Rapid design, analysis and optimisation of cooled Turbine blades

T. Robert Kingston
Technical Lead for Turbine Cooling Methods
Rolls-Royce plc
PO Box 3, Filton, Bristol
BS34 7QE, United Kingdom
Email: Robert.kingston@rolls-royce.com

ABSTRACT

The design and analysis of a cooled Turbine blade involves handling of complex geometry together with the integration of computational tools for various disciplines, that cover everything from aerodynamics and cooling, through to prediction of unit and life cycle cost. As technologies mature, incremental improvements become ever harder to achieve, and this is coupled with a requirement to decrease the life cycle cost of components against a backdrop of reducing project timescales. The challenge is one of ensuring that the concept is firstly placed within a ‘fruitful’ design space, and this is facilitated by having appropriate fidelity (and affordable) modelling at all stages of the design.

In this paper we will concentrate on the cooling part of the process and will describe the hierarchical models available, covering preliminary, 2D and 3D analysis.

NOMENCLATURE

\( h \) Heat-transfer coefficient (HTC)
\( Nu \) Nusselt number
\( Re \) Reynolds Number
\( T_g \) Gas recovery temperature
\( T_{\text{ad},w} \) Adiabatic wall temperature
\( T_w \) Wall temperature
\( \bar{\varepsilon}_f \) Laterally averaged film effectiveness
\( m \) Mass-flow
\( A \) Cross-sectional area
\( A_s \) Surface area
\( d_h \) Hydraulic diameter
\( L \) Average surface length
\( f \) Factor to estimate mean effectiveness
\( Bi \) Biot number
\( t \) Thickness

\( k \) Conductivity

Subscripts

\( o \) Coating
\( c \) Coolant
\( g \) Gas
\( m \) Metal
\( p \) Plan
\( BC \) Bond coat
\( surf \) Surface
\( \text{mean} \) Mean

INTRODUCTION

Even as far back as 2007 full 3D Monte-Carlo thermal analysis of a film cooled Turbine blade was being conducted (Moekel et al). This assessed how the variance in manufactured film cooling sizes affected the life of the component. It placed the emphasis on affordable calculations, with the prime intention of exploring a wide design envelope. Modelling of a heavily film cooled High Pressure Turbine blade still remains challenging; in that not only is it geometrically complicated but it also has difficult modelling challenges to face up to. Any design process has to address the conflicting requirements of higher fidelity modelling with the need to reduce the elapsed design time. A prerequisite is to have appropriate design and analysis tools available, that allow rapid investigations of the design space. Early preliminary tools are able to explore a wide range of engine architectures, and cooling concepts, at negligible analysis cost even before the blading is designed. 2D tools allow detailed assessment of actual profiles, and this is followed by very detailed 3D analysis.

Here we adopt a segregated approach for detailed 2D and 3D thermal analysis, whereby internal and external boundary conditions are exchanged with a FEA for the component. It is desirable to have a simple way of modelling the internal flow and here we use a simple 1D flow solver INCA (INternal...
Cooling Analysis) for design and analysis. Such tools are quick, flexible and can easily be ‘tuned’, based on previous experience, relying on correlations for friction loss and heat-transfer. The internal cooling flow, however, can be complex and is, in certain regions, highly three dimensional. This flow is most easily understood by application of CFD and this constitutes a very important part of the overall design process, especially when combined with the 1D flow calculation.

Design Process

Figure 1 shows the design cycle for Turbine Blade design, and we will focus on the ‘Thermal Analysis’ module.

The thermal analysis requires links to other disciplines (Aerodynamics, Mechanical and Lifing) and suitable data ‘handshakes’ need to be place, to allow smooth transfer of data between each. In this design loop the initial 3D geometry will have been created following extensive preliminary and 2D sectional analysis i.e. by the time we are in the 3D arena we are confident that we have a viable design which we seek to optimise across all disciplines. In this design loop we establish a workflow which is completely scriptable and links all of the disciplines in turn.

The design and analysis process utilises both 1D (‘prelim’), 2D and 3D tools. We start by importing the results of the ‘through-flow axisymmetric’ calculation, which is the basic pre-profile design programme for laying out the engine architecture. Radial profiles of aerodynamic quantities are stored on a series of ‘quasi-orthogonal’ planes throughout the engine, and we will be concentrating on one or more of the blade rows from a cooling design perspective. At all levels of fidelity we set up the model based on a particular design point and can, if appropriate, scale the boundary conditions to other operating points.

Figure 2 shows part of the annulus diagram together with a surfaced representation of the HP blade aerofoil, which sits within the design ‘aero-block’.
• The internal conditions come from a flow network solver (INCA) which is closely coupled to the FEA.

The 2D internal cooling geometry definition can be exported and used to ‘seed’ the geometry construction in 3D. The 3D aerofoil surface will initially have used the same aerodynamic profile (surface built up from stacked streamline sections) as used for the 2D. Subsequently this aerofoil surface will be optimised in 3D and the external BC’s provided by detailed CFD analysis.

Here, at all levels of fidelity, we adopt a segregated approach to the thermal analysis, in that we apply external and internal boundary conditions to either a simple ‘slab’ (1D) or FEA model (2D or 3D). These BC’s are updated during the analysis as they are generally functions of the predicted wall temperature. We provide coolant side $h_c, T_c$ and gas side $h_g, T_{adw}$ conditions. Due to the fact that the coolant is confined, the coolant temperature will be affected by the wall temperature (as well as due to rotation). Figure 3 shows a diagrammatic representation of the interfaces.

The following table summarises the preferred approach at each level of modelling fidelity.

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Internal</th>
<th>External</th>
<th>Film effectiveness</th>
<th>Solid</th>
<th>Compute Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>1D</td>
<td>User specified Aplan, Acfoil</td>
<td>Simple energy equation</td>
<td>Scaled HTC</td>
<td>‘Film coverage’ scaled</td>
<td>ms</td>
</tr>
<tr>
<td>2D</td>
<td>Sectional geometry INCA Boundary Layer solver</td>
<td>Laterally averaged Correlations applied on 2D sections</td>
<td>INCA CFD</td>
<td>As for 2D</td>
<td>2D FEA 5-30s</td>
</tr>
<tr>
<td>3D</td>
<td>Full 3D Geometry INCA CFD</td>
<td></td>
<td>CFD</td>
<td>As for 2D</td>
<td>3D FEA 10-30min</td>
</tr>
</tbody>
</table>

Prelim Analysis

Figure 4 shows a simple boiler-plate (slab) representation of a cooling system. Whilst it might appear that such a simple representation had very little to offer, it does respond appropriately to changes in the applied boundary conditions and can be ‘tuned’ to match the performance of a real cooled design. As well as being useful it is very quick to run a wide range of design studies. Whilst it may in principle be possible to derive minimum and maximum temperatures from such a simple analysis, it is far better to do a more detailed 2D analysis for such details.
Consider a slab of thickness $t_m$ which may have a thermal barrier coating (TBC) of thickness $t_o$. The ‘hot’ side has wetted area $A_{sg}$ and the ‘cold’ side $A_{sc}$. In order to predict a family of temperatures (mean, BC, TBC) for the slab, we require estimates of coolant side $h_c, T_c$ and gas side $h_g, T_{adv}$ conditions.

![Figure 4. Slab representation of a Cooling system](image)

The local thermal effectiveness of the system $\varepsilon$ is given by the following expressions.

$$\varepsilon_{surf} = \frac{T_{adv} - T_{b,surf}}{T_{adv} - T_c} = \frac{1}{Bi_{tot}}$$

$$\varepsilon_{BC} = \frac{T_{adv} - T_{b,BC}}{T_{adv} - T_c} = \frac{1 + Bi_o}{Bi_{tot}}$$

$$\varepsilon_{mean} = \frac{T_{adv} - T_{b,mean}}{T_{adv} - T_c} = \frac{1 + Bi_o + f.Bi_{m}}{Bi_{tot}}$$

Each layer of the slab is characterised by average Biot numbers

$$Bi_o = \frac{h_o t_o}{k_o}, \quad Bi_m = \frac{h_m t_m}{k_m}$$

$$Bi_{tot} = 1 + Bi_o + Bi_m + \frac{h_g A_{sg}}{h_c A_{sc}}$$

If we had a simple slab, with 1D conduction, then the mean temperature would be the average of the inner and BC temperature, in which case the factor $f$ would assume a value of 0.5. For more complicated configurations, representative of real cooling systems, then this factor needs to be established from detailed analysis. Say we have a 2D sectional model, or a sectioned 3D FEA model, then we can post-process the family of predicted temperatures e.g. surface, BC, mean and inner, and formulate the effectiveness for each on every section. We will assume that the factor $f$ is unique for this style of system (at any sectional height). We extract it, together with the radial variation of $Bi_{m}$, from the analysis by noting that

$$\varepsilon_{mean} / \varepsilon_{BC} = \frac{1 + Bi_o + f.Bi_{m}}{1 + Bi_o}$$

We can then derive the ‘effective’ wall thickness of the component, at any particular height. We are now in a position to scale to a new set of boundary conditions, or indeed analyse a new design that features a similar style of cooling system to this one. Thus the approach relies on the fact that at least one detailed analysis of any particular design style has been completed and post-processed. The data reduction from a detailed analysis is a fully automated process that produces a file, containing the important features of the design, which we will refer to as a ‘Salient Features Sheet’ (SFS) file. It is this SFS file of the ‘datum’ component that is subsequently used for the prelim analysis of any ‘study’ design.

Say, for a ‘study’ design a target plan area $A_p$, for any planar section of the design, together with the total coolant flow area $A_{c,tot}$ (on that section), then the load carrying area $A_m$ is given by

$$A_m = A_p - A_{c,tot}$$

This allows estimates of the hydraulic diameter $d_h$, wetted area $A_{w}$ and cross-sectional area $A_c$ to be made. As will be seen we actually require ratios of the geometric parameters between the study and the datum design. Typically, as part of a design study, we will be both investigating the architectural layout of the engine (and hence the aero-block boundary conditions), as well as the taper and cross-section...
of the blade for a range of different cooling schemes.

To predict a family of temperatures (mean, BC, TBC) for the ‘study’ component we require estimates of coolant side $h_{c1}, A_{c1}, T_c$ and hot gas side $h_{w1}, A_{w1}, T_w$ conditions. The HTC boundary conditions will be scaled based on the Reynolds number ratio between those at the ‘study’ and ‘datum’ conditions. Internally we start by initially assuming a required coolant mass flow rate $\dot{m}_c$ from which we estimate the mean Reynolds number $\overline{Re}_c$

$$\overline{Re}_c = \frac{\dot{m}_c d_h}{A_c \mu_c}$$

If we make the assumption that we can estimate the internal cooling heat-transfer coefficient from the internal Reynolds number via Nusselt’s equation, then

$$\overline{h}_c = a \overline{Re}_c^b Pr_c^c k_c$$

The coefficients ‘a’ and ‘b’ depend on the particular features present within the cooling system. These values are mined out of the detailed analysis and stored in the SFS file. We assume that the datum (1) and study (2) cooling systems are ‘similar’ both in the shape (and aspect ratio) of the passages and in terms of their cooling features i.e. the ‘blockage’ of ribs (height and pitch normalised by passage hydraulic diameter). Now we can scale the HTC from the datum design

$$\frac{h_{c2}}{h_{c1}} = \left(\frac{\overline{Re}_{c2}}{\overline{Re}_{c1}}\right)^b \frac{k_{c2}}{k_{c1}} \frac{d_{h1}}{d_{h2}}$$

The ratio of hydraulic diameters comes from the relative size of the cooling passages, assuming that the aspect ratio of the passages is similar.

The internal cooling system is confined and a simple energy equation needs to be solved to update the local coolant temperature $T_c$ at each section. On the hot (gas side) again it is possible to scale from the datum conditions (here assuming turbulent flow)

$$\frac{Nu_{g2}}{Nu_{g1}} = \left(\frac{Re_{g2}}{Re_{g1}}\right)^{0.8}$$

Here the Reynolds number is based on the throat mass-flow $\dot{m}_g$ and throat-area $A_g,t$ and the appropriate length scale, that goes both into the Reynolds and Nusselt numbers is the average suction and pressure side aerofoil length $L$.

$$Re_g = \frac{\dot{m}_g L}{A_g,t \mu_g}$$

$$\frac{h_{g2}}{h_{g1}} = \left(\frac{Re_{g2}}{Re_{g1}}\right)^{0.8} \frac{k_{g2}}{k_{g1}} \frac{L_2}{L_1}$$

The wall thickness of the study design is either specified or scaled from the datum component. The Biot numbers of the metal and coating (if present) are then evaluated based on the predicted mean temperature of the layer (as the thermal conductivity generally varies with temperature). The analysis requires several iterations to reach convergence for a given specified coolant mass-flow rate. The easiest way to undertake trade studies (for a particular aero-block set of boundary conditions) is to simply run a range of plan and coolant flow areas, varying the specified coolant mass-flows, and for each obtain the radial family of temperatures (mean, BC, TBC).

2D analysis

Once we have completed prelim analysis of various cooling design styles, and down selected some viable concepts, then we can move into the 2D arena. Here we have highly automated geometry tools to rapidly create fully featured sectional geometry. The internal flow is predicted using a code called INCA (INternal Cooling Analysis) which also controls the setting of the boundary conditions on to the FEA analysis. The creation, and update, of the 1D flow network is done automatically as the cooling geometry is specified or changed. The sections are tagged, meshed and BC applied automatically suitable for FE analysis. The same tools are used as part of the 3D thermal analysis.

Figure 5 shows a typical flow network (or ‘flow-net’) for a Turbine blade together with the 2D FEA planar sections of the cooling system, consisting
of a “multi-pass” cooling system, shown with film cooling holes and a diagrammatic representation of ribs.

Figure 5. INCA flow-net located with a stack of 2D FEA sections

The flow-net can be manipulated (rotated, zoomed or panned) either on its own or, preferably, in combination with a 2D or 3D FE model. It can be interactively interrogated e.g. clicking on a duct will bring up a panel containing the data that defines that duct, together with the features associated to the duct, and their aerodynamic boundary conditions, whilst clicking on a film-cooling hole will bring up the data for that film row. Shown on the flow-net are ducts (in dark blue), films (green), ribs (light blue, shown pitched at the appropriate distance) and loss features (light blue net).

For INCA a novel pressure correction CFD based approach is adopted which utilises a background mesh that is independent of the ‘associated’ features and represents each and every film cooling hole individually. This approach proves to offer significant improvements in terms of flexibility and speed. The correlations for ribs will normally yield a step function for laterally averaged HTC $\dot{h}$, i.e. the HTC is engineered to be higher on the pressure and suction inner walls, rather than on the webs. Whilst it is possible to include the results of internal CFD into the 2D analysis it is most usually incorporated into the 3D analysis.

The external boundary conditions most usually come from running separate 2D un-filmed boundary layer analyses on each stream-line section. In principle we could map 3D CFD derived BC’s but this is most usually reserved for 3D analysis. The film exit temperatures are used to compute, in combination with a gas recovery temperature distribution and laterally averaged film effectiveness correlations, the local adiabatic wall temperature distribution. The isolated film effectiveness is characterised through both geometric $\frac{x}{d}, \frac{p}{d}, \alpha, \theta$ (i.e. it varies with distance $x$ downstream of the hole) and aerodynamic attributes $B, I$ (blowing rate, momentum flux ratio) of the film row

$$\bar{\varepsilon}_f = fn\left(\frac{x}{d}, \frac{p}{d}, \alpha, \theta, B, I\right)$$

The flow network is solved on demand every thermal time step of the FE analysis, whereby the HTC and coolant temperatures are applied to the FE model and wall temperatures returned. Typical total analysis times are the order of a few seconds. This predicted temperature is then used for rapid mechanical and lifing analysis.

The next step, once a satisfactory design has been achieved, is to start detailed 3D analysis.

3D Analysis

Key to any design system is having a flexible and robust geometry engine and here we use Siemens NX. Most usually for a Turbine blade the initial design and FE analysis will have been undertaken using a series of stacked 2D sections, and this can then be used to ‘seed’ the 3D geometry definition. The cooling core is built in a modular way, based on wall offsets and a camber surface, aided by a variation based on the 2D sectional definition planar cuts – figure 6.
This de-featured core, once removed from the blade solid, and meshed allows a FEA to be conducted. Having the explicit geometry of low blockage ribs is not strictly essential for the purposes of FEA, neither is having explicit CAD representation of the film cooling. However, both of these features must be represented for internal CFD analysis.

The faces of the geometry, when created, are ‘tagged’ for automatic boundary condition application. The next step is to create a 1D flow network (‘flow-net’), which requires coordinates of the duct centre-line, together with a distribution of de-featured flow area, perimeter and aspect ratio perpendicular to the flow direction. As the core solid is built the pertinent 1D geometric information is written out to create the input data for separate INCA ducts. Figure 7 shows the flow-net located within the 3D solid and, as will be seen later, this 1D flow-net will be used to interact directly with the FEA analysis.

It is now necessary to fully complete the flow network to reflect all cooling decorations present in the system. As features are produced in NX a representation is also added to the 1D flow-net so that, at all times, both the 3D geometry and the 1D flow-net representation of it are in step. Of course this simple network solver approach is no substitute for detailed three-dimensional internal CFD analysis of the system, but in terms of speed and flexibility, certainly in routine design work, then there is no comparison between the two!

Typical run times for INCA for a realistic cooling scheme is of the order of tens of seconds. The challenge is one of maintaining the advantages of a simple 1D network but with the possibility to incorporate three-dimensional effects, as predicted by CFD, into the analysis in a practical way. For the purposes of internal CFD we need a fully featured CAD model and this requires explicit ribs and film cooling holes to be added to the core solid, and we have developed NX utilities for both.

**FEA Analysis**

The FE solver for both thermal and structural analysis is a Rolls-Royce proprietary code SC03 and this is used for both 2D stacked sectional as well as 3D analysis. The NX geometry is imported into SC03 and meshed adequately for lifing assessment, and this mesh is then used for thermal analysis as well. We can also adopt some de-featuring of the FE model for analysis simplicity. As the NX model is ‘tagged’, boundary conditions can be automatically applied via scripts. As we are conducting a segregated analysis these boundary conditions are usually of the ‘Convection Zone’ type i.e. a spatial distribution of heat-transfer coefficient and ‘gas’ temperature are applied to internal and external faces of the model. We have distributions of hot ‘gas’ side heat-transfer coefficient (HTC) $h_\text{e}(x,y,z)$ and adiabatic wall temperature $T_{\text{ad},e}(x,y,z)$, together with cold ‘coolant’ side HTC $h_c(x,y,z)$ and coolant temperature (taken as a bulk mean coolant total relative temperature) $T_c(x,y,z)$. These boundary conditions can be derived and mapped from CFD solutions for internal and external or from correlation, or most usually a mix of both.

Figure 8 shows a sectioned 3D FE model with the INCA flow-net (ducts and films) visible.
The thermal FEA proceeds in a time-marched iterative manner (for both steady-state and transient analysis) and at the end of each step a full run of INCA is triggered and boundary conditions exchanged between it and the FEA. Typically, for a steady state analysis, we might have ten such iterations. As the calculation proceeds, the INCA model will update the coolant temperature. At convergence the analysis of both the FE and INCA calculations will be fully converged and in step.

Krueckels et al (2007) showed the advantages of de-featuring the analysis geometry for thermal assessment, in order to permit design flexibility and to allow rapid iteration. In the ‘Volumetric Heat Sink Modelling’ (VHS) approach heat was removed from the FEA model (which has no explicit CAD faces corresponding to the holes), based on the FE mesh, in a spatially varying manner dependent on the proximity to a particular film cooling hole. Here we develop the idea further with a far tighter integration of the INCA code into the SC03 FE code.

Figure 9 shows a simplistic cooling system for an aerofoil - shown as a flow-net, featuring a number of film rows. Also shown are two FEA solutions, firstly without a VHS representation of the film holes and then with VHS applied.

The first requirement for successful application of this approach is to ensure that the FE mesh is fine enough so that we can resolve the film-cooling holes. Domains of the FEA that contain any sinks are flagged, and for all FE volume integration points in these domains a once only vector calculation is performed to see whether the integration point lies ‘within’ any sink hole or not. For the purposes of this exercise each film hole is treated as if it were cylindrical. If the point does lie within a film hole, then both the hole number and fractional length along the hole are stored. To capture more integration points in each hole, a user specified ‘catch diameter’ can be specified, so that a wider search for points is conducted. This approach is widely adopted for routine 3D FEA analysis using SC03 and proves to be advantageous in terms of film optimisation, as the underlying geometry does not need to be updated.

**External CFD**

We use a Rolls-Royce proprietary code HYDRA and this is either run as part of a multi-row or single row calculation, where we typically use a RANS k-omega model. The analysis is run without film cooling on the blade row of interest, but with all other leakages modelled (including disc rim, inter-platform gap and over-tip leakage). This is necessary in order to compute the gas recovery temperature. The CFD mesh will typically be of the order of 10M nodes for a non-film cooled analysis.

We use a series of CFD calculations to extract the external HTC $h_f$ at every surface node. Often the aerodynamic design point (e.g. Cruise) may be different than the Cooling/Lifing condition(s) (e.g. MTO). Either the CFD will be run at individual lifing points or the data can be scaled from one design point to others. For high Reynolds numbers on typical HP blades, especially if film cooled, then running the CFD fully turbulent allows scaling of the HTC based on Nusselt number. This can be
done in the same way as was described for prelim
analysis. The inclusion of films via laterally
averaged film effectiveness correlations follows
the approach adopted in 2D.

**Internal CFD**

The application of CFD to the design and analysis
of internal cooling systems has been used in Rolls-
Royce for many years. It provides a means of
understanding the internal flow structure and to
de-risk the design – in that you need to ensure
that you can get coolant to all parts of the cooling
system and avoid flow starvation. The internal CFD
can further be used to quantify the levels, and
distribution, of heat-transfer within the cooling
system.

Rolls-Royce together with the University of Ox
ford have for many years used large scale static Perspex
models of both stylised and engine representative
gEometries to derive heat-transfer coefficient data
using the Liquid Crystal technique achieving full
non-dimensional similarity to engine conditions –
see Jackson and Ireland (2009). Figures 10(a) -
taken from that paper - show the experimental
setup for the investigation of the flow and heat-
transfer in model of a multi-pass cooling system.
Films are collected into separate plena and the
mass-flow for each metered. The use of the ‘liquid
crystal’ technique allows internal surface HTC
distributions to be derived, which have an
accuracy of typically 8%.

![Figure 10(a) Experimental setup for static PERSPEX test of blade cooling system](image)

**Figure 10(a) Experimental setup for static PERSPEX test of blade cooling system**

![Figure 10(b) Comparison of experimental (left) HTC levels (normalised in the range 0 to 1) against a FLUENT RANS prediction (right).](image)

**Figure 10(b) Comparison of experimental (left) HTC levels (normalised in the range 0 to 1) against a FLUENT RANS prediction (right).**

Whilst the shape of the predicted variation of HTC
number looks reasonable compared to the
experimental results, at least in distribution, the
mean levels are somewhat different. There may be
several reasons for this; firstly the mesh may not
be suitable in terms of structure or fidelity,
secondly the CFD solver may not be fully suitable
for this class of flow. Further work on ribbed
passages of low aspect ratio, under the effects of
rotation, was reported by Pearce, Romero (2015)
and Dhopade, Capone et al (2015) and showed
shortcomings in the ability of RANS based solvers
to model flows in low aspect ratio configurations.
In Tucker et al (2013) from The Whittle Lab,
Cambridge the use of LES for this (and other
problem types) is discussed. It is clear that more
work is needed before a wholly reliable CFD
approach is available for internal flows.

In Krueckels (2007) CFD derived heat-transfer data
was applied to the internal faces of a FEA model.
For ribbed passages the CFD calculation was
performed on a section of passage featuring a
single rib with repeating conditions applied on the
top and bottom faces of the CFD model. This
approach was referred to as ‘RIBCOR’ and was
used to assess the circumferential variation of
heat-transfer coefficient around the perimeter of
the cooling passage. In this way the effects of
rotation and Reynolds number could be
introduced simply into the model i.e. the CFD is
acting as a correlation generator, both in terms of
mean level and circumferential variation, and this
can be used directly in the analysis i.e. the spatial
variation is applied as a boundary condition to the FEA analysis. Nouri (2013) following validation against static experimental data, also used CFD as a ‘correlation generator’ for the effects of rotation in ribbed passages of various aspect ratios and at different passage orientations.

In the SC03/JB56 modelling we use the post-processed HTC from the internal CFD prediction $h(x, y, z)_\text{CFD}$ as an internal boundary condition. The CFD HTC is mapped to the FE surface integration points $h(x, y, z)_\text{FE}$, but is then scaled so that, at any duct position, the mean value from CFD $\bar{h}_{\text{CFD}}$, matches that from correlation $\bar{h}_{\text{COR}}$, and we preserve the underlying ‘shape’ of the distribution.

$$h'(x, y, z)_\text{FE} = h(x, y, z)_\text{FE} \frac{\bar{h}_{\text{COR}}}{\bar{h}_{\text{CFD}}}$$

Remember we are primarily using the CFD to look at flow structure and the distribution of HTC. We can use INCA to predict (and tune) the pressure loss through the blade, together with control of film flows via discharge coefficients, rather than rely on the CFD predictions directly.

CONCLUSIONS

A hierarchy of design and analysis tools, from prelim through to detailed 2D and 3D, allow for rapid design exploration. Tight integration at all levels of fidelity means that fruitful areas of the design space can be identified and investigated rapidly.

To improve the quality of the design it has become essential to incorporate improved physics modelling but this can potentially slow the design process, which initially appears at odds with the requirement to reduce design time frames. We have highlighted the importance of having a robust and flexible geometry engine that can produce either clean or fully featured geometries, and we have identified when and where the use of a de-featured model can significantly increase both the flexibility of the modelling and the analysis speed. We have outlined how a segregated thermal analysis of a cooled Turbine blade can be conducted; featuring mapped internal and external CFD to a 3D FE model, linked to a 1D flow network solver. This segregated approach provides a flexible system for rapid design work.

The integration of tools across a range of disciplines permits rapid and productive exploration of the design space, rather than unnecessarily getting too immersed in exploring the nuances of just a single discipline.

ACKNOWLEDGMENTS

I would like to thank Rolls-Royce plc for their support of this work. I would also like to thank my colleagues at Rolls-Royce; in particular Chris Bradley, Dave Bryant and Timm Janetzke.

REFERENCES


Pearce, R., Romero, E. “Computational and Experimental Study of Heat Transfer in Rotating Ribbed Radial Turbine Cooling Passages”. ASME paper GT2015-43158
