

Integrated CFD–FEA thermal analysis of an aft-fan module for a mini turbojet engine

Göktuğ Karaca¹, Kübra Kurtoğlu Sontay¹, Musa Cenk Özekinci^{1,2},
Altuğ Muslu¹, Doğuş Özkan^{1*}, Mirosław Szala³, Egemen Sulukan⁴

¹ Mechanical Engineering Department, Turkish Naval Academy, National Defense University, Tuzla, 34942, İstanbul, Türkiye

² Mechanical Engineering Department, Atatürk Strategic Studies and Graduate Institute, National Defense University, 34334, Yeni Levent, Beşiktaş-İstanbul, Türkiye

³ Faculty of Mechanical Engineering, Lublin University of Technology, Nadbystrzycka 36D, Lublin 20-618, Poland

⁴ Mechanical Engineering Department, Istanbul Gedik University, 34876, Kartal, İstanbul, Türkiye

* Corresponding author's e-mail: dogus.ozkan@msu.edu.tr

ABSTRACT

This study presented an integrated CFD–FEA thermal analysis of an aft-fan module designed for a mini turbojet engine, employing a hierarchical approach that combines one-dimensional heat-transfer formulations, two-dimensional disc conduction–convection models, and three-dimensional CFD-assisted solid thermal simulations. CFD-derived convection coefficients and secondary air system (SAS) cooling conditions were mapped onto the solid domain to obtain a realistic temperature field under combined hot- and cold-flow conditions. The results indicated a maximum metal temperature of ≈ 979 K on the turbine rotor blades, while the fan-side components remain near ≈ 294 K, establishing a pronounced through-disc thermal gradient driven by the 1045 K turbine inlet gas. The analytical and three-dimensional numerical models produced closely matched temperature distributions along the rotor and casing, demonstrating consistency between the modeling approaches. Materials were selected based on the predicted thermal loads: Inconel 738 for the hot-side components, Ti–6Al–4V for the cold-side aerodynamic surfaces, Inconel 718 for the highly loaded covers, and AISI 410 for the shaft. The presented workflow provides a systematic numerical framework for the thermal assessment of aft-fan modules and offers new insights into heat-load management and temperature-driven design constraints in next-generation mini and micro turbojet propulsion systems.

Keywords: aft fan module, mini turbojet engine, CFD–FEA thermal analysis, CFD-assisted heat transfer, secondary air system, temperature distribution.

INTRODUCTION

Gas turbines remain essential propulsion systems for aircraft across a wide range of operating conditions. Increasing global demand for energy efficiency has driven significant research toward improving thermal performance, reducing specific fuel consumption (SFC), and extending component life (1). Among the factors affecting cycle efficiency, turbine inlet temperature is particularly critical; higher inlet temperatures improve the

thrust-to-weight ratio, but simultaneously impose stringent material and cooling requirements (2). Consequently, numerous studies have examined the thermal behavior, cooling effectiveness, and durability of turbine blades and discs. For example, turbine blade surfaces can withstand temperatures up to 1000 °C, whereas blade tips experience substantially lower allowable limits due to local stress concentration and thermal gradients (3). Taamneh showed that enhanced convective cooling and heat-pipe–assisted designs can

markedly reduce disc temperatures, highlighting the importance of effective thermal management (4). Experimental work has also contributed to understanding the thermo-mechanical deformation of rotating components, as shown by Li et al., who developed a digital image correlation system capable of capturing thermal deformation on high-speed discs with high accuracy (5).

Recent efforts in micro-scale propulsion have focused on integrating fan stages into micro turbojets to improve endurance and reduce SFC. Studies indicated that adding a fan or converting micro turbojets into turbofan-like architectures can significantly enhance performance in small unmanned aerial vehicle (UAV) propulsion regimes. Hwang et al. (6) reported substantial SFC reductions in a turbofan derived from a micro turbojet, while Large and Pesyridis (7) demonstrated that adding a fan module can reduce SFC by more than 60%. Despite these promising results, aft fan configurations have not yet been widely implemented, primarily due to the thermo-mechanical challenges at the hot-cold interface. In aft fan modules, the rotor simultaneously encounters hot turbine gases and a cold fan stream, producing strong through-disc thermal gradients. Ensuring component durability therefore requires realistic modeling of cooling flows, rim-seal behavior, and secondary air system (SAS) performance. Prior research emphasized that inadequate purge flow can lead to hot-gas ingestion, whereas excessive purge penalizes system efficiency (8).

In contrast to the existing studies, which have mainly addressed micro-turbojet to micro-turbofan conversions at the cycle or performance level—focusing on thrust and specific fuel consumption improvements through added fan stages or geared modules—and separate investigations of secondary air systems, rim seals, and gas-turbine disc heat transfer the present work provides an integrated CFD–FEA thermal assessment of an aft-fan module specifically retrofitted to a mini turbojet engine. Building upon the aerodynamic design reported by Özkan et al. (9), which quantified the thrust and SFC benefits of the aft fan module concept, this study coupled (i) 1D radial disc-conduction models, (ii) 2D face–bore–rim heat-transfer formulations and (iii) three-dimensional CFD-derived convection fields from both hot turbine gases and the cold fan stream into a unified framework. By explicitly including SAS purge flows and rim-seal behavior under the thermal boundary conditions—rather than treating them as

simplified or uniform cooling sources—the methodology captures the strong through-disc thermal gradient at the turbine–fan interface under realistic operating conditions. This integrated approach allows the temperature distribution on the rotor, stators, casing, and nozzle to be resolved consistently across all scales, thereby linking system-level aft fan module performance to local thermal loads and material limits. As such, the study bridged a gap between micro-engine performance optimization and detailed CFD-assisted thermal modeling, offering a transferable workflow for the thermal design and numerical assessment of aft-fan modules in next-generation mini and micro turbojet propulsion systems. The present study bridged this gap by providing an integrated hierarchical thermal-analysis framework that combines one-dimensional and two-dimensional analytical formulations with one-way CFD-to-FEA thermal mapping for an aft-fan module integrated into a mini turbojet engine. The main contributions of the present study are as follows:

- Development of a hierarchical thermal-analysis methodology combining one-dimensional conduction–convection calculations, two-dimensional radial disc-conduction modeling, and three-dimensional steady-state solid thermal analysis using spatially varying heat-transfer coefficients obtained from CFD.
- Full quantification of HTC distributions, heat-flux fields, and numerical uncertainties, addressing limitations in uncertainty reporting in prior work.
- Integration of SAS purge flow, rim-seal effectiveness modeling, and temperature-dependent material properties, which are essential for realistic mini-engine thermal prediction.
- Quantitative comparison of analytical and numerical results, including error metrics, radial regression, and mesh-independence grid convergence index (GCI) analysis for both CFD and solid domains.
- This comprehensive methodology provides a verification-oriented thermal-design approach for next-generation mini turbojets and micro turbofans.

MATERIALS AND METHODS

This section describes the methodology used for the thermal calculations of the turbine, fan, casing, and disc. Given that the air entering the fan

blades originates from the compressor, and the air entering the turbine blades consists of combustion gases, the calculations employed forced convection formulas. The aft fan module consisted of an aerodynamic passage that connects the high-pressure turbine (HPT) to low pressure turbine (LPT), hot nozzle, fan, and cold nozzle. After the design of the aft fan through 1D calculations, 2D flow path optimization and streamline calculations, and 3D CFD calculations, the computer aided design (CAD) drawing of the module was carried out as shown in Figure 1. The secondary air system was also designed and is illustrated in the CAD model by the yellow and red arrows. A hybrid polyhedral-prism mesh containing 4.7 million elements was generated for the CFD analyses. The mesh consisted of 12 near-wall prism layers. The first-layer height was adjusted to maintain a y^+ value between 0.8 and 1.2. The shear stress transport (SST) turbulence model was employed. The mesh quality was verified with a global skewness below 0.29 and an orthogonality above 0.32. Turbulence intensity was 5%, turbulence length scale was $0.1 \times$ hydraulic diameter for fan side and for turbine side turbulence intensity was 10% with swirl component included based on the stage outlet v_θ predictions.

The fan and LPT rotor passages were defined as rotating fluid domains in ANSYS CFX, whereas the stator and downstream duct passages were treated as stationary domains. Both rotating domains were assigned a rotational speed of 38,000 rpm about the engine axis. The interaction between the rotating and stationary blade-row domains was modeled using the Stage

(Mixing-Plane) interface approach, in which the flow variables are circumferentially averaged across the rotor–stator interfaces. This steady-state rotating-frame treatment accounts for the effects of rotor motion on the mean flow field without explicitly resolving unsteady blade-passing interactions. All stationary walls were defined using no-slip boundary conditions, while the rotor-blade and hub surfaces were defined relative to their corresponding rotating reference frames.

The hot- and cold-flow passages were modeled as separate steady-state computational domains in ANSYS CFX. The boundary-condition values were obtained from the preceding cycle and AxSTREAM design calculations and were subsequently prescribed in the CFX simulations. For the low-pressure turbine (LPT) domain, a total pressure of 191 kPa and a total temperature of 1045 K were specified at the inlet, whereas a static pressure of 113 kPa was imposed at the outlet. For the fan domain, a mass flow rate of 1.75 kg/s and a total temperature of 288.15 K were prescribed at the inlet, while a static pressure of 101.325 kPa was specified at the outlet. The inlet and outlet boundary conditions, as well as the associated fluid properties are summarized in Table 1, and their locations on the computational domains are illustrated in Figure 2.

Secondary flow system design

The general purpose of a secondary air system (SAS) is to control internal cooling flows, including cooling, seal integrity, pressurization,

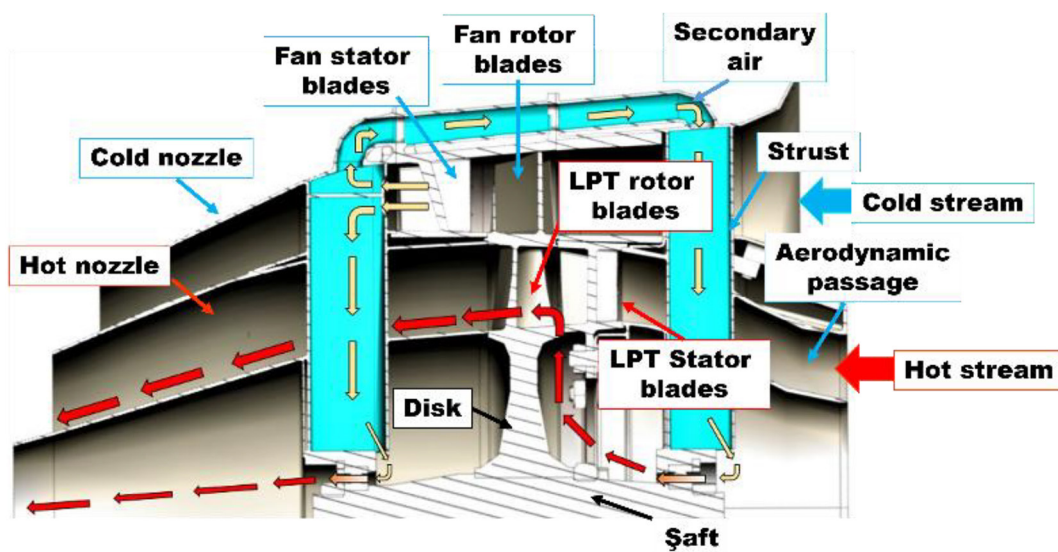


Figure 1. CAD drawing of the aft fan module

Table 1. Boundary conditions and fluid properties used in the ANSYS CFX simulations

Specification	Unit	Fan domain	LPT domain
Inlet boundary specification	–	Mass flow rate	Total pressure
Inlet mass flow rate	kg/s	1.75	–
Inlet total pressure	kPa	–	191
Inlet total temperature	K	288.15	1045
Outlet boundary specification	–	Static pressure	Static pressure
Outlet static pressure	kPa	101.325	113
Specific heat ratio, (gamma)	–	1.400	1.333
Specific heat, (c_p)	kJ/kg·K	1.005	1.148
Gas constant, (R)	kJ/kg·K	0.287	0.287

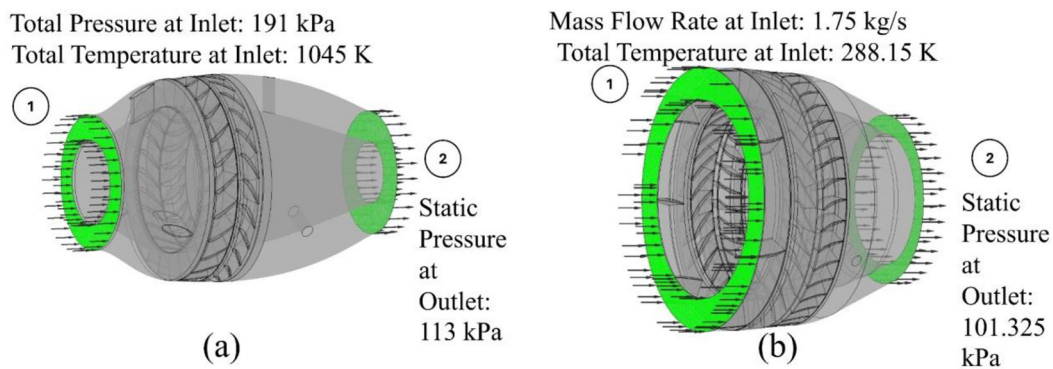


Figure 2. ANSYS CFX computational domains and prescribed inlet and outlet boundary conditions for the (a) LPT hot-flow passage and (b) fan cold-flow passage. Labels 1 and 2 indicate the inlet and outlet boundaries, respectively

and prevention of gas leaks. A well-designed SAS significantly increases engine efficiency and component lifespan. The air taken from the fan outlet of the aft fan module is split into two paths. The first path leads to the secondary flow air channels designed on the outer perimeter of the module, reaching the bearing region between the core engine and the module. Here, the air and fuel mixture, which cools the bearings and hot surfaces, blends with the primary flow through the gap between the turbine stator and rotor before exiting the system. The second path, which goes downwards, follows the connecting component and reaches the bearing located on the nozzle side. Here, the air and fuel mixture, which cools the bearings and hot surfaces, exits the system in the lower cavity of the hot nozzle (see the yellow and red arrows in Figure 1). A labyrinth-type rim seal was selected to prevent gas leakage between the fan and turbine because it reduces hot-gas ingestion while minimizing the required

purge flow. The design of the rim seal was based on Equations 1 and 2, where the ϕ_e represents the entrained air that joins the main flow, coming from the bottom and rising upwards. ϕ_{min} represents the minimum ingress value, which is the air attempting to enter. C_w is the dimensionless flow rate (10).

$$\phi_e = \frac{C_w}{2 \times \pi \times G_c \times Re_{\phi_e}} \quad (1)$$

$$\phi_{min} = \frac{C_{w_{min}}}{2 \times \pi \times G_c \times Re_{\phi_e}} \quad (2)$$

The minimum dimensionless cooling flow rate, $C_{w_{min}}$, was calculated using Equation 3. In this equation, the value of K was determined to be 0.46 based on the seal type. G_c represents the axial gap ratio, which was calculated as the ratio of the stator-rotor axial distance (0.0023 m) to the rotor disc radius (0.046 m), resulting in a value of 0.049.

$$C_{w_{min}} = 2 \times \pi \times K \times G_c \times P_{max}^{1/2} \quad (3)$$

The dimensionless pressure coefficient was calculated by Equations 4–7.

$$P_{max} = 1/2 \times C_{p,max} \times Re_w^2 \quad (4)$$

$$C_{p,max} = \frac{P_{hub,max} - P_{hub,min}}{0.5 \times \rho_1 \times C_1^2} \quad (5)$$

$$Re_w = \frac{C_1 \times \rho \times b}{\mu} \quad (6)$$

$$Re_\phi = \frac{\rho_1 \times \omega \times r_{disc}}{\mu} \quad (7)$$

The purge air mass flow rate was calculated for four stations which were shown in Figure 3. The calculated results for each station were presented in Table 2. The results indicated that the mass flow rate of the purge air was higher in stations 1 and 2, compared to stations 3 and 4. This higher mass flow rate was due to the elevated pressure levels and the increased cooling requirements at stations 1 and 2. The mass flow rates of the purge air obtained from the calculations were utilized in the determination of the secondary air mass flow rates using Equations 8 and 9. The determined secondary air mass flow rates were used in thermal calculations (see Table 2).

$$F_{f,t} = \sqrt{\frac{\gamma \times M^2}{\left(1 + \frac{\gamma-1}{2} \times M^2\right)^{\frac{\gamma+1}{\gamma-1}}}} \quad (8)$$

$$\dot{m}_{SA} = \frac{(A \times F_{f,t} \times P_{s,e})}{\sqrt{R \times T_{s,e}}} \quad (9)$$

1D and 2D thermal calculations

Temperatures were calculated separately for the rotor and stator at the hub, mid-span, and tip sections. To calculate the turbine and fan temperatures, the inlet flow velocity and the rotor and stator surface areas had to be determined first. The incoming airflow velocity to the turbine was specified separately for the hub, mid, and tip, as shown in Table 3. These velocities were obtained from Ansys CFX CFD analysis. The areas used in these calculations were $6,883.36 \times 10^{-6} \text{ m}^2$ and $6,188.96 \times 10^{-6} \text{ m}^2$, respectively. When computing the hub region temperature, the first step involved calculating the Reynolds number. The objective here was to ascertain whether the flow was laminar or turbulent for turbine temperature calculations. The Reynolds number was calculated by using Equation 10.

$$Re = \frac{\rho V D_h}{\mu} \quad (10)$$

where: ρ is the density V is the average velocity, D_h is the hydraulic diameter and μ is dynamic viscosity and Re is the Reynolds number.

If the result is $Re < 2300$ it was considered as laminar flow. In the $Re > 4000$, it was considered as turbulent flow (Çengel and Ghajor, 2015). The dynamic viscosity value used in Equation 10 was selected according to the reference value. The convection heat transfer coefficient of air was calculated with Equation 11.

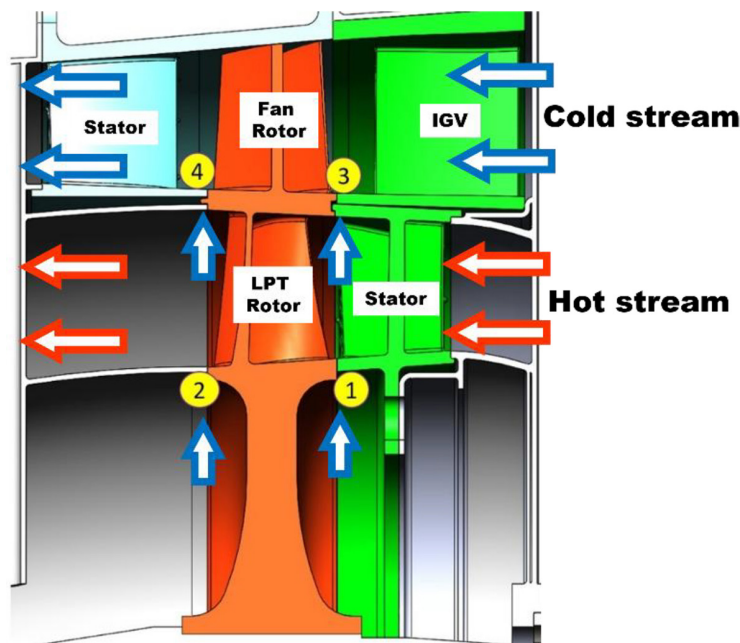


Figure 3. Design methodology of the aft fan module

Table 2. Purge and secondary air mass flow rate calculation results

Stations	$C_{p,MAX}$	R_{ew}	P_{max} (kPa)	Cw_{im}	$Re\phi$	ϕ_{min}	Cw	$\dot{m}_{purge}$ (kg/s)	$\dot{m}_{SA}$ (kg/s)
1	1.85	3.45	11.04	0.47	42.27	0.036	0.47	0.054	0.064
2	0.48	2.19	1.16	0.15	31.15	0.015	0.14	0.0167	0.024
3	13.41	1.44	13.98	0.13	42.27	0.01	0.13	0.015	0.036
4	0.22	4.65	2.37	0.05	92.71	0.002	0.05	0.006	0.0173

Table 3. Gas flow velocity through the turbine and fan annulus

Blade sections	TURBINE		FAN	
	ROTOR Flow velocity (m/s)	STATOR Flow velocity (m/s)	ROTOR Flow velocity (m/s)	STATOR Flow velocity (m/s)
Hub	419.22	180.22	294.228	121.252
Mean	447.60	181.44	332.276	170.135
Tip	453.26	183.02	389.493	143.817

$$h = \frac{Nuk}{D_h} \quad (11)$$

where: k was the thermal conductivity coefficient and D_h was the hydraulic diameter.

The symbol Nu denotes the Nusselt number and was calculated according to laminar and turbulent flow situations, as seen in Equations 12 and 13.

$$Nu = \frac{hl}{k} = 0.664Re^{0.5}Pr^{1/3} \quad (12)$$

$$Nu = \frac{hl}{k} = 0.037Re^{0.8}Pr^{1/3} \quad (13)$$

where: Re and Pr represent Reynolds and Prandtl numbers, respectively.

The flow condition required using Equation 12 while calculating the Nusselt number is laminar. Re and Pr number values should be in the $0.6 < Pr, Re < 5 \times 10^5$ range.

The flow condition required using Equation 13 while calculating the Nusselt number was turbulent. Re and Pr number values should be in the $0.6 < Pr < 60, 5 \times 10^5 < Re < 10^7$ range. The Reynolds number was calculated using Equation 10. The Prandtl number was obtained according to the conditions determined using the properties of air at 1 atm pressure. For the turbine rotor temperature calculations, the Reynolds number was first determined using Equation 10. The hydraulic diameter of the rotor passage was also calculated and used in the subsequent heat-transfer calculations. The hydraulic diameter was found by using Equation 14.

$$D_h = \frac{4A_c}{P_w} \quad (14)$$

where: A_c denotes the flow cross-sectional area and P_w denotes the wetted perimeter.

The blade surface temperatures at the inlet and outlet, together with the static pressure, were calculated using Equations 15–17.

$$T_{in} = T_1 - (Q \times R_{conv}) \quad (15)$$

$$T_{out} = T_1 - (Q \times (R_{conv} + R_{cond})) \quad (16)$$

$$P = \rho \times R \times T \quad (17)$$

where: ρ is the density (kg/m^3), R is the ideal gas constant and T is the temperature (K). Q , R_{conv} and R_{cond} (blades $R_{cond,B}$, other components $R_{cond,C}$) values in these equations express the total heat transfer rate, thermal resistance for convection and thermal resistance for conduction, respectively.

The formulas used in the calculation of these values were expressed in Equations 18–21.

$$Q = \frac{\Delta T}{\sum R_t} \quad (18)$$

$$R_{conv} = \frac{1}{h \times A_c} \quad (19)$$

$$R_{cond,B} = \frac{l}{k \times A} \quad (20)$$

$$R_{cond,C} = \frac{\ln \frac{r_2}{r_1}}{2 \times k \times \pi \times l} \quad (21)$$

where: r_2 and r_1 represent the radius values and the l value represents the length.

The k value represents the thermal conductivity coefficient and Inconel738 material was based on in this study. ΔT is the temperature difference and $\sum R_i$ (K/W) was the total thermal resistance. The symbol h was the value of the convection heat transfer coefficient and was found using Equation 11. A_c represents the area. The adiabatic wall temperature is the temperature attained by a wall under adiabatic flow conditions. Equation 22 was used when calculating the adiabatic wall temperature represented by T_{aw} .

$$T_{aw} = T_{\infty} + \frac{r \times V_{rel}^2}{2 \times c_p} \quad (22)$$

where T_{∞} was the free stream fluid temperature and V_{rel} was the free stream fluid velocity relative to the wall.

The expression r in the equation was related to the value of the Prandtl number. If the flow is turbulent, then $Pr^{1/3}$ was used, and if the flow was laminar, $(Pr)^{1/2}$ taken in consideration. c_p value was the specific heat value at constant pressure.

The 1D heat transfer calculations for the face, bore, and rim regions of the disc, with the fictitious boundary node defined as $i = 1$ at the bore and $i = N+2$ at the rim, are presented in Equations 23–36 (11). The thermal calculation formulas for the face region of the disc are given in Equations 23–28, the thermal calculation for the bore region is provided in Equation 29, and the thermal calculations for the rim region are presented in Equations 30–36.

$$\begin{aligned} \delta m_i c_{m_i} \frac{T_{m_i} - T_{m_i}(\tau)}{\Delta \tau} = & \frac{A_{i_1}(k_{m_i} + k_{m_{i-1}})}{2(r_i - r_{i-1})}(T_{m_{i-1}} - T_{m_i}) - \\ & - \frac{A_{i_2}(k_{m_i} + k_{m_{i+1}})}{2(r_{i+1} - r_i)}(T_{m_i} - T_{m_{i+1}}) - \\ & - h_{i_1} S_{i_1}(T_{m_i} - T_{aw_{i_1}}) - h_{i_2} S_{i_2}(T_{m_i} - T_{aw_{i_2}}) \end{aligned} \quad (23)$$

Equation 23 can be written in the following form:

$$b_i T_{m_{i-1}} + d_i T_{m_i} + a_i T_{m_{i+1}} = c_i \quad (24)$$

where: $i = 2, 3, \dots, N+1$ and;

$$b_i = - \frac{A_{i_1}(k_{m_i} + k_{m_{i-1}})}{2(r_i - r_{i-1})} \quad (25)$$

$$\begin{aligned} d_i = & - \frac{A_{i_1}(k_{m_i} + k_{m_{i-1}})}{2(r_i - r_{i-1})} + \frac{A_{i_2}(k_{m_i} + k_{m_{i+1}})}{2(r_{i+1} - r_i)} + \\ & + h_{i_1} S_{i_1} + h_{i_2} S_{i_2} + \frac{\delta m_i c_{m_i}}{\Delta \tau} \end{aligned} \quad (26)$$

$$a_i = - \frac{A_{i_2}(k_{m_i} + k_{m_{i+1}})}{2(r_{i+1} - r_i)} \quad (27)$$

$$c_i = \frac{\delta m_i c_{m_i} T_{m_i}(\tau)}{\Delta \tau} + h_{i_1} S_{i_1} T_{aw_{i_1}} + h_{i_2} S_{i_2} T_{aw_{i_2}} \quad (28)$$

where: A_{i_1} is area of conduction from node $i-1$ to node i , A_{i_2} is area of conduction from node i to node $i+1$, c_m is temperature-dependent specific heat of disc material, h_{i_1} is the heat transfer coefficient on Face 1 of node i control volume, h_{i_2} is the heat transfer coefficient on Face 2 of node i control volume. k_{m_i} , $k_{m_{i-1}}$ and $k_{m_{i+1}}$ are disc material thermal conductivities evaluated at temperatures T_{m_i} , $T_{m_{i-1}}$ and $T_{m_{i+1}}$, respectively. r_i is the radius of node i , r_{i-1} is the radius of node $i-1$, r_{i+1} is the radius of node $i+1$, S_{i_1} is the convection surface area on Face 1 of node i control volume, S_{i_2} is the convection surface area on Face 2 of node i control volume. T_{m_i} , $T_{m_{i-1}}$ and $T_{m_{i+1}}$ are the disc temperature at node i , $i-1$ and $i+1$, respectively. $T_{aw_{i_1}}$ is the adiabatic wall temperature on Face 1 of node i control volume. $T_{aw_{i_2}}$ is the adiabatic wall temperature on Face 2 of node i control volume. δm_i is the mass of node i control volume, τ is the time and $\Delta \tau$ is the time step.

The convective boundary condition was specified with h_{bore} and $T_{aw_{bore}}$ in Equation 29.

$$\begin{aligned} & \left(\frac{2k_{m_{bore}}}{r_2 - r_1} + h_{bore} \right) T_{m_1} - \\ & - \left(\frac{2k_{m_{bore}}}{r_2 - r_1} + h_{bore} \right) T_{m_2} = 2h_{bore} T_{aw_{bore}} \end{aligned} \quad (29)$$

The convective boundary condition was specified with h_{rim} and $T_{aw_{rim}}$ in Equations 30–32.

$$\begin{aligned} & k_{m_{rim}} \left(\frac{T_{m_{N+1}} - T_{m_{N+2}}}{r_{N+2} - r_{N+1}} \right) = \\ & = h_{rim} \left(\frac{T_{m_{N+1}} - T_{m_{N+2}}}{2} - T_{aw_{rim}} \right) \end{aligned} \quad (30)$$

$$\begin{aligned} & \left(\frac{2k_{m_{rim}}}{r_{N+2} - r_{N+1}} \right) (T_{m_{N+1}} - T_{m_{N+2}}) = \\ & = h_{rim} (T_{m_{N+1}} + T_{m_{N+2}} - 2T_{aw_{rim}}) \end{aligned} \quad (31)$$

$$\begin{aligned} & - \left(\frac{2k_{m_{rim}}}{r_{N+2} - r_{N+1}} - h_{rim} \right) T_{m_{N+1}} + \\ & + \left(\frac{2k_{m_{rim}}}{r_{N+2} - r_{N+1}} + h_{rim} \right) T_{m_{N+2}} = 2h_{rim} T_{aw_{rim}} \end{aligned} \quad (32)$$

Substituting Equation 32 with Equations 25–28 for $i = N+2$ yields Equations 33–36 in this case.

$$b_{N+2} = -\frac{2k_{rim}}{r_{N+2}-r_{N+1}} + h_{rim} \quad (33)$$

$$d_{N+2} = -\frac{2k_{rim}}{r_{N+2}-r_{N+1}} + h_{rim} \quad (34)$$

$$a_{N+2} = 0 \quad (35)$$

$$c_{N+2} = 2h_{rim}T_{awrim} \quad (36)$$

3D Steady state thermal analysis

The present analysis employed a one-way CFD-to-FEA thermal-mapping approach, rather than a fully coupled conjugate heat-transfer solution. First, the hot- and cold-flow domains were solved independently under steady-state conditions in ANSYS CFX. The resulting spatially varying heat-transfer-coefficient and fluid-temperature fields were then exported and applied as convection boundary conditions to the corresponding surfaces of the solid model in ANSYS Mechanical. No iterative or bidirectional data exchange was performed between the fluid and solid solvers; consequently, the changes in the predicted solid temperatures were not fed back into the CFD solution. The term “CFD-assisted solid thermal analysis” was therefore used throughout the revised manuscript to describe the present methodology.

In this study, a three-dimensional steady state thermal analysis was performed on the entire casing utilizing Ansys Mechanical software to investigate the system’s thermal behavior. The calculated convection and conduction coefficients were applied to evaluate thermal distribution on the module parts. Aft fan module geometry was meshed through the tetrahedral mesh with 1984418 elements and 3609624 nodes, as shown in Figure 4. Average skewness and aspect ratio of mesh were found to be 0.35 and 2.41, respectively. To eliminate ambiguity, HTC fields were exported from CFD as spatially varying profiles. Hot side HTC ranged 420–1280 W/m²·K and cold side HTC ranged 80–340 W/m²·K. Furthermore, radial and axial conduction couplings were defined explicitly. Contact conductance between disc–shaft surfaces were found to be 7500 W/m²·K. Although radiation contributes 8–12% of total heat transfer at 1045 K, it was included as

Emissivity (ϵ)= 0.78 (Inconel 738), σT^4 boundary used with view-factor approximation, and effect on metal temperature was –13.4 K correction.

Mesh independence and verification via GCI

A three-level, single-parameter mesh family was used to demonstrate grid independence of the steady state solid thermal solution. Material properties, contact definitions, and the imported HTC were kept identical for all meshes, only the global element size varied. The characteristic mesh scale for 3-D problems was defined with Equation 37.

$$h \approx (1/N)^{1/3} \quad (37)$$

where: N is the total element count. Refinement ratios were found using Equation 38.

$$r_{12} = \frac{h_1}{h_2}, r_{23} = \frac{h_2}{h_3} \quad (38)$$

Quantities of interest (QoIs) were the maximum temperature T_{max} , minimum temperature T_{min} and the temperature spread $\Delta T = T_{max} - T_{min}$.

From Table 4, r_{12} and r_{23} were calculated as 1.246 and $r_{23}=1.274$, and the average ratio was found to be $r=(r_{12}+r_{23})/2=1.260$, respectively. Medium/Fine changes were very small: $|T_{max,3}-T_{max,2}|/T_{max,3}=0.097\%$ and $|\Delta T_3-\Delta T_2|/\Delta T_3=0.136\%$ (Table 5). GCI is a widely accepted method for quantifying numerical uncertainty due to grid discretization in CFD and finite element analysis (FEA) analyses. It is based on Richardson extrapolation and provides a systematic measure of how

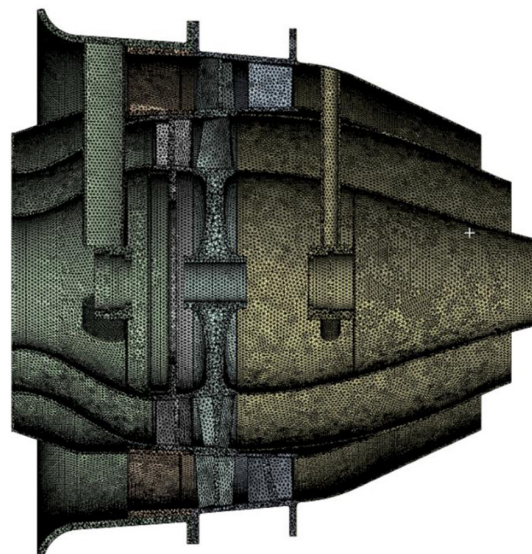


Figure 4. Meshing of the aft fan for the steady state solid-thermal analyses

close a numerical solution is to the asymptotic convergence range as the mesh is refined (12). By reporting GCI values together with refinement ratios and apparent order of accuracy, mesh independence and the reliability of simulation results can be objectively demonstrated.

$$ap = \frac{\ln \left| \frac{\omega_3 - \omega_2}{\omega_2 - \omega_1} \right|}{\ln r} \quad (39)$$

with $r = (r_{12} + r_{23})/2$.

Richardson extrapolation can be calculated using Equation 40,

$$\omega_{ext} = \omega_3 + \frac{\omega_3 - \omega_2}{r^p - 1} \quad (40)$$

and GCI can be determined by Equations 41 and 42 with safety factor (F_s) = 1.25.

$$GCI_{12} = 100F_s \frac{|\omega_1 - \omega_2|}{\omega_2(r^p - 1)} \quad (41)$$

$$GCI_{23} = 100F_s \frac{|\omega_2 - \omega_3|}{\omega_3(r^p - 1)} \quad (42)$$

Asymptotic-range indicator was calculated by using Equation 43. The calculated GCI results are shown in Table 6.

$$R = \frac{GCI_{12}}{GCI_{23}r^p} \quad (43)$$

The very small Medium/Fine differences and $R \approx 1$ for both QoIs confirm that the solution lies in the asymptotic convergence range; the thermal results are mesh-independent for the reported metrics. Therefore, all subsequent analyses and final figures in this study were reported on the medium mesh.

Materials

The efficiency of turbo machines with complex designs is intricately linked to material performance. Therefore, selecting the appropriate material in the design phase is of paramount importance. The components of the turbine, designed with the chosen material, should be capable of functioning under varying conditions of temperature, corrosion, and stress. Particularly, as turbine blades operate under high-temperature, high-stress, and corrosive conditions, ensuring their longevity is of utmost significance (13). In a mini turbojet engine, the blade edges and roots are subjected to fatigue loads throughout the entirety of a flight. Furthermore, there is a potential for fatigue induced by friction in the blade–disc attachment regions. The choice of robust materials is of paramount importance to withstand the stresses experienced in the blade regions. It has been established that through the implementation of a turbofan engine design with a high bypass ratio, the blades can contribute up to 80% of the total thrust. In the design of mini turbojet engine blades, which have earned their place as innovative components in aviation, titanium (Ti) and aluminum (Al) alloy materials are commonly favored (14). This section encompasses the selection of materials for the components, the properties of which are detailed in Table 7. These selections were made by adhering to the temperature values determined through analysis and considering factors such as operational conditions and stress.

The selected materials for the aft-fan module components are presented in Figure 5. When selecting a material for the air duct through which

Table 4. Mesh levels and QoIs

Mesh	Global size, s [mm]	Elements, N		T_{max} [K]	T_{min} [K]
Coarse (1)	2.00	157.937	0.01850	976.16	284.43
Medium (2)	1.48	305.647	0.01485	979.91	284.38
Fine (3)	1.10	945 K	1.04	981.12	284.40

Table 5. Refinement ratios and medium/fine changes

r_{12}	r_{23}	$r_{avg.}$	T_{max} change [%]	Delta T change [%]
1.246	1.274	1.260	0.097	0.136

Table 6. GCI results ($F_s = 1.25$)

Metric (ω)	Apparent order ap	Richardson (ω_{ext})	GCI12 [%]	GCI23 [%]	R
ΔT	0.84	657.87 K	0.97	0.8	1.00
T_{max}	0.54	945 K	1.04	0.92	1.00

air passes at high temperatures and pressures, it is crucial to optimize for a material the maximum stress values of which do not degrade under high-temperature conditions. Inconel 738 was chosen as the material based on the temperatures observed on the surface during the CFD analyses conducted on the geometries. The inlet guide vane (IGV) section serves as the component responsible for regulating the airflow in front of the fan blades, providing cold thrust. Since this part was not subjected to high-temperature conditions, special consideration was not given to the choice of high-temperature-resistant materials. Instead, Ti6Al4V alloy, which boasts high tensile strength and a lower density compared to iron-containing alloys, was chosen as the material for IGV. For the turbine stator, where airflow experiences high temperatures and pressures, selection of a material that maintains maximum stress values at elevated temperatures is imperative. Inconel 738 was selected as the material based on the temperatures observed on the surface

during CFD analyses conducted on the geometries. The fan stator was positioned behind the fan rotor blades, where it contributed to generating cold thrust and extended to cover the rotor blade tips. It also served to connect the intake and nozzle sections. As this part operates under the conditions that do not involve high temperatures, the preference was not for high-temperature-resistant materials. Instead, the fan stator blades were selected from Ti6Al4V alloy, which boasts high tensile strength and a lower density compared to iron-containing alloys. For the nozzle through which air passes at high temperatures and pressures, selection of a material that maintains maximum stress values at elevated temperatures is imperative. Based on the temperatures observed on the surface during CFD analyses, selection of Inconel 738 material for the nozzle part was considered appropriate. AISI 410 stainless steel was chosen for the shaft. The air duct cover part is responsible for supporting the surface where the core engine and the fan module

Table 7. Properties of materials [15]

Materials	Density (g/cm ³)	Solidus/Liquidus (°C)	Yield strength (MPa)	Tensile strength (MPa)
Inconel 718	8.19	1260–1336	1100	1375
Inconel 738	8.11	1230	910	1055
Ti6Al4V	4.43	1604–1660	880	950
410 Stainless Steel (SS410)	7.80	450	415	450
304 Stainless Steel (SS304)	8.00	650	205	515

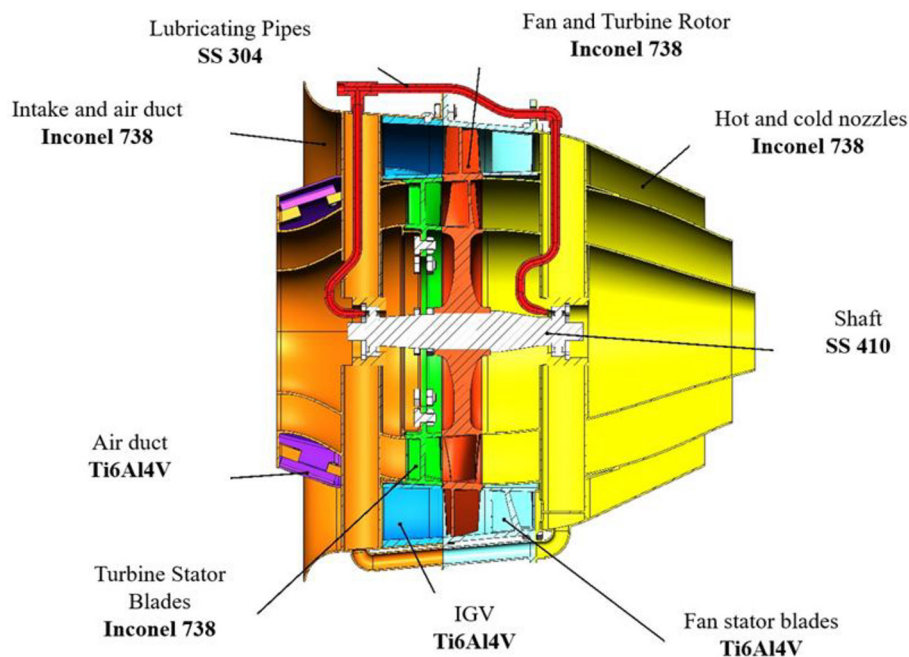


Figure 5. Material selection for aft fan module components

are joined, as well as closing the openings in the geometry to ensure smooth airflow in front of the fan. It was deemed suitable to employ two circular pieces that define each other on the module. The material chosen for this part was Inconel 718 alloy, as it had to withstand high temperatures and stresses. Through the thermal analysis conducted, it was established that the material possesses sufficient strength properties. ANSYS CFX was used to perform the CFD analyses of the hot- and cold-flow domains. The shear-stress-transport $k-\omega$ turbulence model was employed in the simulations. The flow domains were constructed using SOLIDWORKS and AxSTREAM and subsequently transferred to ANSYS SpaceClaim to define the inlet, outlet, wall, and fluid-domain surfaces required for the CFX analyses.

RESULTS AND DISCUSSION

CFD flow-field characteristics

Representative static-pressure contours obtained from the steady-state ANSYS CFX simulations are presented in Figure 6 for the fan cold-flow domain and the low-pressure turbine (LPT) hot-flow domain. In the fan domain shown in Figure 6(a), localized pressure variations occur across the rotor and stator blade rows, followed by a gradual pressure adjustment along the downstream flow passage. The pressure distribution reflects the aerodynamic loading of the fan stage and the subsequent expansion of the cold stream toward the outlet. In the LPT domain shown in Figure 6(b), the static pressure decreases across

the turbine blade rows and continues to decrease along the downstream hot-flow passage. This behavior is consistent with the expansion of the hot gas through the turbine stage and the prescribed outlet-pressure condition. The smooth pressure gradients in the downstream passages and the localized variations near the blade rows demonstrate the physical consistency of the computed pressure fields. These flow-field results also provide the aerodynamic basis for the spatially varying heat-transfer coefficients subsequently transferred to the solid thermal model.

The velocity-vector distributions for the fan and LPT computational domains are presented in Figure 7. In the fan cold-flow domain shown in Figure 7(a), the incoming flow accelerates as it passes through the rotor and stator blade rows. Local velocity gradients are observed within and immediately downstream of the blade passages, followed by a comparatively smooth flow development along the downstream duct. The maximum velocity magnitude in the fan domain reaches approximately 379 m/s. In the LPT hot-flow domain shown in Figure 7(b), the hot gas undergoes pronounced acceleration through the turbine passage and continues to accelerate toward the outlet as the flow area and static pressure change. The maximum velocity magnitude reaches approximately 428 m/s. The vectors remain predominantly aligned with the main flow direction in the downstream sections, while local changes in direction and magnitude occur near the blade rows. Together with the static-pressure distributions presented in Figure 6, these results demonstrate that the ANSYS CFX model

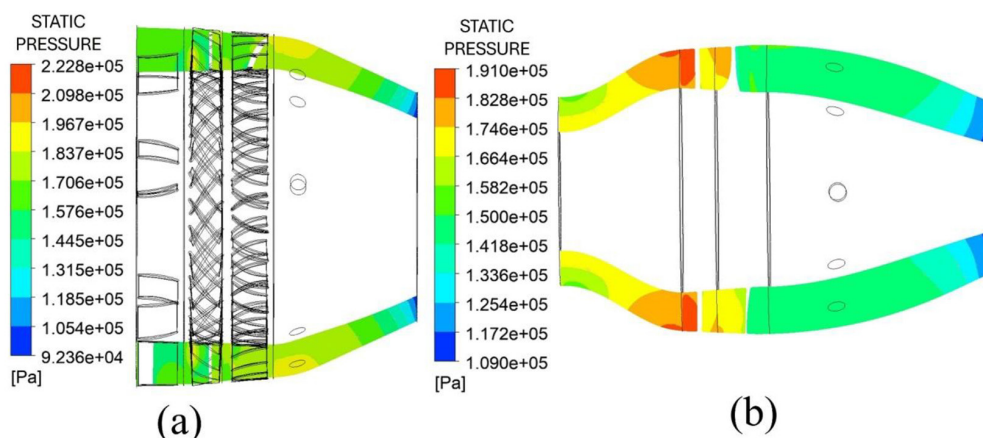


Figure 6. Static-pressure contours obtained from the steady-state ANSYS CFX simulations in axial sections of the (a) fan cold-flow domain and (b) LPT hot-flow domain. Pressure values are expressed in Pa, and each panel is presented using its own scale because of the different operating-pressure ranges of the two domains

captures the principal acceleration and pressure-redistribution mechanisms in both the hot and cold flow passages.

The thermal behavior of the aft fan module was investigated using a hierarchical modeling strategy that combined 1D analytical heat-transfer calculations, 2D radial disc-conduction formulations and three-dimensional CFD-assisted solid thermal simulations. The results reveal the strong interaction between the hot turbine stream, the cold fan flow and the shaft cooling provided by the secondary air system (SAS). To assess the physical credibility of the predictions, all temperatures, gradients and trends were evaluated in the context of previously published turbomachinery and thermal-management studies. The turbine rotor gas inlet and outlet temperature were obtained as 1045 K and 963 K from the cycle and AxSTREAM 2D streamline calculations, respectively. Reynolds number at the inlet and outlet, convection coefficient, thermal resistance, convection resistance and total heat flow were calculated by using the equations which were given in the previous section to calculate the surface temperatures at the blade hub, mid and tip sections. The average surface temperature was found to be 979.64/979.09/978.97 K whereas the outlet was calculated as 977.50/976.93/976.81 K (see Table 8) at the rotor blade hub, mid, and tip sections, respectively. Similarly, the fan rotor blades surface temperatures were found which were on the upside of the turbine blades on the same disc. The air temperature was obtained as 288.15 K and 328.6 K at the inlet and outlet of the fan rotor according to the AxSTREAM streamline calculations. The fan rotor blade surface temperatures were found to be 299.69 K, 301.66 K, and 298.87

K at the hub, mid and tip sections, respectively. Furthermore, turbine and fan inner and outer case temperatures were calculated as shown in Table 9. The turbine inner case surface temperature was 708.49 K whereas the outer case surface temperature was determined to be 534.37 K. Moreover, inner and outer adiabatic wall temperatures were found to be 1000.42 K and 310.07 K, respectively. The outer and inner case surface temperatures were determined to be 534.37 K and 708.49 K, respectively. Additionally, fan outer and inner adiabatic wall temperatures were calculated as 288 K and 366.62 K, respectively (see Table 10).

Similar calculations were carried out for the turbine and fan stator blade sections, including the hub, mid, and tip. At the inlet, the fan stator blade surface temperatures were found to be 299.69 K, 301.66 K, and 298.87 K for the hub, mid, and tip sections, respectively. Meanwhile, at the outlet, the surface temperatures of the stator blades were measured as 301.46 K, 313.57 K and 300.94 K for the hub, mid, and tip sections, respectively. As for the turbine stator blades, temperatures were calculated as 979.64/977.5 K, 979.09/976.93 K, 978.97/976.81 K at the inlet/exit of the hub, mid, and tip sections of the blades. Figure 8 shows the average temperature distribution on the aft fan module.

Temperature distribution on the rotor disc was obtained through slicing the disc into 10 sections. Figure 9 shows the radius and width sections of the aft-fan module rotor disc [16].

Table 11 displays the disc radius and width values along with the calculated thermal resistance and temperature for each section. For the disc temperature distribution calculations, the shaft temperature was assumed to be 400 K due

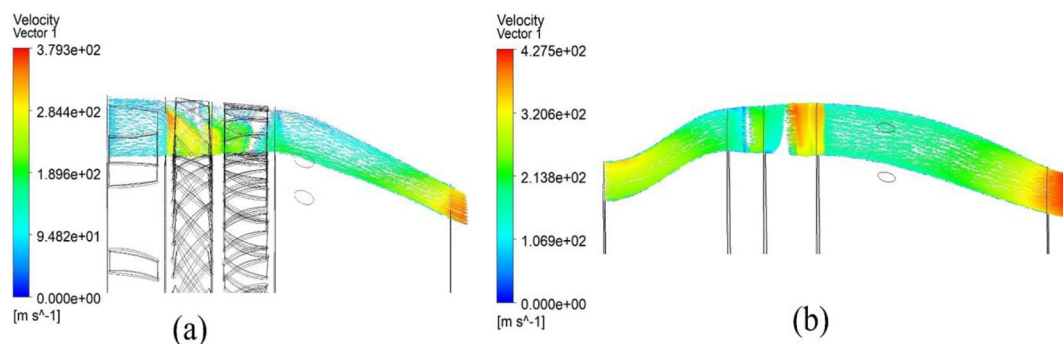


Figure 7. Velocity-vector distributions obtained from the steady-state ANSYS CFX simulations in axial sections of the (a) fan cold-flow domain and (b) LPT hot-flow domain. The vectors are colored according to velocity magnitude, expressed in m/s. Each panel is presented using its own scale because of the different velocity ranges of the two computational domains

Table 8. Turbine stator/rotor blade surface temperature calculations

Blade station	Reynolds number (R_g)	Convection coefficient ($W/m^2 K$)	Total heat flow (W)	Inlet temperature (K)	Outlet temperature (K)
Hub	229777.75 542316.39	221.30 896.59	89.52	979.64	977.50
Mid	231333.20 579029.70	222.06 944.84	90.58	979.09	976.93
Tip	231333.20 586351.60	222.06 954.38	90.73	978.97	976.81

Table 9. Fan stator/rotor blade surface temperature calculations

Blade station	Reynolds number (R_u)	Convection coefficient ($W/m^2 K$)	Total heat flow (W)	Inlet temperature (K)	Outlet temperature (K)
Hub	700410.2 2242853	408.47 880.62	100.33	299.69	301.46
Mid	937286 2300072	541.13 994.05	120.39	301.66	313.57
Tip	830756.6 2969043	468.22 1107.22	117.19	298.87	300.94

Table 10. Turbine/Fan outer and inner case temperatures with adiabatic wall temperatures

Reynolds number (R_e)	Convection coefficient ($W/m^2 K$)	Total Heat flow (W)	Outer case temperature (K)	Inner case temperature (K)	Outer case adiabatic wall temperature (K)	Inner case adiabatic wall temperature (K)
2097087 687167	868.28 1080.62	2012.30	534.37	708.49	310.07 1000.42	
- 4275762	1533.06 1087.66	0.27	288.17	298.84	288.00 366.62	

to the secondary air flow. Moreover, the turbine blade hub temperature was determined to be 977.5 K (see Table 11). Figure 10(a) shows the temperature distribution in the disc, whereas Figure 10(b) displays the temperature distribution on the rotor according to the 2D calculations. Temperature distribution plot showed linear temperature rise from the shaft to turbine rotor blade hub. Shaft temperature was 400 K and disc temperature rose to 998 K in the turbine rotor blade hub with the effect of hot stream. Temperature of LTP rotor blade decreased to 560 K at the hub section and then decreased to 299 K at the fan rotor blade tip

(see Figure 10(b)). The temperature distribution along the rotor disc radius can be expressed with a linear regression equation as (Equation 44):

$$T(R) = 315.40 + 12.70 \times R \quad (44)$$

where: T is the temperature in Kelvin and R is the radius in millimeters.

The regression analysis yielded a Pearson correlation coefficient of $r=0.997$ and an adjusted $R_2=0.994$, confirming an excellent linear relationship between radius and temperature. This result indicates that the temperature

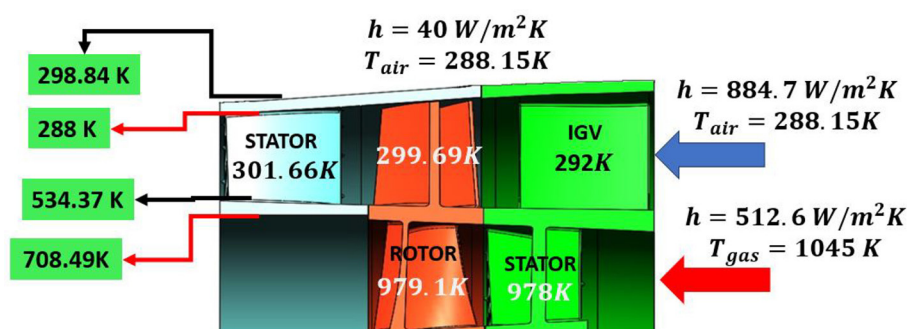


Figure 8. Temperature distribution of aft fan module

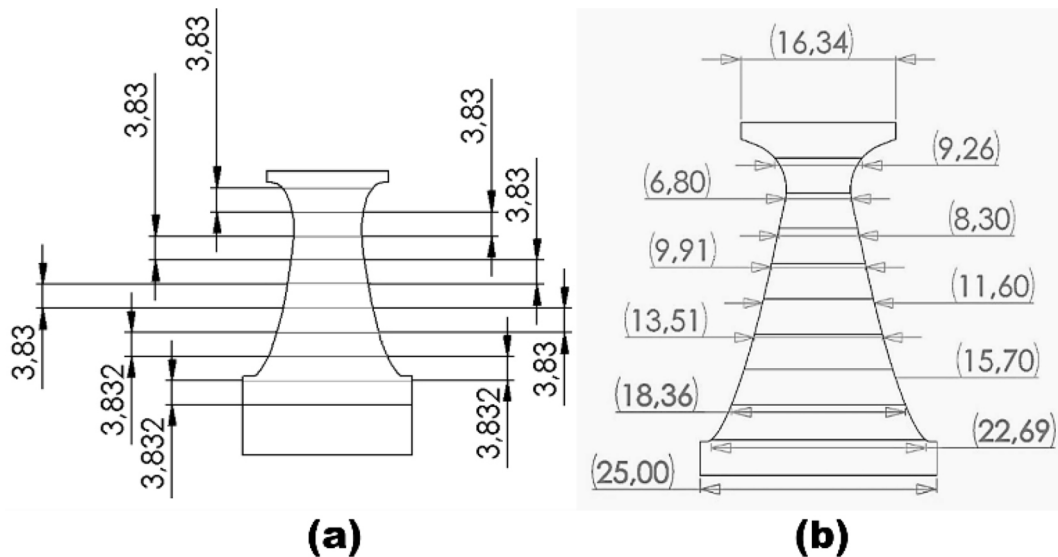


Figure 9. Aft fan rotor disc with sliced sections (a) radius, (b) widths

Table 11. Disc radius, width, thermal resistance, and temperature results

Radius (mm)	Width (mm)	Thermal resistance (K/W)	Temperature (K)
46.322	0.016	0.039	876.400
42.488	0.009	0.075	857.191
38.656	0.006	0.113	820.091
34.824	0.008	0.104	764.290
30.992	0.009	0.098	713.239
27.160	0.011	0.097	664.831
23.328	0.013	0.098	617.176
19.496	0.015	0.103	568.898
15.664	0.018	0.113	518.234
11.832	0.022	0.127	462.693
-	-	-	400.000

increases approximately 12.7 K per millimeter of radial distance, which reflects a strong radial gradient driven by the hot stream acting on the turbine side of the rotor.

3D steady state thermal analyses of aft fan were carried out by using the Ansys Mechanical module. The thermal analysis results were obtained because of CFD convection coefficients import as shown in Figure 11(a). The maximum metal temperature was detected on aerodynamic passage with 1021.1 K (see Figure 11(b)); however, on the fan side, the cold stream reduces the metal temperature to approximately 285 K at the inlet region, producing a strong hot-cold contrast across the module. Such a temperature split is in line with micro-jet measurements, which indicate

rotating blade temperatures around 980–1020 K under similar operating conditions (17). The thermal analysis results for the turbine section of the aft-fan module are shown in Figure 12.

The turbine section of the aft fan module exhibits a maximum metal temperature of 1018.2 K on the stator vanes (Figure 12(a)). For cast IN-738, published alloy data and creep-life studies indicate that long-term operation is feasible up to metal temperatures of about 1050–1100 K, well below its solidus temperature near 1230 °C. The predicted stator temperatures are therefore within the allowable range for the selected material, and the resulting Fs of 1.48 is consistent with typical design margins reported for cooled turbine hot-gas-path components. The rotor solution in Figure 12(b) reveals a peak metal temperature of 979.3 K on the turbine blades, decreasing toward the disc and then to ≈300 K on the fan side. This behavior closely follows the 1D/2D analytical trends and mirrors the through-disc gradients found in earlier conjugate studies of rotor–stator systems and cooled turbine stages (18). The agreement between the present numerical predictions and the thermal trends reported in the literature supports the physical plausibility of the predicted temperature distribution; however, this comparison should not be interpreted as independent experimental validation of the CFD–FEA framework. The disc temperature profile is then advanced radially from an assumed 400 K shaft with simplified through-thickness conduction, which tends to over-predict peak metal temperature near the turbine hub. By

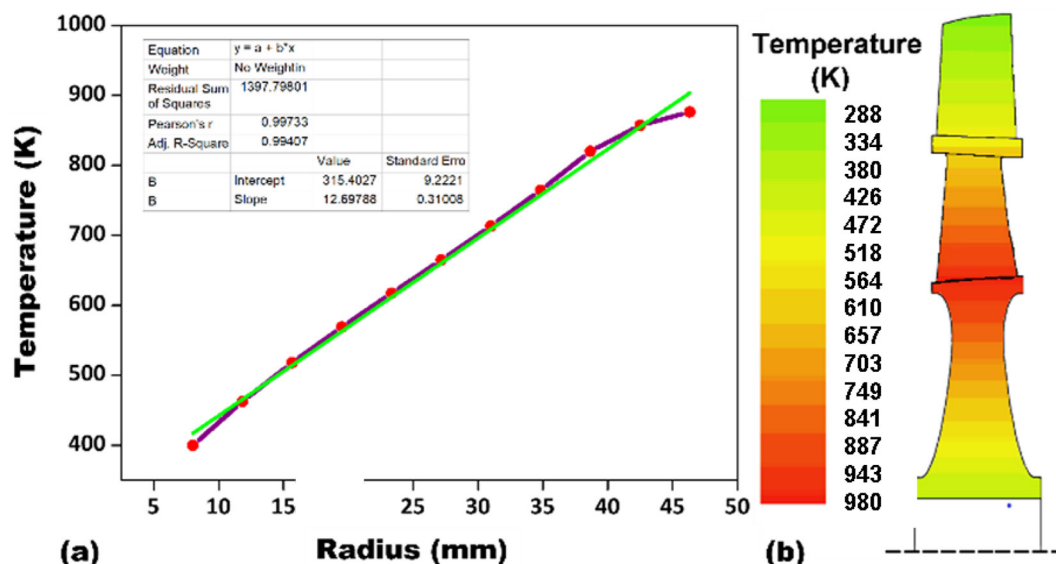


Figure 10. (a) Disc temperature distribution graph, (b) blade and disc temperature distribution

contrast, the 3-D steady state model includes 3-D conduction paths, contact networks, and mapped convection HTC—along with the 400 K secondary-air-system (SAS) cooling around the shaft and bearings—so heat is redistributed more realistically from hot turbine regions into cooler metal and SAS-washed zones, yielding a lower peak metal temperature of 979.28 K [19]. Moreover, temperature deviation can be seen because of heat transfer between hot and cold stream at the shroud and fan hub section fan rotor where temperature changed between 600–750 K. The close agreement between the analytical 1D/2D calculations and the three-dimensional numerical solution indicates internal consistency across the hierarchical modeling levels. This comparison is considered a cross-model verification, rather than an independent validation, because the approaches share common operating conditions, material properties, and boundary-condition inputs. The three-dimensional CFD-assisted thermal model, which includes detailed cooling paths, contact conduction and spatially resolved HTCs, predicts slightly lower peak metal temperatures than the simplified analytical models—a trend also observed in earlier conjugate studies, where multidimensional conduction spreads heat more effectively than 1D formulations assume.

The nozzle exhibited metal temperatures between 904 K and 981 K, the values that are fully compatible with the long-term temperature capabilities of Inconel 738 in hot-gas-path applications. The external shroud temperature decreased to approximately 400 K due to the impingement of

the cold annular flow, highlighting how the aft fan module configuration inherently provides thermal shielding for the nozzle support [20]. Comparable studies on cooled turbine exit nozzles reported metal temperatures in the 900–1000 K range under similar flow and heat-loading conditions, demonstrating that the present model's predictions are consistent with the established thermal behavior of nickel-based superalloy nozzles. The thermal distributions of the fan stator and nozzles are presented in Figure 13.

Table 12 presents a quantitative comparison between the 1D/2D analytical predictions and the three-dimensional CFD-assisted thermal results for the turbine rotor. The temperature deviations across the hub, mid, and tip regions remain remarkably small, within 0.04–0.16%, indicating that the analytical correlations—based on classical Nusselt relations, hydraulic diameter formulation, and adiabatic wall temperature estimates—capture the dominant heat-transfer mechanisms with high fidelity for the present geometry and operating conditions. The strong agreement is further supported by a coefficient of determination of $R^2 = 0.994$, demonstrating that the analytical model reliably reproduces both the magnitude and radial gradient of the thermal field. The slightly higher temperatures predicted by the three-dimensional thermal model are expected, as the 3D solution incorporates multi-dimensional heat-conduction pathways, contact resistances, and spatially varying HTCs that the simplified analytical formulations cannot fully represent. Nevertheless, the small discrepancies

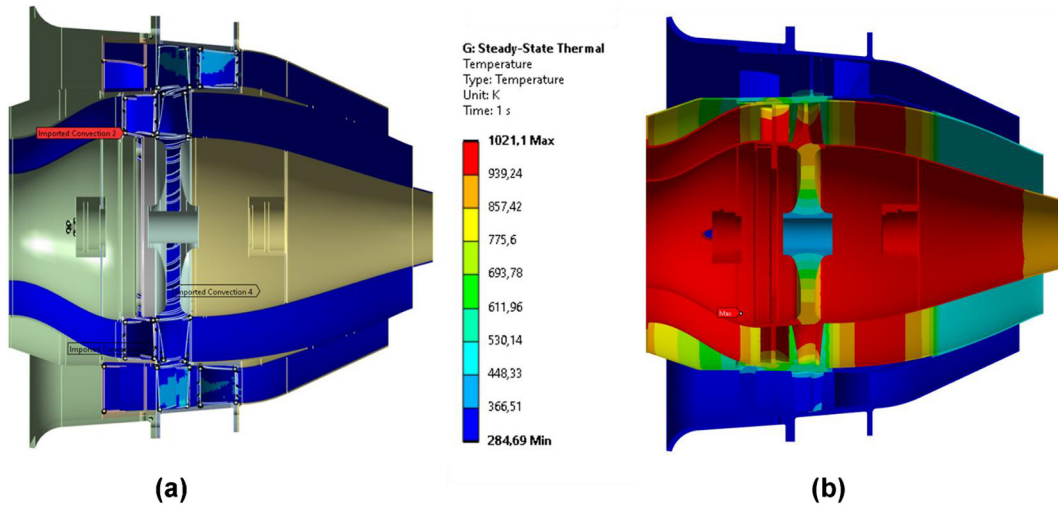


Figure 11. (a) convection coefficient imports from CFD results, (b) steady state thermal analyses of aft fan

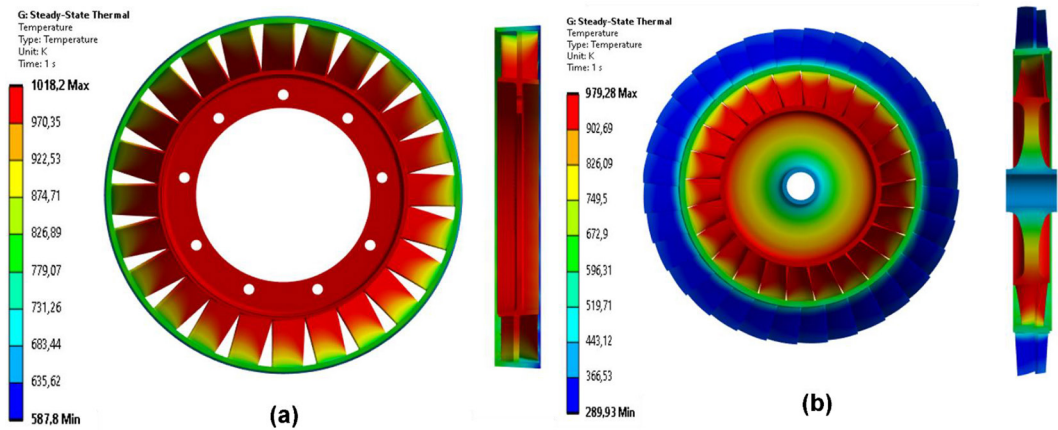


Figure 12. Thermal analyses result of the turbine section (a) stator and (b) rotor

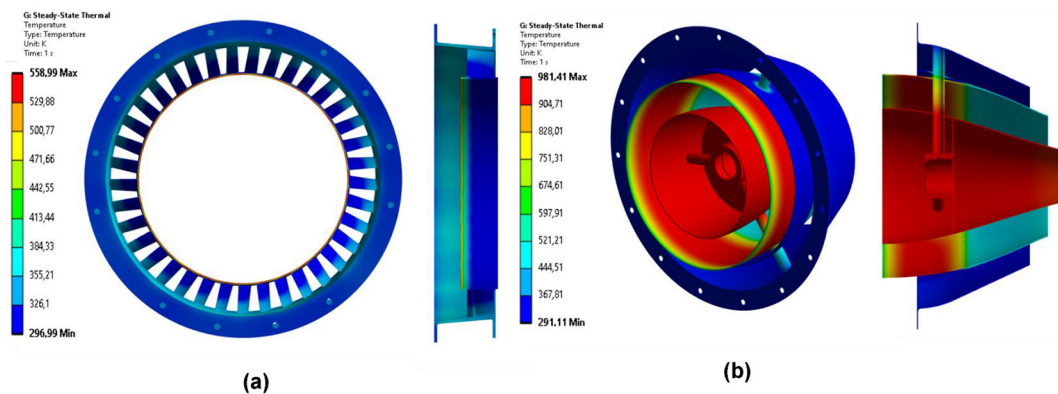


Figure 13. Thermal analyses result of the (a) fan stator and (b) nozzles

indicate that the analytical model reproduces the overall temperature levels and radial trends obtained from the three-dimensional solution. The analytical formulation may therefore be used for preliminary design estimates, whereas detailed thermal assessments should rely on the

three-dimensional model in the regions characterized by strongly non-uniform convection and multidimensional heat conduction. This comparison represents cross-model verification and does not constitute independent experimental validation.

Table 12. Comparison of analytically predicted metal temperatures (1D/2D thermal model) with three-dimensional CFD-assisted thermal results at the hub, mid, and tip sections of the turbine rotor

Region	1D/2D (K)	three-dimensional CFD-assisted thermal (K)	Error (%)
Turbine hub	979.6	981.2	0.16
Mid	979.1	979.9	0.08
Tip	978.9	979.3	0.04

The three-dimensional CFD-assisted thermal results revealed maximum surface heat flux: 1.82 MW/m² on turbine stator suction side while minimum heat flux was obtained as 0.12 MW/m² on fan hub region. Results showed that disc–shaft interface receives ≈ 140 kW of SAS cooling. Maximum thermal gradient was determined as $\Delta T = 681$ K across 46 mm radial distance with a gradient = 14.8 K/mm whereas analytical prediction was 12.7 K/mm with a +13% deviation from CFD results. Additionally, SAS purge effectiveness reduced disc inner radius temperature by ≈ 86 K which prevented ingestion by maintaining $\Delta P_{\text{seal}} = 1.4$ kPa.

CONCLUSIONS

This study presented an integrated one-way CFD-to-FEA thermal analysis of an aft-fan module designed for a mini turbojet engine by combining one-dimensional conduction–convection formulations, two-dimensional disc-temperature modeling, and three-dimensional CFD-assisted steady-state solid thermal simulations within a unified hierarchical framework. By mapping spatially varying heat-transfer coefficients from both the hot turbine stream and the cold fan flow, and by incorporating realistic SAS purge conditions and rim-seal behavior, the model successfully resolved the strong thermal gradients inherent to aft fan module retrofits. The results highlight the presence of a pronounced through-disc thermal gradient of approximately 681 K over a 46 mm radial distance, driven by the 1045 K turbine inlet temperature and the opposing cold fan flow. The predicted turbine-blade metal temperatures (~ 979 – 1018 K) and fan-side temperatures (~ 285 – 300 K) closely align with the analytical 1D/2D predictions, with deviations below 0.2% and an R^2 of 0.994. This close cross-model agreement demonstrates internal numerical consistency and supports the use of the analytical models for preliminary design estimates. Because the analytical and three-dimensional approaches share common input assumptions and boundary

conditions, their comparison is interpreted as verification, rather than independent validation. The three-dimensional CFD-assisted thermal analysis further revealed the spatial distribution of heat fluxes—reaching up to 1.82 MW/m² on turbine stator surfaces—and quantified the cooling contribution of the SAS, which reduced hub temperatures by approximately 86 K and maintained a favorable sealing pressure margin. Material assessments demonstrated that all components operate within the thermal capability limits of the selected alloys, particularly Inconel 738 for the hot gas-path and Ti–6Al–4V for the fan section. The aft fan module configuration inherently provided thermal shielding effects on the nozzle support structure, consistent with the trends reported in earlier cooled turbine and rotor–stator three-dimensional thermal studies. The modeling approach therefore provides a robust and transferable framework for thermal analysis of compact turbomachinery operating under mixed hot-cold environments, a regime where simplified thermal models are typically inadequate.

Overall, the findings establish a systematic numerical methodology for predicting temperature fields, thermal loads, and material constraints in next-generation mini and micro turbojet propulsion systems incorporating aft-fan module architectures. The demonstrated agreement between analytical and numerical solutions, together with quantified uncertainties, provides confidence in the use of this framework for design evaluation, optimization, and thermal risk assessment. Experimental validation was outside the scope of the present study and will be addressed in future work through component-level temperature measurements under representative operating conditions. Future work will extend the current steady state analysis to transient operational phases, such as startup and throttle changes, and will incorporate more detailed modeling of SAS leakage, rotating cavity flows and thermo-mechanical fatigue behavior to further enhance the predictive capability of the methodology.

Acknowledgements

The authors would like to thank to the Turkish Technology Team (T3) Foundation and Turkish Engine Industry company (TEI) for organizing the Teknofest 2022 Jet Engine competition. A special thanks go to Valentine Moroz from Softinway company for their support and assistance with the AxSTREAM software.

REFERENCES

1. Ibrahim TK, Basrawi F, Awad OI, Abdullah AN, Najafi G, Mamat R, et al. Thermal performance of gas turbine power plant based on exergy analysis. *Appl Therm Eng.* 2017;115:977–985. <https://doi.org/10.1016/j.applthermaleng.2017.01.032>
2. Taameh Y. A numerical analysis of gas turbine disks incorporating rotating heat pipes. In: *Proceedings of the ASME International Mechanical Engineering Congress and Exposition (IMECE)*; 2017. p. 61–68. Paper No: IMECE2017-26614.
3. Rajabinezhad M, Moosavi SMR. Corrosion and fatigue failure of gas-turbine blades in an oil-fired power plant. *Materials (Basel).* 2020;13(4):900. doi:10.3390/ma13040900
4. Taameh Y. Thermal analysis of gas turbine disc integrated with rotating heat pipes. *Case Stud Therm Eng.* 2017;10:335–342. <https://doi.org/10.1016/j.csite.2017.06.006>
5. Li G, Bao M, Ding S, Li Y. A system for accurate measuring thermal structure displacement on a high-speed rotating turbine disk using digital image correlation technology. *Appl Therm Eng.* 2017;113:36–46. <https://doi.org/10.1016/j.applthermaleng.2016.11.041>
6. Hwang Y, Kim S, Lee DE, Rhee DH, Chung H, Kang YS. Research of turbofan engine derived from micro turbojet engine. *Int J Gas Turbine Propuls Power Syst.* 2024;15(5). https://doi.org/10.38036/jgpp.15.5_v15n5tp07
7. Large J, Pesyridis A. Investigation of micro gas turbine systems for high-speed long-loiter tactical unmanned air systems. *Aerospace.* 2019;6(5):55. <https://doi.org/10.3390/aerospace6050055>
8. Owen JM, Zhou K, Pountney O, Wilson M, Lock G. Prediction of ingress through turbine rim seals—Part I: Externally induced ingress. *J Turbomach.* 2012;134(3):031012. <https://doi.org/10.1115/1.4003180>
9. Özkan D, Karaca G, Özekinci MC, Aydoğan K. An AFT FAN module design for mini turbojet engine. *Aust J Mech Eng.* 2025:1–13. <https://doi.org/10.1080/14484846.2025.2555143>
10. Sultanian B. *Gas turbines: internal flow systems modeling.* Cambridge (UK): Cambridge University Press; 2018.
11. Owen JM, Zhou K, Pountney O, Wilson M, Lock G. Prediction of ingress through turbine rim seals—Part I: Externally induced ingress. *J Turbomach.* 2012;134(3):031012. doi:10.1115/1.4003180
12. Celik IB, Ghia U, Roache PJ, Freitas CJ, Coleman H, Raad PE. Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. *J Fluids Eng.* 2008;130(7):078001. <https://doi.org/10.1115/1.2960953>
13. Kumar NV, Raju P, Srinivasulu P. Thermal analysis of a gas turbine blade. *Int J Mod Trends Eng Res.* 2016;3(8):2349–9745.
14. Zhang YK, Lu JZ, Ren XD, Yao HB, Yao HX. Effect of laser shock processing on the mechanical properties and fatigue lives of turbojet engine blades manufactured by LY2 aluminum alloy. *Mater Des.* 2009;30(5):1697–1703. <https://doi.org/10.1016/j.matdes.2008.07.017>
15. ASM International. *ASM handbook. Volume 2: Properties and selection—nonferrous alloys and special-purpose materials.* Materials Park (OH): ASM International; 1994.
16. Javiya UR, Wilson M. Coupled FE–CFD thermal analysis for a cooled turbine disk. *J Turbomach.* 2015;137(10):101001.
17. Yan W, Li T, Xing X, Wang X, Liu D. Experimental study on surface temperature and emissivity of rotating turbine blades of a micro turbojet engine. *Appl Therm Eng.* 2025;259:124811. <https://doi.org/10.1016/j.applthermaleng.2024.124811>
18. Okita Y, Yamawaki S. Conjugate heat transfer analysis of turbine rotor–stator system. In: *ASME Turbo Expo 2002: Power for Land, Sea, and Air*; 2002. Paper No: GT2002-30615. p. 1103–1113. <https://doi.org/10.1115/GT2002-30615>
19. Xu C, Amano RS. Aerodynamics and heat transfer in a turbine blade at design and off-design angles of incidence. In: *ASME Turbo Expo 2000: Power for Land, Sea, and Air*; 2000. Paper No: 2000-GT-0210, V003T01A018. <https://doi.org/10.1115/2000-GT-0210>