

Aerothermal Coupling of Leakage Flow in Turbine Endwall Seal Structures: Postprint

Authors: An Yuge, Liu Huoxing, Zou Zhengping

Date: 2017-11-07T00:00:00+00:00

Abstract

This study employs an aerothermal coupling calculation method to investigate the effects of hub sealing coolant and tip leakage flow on turbine performance and thermal load. The results indicate that endwall leakage flow induces flow losses and leads to a reduction in turbine work capacity, and that the effects of different leakage flows on turbine efficiency can approximately satisfy a linear superposition principle. Heat conduction in solid components has negligible effect on blade aerodynamic loading, whereas temperature gradients caused by hub sealing coolant significantly alter wall heat transfer characteristics, necessitating the use of aerothermal coupling calculation to accurately predict thermal load.

Full Text

Aerothermal Analysis of a Turbine with Endwall Leakage Flow

AN Yu-Ge¹, LIU Huo-Xing¹, ZOU Zheng-Ping¹

¹National Key Laboratory of Science and Technology on Aero-Engines Aero-Thermodynamics, School of Energy and Power Engineering, Beihang University, Beijing 100191, China

Abstract

This study employs conjugate heat transfer (CHT) computational methods to investigate the effects of hub seal cooling air and shroud leakage flow on turbine performance and thermal loading. The results demonstrate that endwall leakage flow induces flow losses and reduces turbine work capacity, with the performance penalties from different leakage flow systems following an approximate linear superposition principle. While solid component heat conduction has negligible impact on blade aerodynamic loading, temperature gradients created by hub

seal cooling air significantly alter wall heat transfer characteristics, necessitating coupled aerothermal calculations for accurate thermal load prediction under such conditions.

Keywords: Gas turbine; Computational fluid dynamics; Thermal analysis; Leakage flows

Introduction

Modern aeroengines operate on the Brayton cycle, and increasing turbine inlet temperature represents a critical pathway for enhancing cycle work output and thermal efficiency in the development of high-thrust-weight-ratio advanced propulsion systems. Current turbine gas temperatures substantially exceed the temperature capability of component materials, necessitating widespread adoption of cooling technologies to ensure safe turbine operation. Cooling air bled from the compressor, which can account for up to 20% of mainstream flow, is particularly used to prevent hot gas ingestion in rotor-stator cavities through seal systems and to reduce leakage via shrouded blade tips. Turbine seal cooling alone can consume 6% of overall engine fuel consumption, while shroud leakage flow, which does no work, causes efficiency to decrease linearly with leakage mass flow rate.

Numerous researchers have investigated leakage flow effects on turbine flow fields and thermal loads using increasingly sophisticated methods. Ong et al. [5] employed unsteady experimental and numerical approaches to examine seal flow effects on turbine performance and secondary flows, analyzing how coolant preswirl angle influences cooling effectiveness and additional flow losses. Ameri and Bunker [6] obtained rotor blade tip convective heat transfer coefficient distributions through adiabatic and isothermal wall CFD calculations, demonstrating that tip thermal loading is closely related to leakage flow structure and that accurate flow structure characterization is prerequisite for precise thermal load prediction. Haller and Hilditch [7] investigated convective heat transfer characteristics on shroud outer surfaces. These thermal analysis approaches, which obtain adiabatic wall temperatures or fluid domain boundary heat transfer characteristics through CFD alone and require separate conduction calculations to determine actual component temperatures, are termed uncoupled thermal analysis methods.

With advances in computational capabilities, conjugate heat transfer (CHT) methods have been increasingly applied to turbine blade thermal analysis [8] and internal cooling system optimization design [9], deepening understanding of how turbine endwall seal leakage flows affect mainstream flow fields and thermal loads. However, in actual operation, turbine mainstream flow simultaneously experiences multiple leakage flows from both hub and shroud systems. Under strong temperature gradients induced by seal cooling air, solid domain heat conduction also influences component heat transfer characteristics. As turbine loading continues to increase, interactions between mainstream and leakage flow

systems and coupling effects between flow and temperature fields will intensify, requiring critical examination of the applicability and generalizability of conclusions drawn from traditional single-source leakage studies and uncoupled CFD-based thermal load predictions. The European Sixth Framework Programme's MAGPI [2] and AITEB [10] projects have begun exploring these phenomena, investigating aerothermal characteristics associated with endwall leakage flows in realistic turbine environments.

This research employs conjugate heat transfer computational methods to investigate the effects of hub seal cooling air and shroud leakage flow on the flow field and performance of a single-stage turbine, examining nonlinear superposition effects between different flow systems. The study specifically addresses hub seal cooling air effects on component thermal loading and analyzes the coupling mechanism between turbine component temperature and flow fields under strong temperature gradient conditions, along with its impact on the accuracy of traditional uncoupled thermal analysis methods.

1. Computational Methodology

1.1 Computational Program

This study utilizes the commercial CFD code CFX for conjugate heat transfer calculations. The fluid domain solves the Reynolds-averaged Navier-Stokes equations, closed using the SST (Shear Stress Transport) turbulence model, with the γ -Re transition model employed to predict transition and account for its effects on flow and heat transfer. Heat conduction in the solid domain is simulated by solving the energy equation. For uncoupled thermal analysis, an adiabatic-wall CFD calculation is first performed to obtain the reference temperature T_{ref} distribution. The wall temperature T_w distribution is then calculated using Equation (1), where δT represents a small temperature increase that generates heat flux at the wall. A second CFD calculation is conducted with prescribed wall temperature to obtain the heat flux density q distribution, from which the convective heat transfer coefficient at fluid domain boundaries can be derived using Equation (2). Finally, the solid domain conduction equation is solved using third-type boundary conditions with adiabatic wall temperature as the reference temperature and prescribed boundary heat transfer coefficients to obtain turbine component temperature distributions.

1.2 Computational Model and Mesh

A single-stage transonic turbine with cavity and shroud structures from an aero-engine is selected as the test case, with turbine geometry shown in [Figure 1: see original paper]. The turbine design speed is 17,000 r/min, with a design-point expansion ratio of 1.8 and loading coefficient of 1.95. Cooling air enters the rotor-stator cavity through labyrinth seals, reducing turbine disk thermal loading before being injected into the mainstream to prevent hot gas ingestion. The rotor disk's rear surface is directly exposed to hot gas, creating very strong

temperature gradients within the disk under the combined influence of seal cooling air and mainstream gas, resulting in strong interactions between flow and temperature fields.

Computational boundary conditions specify total temperature and pressure at the mainstream inlet, mass flow rate and temperature at the seal cooling air inlet, and a mixing-plane model between rotating and stationary fluid domain components. Temperature and heat flux continuity are enforced at fluid-solid interfaces, with temperature distributions prescribed at solid domain outer boundaries. The computational mesh is illustrated in [Figure 2: see original paper]. The mainstream flow path employs multi-block structured grids comprising 16 blocks, while hub and shroud seal cavities and the solid domain use unstructured hexahedral grids refined near walls to ensure $y^+ < 1$. The total three-dimensional mesh contains 8.3 million cells. Grid independence verification was performed prior to analysis, demonstrating negligible changes in turbine output work, shroud leakage flow rate, and blade average temperature with further mesh refinement.

2. Results and Discussion

2.1 Effects of Leakage Flow on Performance

To analyze the impact of hub and shroud leakage flows on turbine performance and the interaction between different leakage systems, calculations were performed for ideal mainstream flow, flow with shroud leakage, flow with hub seal cooling air, and flow with both leakage sources. [Figure 3: see original paper] presents the turbine efficiency obtained under these different geometric complexities, quantitatively comparing performance impacts. Notably, while numerous researchers and organizations have investigated turbine efficiency definitions based on various mixing and work extraction models, no consensus has yet been reached [11]. This study employs the industrially widespread efficiency definition shown in Equation (3), where the denominator represents the isentropic expansion work of both gas and coolant, and the numerator is the actual turbine output work (torque multiplied by rotational speed).

The results indicate that hub seal cooling air causes a 1.18% turbine efficiency decrease, while shroud leakage flow induces a substantially larger 2.29% efficiency penalty. When both leakage flows are present simultaneously, the efficiency reduction differs from the sum of individual losses by only 0.02%, demonstrating that the performance effects of hub and shroud leakage flows can be considered linearly additive for this test case. It should be further noted that the shroud leakage flow rate reaches 2% of mainstream flow. According to conventional shroud loss theory, the work capacity loss and efficiency decrease caused by leakage flow equals its mass flow percentage, confirming that shroud leakage represents the dominant loss source in this case.

2.2 Effects of Leakage Flow on Thermal Load

Endwall leakage flow significantly affects turbine component thermal load distribution while altering mainstream flow structure and turbine performance. [Figure 4: see original paper] shows the dimensionless temperature distribution obtained through conjugate heat transfer calculations, where the dimensionless temperature θ is calculated using Equation (4). Temperature differences between seal cooling air and mainstream gas create strong temperature gradients within the turbine disk, particularly at the blade root, adversely affecting structural integrity. Under these conditions, disk heat conduction significantly influences seal air flow and heat transfer characteristics, creating a coupled problem where altered heat transfer characteristics subsequently affect temperature distribution.

[Figure 5: see original paper] compares isentropic Mach number distributions at 5%, 50%, and 95% span sections obtained through conjugate and adiabatic-wall calculations, where isentropic Mach number is calculated using Equation (5) with p_{ref} being the rotor inlet average total pressure. The results demonstrate nearly identical aerodynamic loading distributions between adiabatic and conjugate calculations, indicating that blade heat conduction has minimal impact on aerodynamic loading and can be reasonably neglected in aerodynamic design and analysis.

[Figure 6: see original paper] presents the radial temperature distribution on the cavity side of the rotor disk, revealing that actual disk temperatures significantly exceed adiabatic wall temperatures. Since the rotor disk's rear surface is directly exposed to hot gas, heat conduction raises cavity wall temperature, thereby altering wall heat transfer parameters. [Figure 7: see original paper] shows the dimensionless convective heat transfer coefficient distribution obtained from CHT and uncoupled calculations, demonstrating that uncoupled calculations, which effectively capture heat transfer characteristics near the reference temperature, exhibit deviations from actual values.

Traditional uncoupled thermal analysis methods obtain surface convective heat transfer coefficients or Nusselt number distributions to evaluate turbine component thermal loads, requiring additional conduction calculations to obtain temperature distributions. However, these methods cannot account for real solid wall temperature effects on fluid heat transfer characteristics. [Figure 8: see original paper] details the temperature ratio distribution between rotating components obtained through CHT and uncoupled methods, with CHT results considered more physically realistic. The uncoupled method overpredicts cavity surface temperature distribution, leading to elevated temperature predictions at the blade root where thermal loads are critical for mechanical strength analysis.

To further analyze the conjugate heat transfer problem in rotor disks under large axial temperature gradients, the configuration is simplified as a flat plate of thickness L , as shown in [Figure 9: see original paper], where T_w is the cavity wall temperature and T_r is the rotor disk rear wall temperature. T_g represents

hot gas temperature, while T_f and T_s are fluid and solid temperatures in the first layer of cells adjacent to the wall, respectively, and Δn_f and Δn_s are cell widths. Heat flux density q driven by axial temperature difference can be calculated using Equation (6). Combining Equations (2) and (6) yields the wall temperature expression in Equation (7). The uncoupled method's underestimation of wall convective heat transfer consequently leads to overpredicted wall temperatures.

3. Conclusions

This study employs conjugate heat transfer computational methods to investigate the effects of hub and shroud leakage flows on turbine performance and thermal loading, comparing the accuracy of coupled and uncoupled thermal analysis methods for thermal load prediction. The main conclusions are:

1. Hub and shroud leakage flows cause flow losses and reduce turbine work capacity. The performance effects of different leakage flow systems follow a linear superposition law.
2. Solid component heat conduction significantly affects thermal load distribution, particularly under temperature gradients created by hub seal cooling air. However, heat conduction has negligible impact on blade aerodynamic loading, making the adiabatic wall assumption reasonable for aerodynamic performance engineering design and analysis.
3. When large temperature gradients exist within turbine components, heat conduction affects wall temperature and consequently influences convective heat transfer coefficients. Conjugate aerothermal calculations are essential for obtaining physically realistic temperature distributions.

References

- [1] Chew J W. Developments in Turbomachinery Internal Air Systems[J]. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 2009, 223(1): 189-234
- [2] Dixon J A, Valencia A G, Coren D, et al. Main Annulus Gas Path Interactions—Turbine Stator Well Heat Transfer[J]. Journal of Turbomachinery, 2014, 136(2): 21010
- [3] Bohn D E, Decker A, Ohlendorf N, et al. Influence of an Axial and Radial Rim Seal Geometry on Hot Gas Ingestion Into the Upstream Cavity of a 1.5-Stage Turbine[R]. ASME Paper, GT2006-90453, 2006
- [4] Rosic B, Denton J D, Curtis E M. The Influence of Shroud and Cavity Geometry on Turbine Performance: An Experimental and Computational Study-Part I: Shroud Geometry[J]. Journal of Turbomachinery, 2008, 130(4): 41001
- [5] Ong J, Miller R J, Uchida S. The Effect of Coolant Injection on the Endwall

Flow of a High Pressure Turbine[J]. Journal of Turbomachinery, 2012, 134(5): 51003-51008

[6] Ameri A A, Bunker R S. Heat Transfer and Flow on the First-Stage Blade Tip of a Power Generation Gas Turbine: Part 2—Simulation Results[J]. Journal of Turbomachinery, 2000, 122(2): 272-277

[7] Haller B R, Hilditch M A. External Heat Transfer on a Shrouded HP Gas Turbine Stage[R]. ASME Paper, GT2007-27168, 2007

[8] York W D, Leylek J H. Three-Dimensional Conjugate Heat Transfer Simulation of an Internally-Cooled Gas Turbine Vane[R]. ASME Paper, GT2003-38551, 2003

[9] Eifel M, Caspary V, Honen H, et al. Experimental and Numerical Analysis of Gas Turbine Blades With Different Internal Cooling Geometries[J]. Journal of Turbomachinery, 2011, 133(1): 11018-11019

[10] Janke E, Wolf T. Aerothermal Research for Turbine Project[R]. ASME Paper, GT2010-23511, 2010

[11] Young J B, Horlock J H. Defining the Efficiency of a Cooled Turbine[J]. Journal of Turbomachinery, 2006, 128(4): 707-715

Note: Figure translations are in progress. See original paper for figures.

Source: ChinaXiv – Machine translation. Verify with original.