensional injector. Gaseous simulants are used in this initial program to simulate the pre-ignition oxygen and hydrogen (or methane) flows in a real rocket combustion chamber. Some initial test data are presented. The test configurations have been modeled using the combustion CFD code GEMS at Purdue University.

The combustion stability task was designed to continue the incremental improvement of previous efforts. Two subtasks are in work: 1) development of an injection-coupled response model, and 2) development of a non-linear gas dynamic model. The first model is being created at the NASA MSFC and is intended as a submodel to be used in system response combustion instability analysis packages such as ROCCID, but will also be used in the non-linear gas dynamic model. The second model is an extension of analysis techniques developed over many decades for solid propellant instabilities to liquid rocket engines. This effort is being conducted at the University of Tennessee Space Institute.

VI. Future Work

The current activities in CDIT are planned to be completed by the end of September 2005. Six injection element concepts will be hot-fire tested in the heat transfer rig at The Pennsylvania State University, and the data compared to combustion CFD model predictions from the FDNS code at the NASA MSFC. All ignition configurations will complete testing, and the time-dependent data compared to CFD model predictions from the GEMS code at Purdue University. Final reports from the The Pennsylvania State University and Purdue University will be submitted to the NASA MSFC for the respective activities. A full disclosure of geometrical and operating test conditions and the results of comparisons to CFD model predictions for these two tasks is planned for the 2nd Liquid Propulsion Subcommittee Meeting/53rd JANNAF Propulsion Meeting in Monterey, CA, in December 2005. For the stability tasks, final reports for both tasks – one from NASA MSFC and one from The University of Tennessee Space Institute – will be delivered to the NASA MSFC. Future publication of these developments will be made at a yet undetermined time.

Acknowledgments

The CDIT program for the government fiscal year 2005 is funded by the NASA MSFC Space Transportation Program and Projects Office under the direction of Meg Tuma, Pete Mazurkivich, Rick Ryan, and previously, Terri Tramel. Thanks to Sibtosh Pal, William Marshall, and Robert Santoro of The Pennsylvania State University for providing the figures for the heat transfer test rig, hardware components, and initial test data. Thanks to Charles Merkle, Robert Lucht, Jim Sisco, Scott Meyer, Waruna Kulatilaka, and Charles Tseng of Purdue University for providing the figures for the ignition test rig, hardware components, CFD results, and initial test data. Thanks to Gary Flandro and Joseph Majdalani of the University of Tennessee Space Institute for providing the initial draft of for non-linear instability model section. Larry Jones of Medtherm instrumented the heat transfer combustion chamber spool pieces.

References

[1] Santoro, R.J., Lin, J., Marshall, W.M., Pal, S., West, J., Wang, T.S., Williams, R., Chen, Y.S., and Lee, C.P., "GO2/GH2 Injector Testing and Analysis," Paper 3.2, NASA Marshall Space Flight Center Spring Fluids Workshop, Huntsville, AL, April 13-15, 2004.

[2] Marshall, W.M., Pal, S., and Santoro, R.J., "Benchmark Wall Heat Flux Data for a GO_2/GH_2 Single Element Combustor," AIAA Paper No. 2005-3572, July 2005.

[3] Elam, S.K., "Low Element Density Injector Test Program P329," NASA Marshall Space Flight Center Propulsion Laboratory Report, NASA MSFC, AL, June 7, 1996.

[4] Dexter, C.E., and Best, P.J., "Testing of a Pratt & Whitney LOX/H2 40K Subscale LOX Swirl Coaxial Injector at MSFC TS116," NASA Marshall Space Flight Center Propulsion Laboratory Report TR-EP73-91-03, NASA MSFC, AL, March 18, 1991.

[5] Petersen, E.L., Rozelle, R., and Borgel, P.J., "Characterization and Wall Compatibility Testing of a 40K Pound Thrust Class Swirl-Coaxial Injector and Calorimeter Combustion Chamber," AIAA Paper 91-1873, June 1991.

[6] Gill, G.S., and Nurick, W.H., *Liquid Rocket Engine Injectors*, NASA SP-8089, 1976.

[7] Cheng, G.C., and Farmer, R.C., "CFD Spray Combustion Model for Liquid Rocket Engine Injector Analyses," AIAA Paper 2002-0785, January 2002.

[8] Cheng, G.C., Anderson, P.G., and Farmer, R.C., "Development of CFD Model for Simulating Gas/Liquid Injectors in Rocket Engine," AIAA Paper 97-3228, July 1997.

[9] Chen, Y.S., "Compressible and Incompressible Flow Computations with a Pressure Based Method," AIAA Paper 89-0286, January 1989.

[10] Luke, E.A. and George, T., "A Rule-Based Framework for Parallel Multi-Disciplinary Simulation Synthesis," *Journal of Functional Programming*, Vol. 15, No. 4, 2005.

[11] Cheng, G.C., and Farmer, R.C., "Development of Linearized Real-Fluid Model in Simulating Spray Combustion Flows," January, 2005.

[12] Luke, E.A., Tong, X-L., Wu, J., and Cinnella, P., "CHEM 2: A Finite-Rate Viscous Chemistry Solver—The User Guide," MSSU-COE-ERC-04-07, Mississippi State University, September 2004.

[13] Thakur, S., Wright, J. and Shyy W., "An Algorithm for Chemically Reacting Flows on Generalized Grids Using a Rule-Based Framework," AIAA Paper No. 2005-0875, January 2005.

[14] Lin, J., Chenoweth, J.D., West, J.S., Williams, R.W., and Tucker, P.K., "CFD Code Validation of Wall Heat Fluxes for a GO_2/GH_2 Single Element Combustor," AIAA Paper No. 2005-4524, July 2005.

[15] Meyer, T.R., King, G.F., Martin, G.C., Lucht, R.P., Schauer, F.R., and Dutton, J.C., "Accuracy and Resolution Issues in NO/Acetone PLIF Measurements of Gas-Phase Molecular Mixing," *Experiments in Fluids*, 32, 2002, pp. 603-611.

[16] King, G.F., Dutton, J.C., and Lucht, R.P., "Instantaneous, Quantitative Measurements of Molecular Mixing in the Axisymmetric Jet Near Field," *Physics of Fluids*, Vol. 11, No. 1, Feb. 1999, pp. 403-416.

[17] Li, D., Sankaran, V., Merkle, C.L. and Lindau, J., "A Unified Computational Formulation for Multi-Component and Multi-Phase Flows," AIAA Paper No. 2005-1391, January 2005.

[18] Harrje, D.T., and Reardon, (ed.), *Liquid Propellant Rocket Combustion Instability*, NASA SP-194, 1972, pp 106-115, 233-263.

[19] Muss, J., "Lumped Parameters Injection Response Model (INJ)," in "User's Manual for Rocket Combustor Interactive Design (ROCCID) and Analysis Computer Program, Volume II, Appendices A-K," NASA CR 187110, by J.A. Muss, T.V. Nguyen, and C.W. Johnson, May 1991, pp. J-183 – J-185.

[20] Breisacher, K.J., "Nonlinear Injection Element Theory (LEINJ)," in "User's Manual for Rocket Combustor Interactive Design (ROCCID) and Analysis Computer Program, Volume II, Appendices A-K," NASA CR 187110, by J.A. Muss, T.V. Nguyen, and C.W. Johnson, May 1991, pp. J-175 – J-182.

[21] Priem, R.J., and Briesacher, K.J., "High Frequency Injection Coupled Combustion Instability Program (HICCIP) Status and Plans," *1990 Conf. on Advanced Earth-To-Orbit Propulsion Technology*, NASA, Marshall Space Flight Center, CP 3092, July 1990.

[22]Priem, R.J., and Briesacher, K.J., "Calculations of Combustion Response Profiles and Oscillations," Chapter 17, *Liquid Rocket Engine Combustion Instability*, V. Yang and W. Anderson, editors, Progress in Astronautics and Aeronautics, Volume 169, AIAA, 1995, pp. 455-473.

[23]Hutt, J.J., and Rocker, M., "High-Frequency Injection-Coupled Combustion Instability," Chapter 12, *Liquid Rocket Engine Combustion Instability*, V. Yang and W. Anderson, editors, Progress in Astronautics and Aeronautics, Volume 169, AIAA, pp. 345-355.

[24]Fang, J. J. and Jones, Y. T., "Development of a Generic Combustion Stability Code for Liquid Propellant Rocket Engines," *The 24th JANNAF Combustion Meeting*, Volume 2, Oct 1, 1987, pp. 239-257.

[25]Bazarov, V.G., and Yang, V., "Liquid-Propellant Rocket Engine Injector Dynamics," *Journal of Propulsion & Power*, Vol. 14, No. 5, Sept., 1998.

[26]Bazarov, V., Yang, V., and Puri, P., "Design and Dynamics of Jet and Swirl Injectors," Chapter 2, *Liquid Rocket Thrust Chambers: Aspects of Modeling, Analysis, and Design,"* V. Yang, M. Habiballah, J. Hulka, and M. Popp, editors, Progress in Astronautics and Aeronautics, Volume 200, AIAA, 2004, pp. 19-102.

[27]Miller, K. J., Anderson, W. E., and Heister, S. D., "Investigation of Liquid Rocket Injector Element Stability Margins," *52nd JANNAF Propulsion Meeting*, Las Vegas, NV, May 13, 2004.

[28]Muss, J.A., Nguyen, T.V., and Johnson, C.W., "User's Manual for Rocket Combustor Interactive Design (ROCCID) and Analysis Computer Program," NASA CR 187109, May 1991.

[29]Flandro, G.A., Majdalani, J., and Sims, J.D., "On Nonlinear Combustion Instability in Liquid Propellant Rocket Engines," AIAA Paper No. 2004-3516, July 2004.

[30]Flandro, G.A., Majdalani, J., and Sims, J.D., "Nonlinear Longitudinal Mode Instability in Liquid Propellant Rocket Engine Preburners," AIAA Paper No. 2004-4162, July 2004.