4.2.1
Cooling Design Analysis

Ron S. Bunker

GE Global Research
One Research Circle, K-1 ES-104
Niskayuna, NY 12309

phone: (518) 387-5086
email: bunker@crd.ge.com

4.2.1-1 Introduction

The technology of cooling gas turbine components via internal convective flows of single-phase gases has developed over the years from simple smooth cooling passages to very complex geometries involving many differing surfaces, architectures, and fluid-surface interactions. The fundamental aim of this technology area is to obtain the highest overall cooling effectiveness with the lowest possible penalty on the thermodynamic cycle performance. As a thermodynamic Brayton cycle, the efficiency of the gas turbine engine can be raised substantially by increasing the firing temperature of the turbine. Modern gas turbine systems are fired at temperatures in excess of the material melting temperature limits. This is made possible by utilization of thermal barrier coating materials and by the aggressive cooling of the hot gas path (HGP) components using a portion of the compressor discharge air, as depicted in the aero-engine schematic of figure 1. The use of 20 to 30% of this compressed air to cool the high-pressure turbine (HPT) presents a severe penalty on the thermodynamic efficiency unless the firing temperature is sufficiently high for the gains to outweigh the losses. In all properly operating cooled turbine systems, the efficiency gain is significant enough to justify the added complexity and cost of the cooling technologies employed.

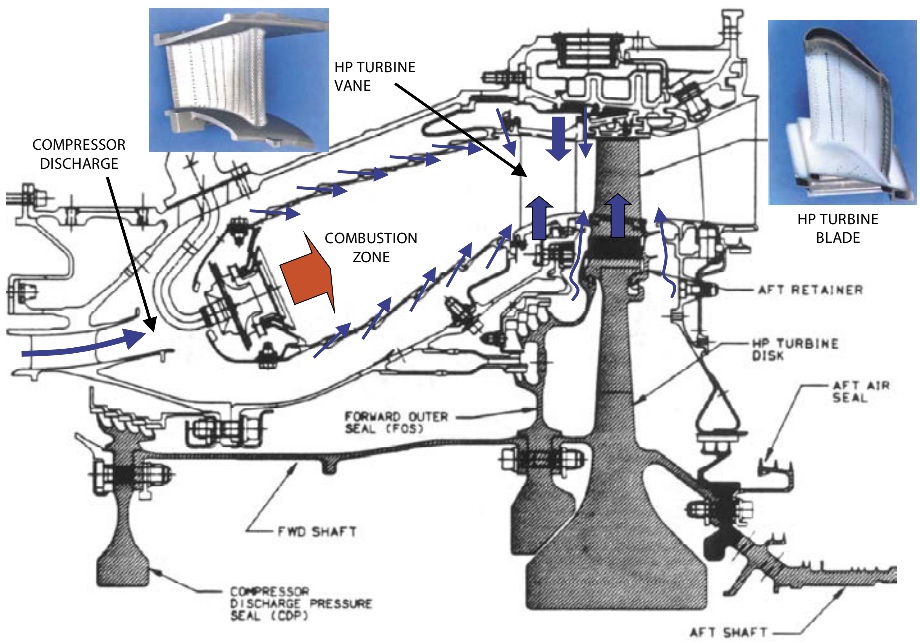


Fig. 1. Aero-engine High Pressure Turbine and Combustor

Cooling technology, as applied to gas turbine components such as the high-pressure turbine vanes and blades (also known as nozzles and buckets), is composed of five main elements: (1) internal convective cooling, (2) external surface film cooling, (3) materials selection, (4) thermal-mechanical design, and (5) selection and/or pre-treatment of the coolant fluid. Internal convective cooling is the art of directing coolant via the available pressure gradients into all regions of the component requiring cooling, while augmenting the heat transfer coefficients as necessary to obtain distributed and reasonably uniform thermal conditions. The enhancement of internal convective flow surfaces for the augmentation of heat transfer has occurred through a myriad of surface treatments and features as well as the forceful direction of flows via diverters, swirl devices, etc. The most common turbine airfoil interior surface features have been rib-rougheners or turbulators, and also pin-banks or pin-fins, which continue to play a large role in today's turbine cooling designs. Film cooling is the practice of bleeding internal cooling flows onto the exterior skin of the components to provide a heat flux reducing cooling layer. Film cooling is intimately tied to the internal cooling technique used in that the local internal flow details will influence the flow characteristics and temperature of the film jets injected on the surface. Materials most commonly employed in cooled parts include high-temperature, high-strength nickel or cobalt-based superalloys coated with yttria-stabilized zirconia oxide ceramics (thermal barrier coating, TBC). The protective ceramic coatings are currently used to actively enhance the cooling capability of the internal convection mechanisms. The thermal-mechanical design of the components must integrate these first three elements into a package that has acceptable thermal stresses, coating strains, oxidation limits, creep-rupture properties, and aero-mechanical response. Under the majority of practical system constraints, this allows for the highest achievable internal convective heat transfer coefficients with the lowest achievable frictional coefficient or pressure loss. In some circumstances, pressure loss is not a concern and the highest available heat transfer enhancements are sought for cooling, while in other applications pressure loss may be so restricted as to dictate a very limited means of heat transfer enhancement. The last cooling design element concerns the correct selection of the cooling fluid to perform the required function with the least impact on the cycle efficiency. This usually is achieved through the use of compressor bleed air from the most advantageous stage of the compressor, but can also be done using off-board cooling sources such as closed-circuit steam or air, as well as intra-cycle and inter-cycle heat exchangers.

In many respects, the evolution of gas turbine internal cooling technologies began in parallel with heat exchanger and fluid processing techniques, "simply" packaged into the constrained designs required of turbine airfoils (ie. aerodynamics, mechanical strength, vibrational response, etc.). Turbine airfoils are after all merely highly specialized and complex heat exchangers that release the cold side fluid in a controlled fashion to maximize work extraction. Actively or passively cooled regions of the hot gas path in both aircraft engine and power generating gas turbines include the stationary vanes or nozzles, the rotating blades or buckets of the HPT stages, the shrouds bounding the rotating blades, and the combustor liners and flame holding segments. Also included are the secondary flow circuits of the turbine wheelspaces and the outer casings that serve as both cooling and positive purge flows. The ever present constraints common to all components and systems include but are not limited to pressure losses, material temperatures, component stresses, geometry and volume, aerodynamics, fouling, and coolant conditions.

An overview of the cooling design analysis system or method is presented in the generic summary diagram of figure 2. For the present purpose, the design analysis method is shown as a three level system, working from Level 1 outwards. Level 1 concerns the conceptual design of the components largely based on nominal target conditions and divorced from the surrounding systems constraints and competing requirements or trade-offs. Level 1 analysis can be performed based on 1D, 2D, or 3D complexities and details, and is primarily used to compare various options in design. Analysis at the conceptual level must still be detailed enough however to allow ranking and down-selection between options. Level 2 cooling analysis is the much more detailed inclusion of surrounding effects and constraints from aerodynamics, material properties, mechanical loads, lifing limitations, clearances etc. as depicted in the design cycle diagram of figure 3. The analyses performed in Level 2 often must be combined thermal-mechanical predictions using very detailed finite element models, sometimes even sub-models of certain component sections. Most Level 2 analyses are performed at one steady-state operating condition, e.g. 100% load. The result of Level 2 analysis, after various alterations and iterations, is the basic system design with balanced choices that satisfy the engine design goals. Level 3 analysis brings in the operational transient aspects to determine if requirements or constraints are violated under conditions such as normal start-up, fast start-up, trips, and hot restarts. Level 3 results can require that additional changes be made with new analyses at Levels 1 and 2.

In all cooling system design analysis levels, engine experience design factors and known engine degradation factors must be included. As examples, such factors may include the use of -3σ material properties, knock-down factors on cooling augmentation, and loss of coatings or metal thickness. In addition, there is a Level 0 analysis not shown in figure 2. Level 0 is the preliminary design of the engine. The preliminary design deals mainly with the mission requirements, such as efficiency, cost-of-electricity, power sizing and number of starts. Level 0 sets the target goals on the cooling system, including the coolant consumption, turbine airfoil life, and inspection intervals.

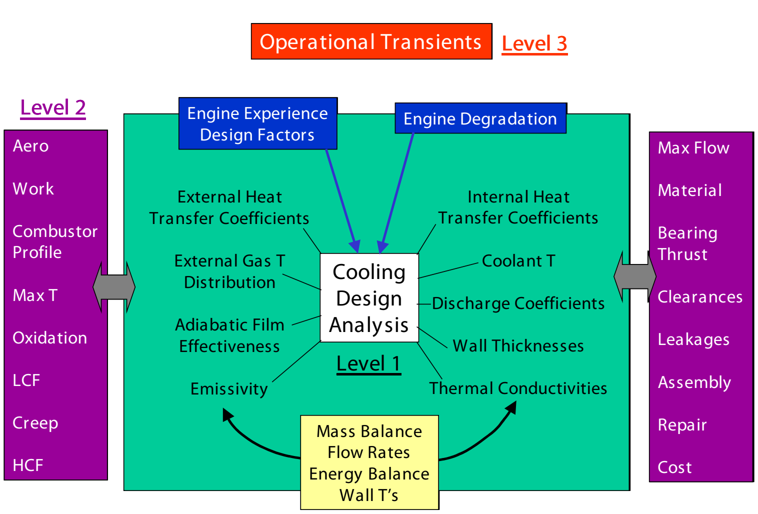


Fig. 2. Cooling Design Analysis System

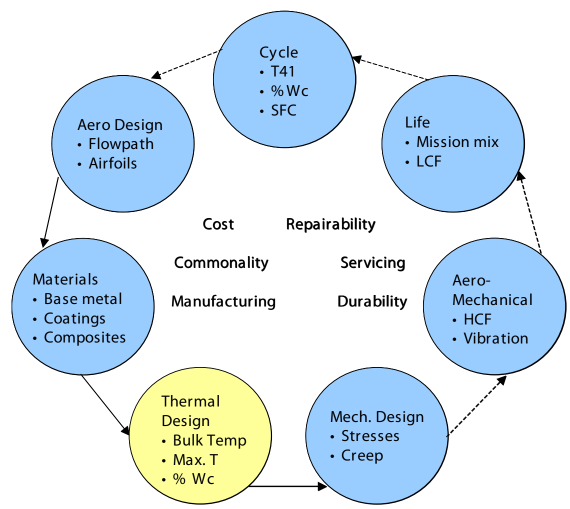


Fig. 3. Turbine Engine Design Cycle

4.2.1-2 Level 0 – Preliminary Cooling Design Analysis

At this very early stage in the definition of an engine design, the cooling system is completely wrapped up in a single set of performance characteristic curves, usually presented in graphical format, known as Cooling Technology Maps. A generic cooling technology performance chart is shown in figure 4 for a turbine airfoil, either a vane or a blade. The technology curves shown on this chart present the gross airfoil cooling effectiveness versus a heat loading parameter, defined as non-dimensional quantities:

Gross Cooling Effectiveness = $(T_{gas} - T_{bulk\ metal}) / (T_{gas} - T_{coolant\ supply})$

Heat Loading Parameter = $(m_{coolant} * C_{p\ coolant}) / 2 * H_{gas} * A_{gas}$

The quantities in these terms are as follows:

T_{gas} = average hot gas temperature (e.g. firing temperature for blade)
$T_{bulk\ metal}$ = average metal temperature of entire airfoil with endwalls
$T_{coolant\ supply}$ = temperature of coolant entering the airfoil
H_{gas} = average external gas heat transfer coefficient (corrected for radiation)
A_{gas} = external gas wetted surface area
$m_{coolant}$ = coolant flow rate to airfoil
$C_{p\ coolant}$ = coolant specific heat.

The heat loading parameter ratios the overall hot gas heat flux (source) delivered to the component against the overall coolant capability to accept heat flux (sink). Since the gas and coolant temperatures are not in this term, the ratio is not unity, but does provide a relative scale for placement of past and current designs. The symbolic points on the chart represent various engine experience data points for different designs. Several curves will generally be present showing major levels of cooling technology. Such maps may also present extrapolated design points based on analysis only, or target design points for new engines.

In this preliminary Level 0 design phase, cooling analysis is simply a matter of looking up the expected or projected coolant flow rates based on the cycle or mission design goals. Temperatures may be altered by various choices of cycles, surface areas by overall power requirements or aerodynamics, coolant specific heat by selection of cooling fluid, airfoil temperatures by cooling mass flow rate, and so forth. All of which lead to differing impacts on overall engine efficiency, emissions, life, and cost.
A similar set of performance curves may be used to examine the effect of wheelspace and casing leakage flows from the secondary cooling circuits. Here, variations may be made in the complexity of seals to obtain lower overall leakages flows with potential consequences such as higher rotor rim material temperatures.

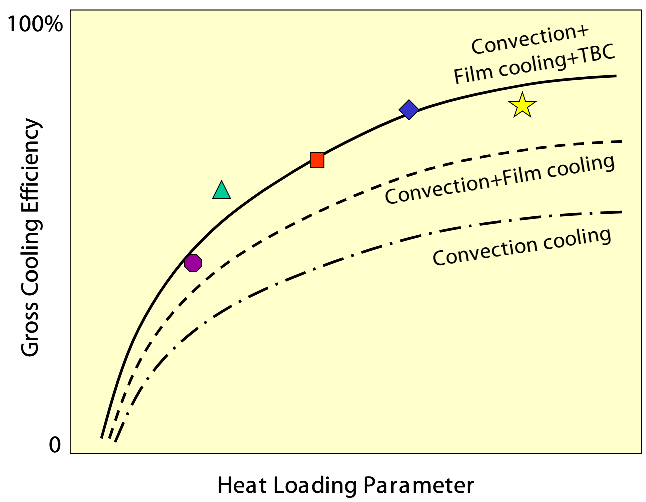


Fig. 4. Cooling Technology Performance Chart

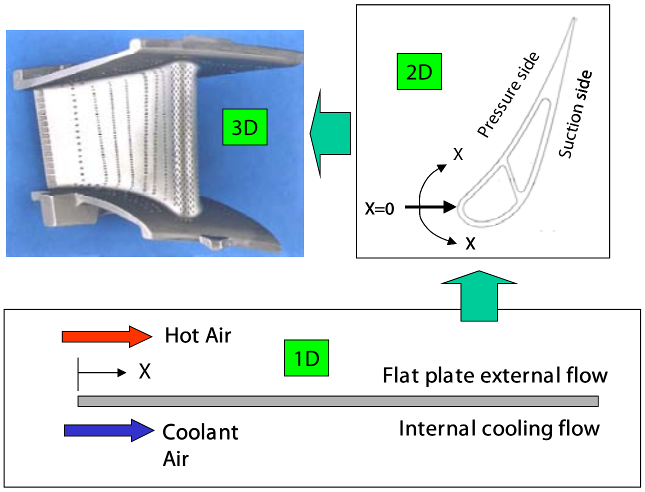


Fig. 5. Simplified to Complex Cooling Design Analysis

4.2.1-3 Level 1 – Conceptual Cooling Design Analysis

Component design may take on one of several depths of analysis, from preliminary estimates, to detailed two-dimensional analyses, to complete three-dimensional computational predictions including the conjugate effects of the convective and radiative environments. Each mode of analysis has its use as the design progresses from concept to reality. Figure 5 shows a three-dimensional vane airfoil and endwalls reduced first to a two-dimensional, constant thickness cross-section of the aerodynamic shape, and then again to a one-dimensional basic flat plate representing flow from the leading edge stagnation point to the trailing edge. Preliminary design uses mostly bulk quantities and one-dimensional simplified equations to arrive at approximate yet meaningful estimates of temperatures and flow requirements. While the actual airfoil / endwall shape involves many complexities of accelerating and decelerating flows, secondary flows, and discrete film injection holes, a good estimate may still be obtained using fundamental flat plate relations. Two-dimensional design incorporates boundary layer analyses, network flow and energy balances, and some thermal gradient estimates to refine the results for local temperature and flow predictions suitable for use in finite element stress modeling. Three-dimensional design may use complete computational fluid dynamics and heat transfer modeling of the internal and external flow fields to obtain the most detailed predictions of local thermal effects and flow losses. Design analyses may of course also mix these methods, such as the use of CFD to predict the hot gas path pressures, velocities, and temperatures for the aerodynamic profile only, while the internal cooling and film cooling are predicted using semi-empirical correlations.

One-Dimensional Analysis – Preliminary Design

The simplest one-dimensional analysis may be best understood as an iterative sequence of several steps leading to an overall model that is approximately optimized for material thicknesses, cooling configuration, and cooling flow. Figure 6 shows the one-dimensional thermal model that applies to any discrete location on the airfoil. These steps include the following:

1. Estimation of the external heat transfer coefficient distribution on the airfoil, which may include effects such as surface roughness and freestream turbulence. This estimate may include thermal radiative heat flux separately, or as part of an effective convective heat transfer coefficient;

2. Calculation of the average adiabatic wall temperature due to film cooling;

3. Calculation of the conductive material thermal resistances, e.g. TBC, bondcoat, and substrate.

4. Estimation of the internal heat transfer coefficients due to cooling;

5. Calculation of the required aggregate cooling flow rate; and

6. Iteration of the solution to achieve target metal temperatures, thermal gradients, material thicknesses, etc., or to comply with target constraints.

The solution is iterative to account for fluid property changes with temperature, both internal and external to the airfoil, as well as temperature rise in the cooling fluid.

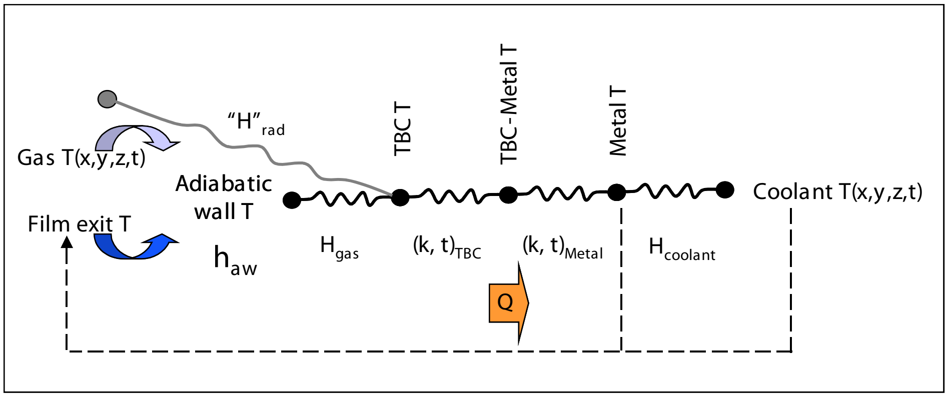


Fig. 6. General Thermal Resistance Model

Two-Dimensional Analysis – Conceptual Design

The simple one-dimensional model is not of much use in conceptual design unless it is knit into a sectional or complete model representing the cooled airfoil. This means applying the simple analysis to many regions of the airfoil (wall elements) making up a 2D sectional view as depicted in figure 5. This is analogous to a finite element model construction, and in many cases can be achieved using a FEM approach. The elements can be disconnected from thermal conduction as a first estimate, or simply connected to include axial conduction effects within the airfoil section. Such conduction effects are more important in regions that are not well modeled by a single wall thickness, like the trailing edge. Taking this a step further, many radial sections of the airfoil may be stacked to form a pseudo-3D model of the nearly complete component (without endwalls, tip, or shank). Again, this can be accomplished with or without complete thermal conduction connections. These are each valid conceptual design modeling approaches with varying levels of accuracy. Note that such approaches do not typically integrate the airfoil and its endwalls, but treat these portions separately by similar analytical means.

One may ask why a FEM approach is not always employed for the conceptual design of cooled airfoils, and also why the airfoil and endwalls are not always integrated into a single component. The answer is the same for both questions, and lies partially in historical design methods and partially in the state-of-the-art computational analysis. Looking at the turbine blade of figure 7, a candidate cooling circuit design can be very complex. In this example, the main portion of the blade is cooled using a turbulated five-pass serpentine circuit, the leading edge is cooled using a radial passage impinging through crossover holes into the concave stagnation region, and the trailing edge is cooled with a radial pin-bank array and aft ejection channels. Film cooling is employed heavily in the leading edge region and tip, with additional rows of film holes on both the pressure and suction sides of the blade. The blade has three distinct cooling circuits isolated in the shank cooling supply. This blade design, and for that matter any other, must be analyzed and modified with the following in mind:

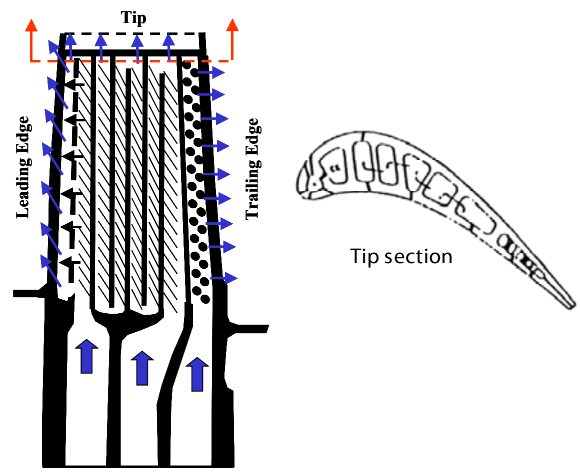


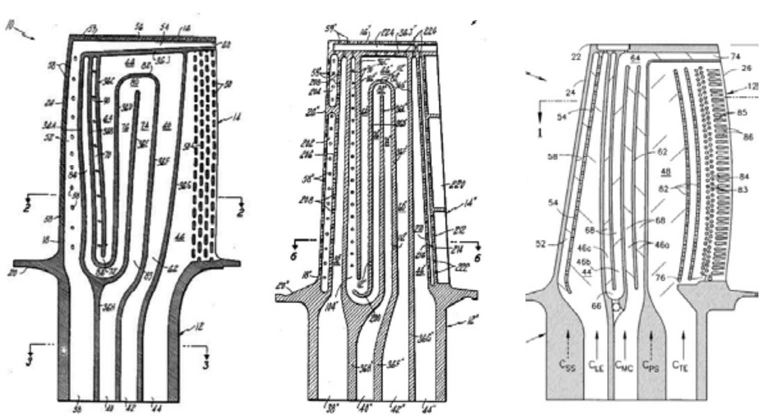
Fig. 7. Cooled HPT Blade (Bucket)

- Typical internal cooling technologies including turbulators, pin-fins, turns, impingement jets, trailing edge holes, swirl cooling, vortex cooling, convoluted passages, tip purge holes, and basic number and sizing of passages must be readily (i.e. easily) manipulated to investigate design options and their effects on performance. Manipulation includes movement to new locations, change of size, change of number or spacing, addition to and subtraction from the component. Performance evaluation usually refers to cooling effectiveness and aerodynamic mixing losses at this stage of analysis.
- Film cooling holes and rows of holes need to be readily moved or altered in the design, including film hole angles and shaping.
- Rotational cooling circuit differences must be evaluated by altering the general passage layouts.
- Balancing of flow rates with coolant temperature rises and pressure losses must be performed readily.
- Changes in the external heat transfer coefficient distributions due to new estimates of freestream turbulence, surface roughness, film injection heat transfer coefficient augmentation, wakes / unsteadiness, hot-streaks / clocking, profile and pattern factors must be accommodated.
- Wall thickness and TBC coating thickness may also be changed in design at virtually any location.

These factors and more dictate that complex FEM and CFD analyses of cooled airfoils at the conceptual design phase are simply not practical. As an example, figure 8 shows three more blade cooling designs, none of which would be easily obtained from some initial generic design if FEM or CFD were used for every change and alteration being investigated[1].

In addition to these design manipulation requirements, the majority of current knowledge concerning internal cooling and film cooling is still contained in empirical and semi-empirical correlations. State-of-the-art computational predictions are as yet not sufficiently advanced to provide prime reliant "data" for the design of cooled airfoils. As such, conceptual design methods must make use of a multitude of design correlations based on experimental data obtained by the original equipment manufacturers (OEM's) and/or contained in open literature.

Putting the foregoing discussion into practice, the two-dimensional or pseudo three-dimensional cooling analysis of the airfoil portion for a vane or blade is typically performed in the following manner.



US patent 4,753,575 US patent 4,753,575 US patent 5,931,638

Fig. 8. Diversity of Cooled Designs

- Given a current prediction of the aerodynamics (static pressure distributions) and gas temperatures surrounding the airfoil sections, the external heat transfer coefficient distribution is calculated for each radial section using either boundary layer analysis or computational heat transfer. The distributions must account for some or all of the influencing factors including:
  - ❖ Airfoil loading
  - ❖ Subsonic boundary layer laminar and turbulent transitions
  - ❖ Bypass transition
  - ❖ Transonic shocks
  - ❖ Surface roughness distribution
  - ❖ Freestream turbulence
  - ❖ Freestream approach swirl
  - ❖ Rotational effects
  - ❖ Boundary layer disturbances due to film coolant injections
  - ❖ Boundary layer disturbances due to coating spallations
  - ❖ Periodic unsteadiness and wake passing
  - ❖ Secondary flow injections in hub and tip regions
  - ❖ Radiative heat flux distributions

- A detailed flow network model of the internal cooling circuits of the airfoil is built using the current known coolant supply pressure and temperature, and the external airfoil static pressure distribution as boundary conditions. The network flow model should allow compressible flow effects, though some models may be sufficient with incompressible flow only. The flow model is executed with an initial solution guess and iterated to convergence based upon the current boundary conditions and internal geometry. This cooling circuit model includes:
  - ❖ Flow area distributions for each passage
  - ❖ Detailed local geometry for each internal feature or repeating feature, such as turbulators, pin-fins, etc.
  - ❖ Cooling passage aspect ratio distributions
  - ❖ Impingement cooling geometry definitions locally
  - ❖ Geometry details for all internal cooling holes
  - ❖ Film cooling extraction locations
  - ❖ Convective heat transfer coefficient correlations
  - ❖ Coefficient of friction correlations
  - ❖ Coefficient of pressure loss correlations for turns, holes, etc.
  - ❖ Cooling fluid properties

- The flow network model can also be arranged to contain a material shell representing the airfoil, such that the model may interact with the external conditions via thermal exchange. This merely requires definition of
  - ❖ Wall thickness distributions for each section
  - ❖ Internal dividing rib thickness distributions
  - ❖ Protective coating thickness distributions
  - ❖ Material property tables

- In addition, the flow model is extended to include the flow and discharge of all film cooling holes. This is done by providing information on
  - ❖ Film hole or film row exit locations
  - ❖ Film hole sizes, shaping factors, spacing, and orientations
  - ❖ Film hole discharge coefficient correlations
  - ❖ Film hole or film row adiabatic effectiveness correlations
  - ❖ Film injection mixing loss correlations
  - ❖ Film hole internal heat transfer coefficient correlations

This total airfoil model can be modified through relatively simple and quick adjustment of the several input distributions and boundary conditions. Execution of the model is straight forward as long as the boundary conditions and geometry parameters are realistic. It must be recognized that such a model contains multiple inlet and exit boundary conditions and parallel flow circuits, of which some flow circuits may be in communication. The complexity of the model must be sufficient to include/resolve all significant pressure losses. The output of the airfoil model can include predictions of all internal heat transfer coefficients, all flow distributions, individual film hole flow rates and mixing losses, total cooling flow rate, the external film temperatures, and of course the local material temperatures. This model can be further coupled to a prediction of the external heat transfer coefficients to update the heat loads for effects of film injection and wall temperature distributions. Once such a model is finalized upon a desired design and result, it may then be exercised to further study manufacturing effects on film hole discharge coefficients and turbulated cooling passages, tolerances for material properties, wall thicknesses, hole diameters, and core shifts, and special considerations for IGCC designs, including surface roughness, TBC spallation, and film hole blockage effects.

Cooling Design Analysis Correlations

A major consideration in the above cooling analysis is the provision of good correlations for both internal flows and film cooling under conditions representative of engines. These correlations are numerous as the variation of internal cooling geometries and film cooling parameters are vast. Because there are so many possible combinations and variations, design analysis is founded on several basic generic correlations from the open literature, and augmented by many geometry-specific correlations determined by OEM research. The following is a list of the primary correlation sources from open literature:

- Impingement jet array heat transfer coefficients (Nusselt numbers) may be obtained from the correlation of Florschuetz et al. for average jet Reynolds numbers typical in engine design[2]. For square arrays of jets at somewhat lower Re numbers, the graphical data of Kercher and Tabakoff may be used[3].

- Impingement cooling that involves the use of individual jets, or slot type jets, or other non-standard configurations, may be determined by correlations in the summary paper of Martin[4].

- Simple fully-developed duct flow turbulent heat transfer may be estimated quite well by the Dittus-Boelter correlation, $Nu = .023\, Re^{0.8}\, Pr^{0.33}$,or other variants on this correlation that can be found in any modern textbook. Care should be taken to account for the wall-to-fluid temperature ratio.

- Most fully-developed turbulated duct flow heat transfer correlations are of the format $Nu = C * Re^{n}\, Pr^{m}$. The basic correlations for stationary turbulated ducts with transverse or angled rib rougheners can be found in Han et al.[5]. This research also includes the coefficients of friction.

- Rotating passage heat transfer data with and without turbulators is contained in the NASA HOST program data sets[6].

- Pin bank internal heat transfer and pressure loss correlations are contained in the works by Metzger et al. and Van Fossen[7].

- Fundamental equations and correlations concerning various cases of idealized slot film cooling, such as might be encountered in various leakage flow paths, are summarized in the review of Goldstein[8].

- The best source of both adiabatic film effectiveness and heat transfer coefficient augmentation factors due to film injection for round and shaped holes is contained in the recent series of studies from the Institute for Thermal Turbomachinery at the University of Karlsruhe, Germany[9]. Such data is generally put into a simplified form to describe the centerline or laterally averaged adiabatic effectiveness as a function of distance and mass velocity ratio. Figure 9 shows several correlation formats that have been used.

- A broad set of data for discharge coefficients of film cooling holes is available from the research of Hay and Lampard and also from the ITS Karlsruhe group[10].

- Aerodynamic film injection mixing losses may be estimated by the use of the method of Hartsel[11].

$$\eta_{aw} = C_1 / (x/Ms)^{n}$$

$$\eta_{aw} = C_1 / (x/Ms + C_2)$$

$$\eta_{aw} = C_1\, Re^{0.2} / (x/Ms)^{0.8}$$

$$\eta_{aw} = C_1 / \{ 1 + C_2 (x/Ms)^{0.8} \}$$

where

$$\eta_{aw} = (T_{rec} - T_{aw}) / (T_{rec} - T_{rec\ coolant})$$

$$M = (\rho V)_{coolant} / (\rho V)_{gas}$$

Re = film jet Reynolds number

s = equivalent film row slot width

Fig. 9. Definitions of Adiabatic Film Effectiveness

Other excellent sources of summarized data and correlations exist in the open literature, but it is up to the design team to determine what to use and how to use it in analysis. One such source is the Lecture Series accumulated by the von Karman Institute of Fluid Dynamics, Brussels. Specific lecture series that cover turbine cooling include Dailey et al., , Harasgama et al., and Glezer et al.[12].

While the above referenced correlations provide a good starting point for the most common methods of cooling, there are dozens of special regions, geometries, and circumstances in cooling design analysis that require case-by-case data. For these cases, the relevant literature is too large to mention here. Most of these cases deal with the so-called "edge" regions of the cooled components, including the endwalls, platforms, airfoil leading and trailing edges, blade tips, interfacial rails, fillets, and any isolated corners. All of these may be treated by the use of similar thermal-flow network models, or integrated into the airfoil model as special regions.

Is this level of cooling analysis detail really required? Figure 10 shows the characteristic uncertainties in engine boundary conditions that affect the complete cooling design analysis of a HPT blade. Also shown is the percentage impact of each boundary condition on the final result (these add to 100%). It should be clear that no detail is unimportant here. Also clear is that the accuracy of certain data, such as the adiabatic film cooling effectiveness distribution, is of very high importance.

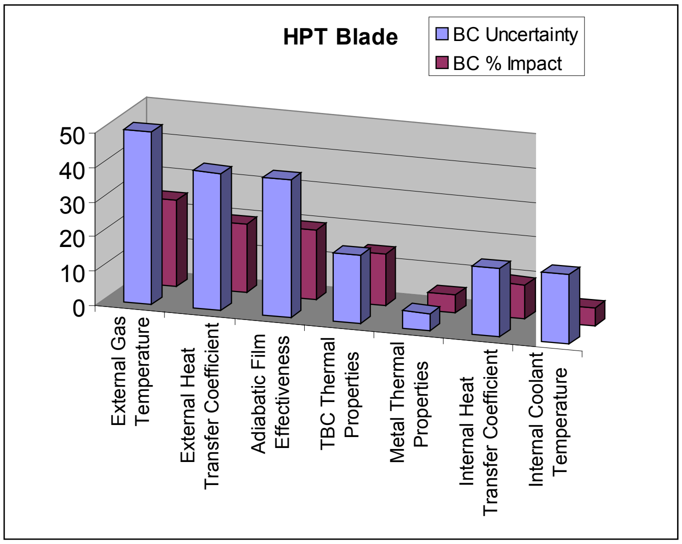


Fig. 10. Impact of Boundary Condition Uncertainties

Additional Factors

Two additional considerations must be incorporated into the cooling design analyses as indicated in figure 2:

1. Engine experience design factors such as film knockdown, coating of film hole interiors, hole spacings, etc; and,

2. Engine degradation factors such as combustor gas profile changes, tip erosion, etc.

These factors account for past experience in both test engines and operational engines that cannot be obtained through research and design activities. These adjustments account for the unknown, or at least poorly understood, conversions from laboratory data and predictions to the reality of complex engine conditions. Another way to look at these factors is as "lessons learned". For the cooling design analyses, experience factors will include film effectiveness realization or knockdown multipliers, film hole diameter reductions due to protective coating applications, minimum allowable hole spacings to avoid hole-to-hole cracking, reduction of internal heat transfer coefficients due to debris collection, typical TBC spallation sizes (if any), surface roughness distribution patterns, and any other generic or design-specific experience gained. Example engine degradation factors will include alterations to the hot gas temperature profiles or magnitudes due to combustor system operation, blade tip erosion, film hole blockages due to deposits, and even modified material properties with exposure at elevated temperatures.

These additional factors are typically incorporated into the design process by one of two methods. First, the data from engine experience can be "data matched" to the design prediction to arrive at the required adjustment factors to be used in the design correlations. Second, modifications due to degradation can be carried through the design analysis in a statistical manner to determine magnitudes of change, as well as sensitivity coefficients.

4.2.1-4 Three-Dimensional Analysis

The two-dimensional or pseudo three-dimensional analysis described above is very similar to the simple one-dimensional analysis in format, but includes all of the required detail to perform design manipulations and trade-off studies to arrive at a "final" cooled component design. Once this iterative process has produced a design that is sufficiently polished a more precise three-dimensional design analysis can be performed. The three-dimensional analysis primarily adds thermal-mechanical detail through the use of a full, accurate FEM of the component. The FEM is executed using mapped convective boundary conditions of local heat transfer coefficients and fluid temperatures from the 2D model results. The FEM solution presents the complete temperature distributions of the materials.

The 3D analysis can also be performed completely through the use of computational modeling, with the prediction of external and internal flow fields and heat transfer coefficients, or coupled with the use of a conjugate model. This method of cooling design analysis must however be in agreement with the conventional design result, since the latter contains a great deal of empirical data and experience factors built in. Sufficient agreement is dictated by the sensitivity of the design to inaccuracies. For example, it is not generally sufficient for the predicted average heat transfer coefficient in a cooling passage to match the average correlation result.

4.2.1-5 Level 2 – Detailed Component and System Cooling Design Analysis

Component Analyses

As indicated in figure 2, Level 2 cooling design analysis is how and where the results of the Level 1 analysis interface with the other component and system goals and requirements. The Level 2 subjects noted in figure 2 do not comprise an exhaustive list, but do represent the diversity of requirements. These aspects of overall design, manufacturing, and operation apply to all of the cooled hot gas path components and their portions – vanes, blades, endwalls, platforms, shrouds, supports, and dovetails. There is no single cooling design analysis method that can be described here. Level 2 analysis must pass and receive results to/from the other engine design analysis packages in an iterative method until an acceptable total design solution is obtained. This may require many changes to the Level 1 design with subsequent re-analysis. As one example of the requirements and complexity of this process, the HPT blade tip region design interaction is considered, as presented in the review of Bunker[13]. (Reprinted by permission of the American Institute of Aeronautics and Astronautics, Inc.)

In designing blade tips, both cooled and uncooled, for proper operation within the larger turbine system one must consider the following major factors (in no particular order):

- Stage and turbine aerodynamic efficiency are greatly affected by the blade tip design in terms of the resulting effective leakage clearance. The effective clearance, which may also be thought of as an effective overall tip discharge coefficient, is determined not only by the tip geometry, but also by the tip aerodynamic distribution, injected cooling flows, tip sealing arrangement, rotational speed, shroud surface treatments, and much more. As a first estimate, each stage can be thought of as having an isolated tip region aerodynamically, but the reality of multistage turbines is that all stages must be designed together to obtain maximum benefit. Another important aspect of the aerodynamic efficiency directly tied to blade tips is the mixing loss associated with the tip leakage flows as they combine with the high momentum suction side passage flow.
- Stage thermal efficiency, and then also overall turbine efficiency, is strongly affected by the amount of chargeable cooling air used to maintain blade tip integrity and life. In highly cooled HPT blades, the tip region alone may account for as much as 20% of the total blade cooling flow.
- Bulk material temperature limits must be considered for the entire blade structure. While the tip region is generally not subject to the same limitations as the rest of the blade in this respect, the tip design does influence the resulting bulk temperatures of the lower blade sections through the overall cooling design. The tip may also present enough weight to require lower bulk temperatures in the main blade sections to avoid creep rupture issues.
- Maximum local material temperatures are typically a major concern for blade tips as these regions are the most difficult to cool. Temperature limits will be placed on the metal substrate, the bond coat, and the thermal barrier coating (TBC) to avoid, for example, excessive oxidation, high coating strains, and melt infiltration of surface deposits, respectively.
- Tip sealing methods vary widely, but all methods attempt to reduce the effective tip clearance. The type of sealing arrangement is intimately tied to the other system design aspects. In many ways, the sealing design is the result of which system design parameters are given the most emphasis.
- Casing out-of-roundness (ie. non-cylindrical) will be transmitted through the structure response to the hot gas flow path roundness bounding the blade tips. This leads to non-uniform tip gaps around the circumference, and potential tip rubs.
- Shroud segment variation, such as bowing, can result from the thermal gradients present in the design, again leading to non-uniform tip gaps either radially and/or axially.
- Approaching and leaving disturbances in the flow around blade tips can affect both the aerodynamics and the cooling. Approaching disturbances are most notably associated with the wakes and shocks being shed from the upstream vane row, which to some degree must influence the tip flow and heat transfer by the introduction of unsteady effects. Approaching and leaving disturbances may be encountered in tip designs that involve shroud recesses and axial flow gaps between the stationary shrouds and attached tip shrouds.
- Gas temperature profiles are the result of the particular combustion system design, the operational point, and mixing through the subsequent stages. The radial gas temperature profile may have severe impact on the blade tip, both in respect to the temperature field itself and the pressure distribution. Stronger radial flows may bring hotter gases to the blade tip than desired, while gas temperatures may drive strong material thermal gradients and cause lower cooling effectiveness.
- Aeromechanics must be considered in the overall blade structural design, and the tip region must be included in this response.
- Stresses, both mechanical and thermal, are key in turbine blade survival. Blade tips must typically deal with very high thermal stresses locally. Higher cooling effectiveness in the tip can alleviate thermal stresses, but must be weighed against the cost to the cycle efficiency. As noted earlier, the blade tip design will influence the weight distribution in the entire blade, which must then be dealt with in the allowable stresses, as well as the low cycle fatigue (LCF) and high cycle fatigue (HCF) responses. This effect will also be transmitted into stress requirements for the blade shank, dovetail, rotor disk posts, and the rotor disk.
- Operating conditions must be considered at various limiting points in the engine cycle, because these change the gas and coolant flow rates, temperatures, and pressures. A blade tip design focused solely on steady state takeoff conditions may not be well suited for cruise conditions. A balanced or optimized cycle design must be sought.
- Transients play a major role in the durability and life of any effective blade tip design. The relative displacements, radial and axial, of the rotor and stator systems during various transients will determine the ultimate steady state operating clearances, as well

as the potential for detrimental interference.

- Durability is desired for both the blade tip and the opposing shroud as a system. In the long term, durability may be associated with oxidation, while in the short term durability is a matter of survival in the face of tip rubs (intentional or unintentional), plugged cooling holes, and thermal stresses.
- Materials and material loss must be planned in blade tip design. Blades and blade tips are not automatically designed with the highest temperature capability material, nor the highest strength material. The tip material may be different from the rest of the blade. The compatibility of the tip and shroud materials must be considered, for example, if a highly abrasive shroud should damage a relatively weak tip material.
- Cumulative damage of blade tips is typically experienced in certain characteristic locations in each design type. Uniform damage or material loss is not the general rule. The change in tip geometry with characteristic damage and loss will alter the aerodynamics and heat transfer, usually leading to accelerated loss.
- Exhaust gas temperature (EGT) is directly and strongly affected by blade tip clearance. Any improvement in effective tip sealing will preserve valuable EGT margin.
- Cost of new parts and cost of repair depend on the complexity of tip design.
- Blade weight impacts the blade root stresses, LCF life, and blade creep. This is not limited to a simple matter of centrifugal stresses, but can also have severe effects on the overall aerodynamic design, changing the reaction and work of the stage.
- Thrust bearing location and bearing housing distortion affect axial motion and disk sag, which in turn are transmitted through the rotor to the blade tip potentially creating larger clearances on one side of the turbine and rubs on the other side.
- Rotor and stator systems should be thermally matched to minimize variations in blade tip clearances during transients. Active clearance control systems can aid in this goal by providing fast thermal response of the shroud radial location.
- Blade tips are commonly at least partially damaged or worn in the course of operation. The ability or inability to repair blade tips becomes an important factor in lifetime cost. The complexity of a blade tip design impacts the decision to provide more or less cooling to balance the cost of repairs. Unrepairable blade tips result in scrapped blades.

While this summary of system design aspects may appear quite detailed and daunting for such a relatively small region of the turbine, there is one requirement that exceeds all others – the blade tip system design must never cause such severe damage as to liberate blades or pieces of blades in operation. As in the other interacting system relationships within the turbine, prior design and operational experience must guide and temper improved designs.

Combustor-Turbine System Analysis

The turbine has a special relationship with the combustion system. Turbine cooling design analysis is directly influenced by the type of combustor system, the combustor exit conditions, and the change in combustor conditions at various cycle points. The combustion system operation and its design relative to the turbine has potential impact in at least six main respects:

- Hot gas temperature profiles
- Hot streak clocking relative to the turbine
- Turbulence characteristics
- Airfoil backflow margins
- NOx emissions
- Fuel type

Each different combustor system design has it's own set of characteristic radial and circumferential gas temperature profiles. The "set" of profiles refers to the fact that the full power radial profile differs from any part-power profile. For example, some systems have annular combustors, some have can-annular combustors, and others have dump combustors. Full annular combustors may be single, dual, or even triple annular systems with respect to the number of fuel nozzle rings present. In such cases, combustor nozzle staging may be used for differing power requirements. Another major difference arises between the low NOx systems of power turbine engines and aero-engines, the former employing very little dilution or film flow injection within the combustors, and the latter utilizing a great deal of dilution and film injection. Most power generation turbines tend toward very flat radial profiles, while aero-engines tend to have more peaked radial profiles that may change peaking location with power condition. Power turbines may also have radial temperature profile changes as operation is changed from diffusion mode to premixed mode. The key for turbine cooling design is to know as much as possible about the combustor system exit conditions for all operating conditions, and to carry this information through to the design for each cycle point.

Combustion systems have circumferential gas temperature and pressure profiles as well, due to the discrete nature of virtually all designs with respect to air/fuel injection and flame holding. While radial profiles are caused by the combined effects of fuel nozzles and combustor dilution / cooling flows, circumferential profiles or pattern factors are caused primarily by the number and spacing of the fuel nozzles. Since the turbine inlet vanes are also of a finite number, this leads to the interesting aspect of hot streak clocking. The combustor hot streak may be aligned directly on a vane leading edge, or midway between two vanes. In fact, the hot streaks may be variable around the entire vane ring depending on the relative count of fuel nozzles and turbine inlet vanes. Different unsteady gas conditions may be incident upon the rotating blade row. The center hot streak may pass through the passage with little vane interaction, while the leading edge hot streak may be greatly modified by interaction with the vane and its cooling flows. There are of course immediate consequences for the vane, but this also translates through to the blade.

As with the hot streak effects, combustion system turbulence and swirl flow are additional complicating factors. The turbulence intensity levels, distributions, and length scales will not be the same as those generated by the grids used in simplified studies. The combustor exit flow, in addition to temperature profiles, may also contain significant swirl content. These factors may not be entirely washed out by the inlet vane row. Some studies have indicated that combustor exit average turbulence intensity over the entire region is as high as 30%.

Of great concern in all gas turbine designs is the attainment of single digit NO_x emission levels. The cooling of low-emissions combustor liners is achieved primarily through the use of convective backside heat transfer, with little or no injection of coolant into the hot gas path. Given the high levels of flow required to perform this cooling, the pressure drop allocated to the combustion system is an important factor. A typical combustion system may use up to 7% of the available pressure from the compressor. This cooling system pressure loss is roughly equivalent to 1% in cycle efficiency, a very significant amount. It is therefore of great concern to designers to achieve the greatest possible cooling effectiveness with the lowest possible pressure loss. It is equally important to the design to achieve a greater cooling effectiveness while matching the pressure loss required by the compressor and turbine design. In this respect, lower pressure loss combustion systems can impose higher loading on the turbine inlet nozzle, and can also present problems in meeting backflow margin requirements. Additionally, since lower NOx emissions can be obtained by "stealing" cooling air from the turbine, this puts pressure on turbine cooling design to use less air.

Fuel flexibility is another clear objective in power turbines, with the desire to use gas, liquid distillates, various syngases, and even heavy oils. The operation of a turbine on multiple fuels presents multiple scenarios for the cooling design analysis. In most cases, this means analysis for the most severe cases. Future turbine systems may conceive of controllable turbine cooling to accommodate such changes in operation.

These several issues concerning the combustor-turbine system all point to the requirement that the cooling design analysis must not only be performed for changing conditions due to the combustor, but in some cases will even lead to vane-to-vane differences in the cooling analysis.

4.2.1-6 Turbine Secondary Cooling Circuit Analyses

While much attention is given to the cooled airfoils of the turbine, the secondary flow cooling circuits deserve equal scrutiny and diligence to arrive at a total engine design solution. Figure 1 shows the secondary flow circuits typical of an aero-engine HPT, and figure 11 shows an example of the secondary flow regions for a heavy frame turbine. Secondary circuits of the turbine include the following:

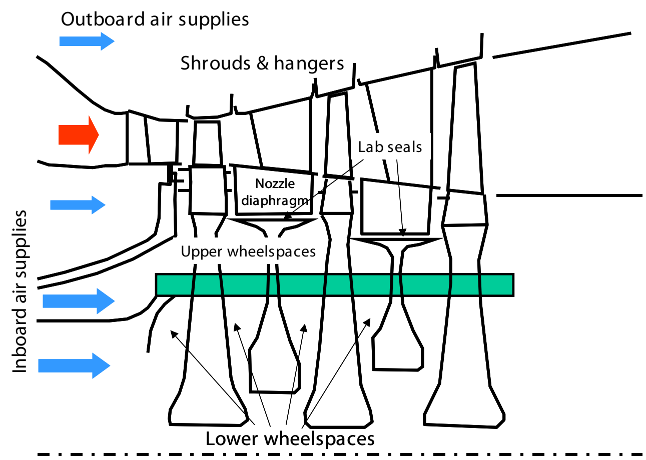


Fig. 11. Heavy Frame Turbine Secondary Flow Regions

- Lower wheelspaces or disk cavities inboard of the hot gas path
- Supply circuits from the compressor discharge region to the inboard turbine flows
- Upper wheelspaces including buffer and trench cavities around the angel wings
- Supply circuits from the compressor discharge to the outer turbine casing flows
- Outboard nozzle and shroud cooling air plenums and connections
- All rotating seals in these areas, e.g. labyrinth and brush seals
- All stationary seals in these areas, e.g. labyrinth and cloth seals
- Component interface leakage pathways and their seals, such as nozzle-to-combustor gaps, nozzle-to-nozzle gaps, shroud-to-shroud gaps, nozzle-to-shroud gaps, and blade-to-blade gaps (spline seals, C-seals, W-seals, leaf seals, etc.)
- Rotating orifices
- Stationary orifices
- Pre-swirl supply nozzles
- Inducers and cover plate systems
- Blade dovetail / shank leakages
- Bolt leakages
- Nozzle support leakages
- Outboard-to-inboard cooling circuits routed through turbine airfoils
- Nozzle diaphragm chambers
- Supply flows bled from earlier compressor stages
- Shroud hanger system flows and leakages

The cooling design analysis for the secondary flow systems is performed in much the same way as that of the turbine airfoils, the main difference being that most of these flow circuits do not directly interact with the hot gas path. Because there is no "external" hot gas flow involved, the thermal-fluid design analysis of these regions becomes an elaborate flow network model with thermal boundary conditions at the hardware surfaces. Just as in the turbine airfoil analysis, the secondary flow models may be simple or complex depending on the design stage.

Ultimately, some regions will require complex CFD analysis to resolve full details. The primary location where this level of design analysis is required concerns the points at which the secondary flows do interact with the hot gas path. One such region is the forward wheelspace sealing cavities between the turbine inlet nozzle and the first stage blade, as depicted in figure 12. A source of cooling air is supplied from the inboard location and routed through the stationary-rotating seal cavities, in this case a buffer cavity and then the trench cavity at the turbine flow path. Aside from this flow circuit, there are several other leakage pathways influencing the region, as depicted in figure 13. In addition, the exit of the flow circuit sees a very three dimensional flow that varies in the circumferential direction due to nozzle wakes and blade leading edge effects. Such interaction regions can involve substantial mixing of the cold and hot flows. A more detailed knowledge or prediction level of the heat transfer coefficients and gas temperatures in these regions is required.

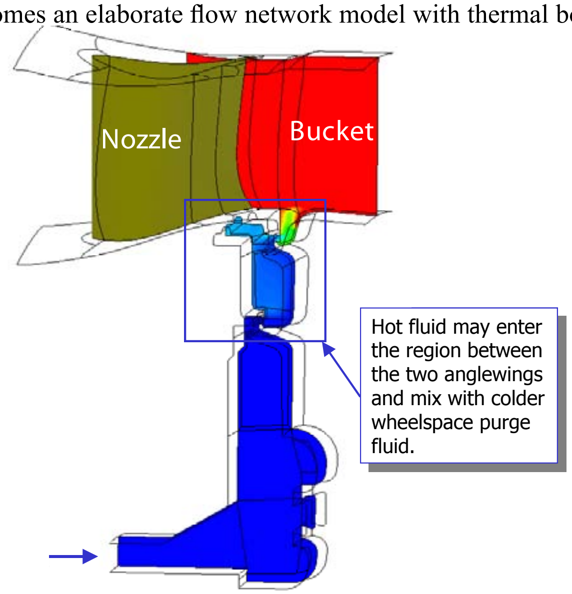


Fig. 12. Purge Flow Circuit for Turbine Wheelspace

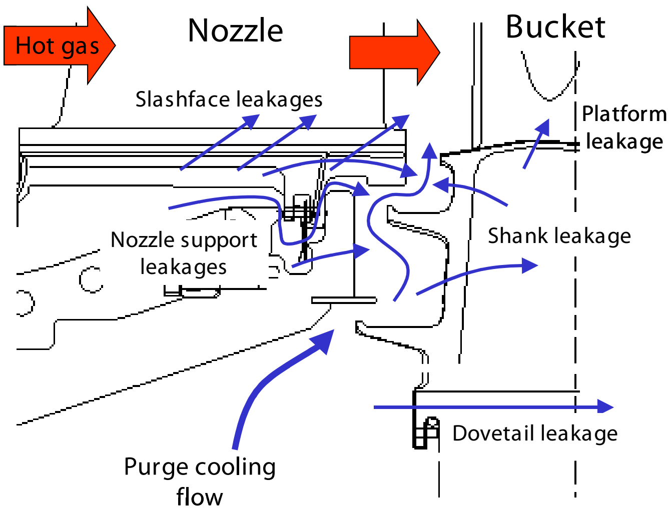


Fig. 13. Cooling Flows and Leakages in Buffer Cavity Region

Secondary flow design analysis begins with overall, large network models representing the compressor discharge and bleeds to the eventual exit flows into the turbine flow path, accounting for all key flow areas, lengths, restrictions, and discharge coefficients, using approximate thermal boundary conditions for heat transfer. More detailed models are made to examine separate portions of the flow circuits and add greater fidelity to the boundary conditions. Open literature sources may be used for most of the required information concerning flow restrictions, friction coefficients, and discharge coefficients. Some commercial flow network solvers contain correlations for much of this information. Heat transfer boundary conditions can be estimated by simple forced duct flow and natural convection correlations in most locations other than the radial disk flow, radial cavity flow, and labyrinth seal regions. A good summary of the flow and heat transfer in radial rotating disk and disk cavity systems for various situations is that of Owen and Rogers and the subsequent literature publications of Owen and co-workers[14]. Labyrinth seal flow and heat transfer data for planar and stepped geometries may be found in the research of the ITS Karlsruhe group, such as that of Waschka et al.[15].

The thermal condition of the hardware surrounding the secondary flow circuits must be included in the final design analysis. These boundaries cannot in most cases be treated as adiabatic. For example, the bucket dovetails are connected to the wheel in the disk-posts. While the forward and aft surfaces of the dovetail and disk-post are exposed to the secondary flows of the upper wheelspace, the primary cooling flow of the bucket is routed between the bottom of the dovetail and the wheel, and the coolant flows inside the dovetail to the airfoil. This forms an additional network that connects the secondary flow circuit and the coolant circuit of the buckets. This internal cooling of the bucket dovetail and shank will thermally affect the response of the disk-post and wheel. Even the cooling of the bucket airfoils and platforms has an influence on the top portion of the wheel, serving to conduct energy from the hot gas path down into the wheel. This latter effect is usually analyzed by applying lumped or equivalent thermal masses to the top of the wheels or bucket shanks to act as heat sources. Detailed thermal models of the airfoils, supports, and wheels are rarely if ever done in the same model. In a similar manner, the outer shrouds and their hangers must be modeled together to provide the complete prediction of flows and thermal response. Individual wheels may be modeled, or the entire turbine rotor system. In fact, at some detailed design level, the entire turbine rotor must be thermally analyzed as one in order to correctly predict all clearances. Going one step further, the so-called "unit rotor", which is the combined compressor-turbine-generator rotor must also be analyzed with thermal boundary conditions, albeit with a less detailed application of conditions.

4.2.1-7 Level 3 – Transient Operational Cooling Design Analysis

All of the foregoing cooling design analyses are commonly applied to steady-state operating conditions at some well defined point in the cycle deck or operational map of the turbine engine. The reality of turbine operation however is that both slow and fast transients must considered in the design process. Slow transients include normal startup and shutdown, or load following operations, while fast transients include quick starts for peaking power, engine trips, and hot restarts. Within even the normally slow startup procedure for a heavy frame turbine, there are several intermediate operating points and transients, such as turning gear operation, low RPM holding, and ~80% power point warm up. Other transients may include specific operating domains dictated by the combustion system, water washing, and power augmentation (e.g. water injection to post-combustion).

Figure 14 shows an approximate transient growth behavior for the turbine rotor and stator during a fast start (< 30 minutes). The transient growth of the rotor is a combination of all portions making up the rotor, with contributions from centrifugal and thermal effects. The transient growth of the stator and casing outboard of the rotor is thermally dominated, occurring at a different rate than the rotor. The cooling design analysis of all transients is performed using a sufficient number of steady-state analyses and their associated boundary conditions. Each steady-state analysis is performed using the Level 2 methods discussed in the pervious section. The boundary conditions of these several operating points, flow rates, pressures, gas temperature profiles, heat transfer coefficients, and film effectiveness, are used to form the anchor points of the transient analysis. Since the number of steady-state analysis points is typically limited, the boundary conditions at several intermediate steps must be obtained by interpolation. As the basic fluid dynamic and thermal domains of the hot gas and cooling flows also change with operating conditions, these interpolations are performed using explicit or ad-hoc rules. The exact nature and definition of these rules are very dependent on the turbine design and operation, and as such are specific to the OEM's.

Transient analyses of individual components, such as the turbine airfoils, follow the same general guidelines. Usually, the concerns associated with these components are not the same as those of the overall turbine stator and rotor systems. Instead, issues with clearances, leakage gaps, binding, and hot gas backflow or ingestion are scrutinized. In addition, transient effects on peak material stress and strain are important, as evidenced by the potential for TBC spallation under severe thermal transients. The transient cooling design analysis for hot gas path components may therefore focus on certain transients, or portions of transients, known to be of greatest concern.

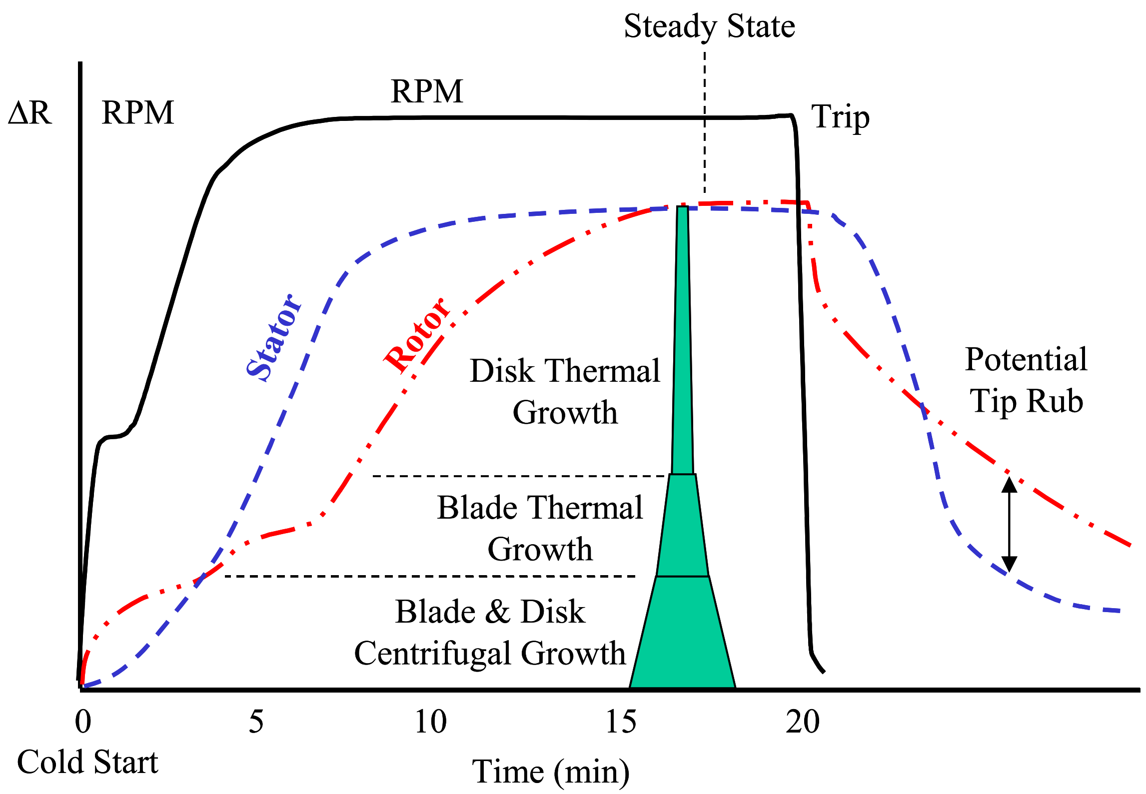


Fig. 14. Transient Rotor and Stator Growth for Fast Startup. (From Bunker[13], reprinted by permission of the American Institute of Aeronautics and Astronautics, Inc.)

4.2.1-8 Notes

1. J. L.Levengood and T. A. Auxier, "Airfoil with Nested Cooling Channels," U.S. Patent 4,753,575, 1988; D. A. Krause, D. J. Mongillo, F. O. Soechting, and M. F. Zelesky, "Turbomachinery Airfoil with Optimized Heat Transfer", U.S. Patent 5,931,638, 1999.
2. L. Florschuetz, C. Truman, and D. Metzger, "Streamwise Flow and Heat Transfer Distributions for Jet Array Impingement with Crossflow," *Journal of Heat Transfer* 103 (1981): 337-342.
3. D. Kercher, and W. Tabakoff, "Heat Transfer by a Square Array of Round Air Jets Impinging Perpendicular to a Flat Surface Including the Effect of Spent Air," *Journal of Engineering for Power*, 92 (1970): 73-82.
4. H. Martin, "Heat and Mass Transfer Between Impinging Gas Jets and Solid Surfaces," *Advances in Heat Transfer* 13 (1977): 1-60.
5. J. C. Han, J. S. Park, and C. K. Lei, "Heat Transfer Enhancement in Channels with Turbulence Promoters", *Journal of Engr. for Gas Turbines and Power* 107 (1985): 628-635.
6. T. J. Hajek, J. H. Wagner, B. V. Johnson, A. W. Higgins, and G. D. Steuber, "Effects of Rotation on Coolant Passage Heat Transfer," NASA Contractor Report 4396 (1991).
7. D. E. Metzger, R. A. Berry, and J. P. Bronson, "Developing Heat Transfer in Rectangular Ducts with Staggered Arrays of Short Pin Fins," *Journal of Heat Transfer* 104 (1982): 700-706; G. J. VanFossen, "Heat Transfer Coefficients for Staggered Arrays of Short Pin Fins," *Journal of Engineering for Power* 104 (1982): 268-274.
8. R. J. Goldstein, "Film Cooling," *Advances in Heat Transfer* 7 (1971): 321-379.
9. M. Gritsch, A. Schulz, and S. Wittig, "Adiabatic Wall Effectiveness Measurements of Film-Cooling Holes with Expanded Exits," Paper 97-GT-164 (IGTI Conference, Orlando, Florida [1997]); M. Gritsch, A. Schulz, and S. Wittig, "Heat Transfer Coefficients Measurements of Film-Cooling Holes with Expanded Exits," Paper 98-GT-28 (IGTI Conference, Stockholm, Sweden [1998]); C. Saumweber, A. Schulz, and S. Wittig, "Free-Stream Turbulence Effects on Film Cooling with Shaped Holes," Paper GT-2002-30170 (IGTI Turbo Expo, Amsterdam, Netherlands [2002]); J. Dittmar, A. Schulz, and S. Wittig, "Assessment of Various Film Cooling Configurations Including Shaped and Compound Angle Holes Based on Large Scale Experiments," Paper GT-2002-30176 (IGTI Turbo Expo, Amsterdam, Netherlands [2002]).
10. N. Hay and D. Lampard, "Discharge Coefficient of Turbine Cooling Holes: A Review," *Journal of Turbomachinery* 120 (1998): 314-319; M. Gritsch, A Schulz, and S. Wittig, "Discharge Coefficient Measurements of Film-Cooling Holes with Expanded Exits," Paper No. 97-GT-165 (IGTI Turbo Expo, Orlando [1997]); M. Gritsch, C. Saumweber, A. Schulz, S. Wittig, and E. Sharp, "Effect of Internal Coolant Crossflow Orientation on the Discharge Coefficient of Shaped Film-Cooling Holes," *Journal of Turbomachinery* 122 (2000): 146-152.
11. J. E. Hartsel, "Prediction of Effects of Mass Transfer Cooling on the Blade Row Efficiency of Turbine Airfoils," AIAA Paper 72-11 (AIAA Aerospace Sciences Meeting, San Diego, California [Jan. 17-19, 1972]).
12. G. M. Dailey, M. Taslim, D. L. Rigby, P. Sagaut, M. Cakan, B. Han, R.J. Goldstein, and J.M. Buchlin, "Aero-Thermal Performance of Internal Cooling Systems in Turbomachines," *Von Karman Institute for Fluid Dynamics Lecture Series VKI-LS 2002-01* (2002); S.P. Harasgama, J.C. Han, S. Dutta, H. Iacovides, G. Rau, J.M. Owen, and M. Wilson, "Heat Transfer and Cooling in Gas Turbines," *Von Karman Institute for Fluid Dynamics Lecture Series VKI-LS 1996-01* (1996); B. Glezer, N. Harvey, C. Camci, R. Bunker, and A. Ameri, "Turbine Blade Tip Design and Tip Clearance Treatment," *Von Karman Institute for Fluid Dynamics Lecture Series VKI-LS 2004-02* (2004).
13. Bunker, R.S., "Axial Turbine Blade Tips: Function, Design, and Durability," to be published in the *AIAA Journal of Propulsion and Power*, March-April, 2006 special issue on Turbine Science & Technology.
14. J. M. Owen and R.H. Rogers, *Flow and Heat Transfer in Rotating Disc Systems*, vol. 1 (Somerset, England: Research Studies Press, 1989); J. M. Owen and R. H. Rogers, *Flow and Heat Transfer in Rotating Disc Systems,* vol. 2 (Somerset, England: Research Studies Press, 1989).
15. W. Waschka, S. Wittig, and S. Kim, "Influence of High Rotational Speeds on the Heat Transfer and Discharge Coefficients in Labyrinth Seals," Paper No. 90-GT-330 (IGTI Turbo Expo Conference, Brussels, Belgium, 1990); W. Waschka, S. Wittig, S. Kim, and T. Scherer, "Heat Transfer and Leakage in High-Speed Rotating Stepped Labyrinth Seals," AGARD Conference Proceedings 527, Heat Transfer and Cooling in Gas Turbines (1993).

BIOGRAPHY

4.2.1 Cooling Design Analysis

Ron S. Bunker

GE Global Research
One Research Circle, K-1 ES-104
Niskayuna, NY 12309

phone: (518) 387-5086
email: bunker@crd.ge.com

Dr. Bunker is an internationally recognized research engineer in the field of Gas Turbine Heat Transfer. Dr. Bunker received his PhD in Mechanical Engineering from Arizona State University in 1988. After a one-year post-doctoral research fellowship from the Alexander von Humboldt Foundation of Germany, Dr. Bunker joined GE Aircraft Engines in Cincinnati. In 1993, Dr. Bunker joined the GE Global Research Center. He has worked on R&D activities focused on turbine vane and blade internal and external heat transfer. The main thrust of efforts during the most recent years has been new technology development for the Advanced Turbine System "H" power plant. Dr. Bunker is a Fellow of the American Society of Mechanical Engineers and Associate Technical Editor for the Journal of Turbomachinery. Dr. Bunker has been awarded 35 US patents and is the author of 75 technical publications.