V ECCOMAS Thematic Conference on the Mechanical Response of Composites
COMPOSITES 2015
S.R. Hallett and J.J.C. Remmers (Editors)

OPTIMISATION OF AN AIR FILM COOLED CFRP PANEL WITH AN EMBEDDED VASCULAR NETWORK

Mark W. McElroy*, **Andrew Lawrie**†, **Ian P. Bond**†

*NASA Langley Research Center
Hampton, VA USA
mark.w.mcelroy@nasa.gov

†University of Bristol
Bristol, UK

Keywords: Carbon fibre-reinforced polymer, film cooling, vascular.

Summary: *This paper summarizes research performed on thermodynamic simulation and design optimisation of a composite panel cooled by an external cool film and an internal vascular network.*

1. INTRODUCTION

The increasing use of lightweight composite materials in primary aerospace structures promises to substantially reduce aircraft non-payload weight, thereby improving fuel consumption and operating profitability. Weight reduction through use of 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, which can be as low as 100°C - 200°C [1, 2] in the case of an epoxy matrix. Throughout the majority of a gas turbine cycle, gas stream temperatures exceed this range by a considerable margin.

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 [3]. 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 [4] 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 [5]. A similar approach may enable use of polymer composites for components in the compressor stage of a gas turbine engine.

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 (2008) examined a stiffened composite panel for satellite electronics systems, containing fluidic microchannels for cooling [6]. This study was developed by Williams (2010) to include structural and thermal analysis [7]. Kozola (2010) 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 [8]. Quantifying the overall efficiency of a component containing an internal vascular network is an important question and in their work, Pierce and Phillips (2010, 2011) included a mechanical evaluation of a panel containing vascular cooling networks [9, 10].

The aim of this study is to explore the extension of carbon fibre-reinforced polymer (CFRP) materials to regions of gas turbine engines previously considered too hot for polymer composites. 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 serve 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.

2. THERMODYNAMIC MODELLING

A numerical model was developed to predict the temperature distribution of a thin flat plate that is subjected to a hot external air flow and is 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 of geometric similarities, a thin plate test model offers a good first approximation to heat transfer properties for a gas turbine compressor blade.

In this paper, the modelling activity is focused only on non-intersecting vascular topologies, 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 vascular flow in this study would introduce highly non-linear heat transfer behaviour to the system and thus create a much more challenging optimisation problem.

Compressible Euler equations accounting for friction are discretised into one dimensional control volumes (CV) 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 CV, with subscripts 1 and 2 representing the locations at the upstream and downstream faces, respectively. From mass conservation, the outlet velocity is given by

$$v_2 = \frac{\dot{m} R_{sp} T_2}{P_2 A_{sec}} \tag{1}$$

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

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

where h is enthalpy, Q is heat, and W is work due to friction (note that h, Q, and W are normalized 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

$$\Delta P_f \;=\; f\frac{\ell}{2r}\frac{\rho v^2}{2} = \frac{64}{Re}\frac{\ell}{2r}\frac{\rho v^2}{2} \tag{3}$$

where, ℓ is the CV 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, (2) can be written as

$$Q = c_p(T_1 - T_2) + \frac{v_1^2 - v_2^2}{2} - A_{wall}\frac{8}{Re}\rho v_1^2 \ell \tag{4}$$

where A_{wall} is the surface area of a control volume. The rate of heat transfer, $\dot{Q}$, is obtained after multiplication of (4) by $\dot{m}$.

The heat transfer rate can alternatively be expressed as

$$\dot{Q} = A_{wall} U (T_1 - T_\infty) \tag{5}$$

where T_∞ is external air temperature. Making the same low-order linearisation previously used to derive (4), inlet temperature T_1 is used to represent average temperature over a control volume. Multiplying (4) by $\dot{m}$ and equating with (5) yields a formulation of the energy equation suitable for sequential numerical evaluation of a single vessel from inlet to outlet given as

$$\dot{m}\left[c_p(T_1 - T_2) + \frac{v_1^2 - v_2^2}{2} - A_{wall}\frac{8}{Re}\rho v_1^2 \ell\right] = A_e U (T_1 - T_\infty). \tag{6}$$

Equation (6) converges to the exact solution as CV length tends to zero. The heat transfer coefficient, U, in (6) was shown by Pierce to vary weakly with mass flux [9]. 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 given by (1), the initial velocity, v_1, is obtained to use in (6). Conservation of momentum provides a third equation to the system, and is expressed as a Rayleigh condition,

$$P_1 A_{sec} + \dot{m} v_1 = P_2 A_{sec} + \dot{m} v_2 + \Delta P_f A_{sec} \tag{7}$$

The unknown variables T_2, P_2, and v_2 are found by solving (1), (6), and (7) simultaneously for each CV. Iteration proceeds until boundary conditions are satisfied and mass flux converges.

A two-dimensional numerical evaluation of Fourier's heat conduction law, $q_{cond} = -kA(dT/dx + dT/dy)$, predicts temperature $T(x, y)$ in the thin plate, where variations in the out-of-plane direction are considered negligible. Future versions of the model may be enhanced to be applicable to anisotropic heat flow by designating different heat transfer rates for the x and y directions based on the lay-up of a laminate.

Heat convection from the surrounding air occurring out-of-plane is accounted for using Newton's law of cooling, $q_{conv} = hA(T - T_\infty)$. An implicit second order finite difference scheme is formed from a standard 5-point stencil, in this instance on a regular grid. Source terms from CVs and external air temperature are linearly interpolated onto the regular grid. The proposed model of external film cooling from leading-edge apertures uses the well-known Blasius solution [11] to the boundary layer equations, $\delta(x) = 1.72\sqrt{\nu x/u}$, where ν is the kinematic viscosity, u is the free airstream velocity, and δ is the boundary layer thickness, to estimate the downstream growth of an insulating layer in the direction normal to the plate. A turbulent entrainment hypothesis is used to approximate transverse mixing of coolant air with the hot gas stream at a rate consistent with a spread angle of $12°$ [12]. An illustration of modelling assumptions with two internal vascules is shown in Figure 1. The temperature of the aperture is assumed to be transferred to the insulating film, and the initial film thickness is set to match the aperture diameter.

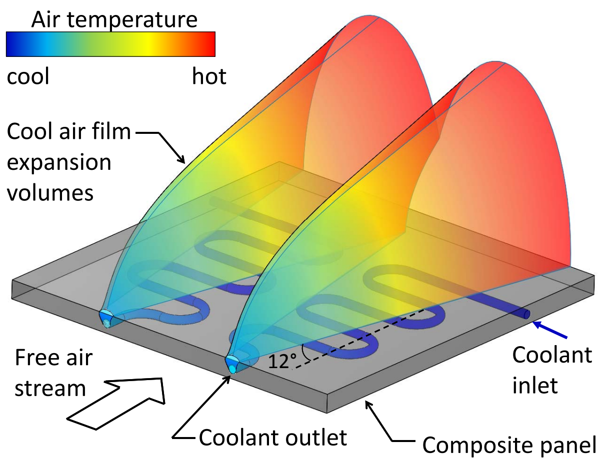

Figure 1. Representative geometry and temperature variation of boundary layer cool air film.

The film air temperature varies along the air stream axis according to

$$T = \left(\frac{A_x - A_o}{A_x}\right) T_\infty + \left(1 - \frac{A_x - A_o}{A_x}\right) T_{exit} \tag{8}$$

where A_o is the initial film section area, A_x is the film section area at a location along the free airstream defined by x, and T_{exit} is the coolant temperature at the aperture. Mixing was assumed to be sufficiently rapid relative to the temperature changes so that the film can be treated as having uniform temperature in a given cross-section.

When the geometry of a cool air film is determined on a panel, the resulting local film temperature replaces the external air temperature source term T_∞ in (6) in a new solution iteration. The updated T_∞ temperature distribution, in turn, then affects the temperature of the internal vascular coolant flow. Since the internal coolant flow supplies the cool film, an iterative loop is performed until the model solution converges as determined by a change in the standard deviation of the panel temperature from one iteration to the next of less than 1%.

3. EXPERIMENTAL SETUP

A flat CFRP panel containing a vascular network was fabricated to validate the numerical model described in Section 2. Because the tests are used for model validation purposes, the vascules are placed far apart in order to obtain internal coolant flows uninfluenced by other nearby vascules. Additionally, in this configuration, the overlap of contributions on panel temperature reduction from adjacent vascules is minimized. By isolating the flow behaviour in each vascule and obtaining an exaggerated panel temperature variance in between vascules, the thermal behaviour is somewhat simplified making the tests well suited for model validation. The effect that including vascules has on mechanical performance parameters such as laminate stiffness and strength was not a part of this study.

The vasculature was connected to a compressed air supply and a hot gas stream was passed over the panel. Once in thermal equilibrium, thermal images were taken of the panel using a NEC San-ei Thermotracer Type TH9100MR camera and compared in detail to predictions from the numerical model whose output closely approximates panel surface temperature. The panel was made from carbon/epoxy prepreg manufactured by Gurit and was oven cured as recommended at 70°C for 16 hrs. The lay-up is given by $[\pm 45, 90, 0_3]_s$ and results in a total panel thickness of 2.4 mm. Four centrally located zero degree plies were cut into patterns such that a linear void would exist and define the geometry for a network of four vessels. During lay-up, 1.1 mm diameter Polytetraflouroethylene (PTFE)® coated wire was lain along four paths that would form the cooling vessels. During curing, molten resin flows around the PTFE® coated wire, and after curing, the wires were pulled out leaving a smooth circular unlined internal void [13]. Fittings were glued to the inlet points of each vessel to connect to an air supply. A photograph and overlaid schematic of the panel and vasculature network geometry is shown in Figure 2a.

A hot gas stream from a Bosch heat gun was guided over the panel by a steel diffuser fabricated to match the panel geometry. Pressurized air at room temperature was supplied to a reservoir manifold, then distributed to the vessel inlets by pneumatic pipes. The heat gun, diffuser, and panel were assembled in a vertical orientation supported in place by clamps and test stands. A photograph of the experiment is shown in Figure 2b. On one side of the panel, six k-type thermocouples were attached to directly measure the air temperature at various locations on the panel. Vessel inlet flow temperature was measured using the thermal image of inlet flow pipes just outside the panel. Air flow velocity at the outlet of each vasculature was recorded using an anemometer, which is assumed to be minimally invasive to the flow. At the time of this paper, experimental validation that is specifically applicable to the boundary layer film cooling behaviour in the model has not yet been performed.

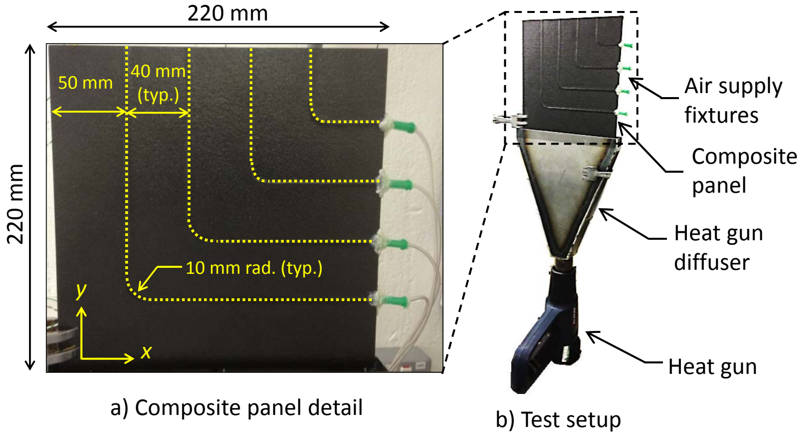

Figure 2. Photograph of composite panel with schematic of embedded cooling vasculature and connected air supply used for numerical model validation.

4. MODEL VALIDATION

Various external gas stream temperatures were used in testing, but in this paper the focus is only on conditions with a heat gun setting of 140°C. To characterize the temperature distribution of heated air flowing across the exterior of the panel such that it can be easily used by the numerical model, thermocouple data were gathered at six panel locations and interpolated using an equation of the form, $y = a * exp(b/(x + c))$, in the streamwise direction and linearly in the cross-stream direction. Using vessel aperture anemometry data and the diameter of the aperture, mass flow rate was estimated for each of the four vessels as

$$\dot{m} = \frac{v_2 P_2 A_{sec}}{R_{sp} T_2} \tag{9}$$

where P_2 is set equal to the ambient room pressure (1 bar). Coolant pressure just upstream of the panel was determined as an input parameter for the thermodynamic model by a calibration process that involved iteratively varying the supply pressure until aperture coolant flow velocity and mass flow rate matched experimental data. Material properties used in the simulations are as follows: conductive heat transfer rate, k = 1.28 W/mK [14]; overall heat transfer coefficient, U = 4.0 W/m^{2}K (est.) [9, 10]; convective heat transfer coefficient, h = 9.0 W/m^{2}K [10]; specific heat of air, c_p = 1005 J/kgK; and kinematic viscosity of air at 20°C, ν = 1.568 m^2/2 x 10^6.

Figure 3 shows a strong correlation between experimental and numerical predictions using a heat gun setting of 140°C and a calibrated inlet pressure of 1.2 bar. In addition to the visual temperature contour images, temperature profiles at cross-sections X-X and Y-Y are shown covering in detail the vessel inlet and outlet regions of the panel. The model's good correlation with the experiment suggests that it may be used as a reliable proxy for the thermodynamics of the physical system in similar panel designs.

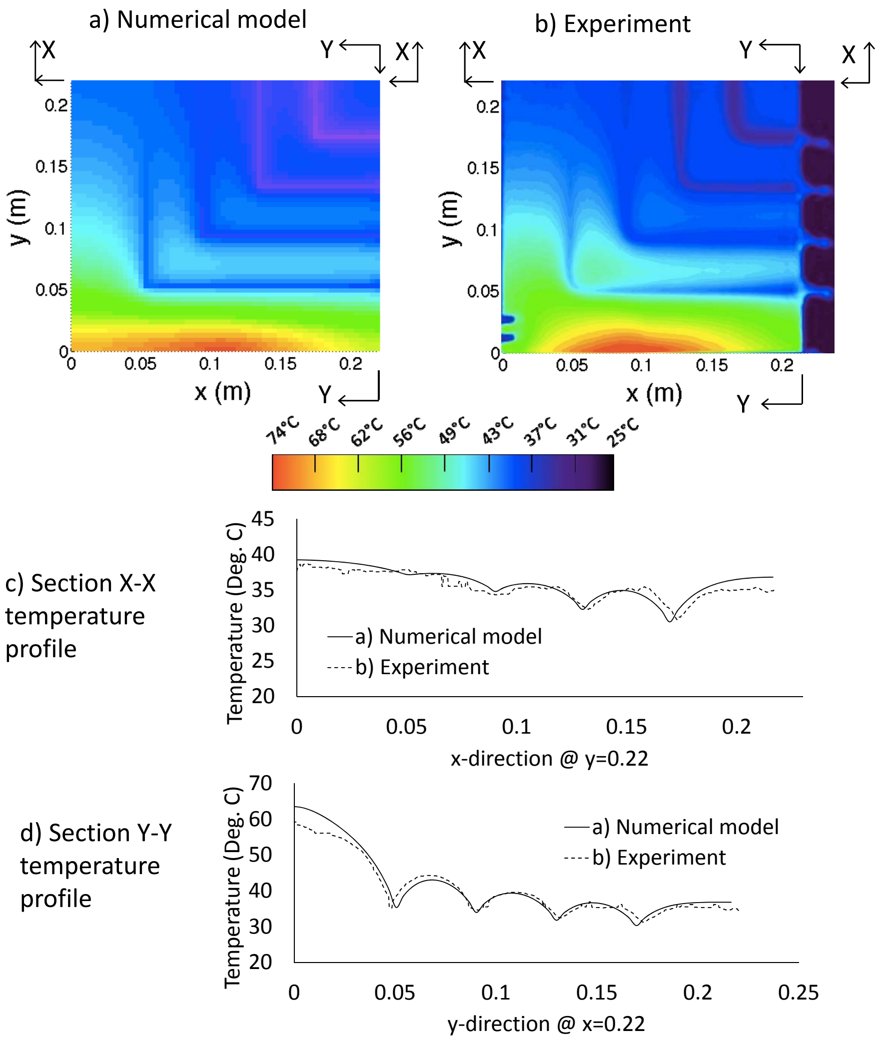

Figure 3. Numerical and experimental data correlation for: inlet pressure $P_{in} = 1.2$ bar and vascule inlet temperatures $[T_{in}]$ = [$T_{in,1}$ $T_{in,2}$ $T_{in,3}$ $T_{in,4}$] = [38 36 33 31]°C.

5. OPTIMISATION

An automatic optimisation procedure is demonstrated in this section to identify suitable performance metrics and establish a thermal design methodology for vasculature topology in a thin panel. The design methodology can be extended to the general case of a gas turbine compressor blade. Design of vascule topology for a combined film-heat exchange cooling system, as described in Section 1, is complex due to the fact that a heat transfer loop is formed between the internal coolant flow and the external film flow. The methodology presented here results in designs where this is taken into account and each cooling mode is utilized without undermining the performance of the other.

In this study, focus is on the thermal behaviour of a 0.1 m x 0.1 m panel containing a single internal vessel. While in any industrial application, one would anticipate several such vessels being embedded in the composite material, modelling this more complex scenario is a relatively straightforward extension of the proof-of-concept shown here. The vessel is defined to take the form of the real part of a complex sinusoid. The design parameter vector [A B C D] is defined according to the function

$$y = Ae^{x}sin(Bx)(1/480) + (1/A)e^{-x}sin(Cx) + sin(Dx) \tag{10}$$

where y is the spanwise ordinate, x increases in the direction of the gas stream, and (1/480) in the first term is an empirically determined scaling constant. The sinosoid defined in equation (10) was chosen in an attempt to minimize the number of design parameters but maximize the number of possible vascule topology configurations. Figure 4 illustrates several distinct vessel topology configurations that can be obtained by varying the four design parameters in equation (10). Additionally, as shown in Figure 4, for all of the simulations performed in the optimisation study, coolant flows in the $-x$ direction, the free airstream flows in the $+x$ direction, and the cool film originates along the edge of the panel where $x = 0$.

The simplest optimisation objective is to seek the lowest maximum temperature, since structural integrity is constrained by the proximity to the material T_g at the most susceptible location. However, simply minimizing the maximum panel temperature may not be a suitable basis for optimal design since other regions of the panel might receive more cooling than necessary to remain below T_g, and one could anticipate incurring an unnecessary penalty in structural efficiency. One can account for the distribution of the structural penalty without feeding back additional objectives (which would greatly restrict the choice of optimisation strategies) by instead considering average temperature across the panel. However, once again this optimisation objective may be vulnerable to suboptimal or invalid designs returning an attractive performance value. A low average value would not necessarily satisfy the independent constraint that temperatures should everywhere remain below the material T_g. It would seem necessary, at least a priori, to invoke second order statistical quantities to constrain the metric sufficiently well to deliver practical and efficient designs. Three panel temperature performance metrics were considered in the optimisation study namely maximum, average, and standard deviation.

The local performance space was evaluated on the 0.1 m x 0.1 m panel using a perturbation technique to estimate gradients and Newton's method of steepest descent to determine the

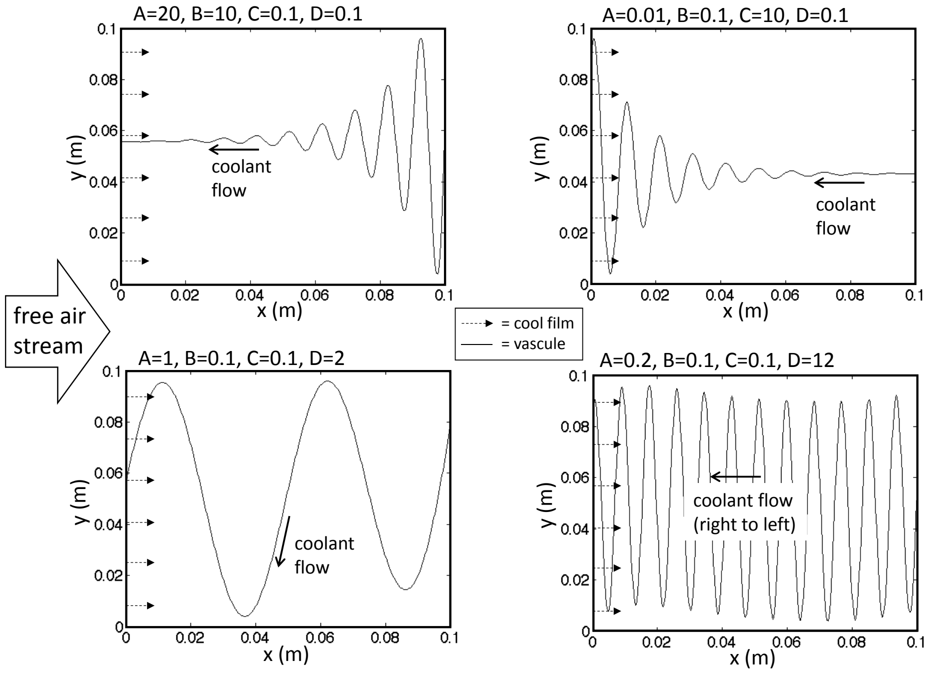

Figure 4. Possible vessel topology using (10). Cool films originate along the entire panel edge coincident with the y-axis.

choice of step size in the parameter space. A refined implementation of this method, known as a dynamic hill-climber [15] was performed using a Rolls Royce proprietary optimisation software package [16] which executes the numerical model described in Section 2 at each iteration. Freestream air temperature, freestream air velocity, coolant inlet temperature, coolant inlet pressure, and vascule section radius were kept constant and defined as 100°C, 400 m/s, 25°C, 1.2 atm, and 0.5 mm, respectively.

Figure 5 shows thermal plots, to a resolution of 1.66 mm, of optimised vacular topologies with and without film cooling using the performance metrics introduced above: minimising the temperature extremum, minimising the average, and minimising the standard deviation, while keeping constant the temperature and mass flux of the coolant and gas stream. In Figure 5, the external gas stream flows from left to right, and the coolant exits over the leading edge in cases where film cooling is included (see Figure 4).

Similar observations can be made from optimal vascular topology designs using all three objectives. It is clear that large amplitude oscillations of the vessel pathway are best, simply because heat is more effectively removed from all regions of the panel. When minimisation of both maximum temperature and average temperature is used as the design objective, there is a reduced need for internal cooling towards the (left) leading edge of the panel, since external cooling is most effective in this region of the panel near its source. The second order statistical moments of the temperature are arguably the most suitable performance metric, since one would expect the temperature variance to be more closely tied to the quality and 'design efficiency' of the vessel topology and less strongly correlated with the particular boundary conditions used in this example. However, the resulting optimal design using this metric is very similar to the others.

The considerations of structural performance degradation caused by the presence of the vasculature have not been incorporated into the automatic process. One approach to gage relative structural robustness is based on total cooling vessel length using the reasoning that if total cooling vessel length is minimized, adverse effects on the structure are also minimized. For all design metrics, an increase in performance is seen and a reduction in vessel length ranging between 7% and 28% is obtained if a cool film is included. This finding indicates that use of a cool film may improve structural efficiency for a vascularized panel. Overall, the specific vascule topology determined in this study is less relevant, however, than the general design methodology presented where the complex heat exchange interaction between the internal and external film cooling is accounted for.

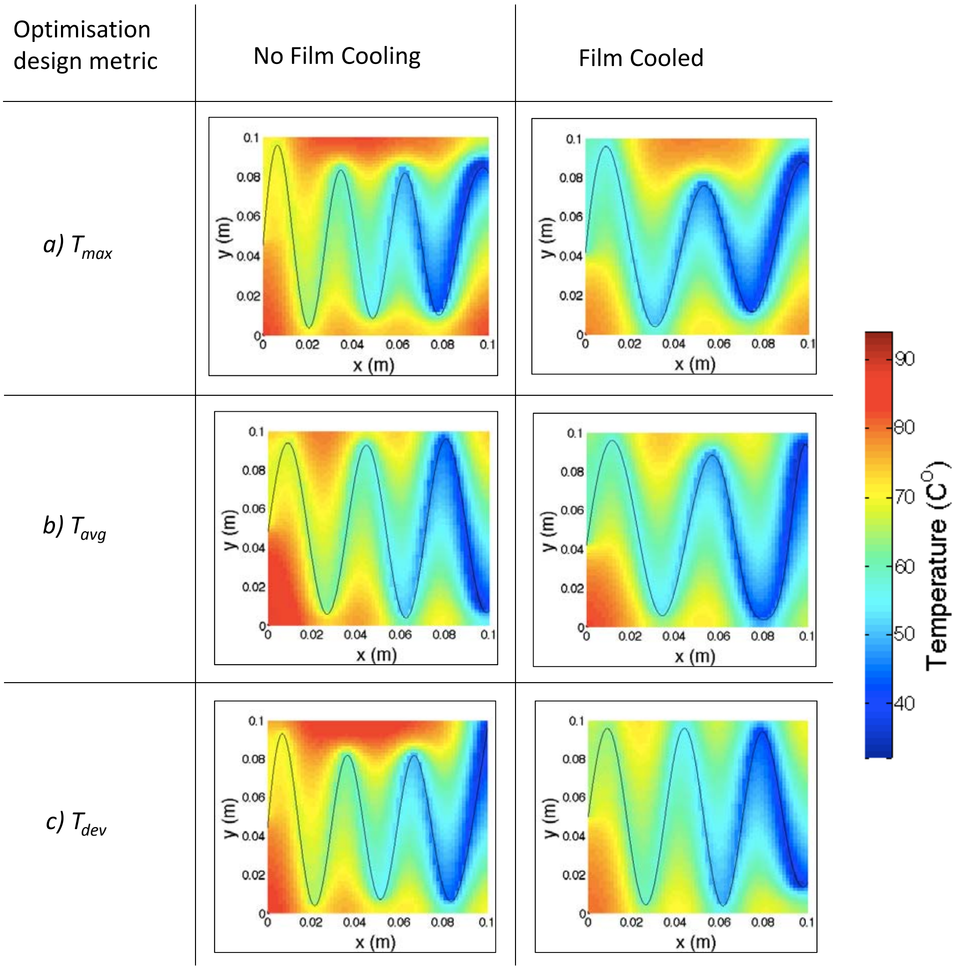

Figure 5. Optimised vascular topologies with and without film cooling.

6. CONCLUSIONS

An investigation was conducted on the use of CFRP composite materials for structural components in gas turbine engines. Use of polymer materials in elevated temperature environments, such as a gas turbine engine, requires an active cooling strategy to ensure they remain below their glass transition temperature. An experimentally validated numerical model was created that is capable of a thermal simulation of a thin composite panel actively cooled by an internal vascular network and an external boundary layer film. An optimisation study was then conducted using the numerical model to study the effect of external film cooling on an efficient internal vasculature topology design. An improvement in performance is observed if film cooling is included, as measured by maximum, average, and deviatory panel temperatures. The improved performance is accompanied by a reduced internal vasculature length. In performing the optimisation study, an effective thermal design methodology was demonstrated for an actively cooled thin composite plate. The numerical model, optimisation findings, and overall design methodology are trivially extensible to a structure with more complex geometry and vasculature topology such as a cooled composite compressor blade in a gas turbine engine.

References

[1] SE 70 low temperature cure high toughness epoxy prepreg system, datasheet level 1, PDS-SE70-0314, http://www.gurit.com, 2014.

[2] Hexply 8552 epoxy matrix (180°C/356°F curing matrix) product data, FTA 072e, http://www.hexcel.com, 2013.

[3] Smith, G.G. *Gas Turbines and Jet Propulsion for Aircraft, 4th Ed.* Aircraft Books Inc., 1946.

[4] Eckert, E.R.G. Transpiration and film cooling. *Heat-Transfer Symposium, 195-210*, 1952.

[5] Treager, I.E. *Aircraft and Gas Turbine Engine Technology, 3rd Ed.* Glencoe/McGraw-Hill, 1995.

[6] Lyall, M.E. and Williams, A.D. and Arrit, B.S. and Taft, B.S. Experimental analysis of a biologically inspired thermal-structural satellite panel. *49th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference, 7-10 April, Schaumburg, IL*, 2008.

[7] Williams, A.D. and Underwood, R.L. and Arritt, B.J. and Busch, G.T. and Taft, B.S. Biologically inspired multifunctional composite panel with integrated thermal control. *51st AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference, Orlando, FL*, 2010.

[8] Kozola, B.D. and Shipton, V.K. and Christensen, K.T. Characterization of active cooling and flow distribution in microvascular polymer. *Journal of Intelligent Material Systems and Structures*, **0**, 1-10, 2010.

[9] Pierce, M.R. Microvascular heat transfer analysis in carbon fiber composite materials. *Master's Thesis, University of Dayton*, 2010.

[10] Phillips, D.M. and Pierce, M.R. and Baur, J.W. Mechanical and thermal analysis of microvascular networks in structural composite panels. *Composites Part A*, **42**, 1609-1619, 2011.

[11] Blasius, H. *Grenzschichten in Flussigkeiten mit kleiner Reibung*, *Z. Math Physik*, **56**, 1-37, 1908.

[12] Cushman-Roisin, B. *Environmental Fluid Mechanics*, John Wiley and Sons, 2014.

[13] Huang, C.-Y. and Trask, R.S. and Bond, I.P. Characterization and analysis of carbon fibre-reinforced polymer composite laminates with embedded circular vasculature. *Journal of The Royal Society Interface*, **7(49)**, 1229-1241, 2010.

[14] CES EduPack software, Granta Design Limited, Cambridge, UK. http://www.grantadesign.com, 2009.

[15] Yuret, D. and De La Maza, M. Dynamic hill climbing: Overcoming the limitations of optimization techniques. *The Second Turkish Symposium on Artificial Intelligence and Neural Networks*, 208-212, 1993.

[16] Shahpar, S. SOFT. Rolls-Royce, Derby, UK, 2013.