International Journal of Heat and Mass Transfer 88 (2015) 284–296

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Optimisation of an air film cooled CFRP panel with an embedded vascular network

M.W. McElroy [a,*], A. Lawrie [b], I.P. Bond [c]

[a] Durability, Damage Tolerance, and Reliability Branch, NASA Langley Research Center, Hampton, VA, USA
[b] Department of Mechanical Engineering, University of Bristol, Bristol, UK
[c] Department of Aerospace Engineering, University of Bristol, Bristol, UK

ARTICLE INFO

Article history:
Received 28 January 2015
Received in revised form 22 April 2015
Accepted 23 April 2015
Available online 14 May 2015

Keywords:
Carbon fibre reinforced polymer
Film cooling
Microvascular
Thermal analysis

ABSTRACT

The increasing use in the aerospace industry of strong, lightweight composite materials in primary structural components promises to substantially reduce aircraft non-pay-load weight, improving fuel consumption and operating profitability. This study explores the extension of composite material to regions of gas turbine engines previously considered too hot for composites with moderate melting points. Throughout the majority of a gas turbine cycle, gas stream temperatures exceed the polymer composite glass transition by a considerable margin. Boundary layer cooling strategies, however, may be adopted in the compression stages to extend the downstream distance that can be constructed using lightweight composites. This paper presents formulation and validation of a numerical model and its use in an optimisation study to develop a systematic process for thermal design of polymer composite structures in 'warm' gas streams. Internal vascular and external boundary layer film cooling strategies are considered.

Published by Elsevier Ltd.

1. Introduction

The increasing use in the aerospace industry of strong, lightweight composite materials in primary structural components promises to substantially reduce aircraft non-pay-load weight, thereby improving fuel consumption and operating profitability. Use of common polymer based composites for gas turbine engine components, however, can prove challenging due to the fact that the maximum operating temperature for a fibre-reinforced polymer composite is constrained by the glass transition temperature (T_g). Copolla [1] showed that for a composite specimen, mechanical performance degrades rapidly at higher temperatures. In the case of a carbon-fibre reinforced polymer (CFRP), an attractive candidate material for aerospace application, T_g lies in the range 100–200 °C [2,3], with precise values dependent on the details of its composition. Throughout the majority of a gas turbine cycle, gas stream temperatures exceed this range by a considerable margin. Boundary layer film cooling strategies may be adopted, however, in the compression stages to extend the downstream distance that can be constructed using lightweight composites.

From the earliest gas turbine engines of the 1940s, turbine blade weakening at high temperature has been mitigated by passing cool air through hollow cavities within the blades [4], transferring heat away from the metal into the coolant flow. In the 1950s, research was initiated on designs that expel the internal cool air flow into the external hot flow through apertures on the blade surface. This allows coolant to be swept back over the blade, coating the blade surface with a boundary film of cool air [5,6] that thermally insulates the metal from the hot stream. Film cooling combined with internal heat exchange has been shown to be an effective active cooling approach for turbine blades [7,8], and modern engine components currently can be designed for gas temperatures downstream of the combustor that exceed the melting point of the metal blades.

While there are a small number of publicly available studies on the use of internal vascular cooling in polymer composite materials, none so far consider the role film cooling can play in improving thermal design. Lyall examined a stiffened composite panel for satellite electronics systems, containing fluidic microchannels for cooling [9]. This study was developed by Williams to include structural and thermal analysis [10]. Kozola explored the use of composite material for a fin with an internal vascular network using water or oil as the coolant. Additionally, a one-dimensional numerical heat transfer model of the fin surface was developed [11]. Quantifying the overall efficiency of a component containing an

* Corresponding author at: 2 W. Reid St, Mail Stop 188E, Hampton, VA 23681, USA. Tel.: +1 757 864 9652.
E-mail address: mark.w.mcelroy@nasa.gov (M.W. McElroy).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.04.071
0017-9310/Published by Elsevier Ltd.

internal vascular network is an important question and in their work, Pierce and Phillips included a mechanical evaluation of a panel containing vascular cooling networks [12,13]. Soghrati et al. looked at strategies for woven composites using microchannels [14], and Bejan et al. [15–18] studied a nature-inspired tree branch geometry.

The aim of this study is to explore the extension of CFRP materials to regions of gas turbine engines previously considered too hot for polymer composites with moderate degradation temperatures. An active cooling strategy consisting of both internal vascular heat exchange and external boundary film cooling is investigated. The vascular network topology in a structural component requires careful consideration to balance thermal efficiency between the vascular and film cooling effects. An optimisation study is performed using a numerical model of a cooled composite panel to better understand this relationship. The optimisation study serves as demonstration of a systematic methodology for thermal design of compressor blades manufactured from polymer composites. Experimental work is included with the purpose of validation of the numerical model. An idealised thin-plate CFRP panel of a thickness, composition, and structural rigidity comparable with low pressure compressor blades is considered.

Sections 2–4 focus on creation of an experimentally-validated, predictive numerical model of thermal behaviour in an actively cooled composite panel. In Section 5, the model is extended for iterative design refinement using an automatic gradient-based optimiser. Proof-of-concept of a thermal design methodology for a vascular/film cooled panel is demonstrated.

2. Thermodynamic modelling

A numerical model was developed to predict the temperature distribution of a thin flat plate subjected to a hot external air flow and actively cooled by an internal vasculature and an external cool film. A compressible finite volume approach is used to simulate the internal vasculatory flow. Fourier's law is solved across the plate to estimate temperature distribution in thermal equilibrium. Film cooling is simulated based on a model derived from the classical solutions for viscous boundary layer growth. The thermal transport from the hot external gas stream towards the plate is estimated from empirical relations governing turbulent entrainment, treating temperature as a passive scalar. Because compressor blades operate in a strongly adverse pressure gradient, curvature of the blades is limited to ensure flows remain attached. It follows then that blade profiles have a small thickness-to-chord ratio and thus a thin flat plate test model offers a good first approximation to their heat transfer properties. More realistic blade profiles would be a trivial extension to the model.

In this paper, the modelling activity is focused only on non-intersecting vascular topologies to reduce the nonlinearity in the parameter space over which an optimal solution may be found. This restriction is a temporary convenience to ensure conclusions are robust and intuitive, but the proposed methodology is fully generalisable to complex network topologies. Another restriction of the current model is that internal vascular flow remains laminar. Including turbulent flow in this study would introduce highly non-linear heat transfer behaviour to the system and thus create a much more challenging optimisation problem. Additionally, while this constraint may normally be satisfied in these cooling flows, a model that accounted for and operated in a regime near turbulent transition could potentially be unduly sensitive to parameter variations that lie within the range of manufacturing tolerances in small cooling ducts. Such sensitivities would curtail the ability, particularly of gradient-based automatic optimisers, to recover robust global performance maxima.

Compressible Euler equations accounting for friction are discretised in one dimension with a first order flux-balance, and the mass flux is adjusted iteratively to satisfy a prescribed downstream exit pressure given imposed upstream pressure and temperature boundary conditions. The following equations are presented with regards to a single control volume, with subscripts 1 and 2 representing the locations at the upstream and downstream faces, respectively. From mass conservation, the outlet velocity is given by

$$v_2 = \frac{\dot{V}_2}{A_{sec}} \tag{1}$$

where A_{sec} is the cross sectional area of the tube. Volumetric flow rate, $\dot{V}_2$, is determined using the ideal gas law and when substituted into (1) yields the form of the mass continuity equation used in the solution given by

$$v_2 = \frac{\dot{m} R_{sp} T_2}{P_2 A_{sec}} \tag{2}$$

where A_{sec} is the cross sectional area of the tube, R_{sp} is the specific gas constant, $\dot{m}$ is the coolant mass flow rate, T is coolant temperature, and P is pressure. Conservation of energy in a steady flow is given by

$$h_1 + \frac{v_1^2}{2} = h_2 + \frac{v_2^2}{2} + Q + W \tag{3}$$

where h is enthalpy, Q is heat, and W is work due to friction (note that h, Q, and W are normalised by mass in this equation). Assuming laminar flow and using a Darcy friction factor, f, the change in pressure as a result of friction, ΔP_f, is given as

$$\Delta P_f = f \frac{\ell}{2r} \frac{\rho v^2}{2} = \frac{64}{Re} \frac{\ell}{2r} \frac{\rho v^2}{2} \tag{4}$$

where, ℓ is the control volume length, r is the control volume radius, v is coolant flow velocity, Re is the Reynolds number, and ρ is coolant density. A low-order approximation is made to the cell centre velocity used for estimating average wall friction, and is set equal to the inlet velocity v_1. Thus (3) can be written as

$$Q = c_p(T_1 - T_2) + \frac{v_1^2 - v_2^2}{2} - A_{wall} \frac{8}{Re} \rho v_1^2 \ell \tag{5}$$

where A_{wall} is the surface area of a control volume. The rate of heat transfer, $\dot{Q}$, is obtained after multiplication of (5) by $\dot{m}$.

The heat transfer rate can alternatively be expressed as

$$\dot{Q} = A_{wall} U (T_1 - T_\infty) \tag{6}$$

where T_∞ is external air temperature. Making the same low-order linearisation previously used to derive (5), inlet temperature T_1 is used to represent average temperature over a control volume. Multiplying (5) by $\dot{m}$ and equating with (6) yields a formulation of the energy equation suitable for sequential numerical evaluation of a single vessel from inlet to outlet given as

$$\dot{m}\left[c_p(T_1 - T_2) + \frac{v_1^2 - v_2^2}{2} - A_{wall} \frac{8}{Re} \rho v_1^2 \ell\right] = A_e U (T_1 - T_\infty) \tag{7}$$

Eq. (7) converges to the exact solution as control volumes tend to zero and boundary conditions are iterated to consistency. The heat transfer coefficient, U, in (7) was shown by Pierce to vary weakly with mass flux [12]. However, it is shown later in Section 4 where U is treated as a constant, that the model performs well.

Using the form of the continuity equation shown by (2), the initial velocity, v_1, is obtained to use in (7). Conservation of momentum provides a third equation to the system, and is expressed as a Rayleigh condition,