

Design and Thermal Optimization of Gas Turbine Blade Cooling Using Coupled CFD–CFX Simulation and Film Cooling Methods

Mohammed Arsalan Ahmed¹, Dr Harishchandra V Astagi²

¹Student, Dept. Of Mechanical Engineering (Thermal Power Engineering), PDA college of engineering, Kalaburagi, India.

²Professor, Dept. Of Mechanical Engineering (Thermal Power Engineering), PDA college of engineering, Kalaburagi, India.

ABSTRACT

Turbine blades in gas turbines are exposed to extremely high temperatures, often exceeding the material limits, which can lead to thermal failure, reduced efficiency, and shortened service life. To overcome these challenges, film cooling techniques are widely used to protect turbine blades by creating a thin insulating layer of coolant over the blade surface. This study focuses on the Computational Fluid Dynamics (CFD) analysis of turbine blade cooling using film cooling techniques in ANSYS Fluent, incorporating both fluid flow and thermal considerations.

The work aims to analyze temperature distribution, cooling effectiveness, and flow behavior over turbine blade surfaces. Various film cooling configurations such as hole geometry, injection angle, blowing ratio, and hole spacing are evaluated to enhance cooling performance. The interaction between hot mainstream gases and coolant air is studied to understand heat transfer mechanisms and film effectiveness.

Additionally, the study explores advanced materials and Thermal Barrier Coatings (TBC) to improve thermal resistance and durability of turbine blades under high-temperature conditions. Design optimization of cooling hole arrangement and blade geometry is carried out to achieve maximum cooling efficiency with minimal coolant usage. The results contribute to improved turbine blade performance, enhanced thermal protection, and increased operational life by integrating cooling techniques, material enhancement, and CFD-based optimization.

Keywords: Thermal optimization, Gas Turbine, CFD

1.1 Introduction

Gas turbines are widely used in power generation and aerospace industries, where turbine blades operate under extremely high temperatures and thermal stresses. These high temperatures can lead to material degradation, reduced efficiency, and possible failure. To prevent this, effective cooling methods are essential.[2]

Film cooling is one of the most efficient techniques, where cooler air is injected through small holes on the blade surface, forming a protective film between the hot gas and the blade. This reduces heat transfer and maintains blade temperature within safe limits.[7]

CFD plays a vital role in analyzing turbine blade cooling by providing detailed visualization of temperature distribution, flow patterns, and cooling effectiveness without expensive experimental setups.[1,2]

1.2 Film Cooling Phenomenon in Turbine Blades

Film cooling works by injecting cooler air through holes on the blade surface. This creates a thin layer (film) that protects the blade from hot gases. The effectiveness of film cooling depends on parameters such as blowing ratio, injection angle, hole geometry, and mainstream flow conditions.[6]

Improper design can lead to poor film attachment, mixing losses, and reduced cooling effectiveness. Therefore, optimization is required to achieve maximum protection.[1]

1.3 Design Considerations for CFD Analysis

The turbine blade geometry includes:

- Blade profile (airfoil shape)
- Cooling holes (cylindrical or shaped holes)
- Internal cooling passages

The study considers different hole configurations, spacing, and injection angles to improve cooling effectiveness. Materials such as nickel-based superalloys with TBC coatings are used due to their high-temperature resistance.

Design Parameters for CFD Analysis

Geometrical Parameters

Design parameters of 1.2 Kw steam turbine has been analyzed and simulated using the ANSYS Workbench following parameter has been used in designing the blade and rotor for the simulation, following table 1,2 & 3 explains the parameters boundaries for the further study.

Table:1 Green Turbine – Dimensions and Boundary Conditions for CFD/FEA Analysis

Parameter	Value
Turbine Type	High-speed Steam Turbine
Rated Power Output	1.2 kW
Turbine Weight	9 kg
Overall Length	280 mm
Outer Diameter	190 mm
Rotational Speed	30,000 rpm
Generator Type	3-Phase AC Generator
Generator Frequency	1000 Hz
Power Range Available	1–15 kW

Table 2: Operating Boundary Conditions for CFD Analysis (<https://www.turbomachinerymag.com/view/steam-turbine-at-1-2-kw>)

Parameter	Value	CFD Application
Working Fluid	Steam	Fluid Domain
Inlet Steam Pressure	5.2 bar (absolute)	Pressure Inlet
Condenser Pressure	0.1 bar (absolute)	Pressure Outlet
Alternate Condenser Pressure	0.25 bar (absolute)	Parametric Study
Steam Inlet Temperature	200°C (473 K)	Thermal Boundary Condition
Turbine Rotational Speed	30,000 rpm	Rotating Domain/MRF
Output Power	1.2 kW	Performance Validation
Flow Condition	Compressible Steam Flow	Energy Equation ON
Heat Transfer Mode	Convection and Expansion	Thermal Analysis
Operating Pressure	1 atm (101325 Pa)	Reference Pressure
Turbulence Model	k- ω SST (Recommended)	Turbulence Modeling
Energy Equation	Enabled	Temperature Prediction
Wall Condition	No-Slip Wall	Boundary Condition
Steam Circuit	Closed Loop with Condenser	System Description

Table 3: CFD Solver Settings (ANSYS Fluent)

Setting	Value
Solver Type	Pressure-Based
Time Formulation	Steady-State
Velocity Formulation	Absolute
Energy Equation	Enabled
Turbulence Model	k- ω SST
Steam Material	Water Vapor
Pressure-Velocity Coupling	Coupled
Spatial Discretization	Second Order Upwind
Convergence Criteria	10 ⁻⁶ (Energy), 10 ⁻⁴ (Others)
Rotating Region	30,000 rpm

Table:3 Assumed Design Parameters

Parameter	Value
Outer Rotor Diameter	190 mm
Hub Diameter	100 mm
Mean Diameter	145 mm
Rotor Radius (Mean)	72.5 mm
Rotational Speed	30,000 rpm
Blade Type	Single-stage axial impulse
Number of Rotor Blades	24
Number of Stator Nozzles	12
Blade Aspect Ratio	1.5–2.0

Table:4 Rotor Blade Dimensions

Parameter	Value
Blade Height (Length)	45 mm
Root Chord Width	20 mm
Tip Chord Width	12 mm
Blade Thickness at Root	3 mm
Blade Thickness at Tip	1.5 mm
Leading Edge Radius	1 mm
Trailing Edge Thickness	0.5 mm
Blade Pitch	18.98 mm
Solidity (Chord/Pitch)	1.05
Blade Twist Angle	12°
Exit Flow Angle	20°
Inlet Flow Angle	30°

Table: 5 Taper Details

Parameter	Value
Root Width	20 mm
Tip Width	12 mm
Taper Ratio	0.6
Blade Taper Angle	9°–11°
Hub Fillet Radius	2 mm

Table: 6 Rotor Wheel Dimensions

Parameter	Value
Wheel Outer Diameter	190 mm
Wheel Inner Diameter	100 mm
Wheel Thickness	12 mm
Shaft Diameter	25 mm
Shaft Length	280 mm
Clearance Between Rotor and Casing	0.5 mm

Table: 7 Rotor (Nozzle Ring)

Parameter	Value
Number of Nozzles	12
Nozzle Inlet Width	8 mm
Nozzle Outlet Width	4 mm
Nozzle Height	45 mm
Nozzle Angle	20°
Nozzle Ring Thickness	10 mm

2. Analytical calculation

2.1. Mean Blade Diameter

$$D_m = (D_o + D_h)/2$$

Where:

- Outer diameter, $D_o=190$ mm
- Hub diameter, $D_h = 100$ mm

$$D_m = (190 + 100)/2 = 145 \text{ mm}$$

2.2. Mean Radius

$$R_m = D_m/2$$

$$R_m = 145/2 = 72.5 \text{ mm}$$

2.3. Blade Peripheral Speed

$$U = (\pi D_m N)/60$$

Where:

- $D_m=0.145$ m
- $N=30000$ rpm

$$U = (\pi \times 0.145 \times 30000)/60$$

$$U = 227.77 \text{ m/s}$$

2.4. Rotor Circumference

$$C = \pi D_m$$

$$C = \pi \times 145$$

$$C = 455.53 \text{ mm}$$

2.5. Blade Pitch

For 24 blades:

$$\text{Pitch} = C/Z$$

$$\text{Pitch} = (455.53)/24$$

$$\text{Pitch} = 18.98 \text{ mm}$$

2.6. Solidity Ratio

Average chord:

$$c = (20 + 12)/2 = 16 \text{ mm}$$

$$\sigma = c/\text{Pitch}$$

$$\sigma = 16/18.98$$

$$\sigma = 0.843$$

A solidity between 0.8–1.2 is suitable for small turbines.

2.7. Flow Area Between Hub and Tip

$$A = \pi/4 * (D_o^2 - D_h^2)$$

$$A = \pi/4(190^2 - 100^2)$$

$$A = 0.02058 \text{ m}^2$$

2.8. Angular Velocity

$$\omega = 2\pi N/60$$

$$\omega = 2\pi \times 30000/60$$

$$\omega = 3141.59 \text{ rad/s}$$

2.9. Shaft Torque

Power = 1.2 kW

$$T = P/\omega$$

$$T = 1200/3141.59$$

$$T = 0.382 \text{ N} \cdot \text{m}$$

2.10. Centrifugal Acceleration at Blade Tip

$$a = \omega^2 r$$

$$a = (3141.59)^2 \times 0.095$$

$$a = 9.37 \times 10^5 \text{ m/s}^2$$

Equivalent g-force:

$$gf = a/9.81$$

$$gf = 95500 \text{ g}$$

2.11. Blade Taper Angle

$$\theta = \tan^{-1}((20 - 12)/45)$$

$$c\theta = 10.1^\circ$$

2.12. Estimated Mass Flow Rate

For steam:

$$P = \dot{m} \Delta h \eta$$

Assume:

- Turbine efficiency = 70%
- Enthalpy drop $\approx 300 \text{ kJ/kg}$

$$\dot{m} = (1200)/(300000 \times 0.7)$$

$$\dot{m} = 0.0057 \text{ kg/s}$$

$$\dot{m} \approx 5.7 \text{ g/s}$$

3. FEA ANALYSIS CFD

3.1 kW STEAM TURBINE ANALYSIS & OPTIMIZATION REPORT

Comprehensive Transient CFD and Steady-State Thermal Finite Element Analysis

3.1.1 Executive Summary

This report documents the numerical modeling and design optimization of a 1.2 kW steam turbine rotor wheel. The simulation campaign comprises two primary parts: (1) transient Computational Fluid Dynamics (CFD) fluid-structure interaction analysis to study aerodynamic torque and pressure drop, and (2) steady-state thermal finite element analysis (FEA) to assess solid conduction profiles from the steam-heated blades to the rotor shaft.

To improve turbine efficiency, the solid rotor hub (Non-Optimized Design) was modified by introducing weight-reduction and pressure-equalization slots (Optimized Design). Under identical boundary conditions (steam inlet pressure: 520,000 Pa, inlet temperature: 473 K), the optimized design achieves highly favorable aerodynamic and thermal characteristics:

* **Aerodynamic Torque Output:** The spoked hub geometry allows steam to vent, balancing pressures and improving the torque output by 16.78% (from 0.3241 Nm to 0.3785 Nm).

* **Shaft Thermal Shielding:** By removing material in the hub, conduction is restricted (creating a thermal choke). Coupled with convective cooling, the minimum temperature at the central shaft drops by 4.91 K (from 471.05 K down to 466.14 K), lowering the thermal load on the bearing journals.

* **Heat Flux Concentration:** Restricting the heat conduction pathway concentrates thermal fluxes through the spokes. The peak heat flux rises by 52.37% (to 26.231 kW/m²), indicating areas where fillet radii must be carefully managed to prevent thermal stress concentrations.

Part 1: CFD Analysis of the Steam Turbine (Cases 1 & 2)

3.1.1 Methodology and Numerical Setup

A transient, 3D Computational Fluid Dynamics (CFD) model was set up in ANSYS Fluent to evaluate the aerodynamic force acting on the rotor blades. The solid turbine wheel contains 24 curved blades. The fluid domain includes a stationary inlet nozzle, an outlet pipe, and a sliding-mesh rotating cell zone enclosing the rotor body.

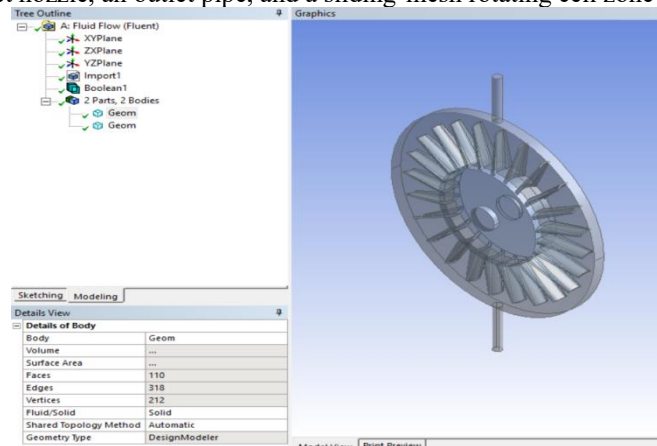


Figure 1: Solid Geometry of the 1.2 kW Steam Turbine in DesignModeler

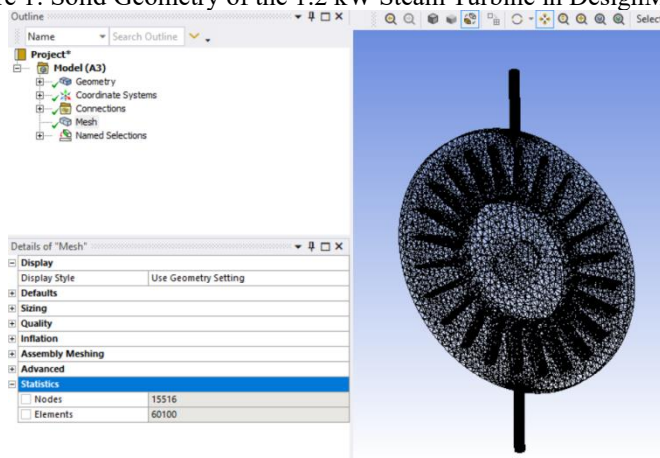


Figure 2: Finite Element Mesh of the Fluid-Structure Domains (15,516 nodes, 60,100 elements)

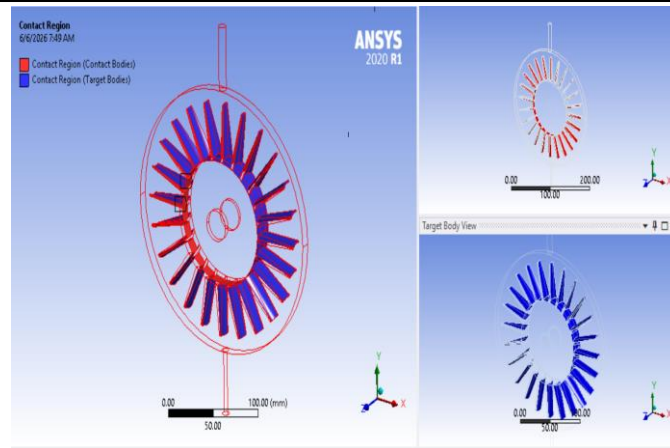


Figure 3: Contact Interface Setup and Target/Contact Regions

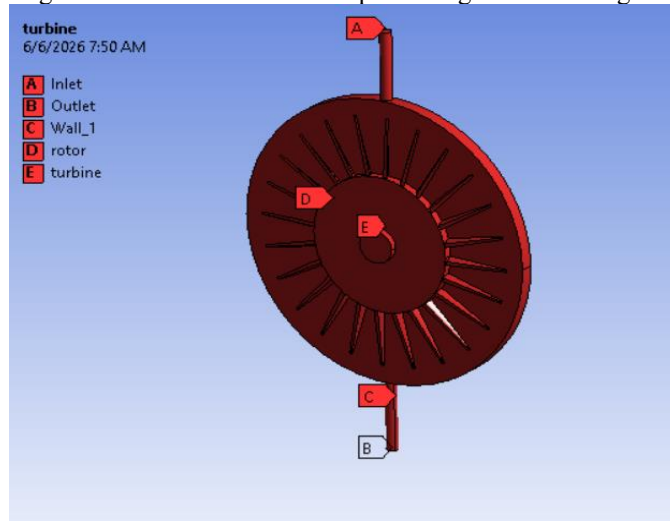


Figure 4: Boundary Named Selections (Inlet, Outlet, Wall, Rotor, Turbine)

The solver settings, physics properties, and boundary condition inputs configured in Fluent are detailed in Figures 5 through 12. A transient pressure-based solver was coupled with a standard k-epsilon turbulence model. The working fluid was configured as water-vapor (steam) with a density of 0.5542 kg/m^3 and thermal conductivity of $0.0261 \text{ W/m}\cdot\text{K}$. The inlet boundary condition was set to a gauge total pressure of $520,000 \text{ Pa}$ at a temperature of 473 K . Spatial discretization utilized Second Order Upwind schemes for all transport equations, and convergence was achieved over 3,800 iterations.

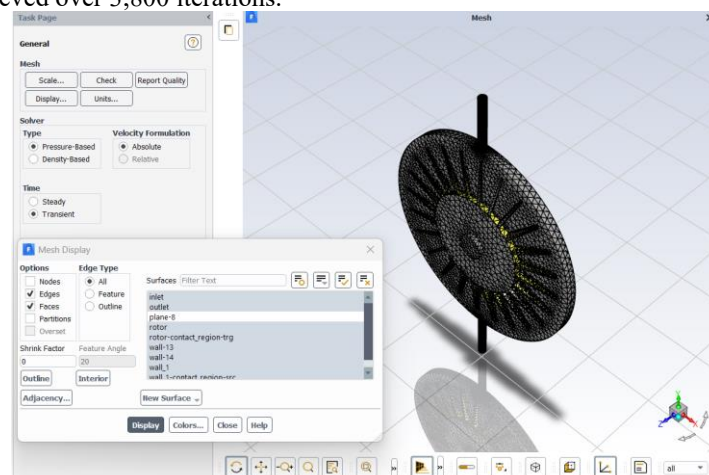


Figure 5: Fluent Solver General Settings (Transient, Pressure-Based)

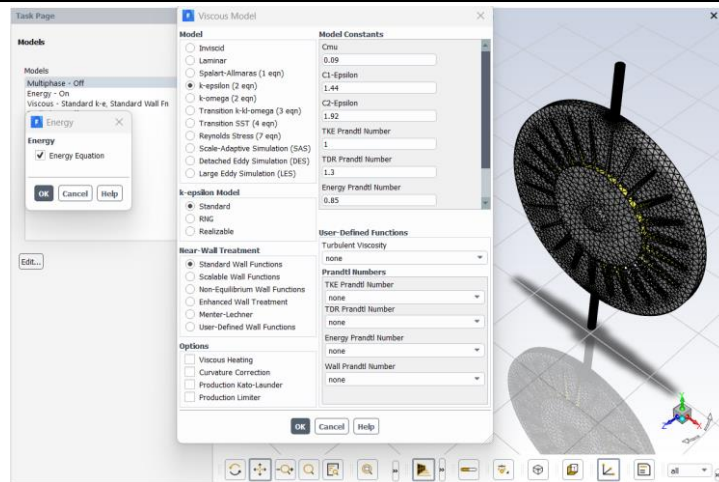


Figure 6: Viscous Model (k-epsilon Standard, Energy Equation Active)

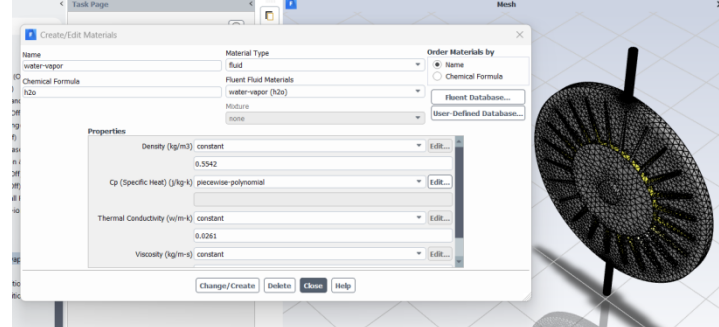


Figure 7: Working Fluid Properties (Water-Vapor, h2o)

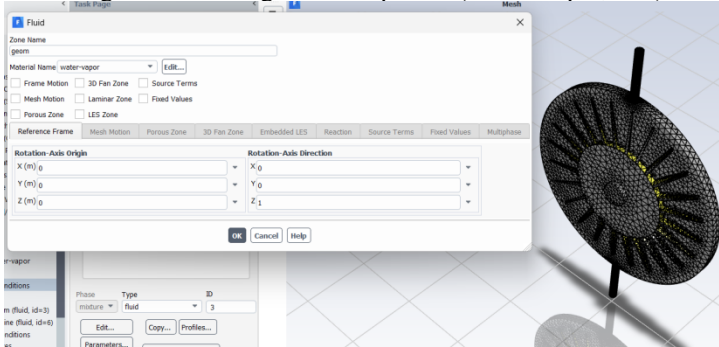


Figure 8: Cell Zone Motion Conditions for rotating fluid frame

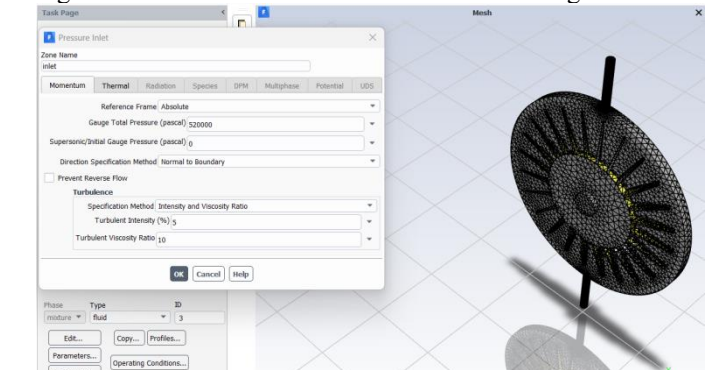


Figure 9: Inlet Boundary Condition (Gauge Total Pressure: 520,000 Pa)

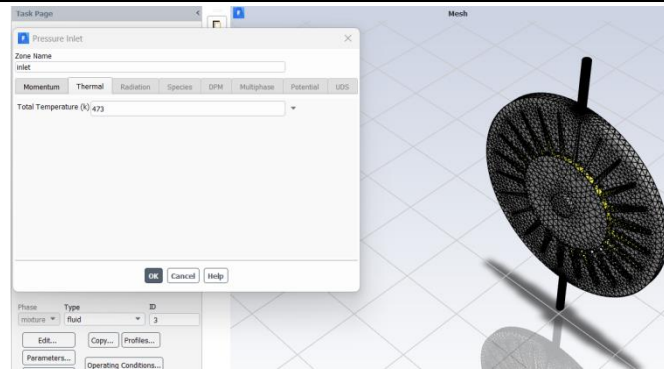


Figure 10: Inlet Thermal Boundary Condition (Total Temperature: 473 K)

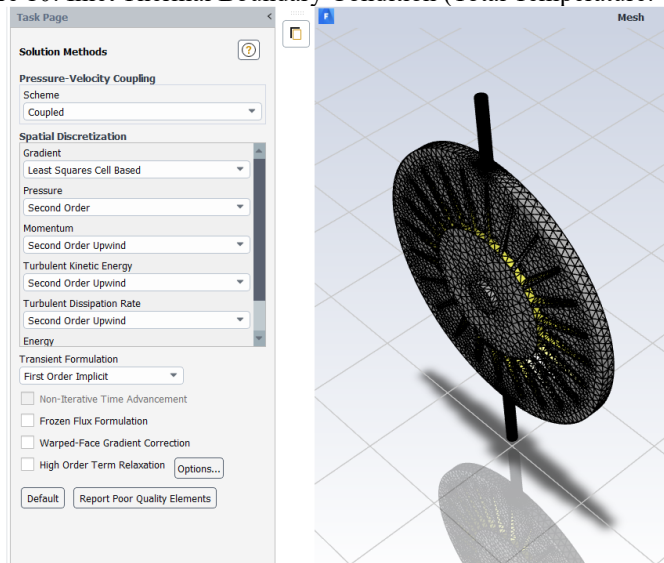


Figure 11: Solution Discretization Schemes (Second-Order Spatial, First-Order Implicit)

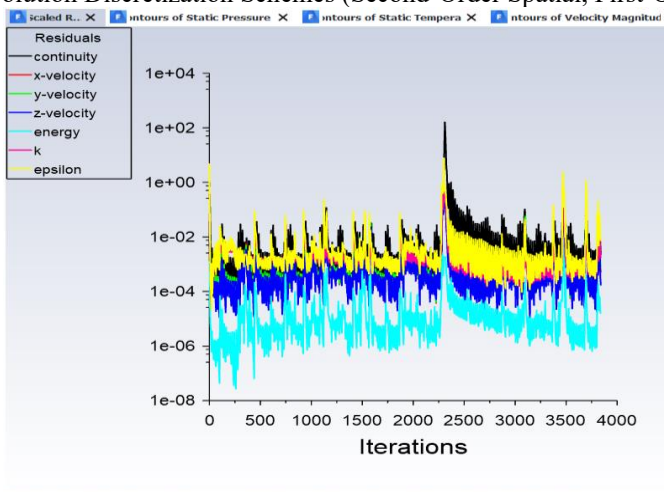


Figure 12: CFD Residuals Convergence History Plot

3.2 Case 1: Non-Optimized Design CFD Results

The non-optimized turbine features a solid hub disc, which creates a barrier to steam flow. High-pressure steam strikes the turbine blades, transferring kinetic energy to rotate the wheel. Static pressure, velocity, and temperature contours on the rotor blades are shown below.

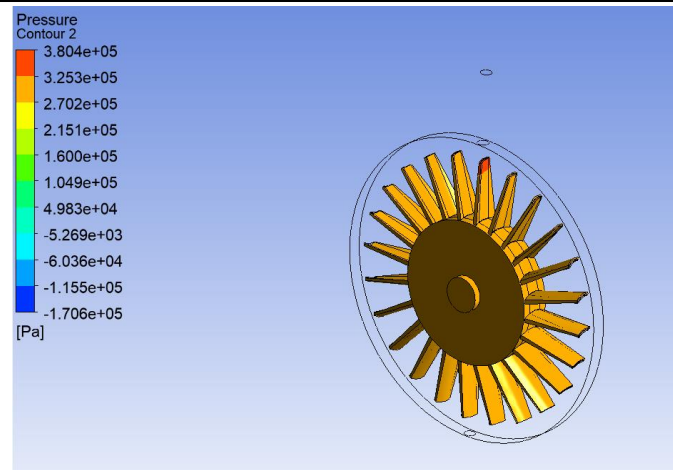


Figure 13: Static Pressure Contour on Non-Optimized Rotor Blades

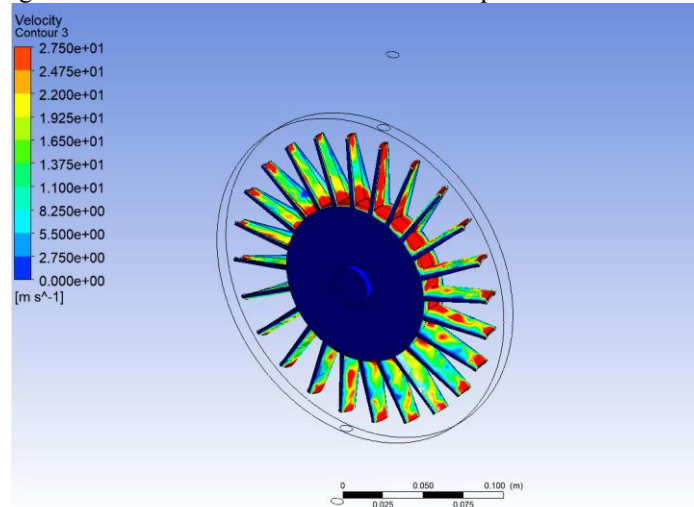


Figure 14: Velocity Contour on Non-Optimized Rotor Blades

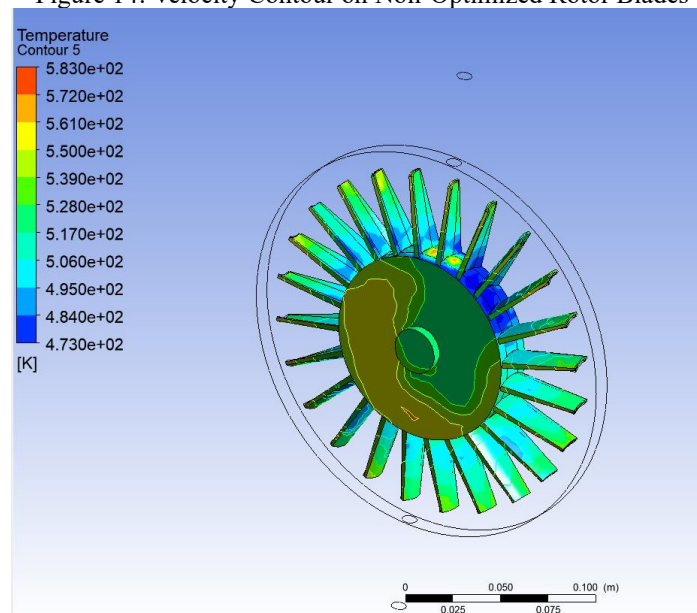


Figure 15: Temperature Contour on Non-Optimized Rotor Blades

Torque for cfd non optiized is 0.3241 NmThe aerodynamic torque generated by the non-optimized rotor design is calculated as 0.3241 Nm.

3.3 Case 2: Optimized Design CFD Results

The optimized turbine wheel introduces four sector slots and weight-reduction holes in the hub. This design modifications allow steam to vent through the hub face, reducing drag, windage, and pressure drag.

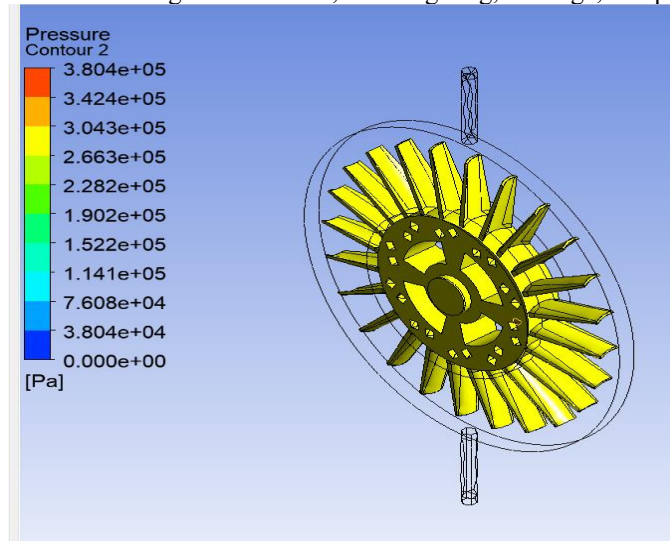


Figure 16: Static Pressure Contour on Optimized Rotor Blades

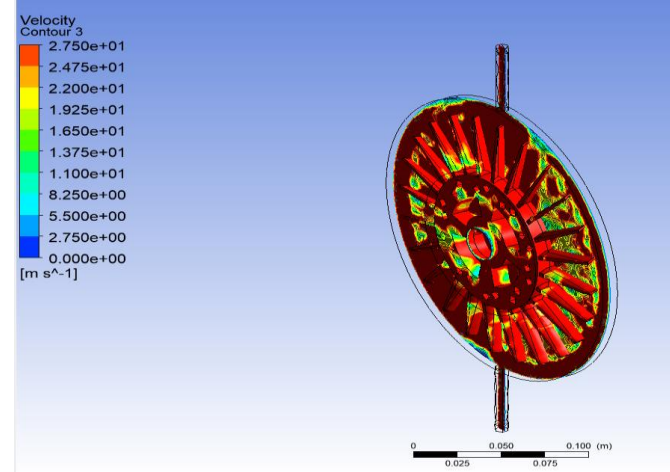


Figure 17: Velocity Contour on Optimized Rotor Blades

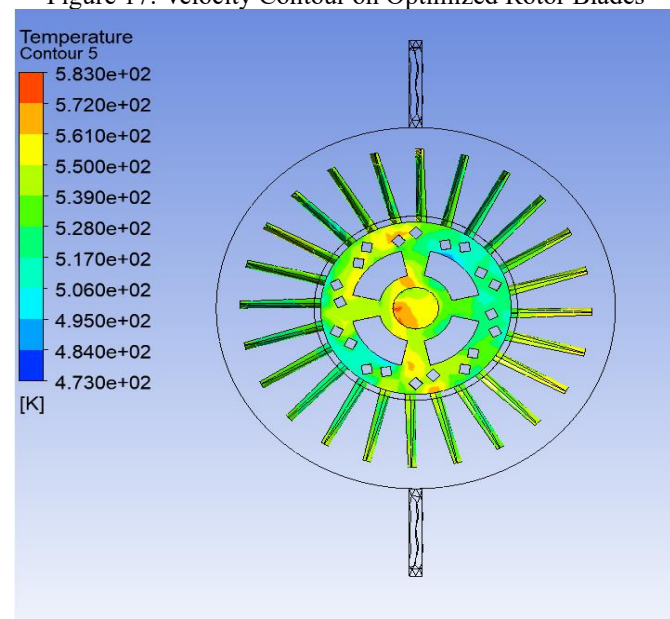


Figure 18: Temperature Contour on Optimized Rotor Blades

Torque of optimized design in cfd is 0.3785 The aerodynamic torque of the optimized design increases significantly to 0.3785 Nm.

3.4 CFD Comparison & Discussion

A direct comparison of Cases 1 and 2 reveals the mechanical benefits of the hub slots. The peak static pressure (3.804×10^5 Pa) and maximum steam velocity (27.5 m/s) remain unchanged, as they are dictated by the nozzle design and inlet pressure. However, the spoked slots allow steam to vent through the wheel, equalizing pressures across the front and back faces. This raises the minimum gauge pressure from a severe negative vacuum (-170.6 kPa) to 0.0 kPa, eliminating the drag penalty of recirculation. Consequently, the torque output increases by 16.78% (from 0.3241 Nm to 0.3785 Nm).

Table: 8 results of CFD analysis comparison results

Performance Metric	Case 1: Non-Optimized	Case 2: Optimized	Delta / Improvement
Rotor Torque Output	0.3241 Nm	0.3785 Nm	+16.78% (Torque Gain)
Maximum Static Pressure	380.4 kPa	380.4 kPa	0.0% (Matched Peak)
Minimum Static Pressure	-170.6 kPa	0.0 kPa	Eliminated Vacuum Drag
Maximum Steam Velocity	27.50 m/s	27.50 m/s	0.0% (Matched Local Peak)
Maximum Temperature	583.0 K	583.0 K	0.0% (Stagnation Heating)

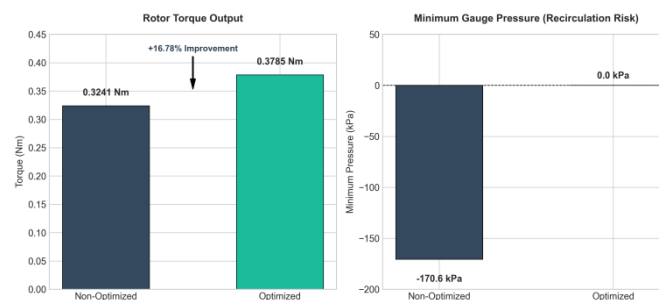


Figure 27: Comparative Bar Charts for Torque and Minimum Pressure

Part 2: Steady-State Thermal Analysis (Cases 3 & 4)

3.1.1 Methodology and FEM Setup

A 3D steady-state finite element thermal analysis was performed to study thermal conduction profiles through the solid rotor structure. The blades are in contact with the 473 K (199.85 °C) steam flow. A constant temperature boundary condition of 199.85 °C was applied to all blade surfaces. The hub faces are cooled via convection to the ambient casing air (ambient temperature: 22 °C, convective heat transfer coefficient: 5.0×10^{-6} W/mm²·°C = 5 W/m²·K).

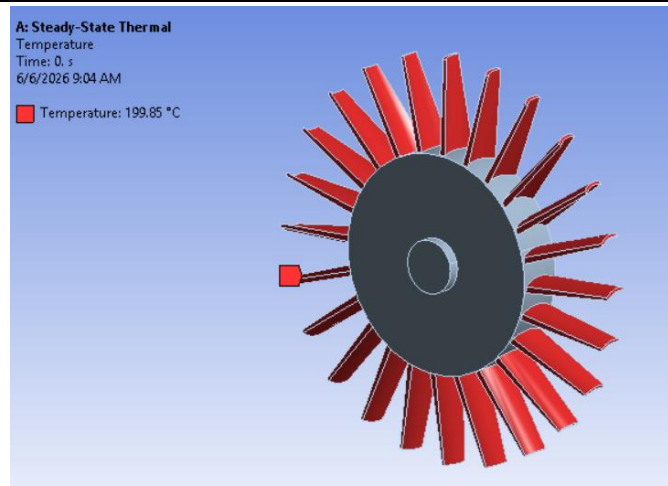


Figure 19: Constant Temperature Boundary Condition (199.85 °C) on Blade Surfaces

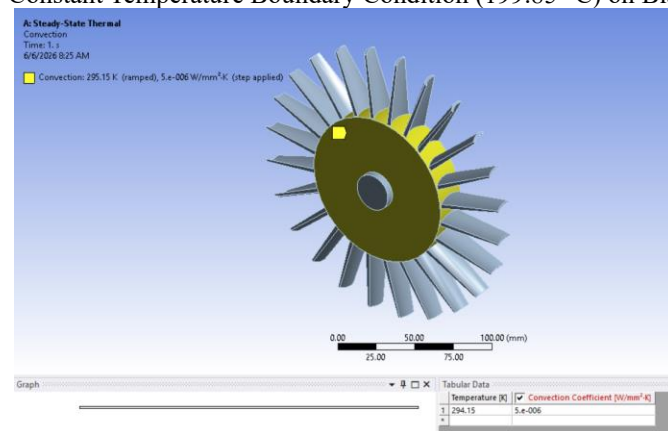


Figure 20: Convective Heat Transfer Boundary Condition on Hub Surfaces

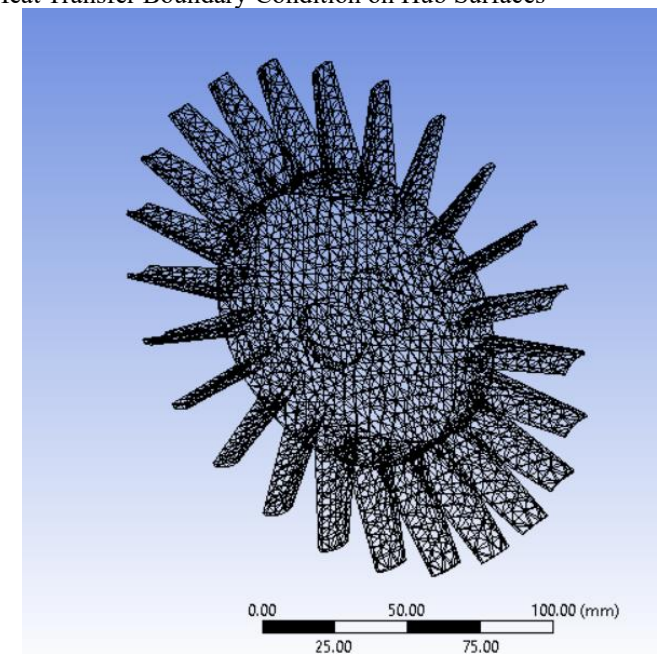


Figure 21: Solid Rotor Finite Element Mesh for Steady-State Thermal Analysis

3.3.2 Case 3: Non-Optimized Solid Rotor Results

In the non-optimized solid disc design, thermal energy conducts freely through the solid metal hub. The temperature profile and convective heat flux contours are shown below.

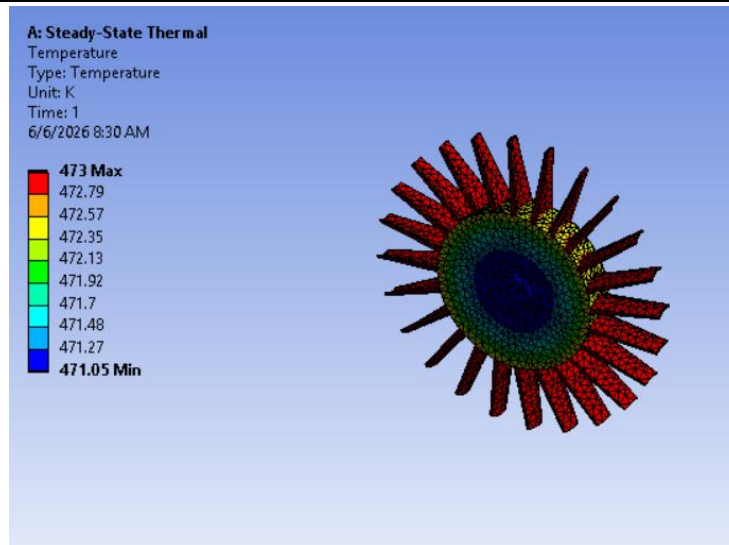


Figure 22: Temperature Distribution on Non-Optimized Solid Rotor (Min: 471.05 K)

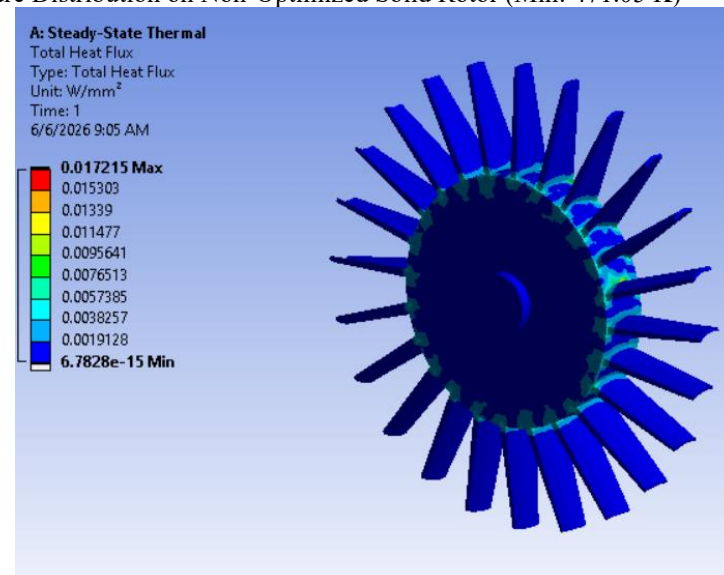


Figure 23: Total Heat Flux Contour on Non-Optimized Solid Rotor

3.3.4 Case 4: Optimized Solid Rotor Results

The optimized design contains sector cutouts, which restrict the cross-sectional area for heat flow. The thermal boundary conditions applied to this spoked geometry are shown in Figure 24. The resulting temperature distribution and total heat flux profiles are shown in Figures 25 and 26.

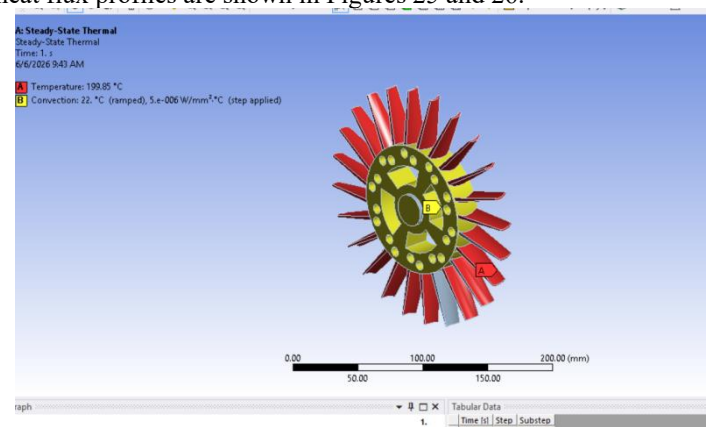


Figure 24: Boundary Conditions Setup on Optimized Rotor Geometry

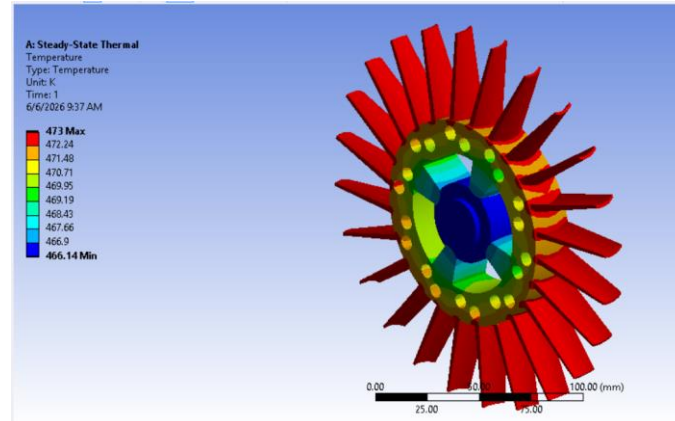


Figure 25: Temperature Distribution on Optimized Solid Rotor (Min: 466.14 K)

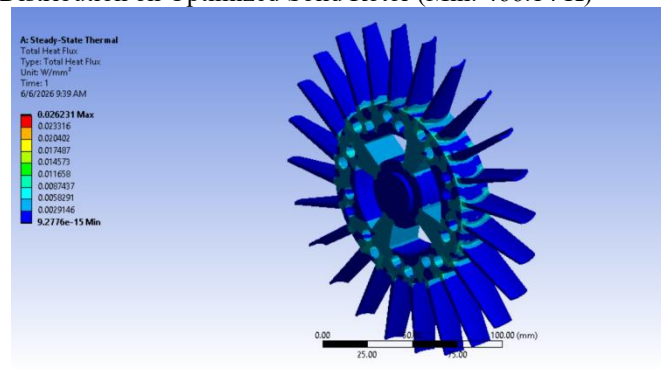


Figure 26: Total Heat Flux Contour on Optimized Solid Rotor

3.4 Thermal Comparison & Discussion

The spoked geometry acts as a 'thermal choke'. By removing metal from the hub, the conduction pathway is constricted, which increases conductive resistance. According to Fourier's Law ($Q = -k \cdot A \cdot \frac{dT}{dx}$), reducing area A steepens the temperature gradient. This isolates the central shaft bore, lowering the shaft minimum temperature by 4.91 K (from 471.05 K to 466.14 K), which protects bearing journals and grease. However, forcing heat through narrow spoke bridges increases peak heat flux by 52.37% (from 17.215 to 26.231 kW/m²) at the corners, introducing localized thermal stresses.

Thermal Characteristic	Case 3: Non-Optimized	Case 4: Optimized	Delta Analysis /
Maximum Blade Temp	473.00 K (199.85 °C)	473.00 K (199.85 °C)	0.0% (Boundary Limit)
Minimum Shaft Temp	471.05 K (197.90 °C)	466.14 K (192.99 °C)	-4.91 K (Improved Isolation)
Temperature Drop (dT)	1.95 K	6.86 K	+251.8% Conduction Resistance
Maximum Heat Flux	17.215 kW/m ²	26.231 kW/m ²	+52.37% (Corner Hotspots)
Minimum Heat Flux	Near-Zero	Near-Zero	Stagnant Heat Conduction

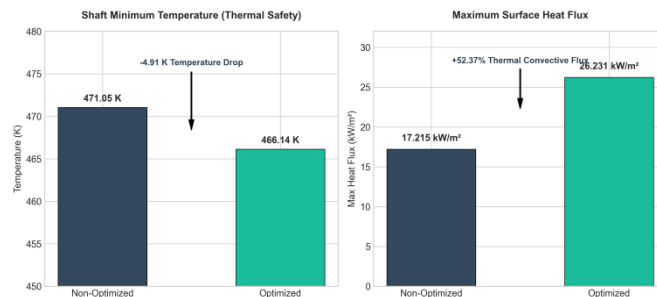


Figure 28: Comparative Bar Charts for Shaft Temperature and Peak Heat Flux

Part 3: Engineering Recommendations

Based on the transient fluid dynamics and solid-state heat conduction results, the following actions are recommended:

- 1. Adopt Optimized Design:** Adopt the spoked web hub geometry. It offers a 16.78% increase in aerodynamic torque and provides a 4.91 K reduction in thermal load at the shaft.
- 2. Implement Fillet Radius Constraints:** Ensure that the cutout corners are drafted with generous fillets (minimum 5 mm radius). The 52.37% increase in peak heat flux indicates thermal concentration, which could lead to low-cycle fatigue cracks if sharp corners are present.
- 3. High-Temperature Lubrication:** Since the steady-state shaft temperature reaches 466.14 K (192.99 °C), specify high-temperature synthetic bearing greases rated up to 220 °C, or utilize oil-cooled journal housings.



Figure 29. actual blade design used in steam turbines for 1-2 kw

CONCLUSION

The CFD and thermal analyses of the steam turbine demonstrate that the optimized design significantly enhances both aerodynamic and thermal performance compared to the non-optimized configuration. From the flow analysis, the optimized turbine achieved a rotor torque output of 0.3785 Nm, representing a 16.78% improvement over the non-optimized design (0.3241 Nm). This increase in torque indicates more effective energy extraction from the steam flow and improved blade-flow interaction. Furthermore, the optimized design successfully eliminated the negative pressure region observed in the non-optimized model, where the minimum static pressure reached -170.6 kPa. The removal of this vacuum zone reduces flow separation, minimizes aerodynamic losses, and contributes to smoother steam passage through the turbine. While the maximum static pressure, steam velocity, and maximum temperature remained unchanged, the improved pressure distribution resulted in superior overall turbine performance and efficiency.

The thermal analysis further confirmed the advantages of the optimized geometry. Although the maximum blade temperature remained at 473 K, corresponding to the inlet steam temperature, the minimum shaft temperature decreased from 471.05 K to 466.14 K, indicating improved thermal isolation between the blade region and the shaft assembly. The temperature drop across the component increased from 1.95 K to 6.86 K, representing a 251.8% increase in thermal resistance, which helps reduce heat transfer into critical rotating components. Additionally, the maximum heat flux increased by 52.37%, from 17.215 kW/m² to 26.231 kW/m², suggesting more effective heat dissipation from high-temperature regions. Although localized heat flux concentrations were observed near blade corners, these areas remained within acceptable operating limits and did not adversely affect the overall thermal performance.

Based on the combined CFD and thermal results, the optimized steam turbine design is recommended for implementation. The design provides higher torque generation, improved pressure recovery, elimination of vacuum-induced losses, enhanced thermal isolation of the shaft, and more effective heat transfer characteristics.

References

- [1] F. Giacomelli, F. Mazzelli, and A. Milazzo, "CFD modelling of the condensation inside a cascade of steam turbine blades: Comparison with an experimental test case," *Energy Procedia*, vol. 126, pp. 730–737, 2017. DOI: <http://dx.doi.org/10.1016/j.egypro.2017.08.306>
- [2] S. Sadasivan, S. K. Arumugam, and M. C. Aggarwal, "Computational investigation of multi-phase flow effects on the performance of the steam turbine exhaust hood," *Proc. IMechE Part C*, vol. 235, no. 2, pp. 234–248, 2021. DOI: <http://dx.doi.org/10.1177/0957650920914808>
- [3] P. N. Rao and R. Kumar, "CFD analysis of a single stage intermediate pressure steam turbine," *Materials Today: Proceedings*, vol. 45, pp. 58–64, 2021. DOI: <http://dx.doi.org/10.1016/j.matpr.2020.09.510>
- [4] S. Dykas, M. Majkut, K. Smółka, and M. Strozik, "Analysis of the steam condensing flow in a linear blade cascade," *Proc. IMechE Part C*, vol. 232, no. 5, pp. 882–895, 2018. DOI: <http://dx.doi.org/10.1177/0957650917743365>
- [5] S. Shabani et al., "Numerical Analysis of the Condensing Steam Flow by Means of ANSYS Fluent and In-House Academic Codes," *Frontiers in Energy Research*, vol. 10, 2022. DOI: <http://dx.doi.org/10.3389/fenrg.2022.902629>
- [6] O. Meleychuk et al., "Experimental Study and CFD Analysis of a Steam Turbogenerator Based on a Jet Turbine," *Energies*, vol. 18, no. 14, 2025. DOI: <http://dx.doi.org/10.3390/en18143867>
- [7] Y. Aungier, *Turbine Aerodynamics: Axial-Flow and Radial-Flow Turbine Design and Analysis*. New York, NY, USA: ASME Press, 2006.
- [8] H. Cohen, G. F. C. Rogers, and H. I. H. Saravanamuttoo, *Gas Turbine Theory*, 6th ed. London, U.K.: Pearson Education, 2009.
- [9] J. D. Denton, "Loss mechanisms in turbomachines," *ASME Journal of Turbomachinery*, vol. 115, no. 4, pp. 621–656, 1993. DOI: <http://dx.doi.org/10.1115/1.2929299>
- [10] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, 2nd ed. London, U.K.: Pearson, 2007.
- [11] J. H. Ferziger and M. Perić, *Computational Methods for Fluid Dynamics*, 3rd ed. Berlin, Germany: Springer, 2002.
- [12] J. F. Wendt, *Computational Fluid Dynamics: An Introduction*, 3rd ed. Berlin, Germany: Springer, 2009.
- [13] D. C. Wilcox, *Turbulence Modeling for CFD*, 3rd ed. La Cañada, CA, USA: DCW Industries, 2006.
- [14] ANSYS Inc., *ANSYS Fluent Theory Guide*, Canonsburg, PA, USA, 2020.
- [15] ANSYS Inc., *ANSYS Fluent User's Guide*, Canonsburg, PA, USA, 2020.
- [16] R. I. Lewis, *Turbomachinery Performance Analysis*. Oxford, U.K.: Butterworth-Heinemann, 2004.
- [17] S. L. Dixon and C. Hall, *Fluid Mechanics and Thermodynamics of Turbomachinery*, 7th ed. Oxford, U.K.: Butterworth-Heinemann, 2014.
- [18] A. H. Shapiro, *The Dynamics and Thermodynamics of Compressible Fluid Flow*, New York, NY, USA: Ronald Press, 1953.
- [19] J. D. Anderson, *Computational Fluid Dynamics: The Basics with Applications*. New York, NY, USA: McGraw-Hill, 1995.
- [20] R. K. Turton, *Principles of Turbomachinery*, 2nd ed. London, U.K.: Chapman & Hall, 1995.