S26-20
173487

1996
NASA/ASE SUMMER FACULTY FELLOWSHIP PROGRAM

MARSHALL SPACE FLIGHT CENTER
THE UNIVERSITY OF ALABAMA

COOLING DUCT ANALYSIS FOR TRANSPIRATION/FILM COOLED LIQUID PROPELLANT ROCKET ENGINES

Prepared by:	Gerald J. Micklow, Ph.D.
Academic Rank:	Assistant Professor
Institution and Department:	The University of Alabama Department of Mechanical Engineering

NASA/MSFC:

Laboratory:	Propulsion Laboratory
Division:	Propulsion Systems
Branch:	Combustion Devices and Propulsion Design

MSFC Colleague:	Charles Cornelius, Ph.D.

Introduction

The development of a low cost space transportation system requires that the propulsion system be reusable, have long life, with good performance and use low cost propellants. Improved performance can be achieved by operating the engine at higher pressure and temperature levels than previous designs. Increasing the chamber pressure and temperature, however, will increase wall heating rates. This necessitates the need for active cooling methods such as film cooling or transpiration cooling. But active cooling can reduce the net thrust of the engine and add considerably to the design complexity. Recently, a metal drawing process has been patented where it is possible to fabricate plates with very small holes with high uniformity with a closely specified porosity. Such a metal plate could be used for an inexpensive transpiration/film cooled liner to meet the demands of advanced reusable rocket engines, if coolant mass flow rates could be controlled to satisfy wall cooling requirements and performance. The present study investigates the possibility pf controlling the coolant mass flow rate through the porous material by simple non-active fluid dynamic means. The coolant will be supplied to the porous material by series of constant geometry slots machined on the exterior of the engine.

Numerical Analysis

First, the flow through the rectangular slot on the exterior of the engine will be discussed. The flow was assumed to be one dimensional and adiabatic in a constant area duct with friction. Starting with a differential control volume with a constant area cross-section, the equation for the conservation of mass is as follows:

$$\rho V = (\rho + d\rho)(V + dV) \tag{1}$$

where ρ is the density and V is the velocity. Neglecting higher order terms gives;

$$V d\rho = -\rho dV \tag{2}$$

From conservation of energy the following equation is obtained;

$$h + \frac{V^2}{2} = h + dh + \frac{(V + dV)^2}{2} \tag{3}$$

where h is the enthalpy. Neglecting higher order terms gives;

$$V dV = -dh \tag{4}$$

With the continuity equation given by eq (2), this expression becomes;

$$V dV = -dh = -V^2 \frac{d\rho}{\rho} \tag{5}$$

Now to investigate the linear momentum equation. From a summation of forces on the control volume the following expression results;

$$pA - \tau_w A_{wet} - (p + dp)A = \dot{m}(V + dV - V)$$

or

$$\tau_w A_{wet} - dpA = \dot{m}dV \tag{6}$$

where τ_w is the wall shear stress, A is the duct cross-section area, p is the pressure and $\dot{m}$ is the mass flow rate. Now the wall shear stress can be expressed as;

$$\tau_w = \frac{f\rho V^2}{8} \tag{7}$$

where f is the Darcy friction factor. Combining eqs (6) and (7) gives the following;

$$-dp = \frac{f\rho V^2 A_{wet}}{8A} + \frac{\dot{m}}{A}dV \tag{8}$$

The wetted duct area can be expressed as;

$$A_{wet} = P_w dx \tag{9}$$

where P_w is the wetted perimeter of the duct. Defining the hydraulic diameter as;

$$D_H = \frac{4A}{P_w} \tag{10}$$

and substituting this expression along with eq (9) into eq (8) gives;

$$-dp = \left(\frac{f\rho V^2}{2D_H}\right)dx + \rho V dV \tag{11}$$

For a rectangular cross-section the hydraulic diameter is given by

$$D_H = \frac{2bH}{b + H} \tag{12}$$

where b is the slot base dimension and H is the slot height. For the circular duct the hydraulic diameter is the duct diameter.

Equations (2), (5), and (11) represent a set of equations which can be used to describe the flow field. Two issues must be addressed however. First, the thermodynamic properties of the working fluid are required. For the current problem, the coolant fluid is liquid hydrogen and a computer program was developed to produce the thermodynamic properties. Second, the problem is non-linear, therefore an iterative solution is required to obtain the flow field where the coefficients are lagged to linearize the problem. A space marching technique will be utilized for the solution.

The solution technique is as follows:

1. The inlet mass flow rate, pressure and temperature, along with the slot dimensions are specified.
2. The inlet thermodynamic properties, velocity, Reynolds number and friction factor are calculated.
3. The change in velocity at the first section is guessed.
4. From the continuity equation, eq (2), the density change is calculated based on the guessed velocity change.
5. The average density for the control volume is calculated.
6. The momentum equation, eq (11), is used to calculate the pressure.
7. The energy equation, eq (4) is used to update the enthalpy.
8. Based on the updated pressure and enthalpy, the thermodynamic property subroutine is used to update the density.
9. The relative change in the pressure is calculated and if below a specified tolerance the thermodynamic properties are updated and the calculation proceeds to the next axial location where the procedure is repeated. If the relative change in the pressure is not below the specified tolerance, steps 3-9 are repeated.

For the cooling flow through the constant diameter circular ducts into the engine gas path, a similar analysis is performed. However, the upstream and downstream boundary conditions are changed. Rather than specify the inlet mass flow rate and calculate the exit pressure, as in the rectangular exterior slot, the inlet pressure is specified from the rectangular slot flow field calculation and the inlet velocity is guessed. The calculation procedure previously discussed is implemented and the exit pressure into the engine gas path is calculated. This pressure must match the pressure at the appropriate location in the engine gas path. If it does not, the inlet velocity is changed and the calculation is repeated. This is continued until the pressure at the circular duct exit matches the pressure in the engine gas path within a specified tolerance. At this point the cooling mass flow rate through the circular duct is calculated.

Results

First, the engine external slot to transport the liquid hydrogen to various engine gas path locations was analyzed. Figure 1 gives the resulting slot pressure distribution for various rectangular slot height/width ratios. It is clearly seen that as the slot width/height ratio or aspect ratio is increased for the same slot width that the pressure drop in the duct increases. This will have a marked effect on the coolant mass flow rate in the engine gas path through the porous wall liner. Figures 2-4 give the circular hole velocity distribution for the porous wall liner for various hole diameters at a mid-combustion chamber location. The inlet boundary condition for the hole is the pressure set by the exterior rectangular duct location. The exit boundary condition is the pressure set by the engine gas path. It is clearly seen that for a specified hole diameter that as the rectangular slot aspect ratio increases, the velocity magnitude in the circular hole decreases. This will have a marked effect on the coolant mass flow rate into the engine gas path. Further, from figures 2-4 it is seen that as the hole diameter is increased for a set exterior slot aspect ratio that the duct velocity increases. Increased duct velocity along with increased flow cross-sectional area, caused significant increases of coolant mass flow rate into the gas path.

Figures 5- 7 give the circular hole velocity distributions as a function of exterior slot aspect ratio for various hole diameters at the nozzle throat location. As before, increasing slot aspect ratio decreased the velocity in the circular hole, while increasing hole diameter increased the hole velocity. Further, the velocity magnitude has increased in comparison to the similar geometry for the combustion chamber location. Similar trends were observed for other downstream gas path locations.

Conclusions and Recommendations

For the exterior rectangular slot, the duct pressure drop increased as the slot aspect increased. As slot aspect ratio is increased, the duct surface area/cross-section area increases causing the effect of wall friction on the flow to increase. This results in a higher pressure drop due to friction. For the circular holes in the porous wall liner providing the coolant passage from the rectangular exterior slot to engine gas path, the exterior slot aspect ratio had a marked effect on the magnitude of the hole velocity distribution. As the rectangular slot aspect ratio is increased the duct pressure drops. Therefore, for a fixed engine gas path location which fixes the circular hole exit pressure, increasing slot aspect ratio will decrease the hole inlet pressure. Decreasing the inlet pressure for a fixed exit pressure will decrease the inlet velocity and therefore the coolant mass flow rate. Increasing the hole diameter increases flow cross sectional area and for a fixed exterior duct aspect ratio increased the circular hole inlet velocity. This results in a substantial increase in the coolant mass flow rate since for fixed inlet density, both the velocity and flow area increased. The reason for the increased duct velocity is due to the decreased wall surface area/flow cross-sectional area which reduces the effects of friction and a larger percentage of the hole pressure drop(which is fixed) can be associated with a velocity increase.

It is seen that the coolant passage geometry, i.e. exterior duct aspect ratio and porous wall circular hole diameter can result in a large variation of the coolant mass flow rate. Therefore, proper selection of the wall coolant circuit geometry could result in an inexpensive and effective wall cooling scheme for a re-usable launch vehicle engine. To obtain the correct coolant passage geometry, a detailed computational study is required to define the cooling requirements and assess the effect of coolant injection on overall engine performance. This will require the solution of the three-dimensional compressible turbulent Navier-Stokes equations with combustion to define the interaction between the coolant flow the the main engine gas path.

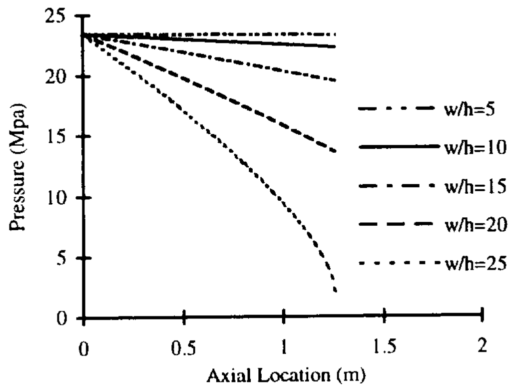

Figure 1 Rectangular Slot Pressure

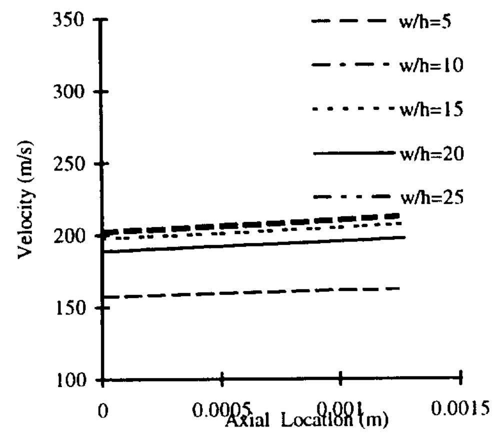

Figure 2 Velocity Distribution - 30 micron Hole

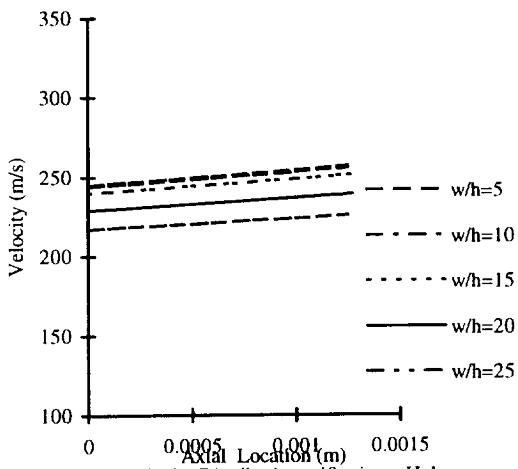

Figure 3 Velocity Distribution - 40 micron Hole

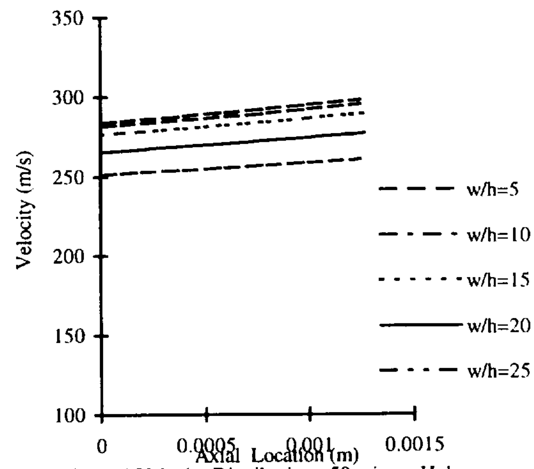

Figure 4 Velocity Distribution - 50 micron Hole

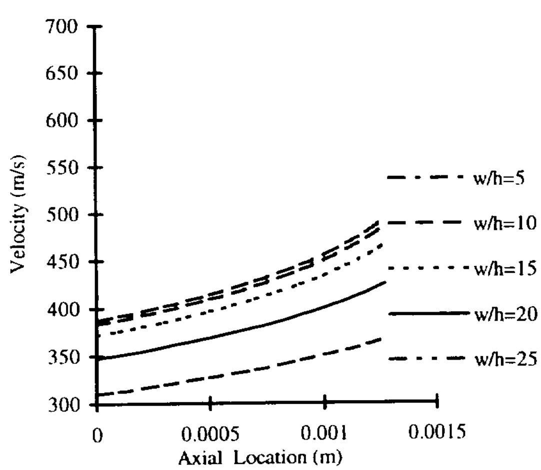

Figure 5 Velocity Distribution - 30 micron Hole

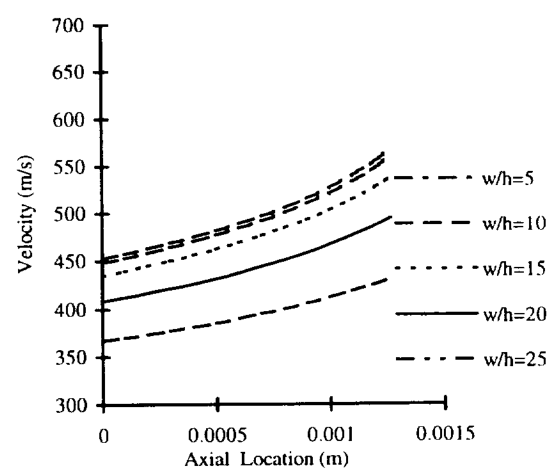

Figure 6 Velocity Distribution - 40 micron Hole

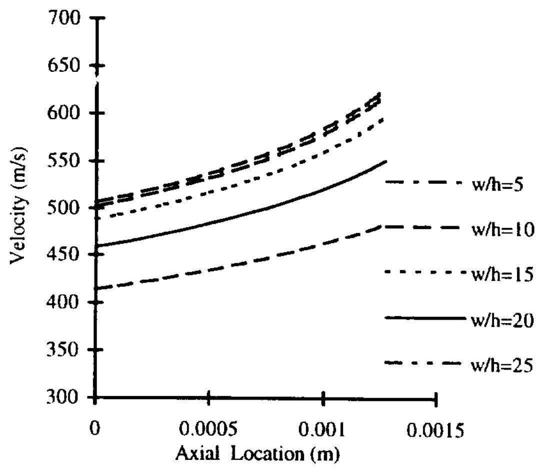

Figure 7 Velocity Distribution - 50 micron Hole