A High Heat Flux Facility Design for Testing of Advanced Hydrocarbon Fuel Thermal Stability

E. Maas, S. Irvine, R. Bates, and T. Auyeung
Air Force Research Laboratory Propulsion Directorate, Edwards AFB, CA 93524

The Air Force Research Laboratory's Propulsion Directorate is developing higher energy density hydrocarbon fuels for application in reusable liquid rocket engines. For increased performance and operability, next generation engines will require better thermal stability understanding of hydrocarbon fuels under high heat fluxes. Of the existing thermal stability test rigs, none have the ability to accurately simulate the high heat flux conditions that will exist in the cooling channels of these new high-pressure hydrocarbon engines. To design and test fuels to meet the high reliability and reusability requirements proposed for these engines, the Air Force Research Laboratory (AFRL) at Edwards AFB has designed the High Heat Flux Facility (HHFF) using experience gained from past thermal stability test rig experiments. In order to design a facility capable of simulating the higher heat fluxes expected in the channels, CFD++, a Metacomp Technologies Inc. computational fluid dynamics software suite, was employed to optimize the design prior to manufacture. Conjugate heat transfer calculations were performed in a single computational domain containing the copper heater block and the fluid channel of the new test rig design. The parameters of interest during each experiment will be heat transfer coefficient, degree of coking and corrosion in the channel, and pressure drop as a function of heat flux, wall temperature, Reynolds number, channel material, fuel composition and pressure. AFRL's HHFF will be an important tool for facilitating the development and transition of new advanced hydrocarbon fuels.

I. INTRODUCTION

Hydrocarbon fueled launch systems offer the potential for decreased vehicle size and enhanced operability compared with the larger hydrogen fueled systems. With smaller vehicles historically having cheaper operating and maintenance costs, it reiterates the benefits of hydrocarbon engines. Along with increased operability, there is a desire for increased performance in volume limited launch systems. These limitations require possible new fuels to mimic current ones regarding storage and life, while improving on payload and range. To accomplish increases in performance there is a push to increase the chamber pressure along with utilizing advanced hydrocarbon fuels with higher energy densities. These modifications result in engine conditions that increase heat flux across the engine walls, producing an even more challenging regenerative cooling environment. Due to these modifications, the fuels need to withstand higher heat fluxes without thermally decomposing and producing undesirable deposits inside the cooling passages. To simultaneously accomplish performance increases and a high degree of reusability in hydrocarbon-fueled engines, a better understanding of the fluid behavior in the cooling channels is required.

The most simulative approach to study these processes would be to build a highly-instrumented full-scale engine and conduct extensive, full duration ground tests. This is, of course, cost prohibitive, and fails to allow timeliness and flexibility, which are important during the initial stages of research and development. The key, then, is to find a way to closely simulate the actual conditions encountered by the fuel, while maintaining a cost-effective and quantitative experimental apparatus. The conditions typically found in regenerative cooling passages of modern rocket engines include high initial fuel pressures (3000-7000 psi) and low initial fuel temperatures (from the initial vehicle tank temperature to slightly above) [1]. Fuel velocities vary accordingly with the test channel cross-sectional area, providing a balance between channel pressure drop and the increased heat transfer at the highest heat flux locations of the engine. These variations to the channel dimensions can introduce secondary flow structures within the channel, and can affect convective heat transfer rates. Wall temperatures, from 500-1100 °F for copper walls, can vary from mild to pushing the safety factor for the structural integrity of the wall material and heat flux rates vary from 1 to 100 BTU/in^2/sec, producing a range of boundary layer thicknesses and potential striations.

Facilities used for studying heat transfer behavior and fuel thermal stability in the United States are resistively heated tube facilities (such as NASA Glenn Research Center's Heated Tube Facility [2-4], AFRL's Phoenix

Report Documentation Page	*Form Approved* *OMB No. 0704-0188*

1. REPORT DATE **DEC 2004**	2. REPORT TYPE	3. DATES COVERED **-**

4. TITLE AND SUBTITLE **A High Heat Flux Facility Design for Testing of Advanced Hydrocarbon Fuel Thermal Stability**	5a. CONTRACT NUMBER
	5b. GRANT NUMBER
	5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S) **Elizabeth Maas; Solveig Irvine; Ronald Bates; Tim auyeung**	5d. PROJECT NUMBER **4847**
	5e. TASK NUMBER **0244**
	5f. WORK UNIT NUMBER
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) **Air Force Research Laboratory (AFMC),AFRL/PRSA,10 E. Saturn Blvd.,Edwards AFB,CA,93524-7680**	8. PERFORMING ORGANIZATION REPORT NUMBER
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)	10. SPONSOR/MONITOR'S ACRONYM(S)
	11. SPONSOR/MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution unlimited

13. SUPPLEMENTARY NOTES

14. ABSTRACT
The Air Force Research Laboratory's Propulsion Directorate is developing higher energy density hydrocarbon fuels for application in reusable liquid rocket engines. For increased performance and operability, next generation engines will require better thermal stability understanding of hydrocarbon fuels under high heat fluxes. Of the existing thermal stability test rigs, none have the ability to accurately simulate the high heat flux conditions that will exist in the cooling channels of these new high-pressure hydrocarbon engines. To design and test fuels to meet the high reliability and reusability requirements proposed for these engines, the Air Force Research Laboratory (AFRL) at Edwards AFB has designed the High Heat Flux Facility (HHFF) using experience gained from past thermal stability test rig experiments. In order to design a facility capable of simulating the higher heat fluxes expected in the channels, CFD++, a Metacomp Technologies Inc. computational fluid dynamics software suite, was employed to optimize the design prior to manufacture. Conjugate heat transfer calculations were performed in a single computational domain containing the copper heater block and the fluid channel of the new test rig design. The parameters of interest during each experiment will be heat transfer coefficient, degree of coking and corrosion in the channel, and pressure drop as a function of heat flux, wall temperature, Reynolds number, channel material, fuel composition and pressure. AFRL's HHFF will be an important tool for facilitating the development and transition of new advanced hydrocarbon fuels.

15. SUBJECT TERMS

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT **unclassified**	b. ABSTRACT **unclassified**	c. THIS PAGE **unclassified**		**13**	

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std Z39-18

Rig, and UTRC[5,6] and Rocketdyne's[7,8] heated tube rigs), conductively heated thermal concentrators (such as Aerojet's Carbothermal rig[9,10]), combustion, arc lamp and laser heating facilities, with the bulk of the data generated coming from the heated tube and conductively heated concentrators[1-12]. For the heated tube facilities, direct ohmic heating of both pure metal and bimetallic co-annular tubes is used to produce high wall temperatures and circumferential heat transfer to the fuel coolant. External wall temperature measurements and electrical power consumption are used to infer the internal wall temperature[2-8,11,12]. The implementation is relatively simple and allows for easy post-test specimen interrogation of coking and corrosion behavior. Unfortunately when the tube metal is copper or one of its alloys, simulative to the engine cooling channels, there are extreme power input requirements reducing the typically achievable heat fluxes to less than 10-15 BTU/in ^{2}/sec. Due to the power requirements, it is critical to achieve electrical isolation of thermocouples without thermal contact loss or significant thermal resistance. In the mid-eighties, Cook reported that the Rocketdyne facility had generated a heat flux of 84 BTU/in^2/sec using a bimetallic tube configuration[8], but it required a current use of 4000 Amps and introduced extremely ambiguous results. Electrically driven chemical phenomena, as well as potential degradation of the tube materials, could not be ruled out as contributing factors in the results observed. It was also reported that investigations into asymmetric heating of square tubes using heating strips applied to one side was accomplished at Rocketdyne. Conditions studied ranged from 0.27 to 2.36 BTU/in^2/sec with wall temperatures approaching 500 °F. Work at UTRC's heated tube included bimetallic tube configurations and formed much of the early understanding of RP-1 coking and the role of sulfur compounds in copper corrosion[7,8]. At NASA Glenn Research Center's Heated Tube Facility, Meyer and coworkers are actively investigating thermal decomposition and heat transfer behavior under rocket-like conditions up to 15 BTU/in^2/sec[2-4]. Channel curvature and aspect ratio effects have been studied as well[3]. The facility was recently modified in collaboration with AFRL to study sub-cooled hydrocarbon propellants.

Aerojet's Carbothermal Rig was based on a large copper conduction block internally heated by cartridge heaters[9,10]. The block transferred the thermal energy onto a small test section slab with a milled channel and braised closeout panel, which was simply machined off to expose the wetted wall for interrogation. Pressures up to 3500 psi were studied using a variety of hydrocarbon fuels, including methane, propane, RP-1, and simulants of RP-1 doped with impurities. The rig generated heat fluxes up to 53 BTU/in ^{2}/sec and a range of heat transfer, coking, sulfur corrosion, channel coating and chemical refurbishment data. Unfortunately, it is no longer in existence.

Of the heat transfer rigs that have been used in the past, conduction heated rigs offer the closest simulation of the high heat flux conditions encountered during regenerative cooling of the rocket engine, with minimum complexity and maximum flexibility[9,10]. Asymmetric heating and easy instrumentation access of the channel are all advantages of this facility. Channel geometry and material can be freely adjusted without impact on the electrical input characteristics. Issues with the extremely large currents (> 4000 Amps) required by the resistively heated tube facilities to develop high heat flux and wall temperatures are avoidable. Thus, from the degree of simulation achievable, and the advantages of quantitative measurement and control of experiments, the conductively heated design was chosen. The Air Force Research Laboratory (AFRL) High Heat Flux Facility (HHFF) was designed as an extension of the Aerojet Carbothermal rig to achieve heat fluxes up to 100 BTU/in ^{2}/sec.

II. COMPUTATIONAL DESIGN EFFORT

CFD++ is a computational fluid dynamics software suite developed by Metacomp Technologies Inc. that was employed to perform fluid flow and heat conduction calculations on the new HHFF test section assembly design. CFD++ has the ability to handle the conjugate heat transfer (CHT) problem between the heater body and the fluid flow within the test section. In this mode, the full Navier-Stokes equations are solved for the fluid, and a heat conduction equation is solved within the solid. The key advantage of solving the full CHT problem in CFD++ is that complex geometries can be examined within a single computational domain.

A. Initial Verification

With Aerojet's Carbothermal rig as the basis for the HHFF design, the first design step was to qualitatively reproduce Aerojet's experimental results. Using the full CHT mode of CFD++, experiments conducted by Aerojet were modeled in a representative Carbothermal rig, modifying the fluid step flow into the test channel, using propane fuel properties, small copper channels, and under high heat flux. These results gave numerically predicted heat transfer coefficients within 10% of Aerojet's measured values, increasing the confidence in CFD++'s prediction capabilities. The next step was to modify the initial test conditions from the previous runs to achieve greater heat transfer coefficients. These modifications showed significant separation bubbles and recirculation of fluid near the hot wall of the exit plenum, as seen in the test section velocity contours in Figure 1. Realizing that the representative models used in these simulations did not accommodate for the high channel velocities required to sustain the higher heat fluxes desired in the High Heat Flux Facility, the test section was redesigned to incorporate a straight flow path with inlet and outlet venturis, therefore avoiding separation for high fluid velocities. The CFD++ results show no separation with the straightened outlet venturi, Figure 2.

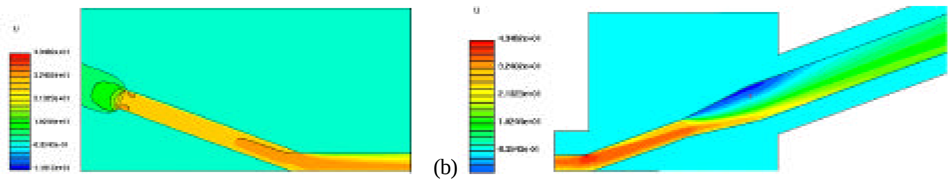
(a) (b)

Figure 1. Velocity (m/s) profiles showing separation in a representation of an Aerojet (a) inlet and (b) exit

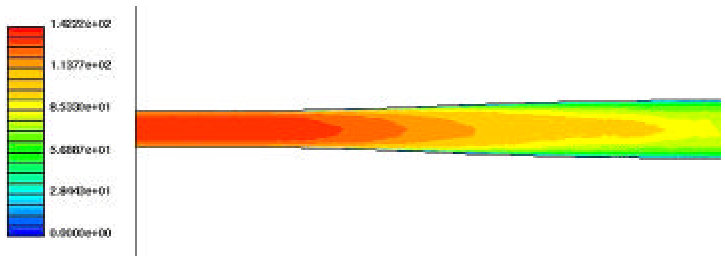

Figure 2. CFD results showing no separation in straightened outlet venturi

B. Heater Block Design

Once initial test channel modifications were complete and CFD++ was verified as capable of modeling conditions near that of the actual test conditions, the focus next turned to the heater body design. While CFD++ was chosen for its ability to simultaneously solve the fluid and heat conduction problems, the computational suite is also capable of solving each problem independently. For the heater block design, the heat conduction through the solid was solved with the fluid effect modeled as a convective heat transfer rate boundary condition. Modeling the heater block in this fashion allowed for multiple design concept iterations with minimal cost and rapid feedback.

Using this method, the heat conduction problem was solved, using CFD++, for three different heater body configurations. These configurations only differed in the shape of the thermal mass mounted on the test section and the number of heaters in the heater blocks, as shown in Figure 3. Solids 1 and 2 are geometrically focused at the bottom to allow for focusing of the thermal wave directly onto the test section. Only Solid 3 incorporates an embedded test section, similar to the original Aerojet Carbothermal rig, which is meant to allow for rapid replenishment of the heater's thermal reservoir region nearest the test section. Solid 1 was configured with a square cross section, while Solids 2 and 3 have circular cross sections. All three configurations have heater lengths of 10 inches and a sufficient thermal diffusion region just below the cartridge heater section to allow for smooth thermal wave propagation to the test section.

Solid 1 Solid 2 Solid 3

Figure 3. Solid model variations of the heater block

Each heater block configuration was created using a tetrahedral mesh, consisting of approximately 4.4, 3.9 and 4.9 million cells respectively. All three heater block configurations were run under the same initial conditions. An initial temperature of 300K (81°F) and a convective heat transfer coefficient of 395 W/m^2K were used. The wetted walls of the inlet and outlet pipes, converging and diverging sections, and test section used a convective heat transfer coefficient of 400 W/m^2K and a T_8 of 160 K (-172 °F) as the boundary wall conditions. Heater cartridge walls were simulated as isothermal walls with a set point of 1400 K (2060°F) and all exposed surfaces were set as adiabatic walls.

After the heat conduction simulations were completed, temperature distribution results from the test section inner core were analyzed. Figures 4, 5 and 6 show the temperature distributions of Solids 1 through 3, respectively. In Figure 4, Solid 1 can be seen to provide a more uniform temperature at the test channel upper wall while Solid 3, Figure 6, provides the greatest heat transfer rate to the fluid.

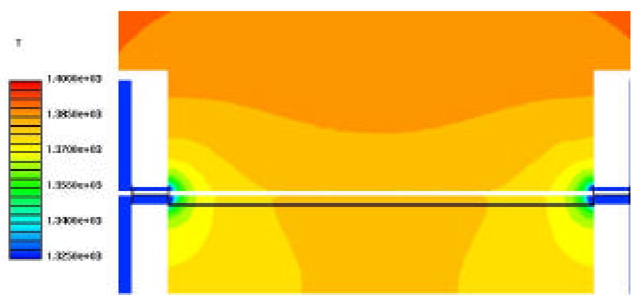

Figure 4. Solid 1 - Steady state temperature (K) profile of coupon section on XZ cut plane (y=0m)

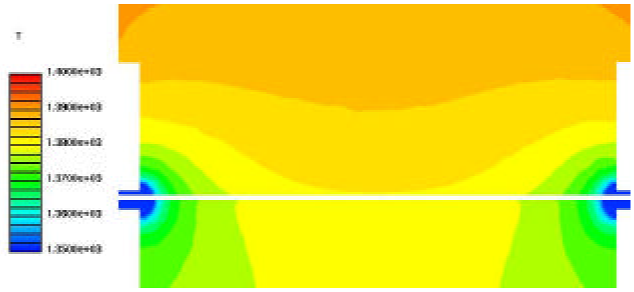

Figure 5. Solid 2 - Steady state temperature (K) profile at coupon section on XZ cut plane (y=0m)

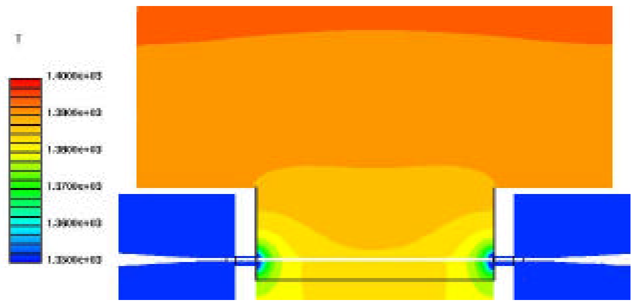

Figure 6. Solid 3 - Steady state temperature (K) profile of coupon section on XZ cut plane (y=0m)

For each heater body design, the fluid flow through the test section was also modeled to determine if these changes in body shape had an effect on fluid flow and if there were any fluid separation issues. This was done by modeling the fluid/wall interaction via a simple heat transfer boundary condition. The 3-D geometric model used was composed of an inlet pipe, a converging section, a square test section with cross sectional area, a diverging section and an outlet pipe. The model was constructed using a hexahedral mesh consisting of 176,128 cells. The simulations were based on liquid propane physical properties with boundary conditions including an inlet mass flow rate of 0.02 kg/s (0.044 lb_m/s) and a set temperature of 160 K (-172 °F). Outlet back pressure was used for simulated flow control, convective heat transfer wall temperatures set to 1400 K (2060°F) and a heat transfer coefficient of 400 W/m^2K set as the test section upper surface. The steady state results obtained, showed no flow separations, and further validated the heater body design improvements.

Based on the heater body solid heat conduction model results, as well as further modeling with higher heat transfer coefficients, a hybrid geometry of Solids 1 and 3 (as shown in Figure 7) was selected for the final heater body design. This design incorporated the strengths of geometric focusing from Solid 1, which improved the upper test section wall temperature uniformity by allowing faster thermal replenishment, as well as embedding of the test section within the heater block, from Solid 3. This increases the amount of thermal mass in close proximity to the test section. Both heat flux and temperature uniformity are improved as a result of this design. The solid copper length of the heater block, directly below the cartridge heaters, was also reduced as it was determined from previous design iterations that the section was longer than necessary to achieve smoothing of the thermal wave from the cartridge heater section. A full conjugate heat transfer analysis was performed to verify that the new heater body design resulted in a uniform heat flux to the test section and no fluid separation occurred.

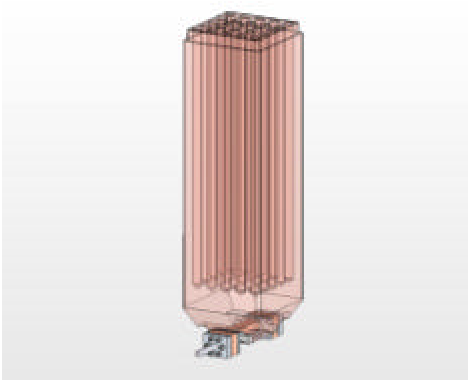

Figure 7. Final Heater Block Geometry

C. Test Section and Cradle Design

After completion of the heater body design, the focus moved next to the test section and cradle design. The test section was designed to allow for flexibility of channel geometry as well as varying test section dimensions, including lengths from two to six inches, channel widths from 0.020 to 0.090 inches and channel heights from 0.020 to 0.200 inches. There is also potential in the future for multi-channel test sections with varying land widths. Figure 8 shows 3-D views of the final test section assembly including an exploded view showing the test section components independently. The test assembly includes the test section, insulation to act as a thermal choke and a venturi section which allows for transition from the circular tubing to the square test channel.

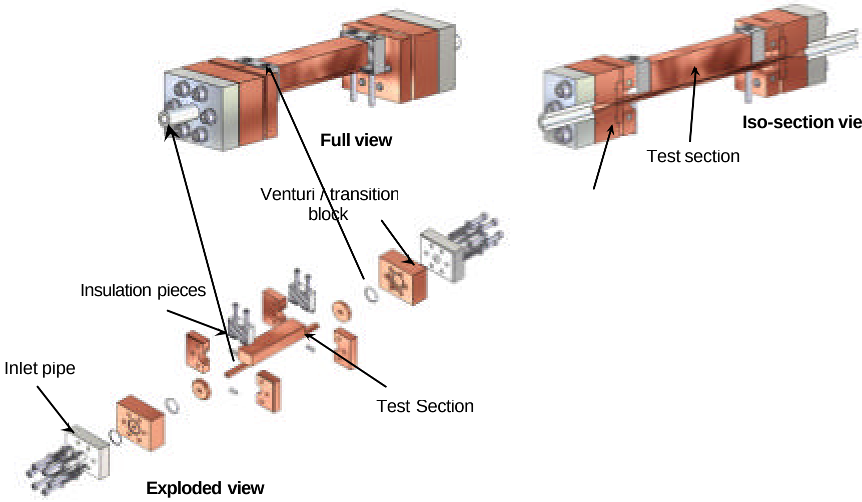

Figure 8. Test Section Assembly Design (Full view, iso-section view, and exploded view)

In order to accurately measure the test section and heater block temperatures, a highly instrumented test section cradle was designed (Figure 9). The cradle design incorporates two sections, an inner section to reduce thermal losses due to contact with the test section, and an outer section designed to support the upper cradle and test section weight, along with any extra force applied by the heater block. The inner section is constructed out of

Marinite A, a machinable ceramic insulation, which is capable of handling high temperatures without out-gassing or loss of structural integrity. The outer section is composed of aluminum, ensuring structural rigidity as well as ease of manufacturing. The cradle sides mimic the thermal reservoir shape to allow proper measurement of the thermal wave which is passing down to the test section. Once the design was complete, it was necessary to verify that heat loss to the test cradle would be minimal. Results of this CFD++ simulation are seen in Figure10.

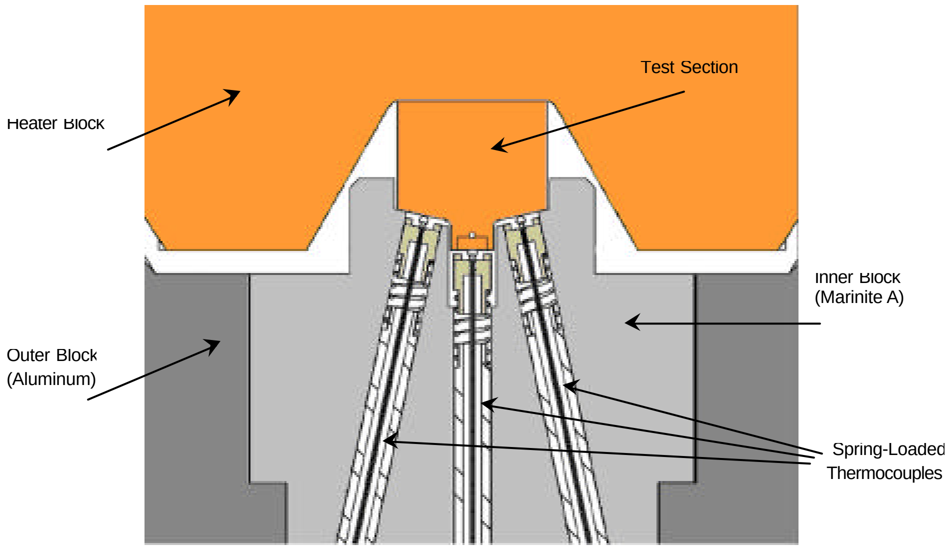

Figure 9. Test Section Cradle with Inner (Marinite A) and Outer (Aluminum) blocks

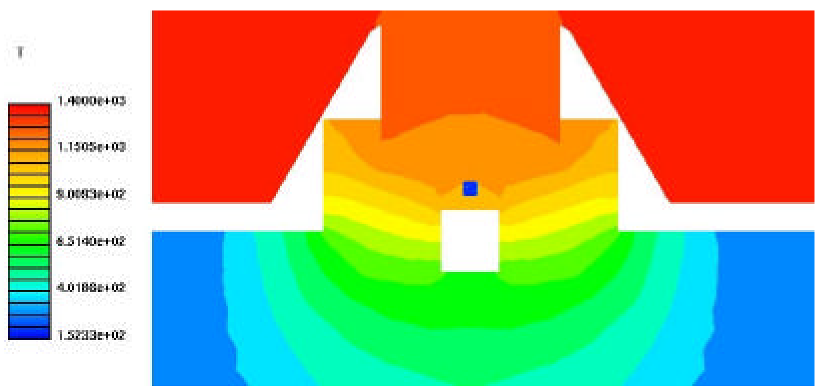

Figure 10 Temperature Profile of Cross-Sectional View of Test Section on Thermocouple Cradle

III. FACILITY DETAILS

After determining that the test section and heater block design will achieve a uniform, high heat flux, the supporting facility design was undertaken. The overall goal of the HHFF is to realistically simulate the conditions fuels encounter while cooling rocket engine passages, therefore a maximum wall temperature of 1200 °F and maximum test section pressure of 4500 psi are desired. These conditions are significantly higher than what other facilities are currently achieving and were selected from a compromise between component cost and availability, and the desire to keep experimental flexibility for a range of fuels.

With the desire to have such high wall temperatures, there was a need to transfer as much of the heat from the heater block directly into the test section as possible. To accomplish this, the test section assembly and copper heating block are fully enclosed in an altitude chamber, both to reduce convective losses and subsequently to allow for more quantitative heat transfer calculations. The chamber assembly is shown in Figure 11.

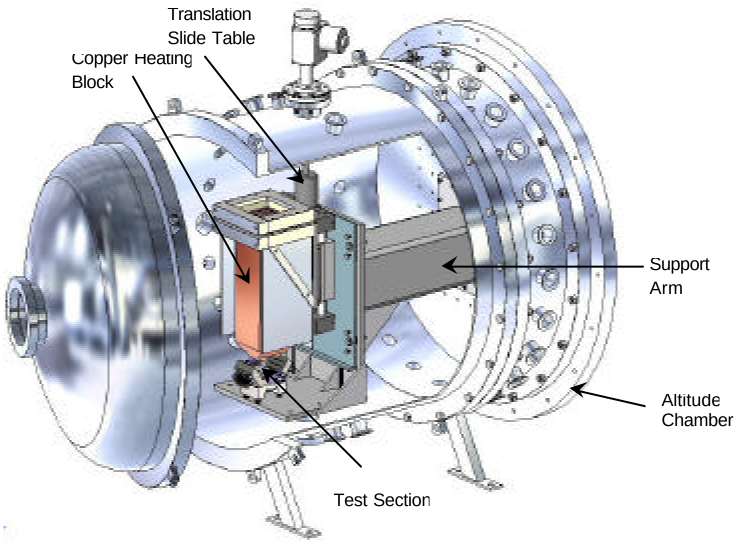

Figure 11. Altitude Chamber Configuration

The heater block, containing 25 cartridge heaters for a two inch test section, is mounted on a linear slide table and surrounded by mirror-polished aluminum plates. During a test, it will be heated and then lowered directly onto the test section using the linear slide table. This will allow both the fluid flow within the fuel channel and the copper heater block to reach steady state before contact occurs. With the heater block mobile, it also allows for the block and the test section to separate once the test is complete, eliminating residual heat transfer from the block to the test section during shutdown. In this manner, transient behavior during startup and shutdown of a given test can be controlled.

Both the heater block and test section temperatures are measured using spring-loaded thermocouples located throughout various slots in the test section cradle sides and bottom (Figure 12). This placement was chosen in order to obtain the most accurate temperature measurements of the test channel and heater block without disrupting the fluid flow. Thermocouple placement within the channel may disturb the flow and cause unpredictable fluid behavior during the tests as well as possibly affecting the corrosion and deposition of the system.

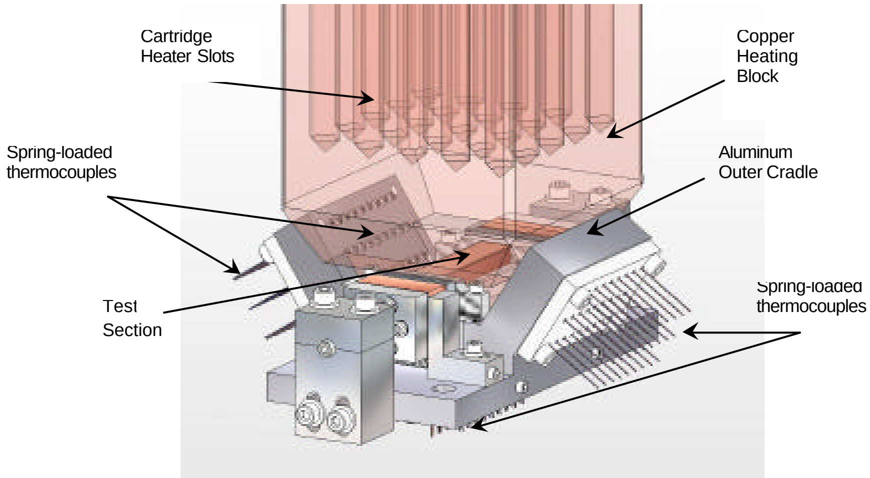

Figure 12. Front View of the Test Section Cradle with thermocouples locations

Due to pressure losses throughout the system and the desire to reach a test section fuel pressure of 4500psi, it is necessary to have system inlet pressures set at 6000 psi. Fuel pressurization is accomplished using bladder tanks in order to eliminate diffusion of the gaseous tank pressurant, into the fuel.

System flow is controlled a Coriolis flow meter, and measured by both a cavitating venturi, and an electronic flow control valve. A Coriolis flow meter was chosen for its accuracy to 0.10% of total flow and its ability to measure both mass flow rate and fuel density. The flow meter is the first instrument the fuel passes through after it exits the bladder tanks. The cavitating venturi, located just prior to the test section, eliminates any potential for feed system coupled oscillations that could affect flow through the test section. The final system flow control component, located just past the altitude chamber, is an electronic control valve that has precise fractional flow control for regulating system backpressure.

After the fuel flows through the Coriolis flow meter, it enters a heat exchanger which controls the inlet bulk temperature into the test section and allows for greater flexibility in producing a varied thermal history for each fuel. This heat exchanger, or “preheater”, consists of dual, coiled tubes cast in aluminum with six, 4 kW (3.8 Btu/s), heating elements. This unique design was chosen for its flexibility in using either the coiled tube heat exchanger or the cast-in heater elements, giving a total controlled temperature range of –4 to 500 °F. The coiled tube design allows for the preheater to function in either co-flow or counter-flow and incorporates a glycol recirculating system to provide continuous heating or cooling in the low to mid-temperature range from –4 °F to 176 °F. For higher operating temperatures, up to 500 °F, the heater elements are configured into three zones for multi-level heating and to prevent hot spots from causing fuel coking and deposition inside the cast-in heat exchanger. This system is critical for maintaining constant fuel inlet temperature without being affected by the temperature changes of the surroundings.

Located downstream of the test section and outside the altitude chamber are several data collection components. One is a filter for collection of free floating particles discharged from the test section. A sample vessel is also located downstream of the chamber and will be used to remotely collect a post-test fuel sample during experimentation. Samples collected from both of these will be used for additional studies of fuel degradation under extreme conditions.

Pressure transducers and thermocouples are located throughout the system, in the fluid flow prior to entering the altitude chamber, as well as inside the chamber. These pressure and temperature measurements will provide feedback regarding the fluid behavior within the test channel and the overall functionality of the system.

To provide control over the dissolved oxygen content within the fuel, a sparging system was incorporated. The fuel will be placed in a sparging tank fed with low pressure (>100 psi) gaseous Nitrogen, a stir bar with magnetic stirrer, and three oxygen sensors. Dissolved oxygen is removed from the fuel by slowly bubbling gaseous nitrogen up through the fuel, while it is gently agitated by the stir bar to prevent potential non-uniformities. The oxygen sensors are located at various distances along the height of the tank and are capable of detecting dissolved oxygen as low as six parts per billion. By incorporating the sparging system into the facility, it allows for a thorough study of dissolved oxygen and the ability for fuel additives to reduce thermal-oxidative decomposition.

In the HHFF, a programmable logic controller (PLC) is used to remotely operate and control the facility. The five areas the PLC controls are remote valve operation, maintaining fuel flow, fuel heating and cooling processes, and initialization of data recording. This is achieved by utilizing digital and analog interface modules with advanced logic instructions for intelligent facility control. Along with the PLC, are a data acquisition system and two heater control systems which all operate simultaneously. The PI 6000 data acquisition system, manufactured by Pacific Instruments, is a mainframe system with integrated signal conditioning providing 16-bit measurement resolution. A total of 80 analog input channels are set up for facility pressure and temperature measurements. A digital interface module with 16 input and 16 output channels has also been added to the system for communication with the PLC and other digital interface devices. A 32-zone temperature controller panel, custom-built by Watlow, is responsible for control of the heater cartridges, and includes a built-in wattmeter to measure the total power consumption. Global alarm outputs from this controller are directly connected to the PLC for safe operation.

A variety of software applications were evaluated for facility hardware integration; RSView32, a human-machine interface (HMI) type application developed by Rockwell Automation, was selected. PI 660, the data acquisition software developed by Pacific Instruments for use with their mainframe data system, was the main reason for selecting the previously mentioned PI 6000 data acquisition system. Software integration between the two applications is accomplished by sharing real-time data collected from the PI 6000 with RSView32 for display and PLC process control. An add-on health monitoring program was integrated into the PI 660 software to monitor critical parameters for out-of-limit conditions. Both data and control systems are designed with future expansion in mind, giving the facility flexibility for future modifications. Anawin, a software package developed by Watlow to accompany the temperature controller, supports controller functions such as monitoring thermal distribution and maintaining high heat flux conditions using either built-in PID control functions or by predetermined ramp and soak profiles. An overview of the entire data acquisition and control system architecture is shown in Figure 13.

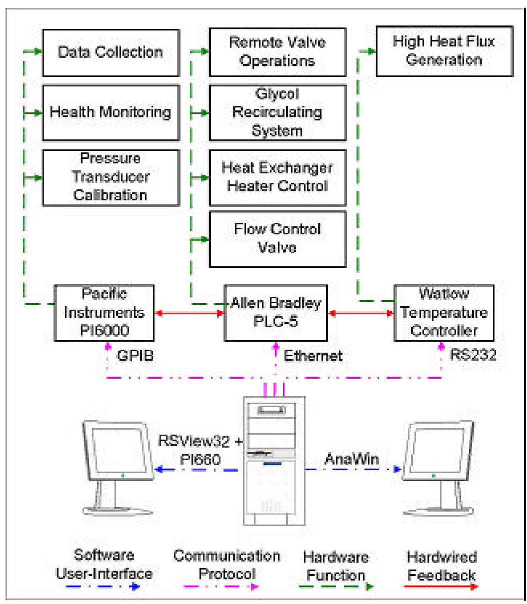

Figure 13. Data Acquisition and Experiment Control System

IV. TEST AND ANALYSIS

Using RSView, a typical test begins with the test section installed in the altitude chamber and under vacuum. The heater block is located a few inches above the test section, out of thermal contact. Preset cartridge heater temperatures are initiated and the entire heater block is allowed to reach steady state. Fuel flow is then started, utilizing the bypass line, to reduce fuel effects on the test channels walls before steady state is reached, and the pre-heater is activated. When steady state fuel inlet conditions are reached, the fuel is switched to flow through the test section and the heater block is lowered onto the test section by means of the linear slide table. Once the heater block makes thermal contact with the test section, time zero for the test begins. This minimizes the initial transients and forces the fuel to never see wall temperatures higher than during its planned test time, which is critical to quantitative post-test analysis. Parameters, such as cartridge heater temperature, electrical power input, mass flow rate, and inlet pressure and temperature, are monitored throughout the test. Test section temperatures and pressure drops are observed and used as early indicators of coking and/or flow blockage. When the test duration, from ten minutes to one hour, is met, the linear slide table lifts the heater block off the test section, breaking thermal contact, and allowing the test section to cool. Fuel continues to flow, allowing for faster test section cooling and preventing thermal soak back and coking of stagnant fluid on the test section walls. When the test section temperature falls to an acceptable level, the fuel flow is stopped and the test section is purged with nitrogen. Once the test section and heater block have cooled, the test section is removed for further analysis.

For internal wetted surface analysis, each test section will be cut apart and sectioned to expose the fluid channel. Each individual section will then be examined by three different methods. Sections will first be examined by scanning electron microscope (SEM) to determine morphology of deposition, chemical attack of the channel walls, and diffusion of species into the substrate. A comparison will be made between each experimented test channel and a standard, in this case a virgin test section, seen in Figure 14. The virgin test section shows some surface roughness, which will allow for studies of these affects on coking and corrosion on the channel walls. Overall, the SEM will allow for a visual characteristic comparison of the deposits formed on the test section walls to the virgin test channel.

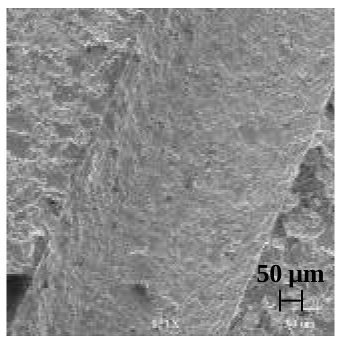

Figure 14. SEM of virgin test section

Also, during SEM analysis, Energy Dispersive X-Ray scattering (EDX) will be used to determine composition of deposits, examine grain boundaries, and examine exposed metal surfaces. Lastly, carbon deposition rates will be determined by LECO analysis of approximately ¼-inch test channel sections. In addition to enabling understanding of fuel coking behavior, these analyses will also demonstrate various compound interactions, including sulfur-containing compounds, with the test channel wetted material. This will help quantify fluid channel corrosion by sulfur species. The results will also aid in determining and comparing thermal stability of both current and newly created fuels.

A post-test analysis of all measurements obtained during experimentation, including pressures, temperatures, mass flow rates, and electrical energy supplied, enables quantitative calculation of the experimental conditions. Determination of overall heat transfer coefficient, pressure drop along the test channel, and axial temperature profiles throughout the length of the section, will help establish wall and core bulk temperatures for each test. These parameters are of value in determining fuel cooling efficiency. Comparisons will be made with computational CFD++ results of each experiment, paying particular attention to the very thin boundary layers encountered during high heat flux cooling.

Along with varying the flow rate and fuel type, which will initially include various grades of RP-1 and in-house alternate hydrocarbon fuels, the HHFF is capable of testing various test section materials. Some of the materials scheduled to be tested in this system are Haynes 188 and 214, GRCop84, NarloyZ, and OFHC copper. By testing these materials, the chemical interaction between the fuels and the channel material will be identified, along with the ability for the test section material to handle the facility's high temperatures and pressures.

V. SUMMARY AND CONCLUSIONS

The design of a unique thermal stability facility was undertaken to facilitate the development and transition of advanced hydrocarbon fuels. The High Heat Flux Facility (HHFF) is capable of achieving 100 BTU/in ^{2}/sec and operates up to 4500 psi, closely simulating the actual conditions found in the cooling passages of a modern rocket engine. Using experience gained from past and present thermal stability test rigs in conjunction with CFD++, a Metacomp Technologies Inc. computational fluid dynamics software suite, conjugate heat transfer calculations were performed in a single computational domain containing the copper heater block and the test section fluid passages to optimize the design. A facility description and experimental parameters of interest (the heat transfer coefficient, the degree of coking and corrosion in the channel, and the pressure drop as functions of heat flux, wall temperature, Reynolds number, channel material, fuel composition and pressure) were given. Asymmetric heating, easy instrumentation access, and flexibility of the channel geometry are all advantages of this facility.

FUTURE WORK

The HHFF will be used in the future to characterize the thermal performance, coking and corrosion behavior of RP-1, new grades of RP-1, new advanced synthetic hydrocarbon rocket propellants, sub-cooled hydrocarbons propellants, and additives for enhanced thermal stability, storability, and performance. The facility may also be used to study the effects of channel geometry, surface roughness, coatings, channel materials, and heat-transfer aids such as riblets, dimples, and bumps.

ACKNOWLEDGMENTS

The authors would like to thank Mr. Philip Rice, Ms. Claudia Heflin, and Mr. Wesley Mosier of Applied Engineering Services, Inc. (AES), Mr. R. Mike Griggs, Mr. David Hill, Mr. Earl Thomas, Mr. Randy Harvey, and Mr. Toya Johnson of Engineering Research and Consulting, Inc. (ERC), Mr. Mark Wilson of ERC, Inc. (ERC/ROSS), Mr. Mark Pilgram, Mr. Matt Kracke and Mr. Michael High of Sverdrup/ROSS for their support throughout the course of this project. The authors also wish to thank and acknowledge the technical discussions and advice of Major Abdollah Nejad (AFRL/PRS) and Mr. Jay Levine (AFRL-ret).

REFERENCES

1. Bates, R. W., Edwards, J. T., and Meyer, M. L., ***Heat Transfer and Deposition Behavior of Hydrocarbon Rocket Fuels***, Aerospace Sciences Conference AIAA 2003-0123, Reno, NV (Jan 2003)

2. Green, J. M., Pease, G. M., and Meyer, M. L., ***A Heated Tube Facility for Rocket Coolant Channel Research***, 31st Joint Propulsion Conference and Exhibit AIAA-95-2936, San Diego, CA (Jul 1995)

3. Meyer, M. L., ***Electrically Heated Tube Investigation of Cooling Channel Geometry Effects***, 31st Joint Propulsion Conference and Exhibit AIAA-95-2500, San Diego, CA (Jul 1995)

4. Stiegemeier, B., Meyer, M. L., Taghavi, R., ***A Thermal Stability and Heat Transfer Investigation of Five Hydrocarbon Fuels: JP-7, JP-8, JP-8+100, JP-10, and RP-1***, Joint Propulsion Conference and Exhibit AIAA 2002-3873, Indianapolis, IN (Jul 2002)

5. Giovanetti, A., Spadaccini, L., Szetela, E., ***Deposit Formation and Heat-Transfer Characteristics of Hydrocarbon Rocket Fuels***, Journal of Spacecraft and Rockets, Vol. 22, No. 5, 1985

6. Marteney, P., Colket, M., Vranos, A., NASA CR-168027, ***Experimental Study of the Thermal Stability of Hydrocarbon Fuels***, Contractor Report, East Hartford, CT (Feb 1983)

7. Hines, W. S., ***Heat Transfer to RP-1 Kerosene in Turbulent Flow Under Asymmetric Heating Conditions***, Chemical Engineering Progress Symposium Series, Heat Transfer, pp 193-200

8. Cook, R. T., and Kirby, F. M., NASA CP-2437***, Advanced Earth-to-Orbit Propulsion Technology 1986***, Vol. 2, pp 461-473, 1986.

9. Gage, M. L., Rosenberg, S. D., NASA CR-185203, ***Hydrocarbon-Fuel/Combustion-Chamber-Liner Materials Compatibility***, Interim Final Report, Cleveland, OH (Apr 1990).

10. Rosenberg, S. D., Gage, M. L., Homer, G. D., and Franklin, J. E., ***Hydrocarbon-Fuel / Copper Combustion Chamber Liner Compatibility, Corrosion Prevention, and Refurbishment***, Journal of Propulsion and Power, Vol. 8, No. 6 (Nov-Dec 1992)

11. Striebich, R. C., Lawrence, J., ***Thermal Decomposition of High Energy Density Materials at High Pressure and Temperature***, Journal of Analytical & Applied Pyrolysis, 70, 339, 2003.

12. Wohlwend, K., Maurice, L. Q., Edwards, T., Striebich, R. C., Vangsness, M., Hill, A. S., ***Thermal Stability of Energetic Hydrocarbon Fuels for Use in Combined Cycle Engines***, Journal of Propulsion and Power, Vol. 17, No. 6, (Nov-Dec 2001)