Heat Transfer During Burner Rig Thermal Fatigue of Ceramic Matrix Composites", T. Erturk, J. McKelliget, *Ceram. Eng. Sci. Prod.*, 16[4],1995, pp. 95-104

HEAT TRANSFER DURING BURNER RIG THERMAL FATIGUE OF CERAMIC MATRIX COMPOSITES

T. Erturk[*], J. McKelliget[**]
University of Massachusetts at Lowell, Lowell, MA 01854
[*] Department of Chemical Engineering
[**]Department of Mechanical Engineering

ABSTRACT

A two-dimensional transient heat transfer model was developed to calculate temperatures during the heating and cooling of hot pressed SCS-6 and SCS-9 SiC fiber reinforced ceramic composite specimens in the NASA Lewis Mach 0.3 atmospheric pressure burner test rig. The specimens modeled were thermally cycled under an impinging jet fuel flame in the temperature range 500 C to 1350 C under a constant applied tensile stress. An implicit finite difference procedure was employed that included the effect of forced convection, natural convection, and thermal radiation between the burner flame and the specimen. The predicted temperature distributions may be used to predict thermal stresses in the specimens.

INTRODUCTION

A primary constraint preventing the application of ceramic matrix composites in advanced gas turbine engines is a lack of adequate data and predictive modeling techniques to substantiate life limiting mechanisms such as creep, fatigue, thermal shock induced rupture, thermal fatigue under constant applied stress, thermo-mechanical fatigue, and corrosion and oxidation. Although previous investigations have evaluated the elevated temperature tensile fatigue and creep behavior of hot pressed SiCf/Si3N4, research data simulating the actual service conditions of gas turbine engines are scarce.

As part of a continuing program of research(1) the thermal fatigue behavior of crossply HP-SiCf/Si3N4 ceramic composites was examined for the first time in a burner rig environment in thermal cycles between 500 C and 1350 C. The high velocity jet fuel flame impingement configuration used simulates gas turbine hot path conditions well.

SYSTEM TO BE MODELED

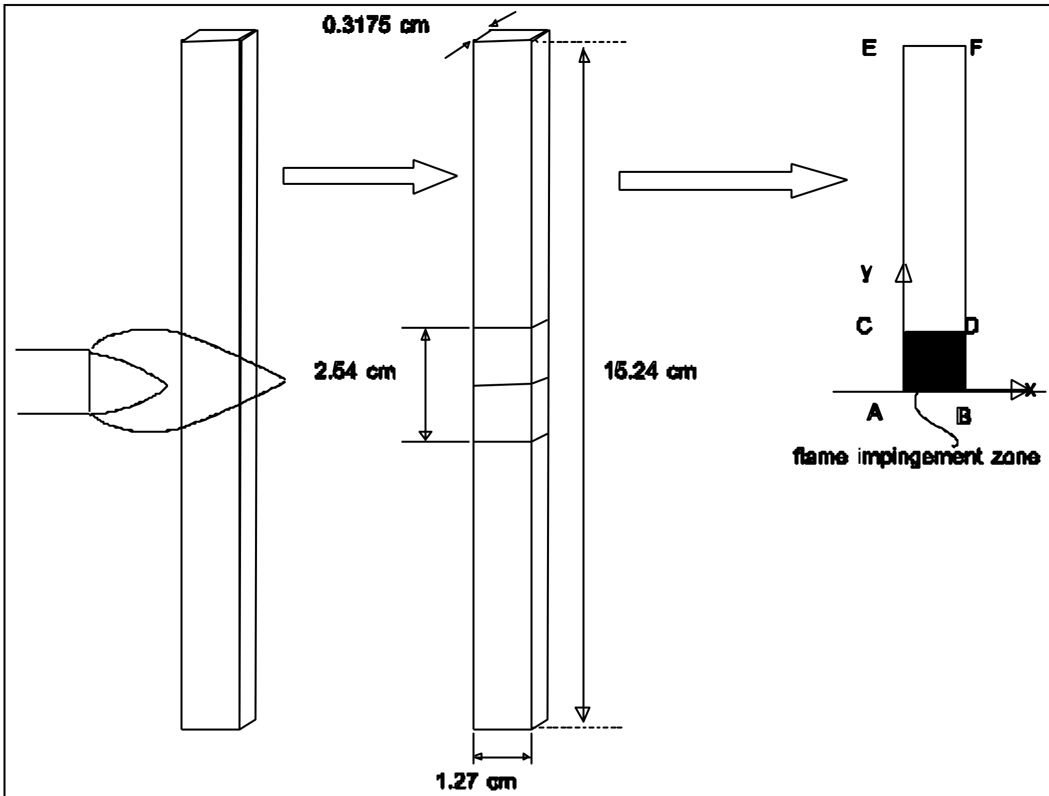

Figure 1 Schematic representation of burner rig configuration.

To simulate the environment of the high pressure nozzle of a gas turbine engine a jet fuel (JP4) flame was directly impinged onto 3.175x152.4x12.7 mm rectangular composite specimens at high velocities (Mach 0.3) as shown schematically in Figure 1. The temperature excursions obtained by rotating the burner and flame on and off the specimen simulate the potential quick temperature bursts experienced during vehicle takeoff and rapid cooling during engine shutdown. Thermal cycles consisting of

60 s flame impingement followed by 60 s of radiative and natural convective cooling were performed under a constant applied tensile stress. The specimen was located 25.4 mm away from the flame exit nozzle, and approximately 27.0 mm of the leading edge of the test specimen was impinged. The flame yielded a symmetrical temperature profile above and below the centerline of the specimen.

Using isentropic compressible flow theory and measured pressure drops the velocity of the flame around the specimen was calculated to be 231 m/sec. The temperature of the flame was measured using a thermocouple to be 1350 C.

The current paper describes a heat transfer model of the burner rig experiments which calculates the temperature history of the specimen during the heating and cooling process. As described elsewhere(1) the predicted temperatures are then used to calculate the thermal stresses in the specimen.

MODEL FORMULATION

A diagram of the region modeled and the coordinate system used are included in Figure 1. For modeling purposes the specimen was divided into two regions. In the flame impingement region (ABCD) the specimen is exposed to the flame during the heating cycle and to room temperature air during the cooling cycle. Although the shape of the boundary CD of the flame impingement zone can be varied to match the shape of the flame, the calculations in this paper were performed with the rectangular region shown. Outside the flame impingement region the specimen is assumed to be exposed to room temperature air during both the heating and the cooling cycles.

The following basic assumptions were made:

(i) Temperature variations in the z direction (through the minimum dimension of the specimen) are negligible. The maximum Biot number in this direction is about 0.07 at the leading edge of the flame impingement zone, which is within the lumped heat capacity regime ($Bi<0.1$).

(ii) The temperature field is symmetric about the plane AB at the center of the specimen. There is a minor violation of this assumption during the cooling cycle when the natural convection boundary layer starts to form at the bottom of the specimen.

(iii) The flow field around the specimen is laminar. The maximum Reynolds number, occurring at the trailing edge of the specimen in the flame impingement zone, is 6500 which is well below the turbulent transition for flow over a flat plate (5 x 10^5).

Under these assumptions the temperature field in the specimen is governed by the two-dimensional transient heat conduction equation expressed in the form;

$$\frac{\partial H}{\partial t} = \frac{\partial}{\partial x}\left(k_{s,x}\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k_{s,y}\frac{\partial T}{\partial y}\right) + S \qquad \mathbf{1}$$

H is the specific enthalpy of the specimen defined as

$$H(T) = r_s\int_0^T C_s(T)dT \qquad \mathbf{2}$$

and is introduced to account for the temperature dependent specific heat inside the specimen. In Equation 2 $k_{s,x}$ and $k_{s,y}$ are the thermal conductivities of the specimen in the x and y directions respectively. r_s 1 and C_s are the density and specific heat of the specimen respectively.

The heat flux into the external surface of the specimen, q, is calculated from heat transfer coefficients, h, as;

$$q = h\,(T_\infty - T_s) \qquad \mathbf{3}$$

where T_s and T_∞ 2 are the temperatures at the surface of the specimen and in the surrounding gas respectively. During the heating cycle T_∞ 3 is set equal to the flame temperature (1350°C) in the flame impingement zone, and equal to room temperature (20°C) outside this zone. During the cooling cycle T_∞ 4 is set to room temperature over the entire specimen.

Since temperature variations through the thin dimension of the specimen have been ignored heat conduction in this direction is modeled through the source term, S, in Equation 1, which is given by

$$S = 2q/\Delta \qquad \textbf{4}$$

where Δ5 is the thickness of the specimen.

The boundary conditions over surfaces AE, BF, and EF are;

$$k_{s,n}\frac{\partial T}{\partial n} = q \qquad \textbf{5}$$

where n is a coordinate normal to the surface of the specimen and directed outward. A zero temperature derivative is imposed at the symmetry plane AB.

The heat flux into the specimen consists of radiative transfer with both the flame and the room, and convective transfer from the gases surrounding the specimen. Both effects are modeled using appropriate heat transfer coefficients;

$$h = h_{conv} + h_{rad} \qquad \textbf{6}$$

Taking the view factors between the specimen and the surrounding gas to be unity the radiation heat transfer coefficient becomes;

$$h_{rad} = s\, e_s (T_s^2 + T_\infty^2)(T_s + T_\infty) \qquad \textbf{7}$$

Here s 6 is the Stefan-Boltzman constant, and e_s 7 is the emissivity of the specimen.

Convective heat transfer consists of forced convection in the flame impingement zone (ABCD) during the heating cycle, and natural convection over the entire surface during the cooling cycle.

Forced convection over the x-y faces is modeled as laminar flow over a zero incidence flat plate(2)

$$Nu_x = \frac{h_{conv} x}{k_{flame}} = 0.332\, Re_x^{\frac{1}{2}} Pr^{\frac{1}{3}} \quad \textbf{8}$$

where the Reynolds number and Prandtl number are defined as

$$Re_x = \frac{V_{flame} x}{n_{flame}} \quad ; \quad Pr = \frac{k_{flame}}{r_{flame} C_{P,flame}} \quad \textbf{9}$$

The horizontal coordinate, x, is measured from the leading edge of the specimen.

The convective heat transfer coefficient at the leading edge of the flame impingement zone (AC) was calculated using a computational fluid flow model of a planar jet impingement zone, and is given by the relation

$$\frac{h_{conv} R}{k_{flame}} = 40 \quad \textbf{10}$$

where R is the radius of the flame nozzle (1.27 cm). At the trailing edge flame impingement zone (BD) the convective heat transfer coefficient was neglected and only radiative transfer was included.

Natural convection is modeled as laminar flow over a vertical flat plate(2).

$$Nu_y = \frac{h_{conv} y}{k_{air}} = 0.59(Gr_y\, Pr)^{\frac{1}{4}} \quad \textbf{11}$$

where the Grashof number and Prandtl number are defined as

$$Gr_y = \frac{g\, b |T_s - T_\infty| y^3}{n_{air}^2} \quad ; \quad Pr = \frac{k_{air}}{r_{air} C_{P,air}} \quad \textbf{12}$$

The vertical coordinate, y, is measured from the plane CD during the heating phase and from the bottom of the plate during the cooling phase. The thermal expansion coefficient of the gases, *b*8, is assumed to be the reciprocal of the absolute gas temperature.

In Equations 8 to 12 the thermodynamic and transport properties of the flame and the air are evaluated at the mean film temperature.

The initial conditions for the problem are obtained by setting the temperature of the entire specimen to room temperature at the beginning of the first heating cycle.

SOLUTION TECHNIQUE

The governing equations are solved using a control volume finite difference formulation. The time integration is performed using a fully implicit scheme. A rectangular mesh consisting of 20 grid points in the x direction and 40 grid points in the y direction is used.

As a result of non-linearities resulting from the temperature dependence of the material properties, and from the radiation terms, it is necessary to perform an iterative loop at each time step. The solution is iterated until the overall energy conservation within the specimen is satisfied to within 0.1%. Using a time step of 0.1 s. a complete heating and cooling cycle requires 500s on a 486, 66MHz PC.

PROPERTIES

The thermal properties for the specimen are taken from the literature(3).

Although the model can easily handle temperature dependent flame properties, a lack of data requires that constant properties be used. These correspond to the flame at a mean film temperature of 925^{o}C and are summarized in Table I.

Table I
Properties of Flame at 925^{o}C

Property	Value	Units

Kinematic Viscosity	4.285 x 10^{-4}	$m^2.s^{-1}$
Thermal Conductivity	0.112	$W.m^{-1}.K^{-1}$
Prandtl Number	0.774	-

The temperature dependent properties of air are taken from standard heat transfer texts(2).

RESULTS AND DISCUSSION

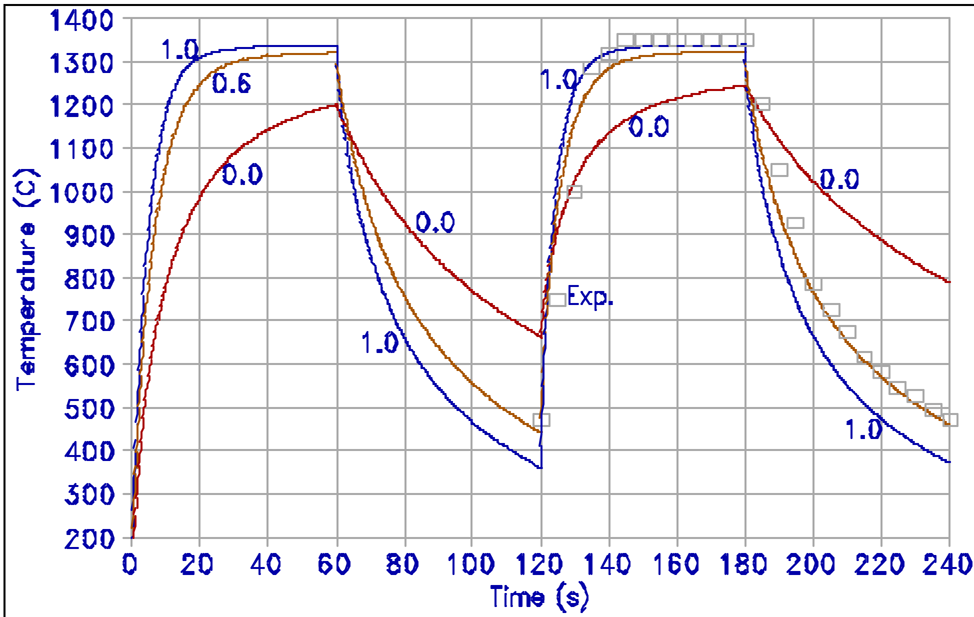

Figure 2 Predicted and experimental stagnation point temperatures for the first two cycles, as a function of specimen emissivity.

The predicted temperature history of the center of the impingement zone (point A in Figure 1) is given in Figure 2 for the first two heating cycles, along with experimental observations made with a two-color pyrometer. Since no exact measurements are available for the specimen emisivity, calculations were performed for three values of this parameter. An emissivity of 0.0 corresponds to no radiative transfer, while an emissivity of 1.0 corresponds to the maximum, or black body, radiation. It is evident from Figure 2. that radiation is an important exchange mechanism in this system.

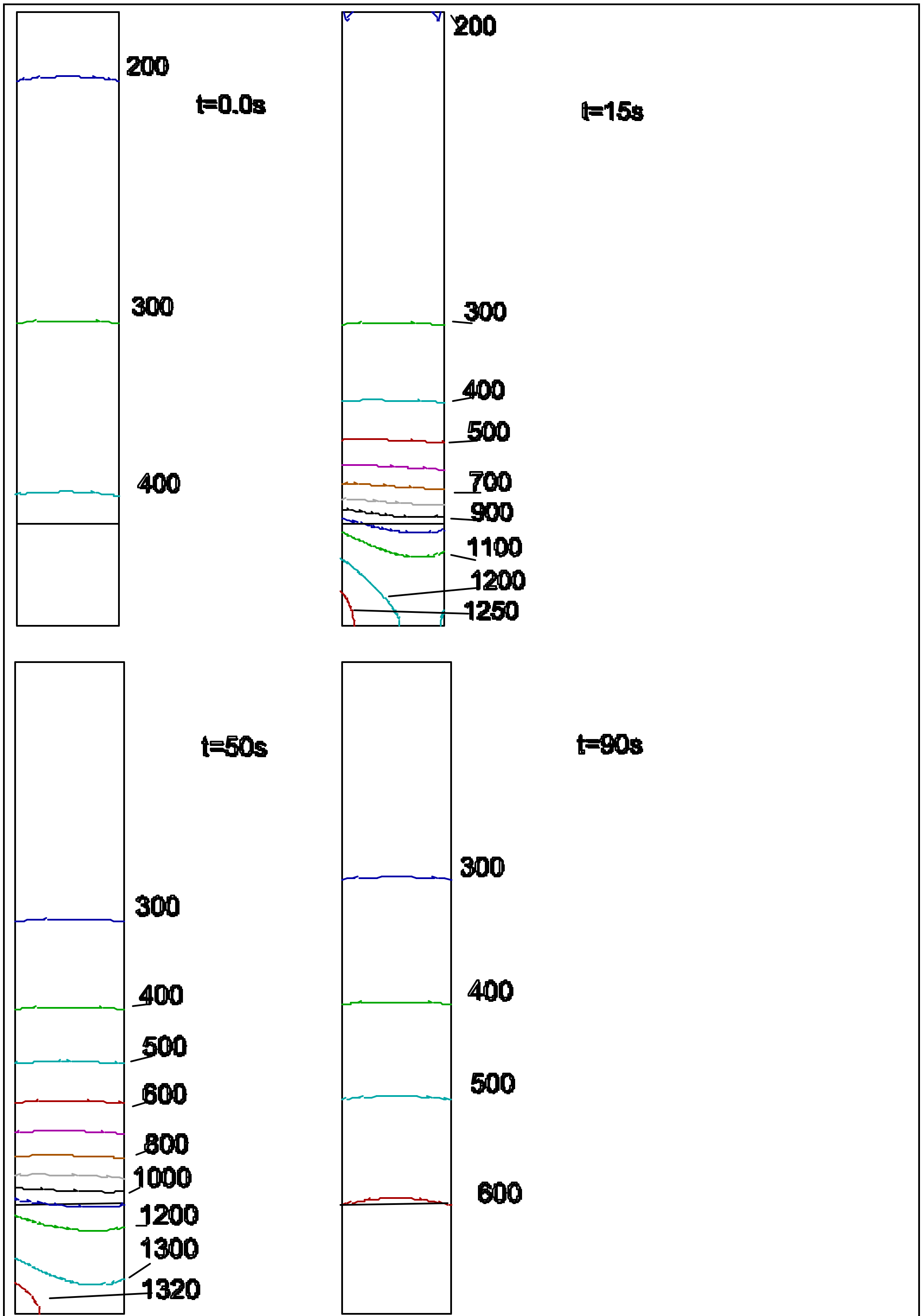

Figure 3 Calculated temperature distribution (C) in the second cycle.

Temperature distributions in the upper half of the specimen during the second heating cycle are given in Figure 3 for a specimen emissivity of 0.6. At t=0.0s, which corresponds to the end of the previous cooling cycle, there is a variation of only about 200 C from the center to the top of the specimen, and the temperatures are uniform in the horizontal direction. At t=15.0s the vertical variation in temperature is much greater, about 900 C, and there is a 60^{o}C horizontal variation in the flame impingement region. At t=50.0s, which is close to the steady state situation, the temperature variations in the horizontal direction have largely been smoothed out by conduction, and are only about 14^{o}C. At t=90.0s, which corresponds to halfway through the cooling cycle, the temperatures are low, and uniform in the horizontal direction.

In interpreting the physical significance of the computed temperature fields it is necessary to consider their effect on the generation of thermal stresses in the specimen. In the burner test rig (and probably also in the turbine engine) the specimen is free to expand in the y direction as the temperature increases. Consequently, the large vertical temperature gradients predicted by the model will not produce appreciable thermal stresses. It seems unlikely that the horizontal gradients will produce appreciable thermal stress, except early in the initial heating phase.

It is necessary to improve the model in a number of areas in the immediate future. More attention should be paid to the spacial variations of jet velocity and temperature over the specimen. Also, precise temperature dependent gas flame properties should be incorporated. Finally, the predicted temperatures should be related to the thermal stresses developed in the specimen.

REFERENCES

1. 'Thermomechanical Fatigue of Crossply SiCf/Si3N4 Ceramic Composites Under Impinged Kerosene-Based Flame.' G. St. Hilaire, D.Eng. Thesis, Dept. of Mechanical Engineering, University of Massachusetts at Lowell, April 1993.

2. 'Heat Transfer', J.P. Holman, McGraw-Hill, Sixth Ed. 1986.

3. 'Thermal Conductivity of SiC Reinforced Si3N4', R.E. Taylor, H.Groot, and J. Larimore, TPRL 880, Thermophysical Properties

Research Laboratory, School of Mechanical Engineering, Purdue University, West Lafayette, Indiana, October 1989.