Turbulent combustion studies in a model turbine burner

Srivatsava V. Puranam[1], **Nicola Sarzi-Amade'**[1]
Derek Dunn-Rankin[1]
[1]Department of Mechanical and Aerospace Engineering,
University of California Irvine, Irvine, CA 92697 U.S.A.

May 28th 2007

1 Motivation

Thermal cycle analyses[1, 2] of gas turbine engine cycles have shown that energy addition in the turbine stages can considerably increase the specific power(thrust) and power(thrust)-to-volume ratio of a gas turbine engine, while decreasing specific fuel consumption. Combustion in a system such as in a turbine vane where it is possible to achieve energy addition simultaneously with work extraction, can keep the temperature below the material limits.

One way of achieving combustion in turbines would be to have one or more combustion chambers located in the turbine system. These combustion chambers can be located in the transition duct between high pressure and low pressure stages of the turbine, in the turbine stator stage or in the turbine vanes. This type of a combustion system has been called a "turbine burner" [1, 2] or inter turbine burner[3] in the literature. The flow in the system is multi-dimensional with mixing and chemical reactions. Literature and research with this type of flow at high pressures and transonic flow is relatively scarce [1, 4, 5].

Related work done on reacting laminar mixing layer flows suggests that, for flows with acceleration, the peak temperature decreases with downstream distance [1, 4, 5]. This leads to the conclusion that NO_x formation would be less than that which occurs in a flow without acceleration. The reduction in peak temperatures due to acceleration results in the promise of reduced heat transfer losses in many other combustion applications. A mixing and exothermic chemical reaction in the accelerating flow through the turbine passage offers, therefore, an opportunity for a major technological improvement. This work in progress examines various aspects of non-reacting flow in a turbine burner.

2 Simulations

As a precursor to the experimental study, 2-D simulations were done on the proposed setup using a CFD software "ESI-CFD" . The simulations were done on two different configurations, where the cavity is placed on two different sides of the curving duct. The results of this simulations are promising. The flow in the main duct is fairly undisturbed by the cavity. The flow in the cavity has a maximum speed which is 10% of the flow in the main channel just upstream of the cavity. The recirculation zone has good mixing properties, i.e., the eddy formed is very stable and encompasses most of the cavity.

Correspondence to : vpuranam@uci.edu

3 Experiments

The experimental study of the turbine burner system is being done in two steps. The first step is to study the flow in a turbine burner system without combustion and the second step is to study the system under reacting conditions. In order to achieve similar conditions to practical devices, the experiments will operate at pressures up to 5 atmospheres. At the maximum mass flow rate possible with the air-supply system, the inflow velocity and therefore the g-factor increases as pressure decreases and as inflow temperature increases. While any practical device will operate at much higher pressure, at this stage, there are sufficient fundamental questions regarding mixing, combustion, and flame-holding that are nominally pressure-independent, which can be investigated at low pressure.

The first set of experiments will be used to get an insight into the flow behaviour inside the test section, the effect of main flow on the flow in cavity, and fuel droplet evaporation and mixing. For the cold flow experiments the test-section used is made of a transparent plastic, lexan. Lexan is transparent in the visible region and is useful because it gives complete optical access to the test section.

The second set of experiments will be done for the reacting flow case. The test section for this case is made of steel. The test section has 14 quartz windows for optical access, throughout the test section. Turbine materials cannot accept the full thermal load of stoichiometric combustion. Heat addition in the turbine will therefore need to be fuel lean (at least overall).

3.1 Current Experimental Setup

The first experimental rig involves cold flows (reacting flows will be considered in a metal chamber of equivalent design). In the current experimental design, the apparatus consists of a large PVC tube (Fig.2) for air delivery connected to a completely transparent channel made of lexan. This material has greater resistance compared to other kinds of plastics, it can handle high pressure more easily, and is more durable when in contact with liquid fuels. The flow is provided by our high-capacity house air-compressors, and it passes through a series of pipes that convey the air through a flow straightener and screen section in the PVC tube and ultimately inside the channel. A butterfly valve in the line allows variation of the airflow, and is easily controllable through a variable switch. After the valve, at a distance large enough for allowing homogeneity of the flow field (about 1.5 meters), a Nice Instrumentation FVP vortex flow meter is installed to measure the main airflow rate (volumetric). The value of the flow rate is read on the digital display of the meter. A pressure gage has been added in the line to measure the pressure in three locations: before the flow meter, right after it, and at the exit of the big PVC tube (i.e., entrance of the channel). Through the measured pressure values, the value of the air density across the flow meter can be calculated and included in a calibration that provides the mass flow rate at any time.

After the air leaves the flow meter, it is delivered to the PVC tube by a pressure-resistant rubber hose . At the entrance of the PVC tube, an air filter has been added to prevent coarse dirt particles in the air from entering the tube. Inside the tube, two circular screens at a distance of one foot (30.48) from each other have been put to filter the air further and straighten it. After the two screens, inside the tube a honeycomb has been inserted to complete the straightening of the air. Several small 18-inch-long (46 cm) PVC tubes constitute the honeycomb. They are firmly glued together and screwed to the big tube from outside to prevent movement. After the air leaves the honeycomb, it enters the lexan channel connected on the other side. The channel has curved contracting walls (25 cm radius of curvature) producing a rectangular cross section that contracts from 10 x 5 cm at the inlet to 10 x 1 cm at the outlet over a 70 degree arc and a 30 cm length along its centerline.

A removable small auxiliary cavity can be attached to the curved section (both on top or bottom, interchangeably) to provide a recirculating low speed zone, in which liquid spray can be injected via small simplex-atomizers. The atomizers are designed to produce spray droplets with mean diameter less than 100 microns. The cavity is as wide as the channel (10 cm) and its other dimensions are 5.5 cm in both directions. It is located at approximately one third of the channel length. There are three possible

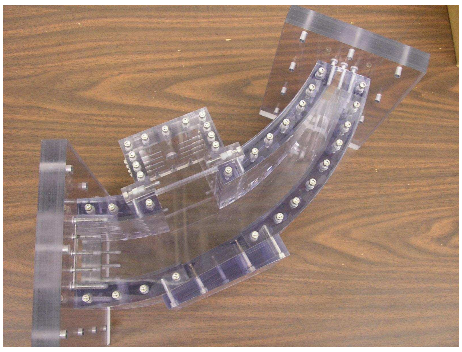

Figure 1: Turbine burner test section

injection locations in the cavity, one on the top, and two on the sides that are perpendicular to the main stream direction. Any of these locations may also be used for additional air injection, to help push the mixture out of the cavity and enter the main stream. Two additional injection points are present in the channel itself, before the cavity, one on the top and one on the bottom respectively. A theoretical 1-D model has been implemented to predict the choking conditions for the flow at different airflows and pressures, and the model can predict behavior of the hot flow as well as of non-reacting air flow. Air preheating plus heat addition provided by combustion are incorporated into the model. The results of this calculation have guided the design of different choked nozzles that will be implemented for high pressure tests by connection to the flange at the exit of the channel.

4 Cold flow experiments

4.1 Velocity calibration and boundary layer estimation

Exit and entry velocity profiles were measured for the test chamber using a pitot tube. Eleven points were surveyed along the exit and entrance sections of the chamber. Test airflow inside the channel was 0.17 kg/s. Figures 2 and 3 show that, the mean flow is very uniform at both the entrance and exit of the channel. At this mass flow rate, the average entrance velocity is 26.4 m/s, and the average exit velocity is 86.1 m/s. The velocity at the exit is underestimated since the accuracy of our manometer decreases at these high velocities. The results indicate that the flow can be approximated as being 2D inside the channel (we suppose that it is 2D everywhere inside the channel, since it is at the entrance and exit). The other important result from this preliminary test is that the thickness of the boundary layer at the exit section is small enough to prevent significant choking of the flow. Even though the diameter of the Pitot tube we used (2 mm external and 0.75 mm internal) is not small enough to give accurate boundary layers profiles, we estimate that the thickness of the boundary layer at the exit is less than 0.5 mm.

4.2 Flow visualization

4.2.1 Smoke test

To visualize the flow inside the channel, the flow was seeded with smoke generated from dry ice. The smoke exits four rubber hoses attached to four injection points, in the chamber and cavity, at a speed of about 2 m/s. This speed is small enough to not perturb the actual flow in the chamber or the cavity. The air mass flow rate was about 0.04 Kg/s, with an entrance velocity of about 7 m/s. A laser sheet

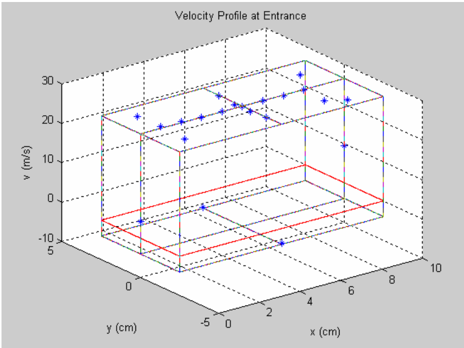

Figure 2: Velocity profile at entrance

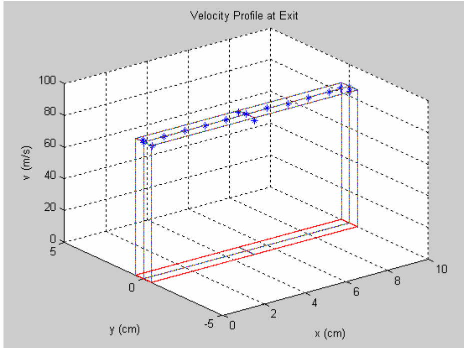

Figure 3: Velocity profile at entrance

from a frequency doubled Nd:Yag laser was used to illuminate the flow and the pictures were recorded by a CCD camera synchronized with the laser firing rate.

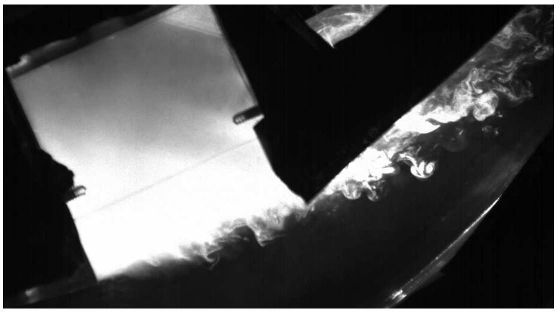

Figure 4: Smoke test with laser illumination

Figure 5: Smoke test without laser illumination

Flow images with and without laser illumination are shown in figures 4.2.1 and 4.2.1. Figures 4.2.1 and 4.2.1 show that flow is being drawn from the main flow into the cavity and that the cavity is trapping a vortex. The image acquisition speed is known and the size of the cavity is known; hence the average residence time of the vortex was estimated. The speed inside the cavity is estimated by noticing that the flow covers half the perimeter of the cavity between two consecutive images.

$$\tau_{\mathrm{res}} = \frac{\text{circumference of cavity}}{\text{speed in the cavity}} \approx 0.4s \tag{1}$$

The residence time inside the main flow is calculated as:

$$\tau_{\mathrm{res}} \approx \frac{\text{Length of channel}}{\text{average speed}} \approx 0.012s \tag{2}$$

Equations 1 and 2 suggest that the flow in the main channel is about 33 times faster than is the cavity flow. The flow in the channel depends on the shear layer flow caused by the main flow boundary layer, when it encounters a sudden step in the flow due to the channel. As the flow Reynolds number increases, the part of the main flow going into the cavity is reduced due to thinning of the shear layer. Hence, at high main flow speeds the cavity flow is expected to be two orders lower than the main flow. This prediction is in agreement with simulation results.

4.2.2 Streamers test

Another qualitative measurement was done for the non-reacting flow to verify the results obtained in the smoke test and the flow velocity profile calibration. Streamers were attached to the chamber and the cavity at strategic positions. Tests were done for two flow conditions 0.05 Kg/s and 0.123 Kg/s with entrance velocities 9 m/s and 21 m/s.

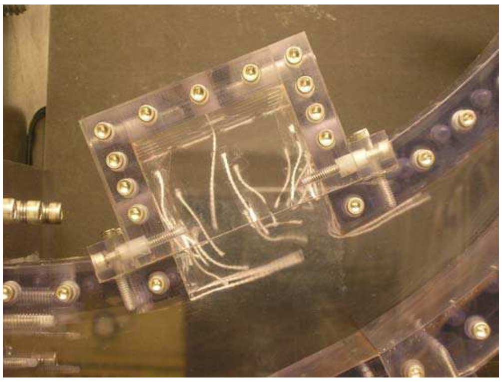

Figure 6: Streamers test for 0.05 Kg/s

For the higher mass flow rate the streamers vibrated more than they did for the low mass flow rate, but the attitude of the streamers remains same. Hence, the test confirms that a vortex is trapped in the cavity and that the recirculation zone in the cavity is a single large eddy.

The streamers at the exit of the section indicate that the boundary layer is relatively thin. This precludes the possibility of flow chocking due to the boundary layer, a conclusion drawn from the velocity calibration for the channel.

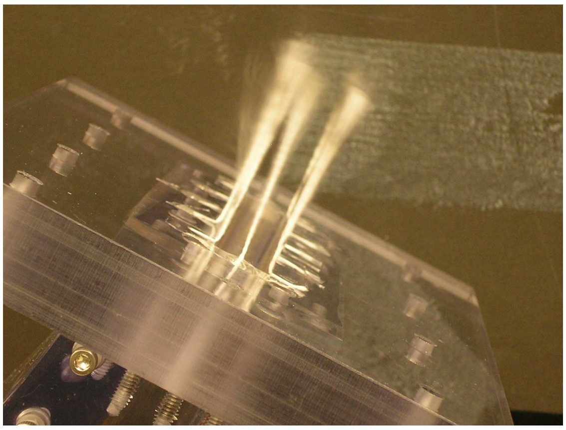

Figure 7: Exit section

5 Progress and proposed work

Experiments for the cold flow case are almost complete. Test for fuel evaporation and for fuel injection measurements are in progress. Flow velocity, temperature and pressure measurements are planned for the future.

References

[1] W.A. Sirignano and F. Liu. Performance increases for gas turbine engines through combustion inside the turbine. *Journal of Propulsion and Power*, 15(1):111–118, Jan - Feb 1999.

[2] F. Liu and W.A. Sirignano. Turbojet and turbofan engine performance increases through turbine burners. *Journal of Propulsion and Power*, 17(3):111–118, May - June 2001.

[3] J. Zelina, J. Ehret, R. Hancock, W. Roquemore, D. Shouse, and G. Sturgess. Ultra compact combustion technology using high swirl for enhanced burning rate. *38th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit*, 2002.

[4] X. Fang, F. Liu, and W.A. Sirignano. Ignition and flame studies for an accelerating transonic mixing layer. *Journal of Propulsion and Power*, 17(5):1058–1066, Sep - Oct 2001.

[5] C. Mehring and W.A. Sirignano F. Liu. Ignition and flame studies for a turbulent accelerating transonic mixing layer. Technical Report 2001-0190, 39th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, January 2001.