Extended Abstract in English version

Title of thesis: Analysis of the blade geometries for a highly efficient wet steam turbine stage

Author:

Mgr inż. Sima Shabani

Silesian University of Technology
Faculty of Energy and Environmental Engineering
Department of Power Engineering and Turbomachinery
Konarskiego 18
44-100 Gliwice
Poland

Supervisor:

Dr hab. inż., Mirosław Majkut , prof. PŚ

Silesian University of Technology
Faculty of Energy and Environmental Engineering
Department of Power Engineering and Turbomachinery
Konarskiego 18
44-100 Gliwice
Poland

Chapter 1: Introduction

Axial steam turbines play a vital role in various industries, especially in power generation, oil and gas, and petrochemical sectors, due to their high efficiency and reliability. They convert steam energy into mechanical work, commonly operating in fossil fuel, nuclear, and renewable energy plants. The low-pressure (LP) stages of these turbines are particularly important because during steam expansion in these stages, condensation occurs, leading to the formation of wet steam—a two-phase mixture of vapor and liquid droplets. This phenomenon causes efficiency losses and mechanical issues such as blade erosion, where larger droplets impact turbine blades, damaging them and reducing turbine performance.

The process of steam expanding from a superheated state to wet steam inside turbine blades involves complex thermodynamic and aerodynamic changes. As steam passes through the blade channels (Fig. 1.1), it expands, cools, and crosses the saturation line, initiating nucleation and droplet growth. The condensation may occur homogeneously, without external particles, and the interaction of droplets with shock waves further complicates the flow dynamics. Understanding and accurately simulating these phenomena are crucial for improving turbine design and efficiency.

The motivation behind this thesis lies in addressing the challenges of modeling wet steam flow numerically and validating these models against experimental data. By investigating various condensation models and equations of state, the research aims to identify the most accurate approaches for simulating steam condensation. The focus is on the last stage stator blades of a 200 MW steam turbine, where significant energy losses occur. Using advanced computational fluid dynamics (CFD) simulations, this thesis seeks to enhance the blade configuration, thereby reducing losses and improving turbine performance.

The main objectives of this work include:

1. Evaluating different condensation models for simulating steam flows through Laval nozzles and linear cascades.
2. Validating the numerical simulations with experimental data to ensure model accuracy.
3. Extending the simulations to the three-dimensional geometry of the last turbine stage.
4. Proposing improvements to the stator blade design based on CFD results to minimize energy losses.

Ultimately, this research bridges numerical modeling with practical engineering applications, aiming to enhance the efficiency and sustainability of steam turbines.

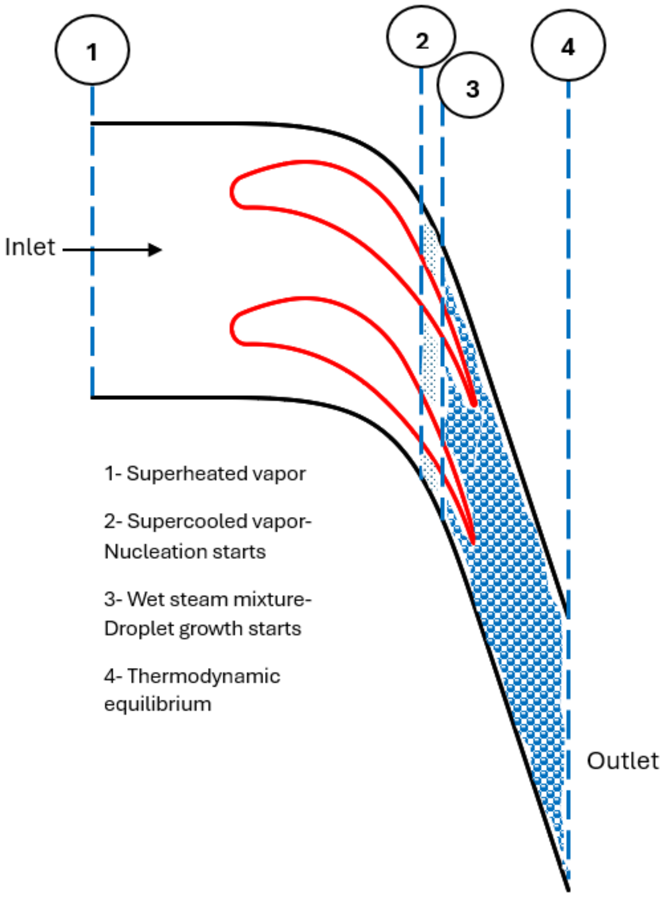
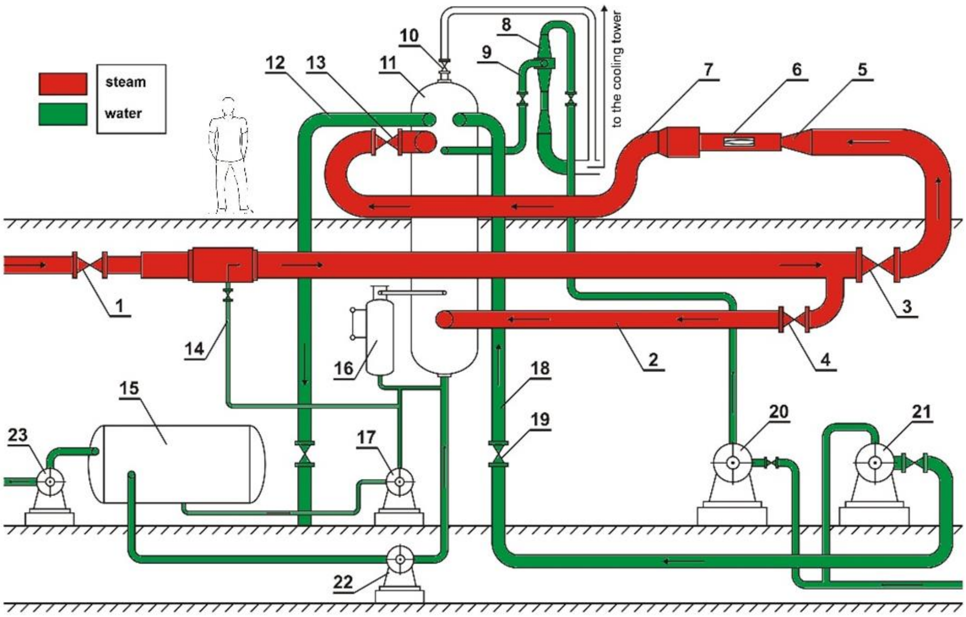


Fig. 1.1. The expansion process within the blades of LP section of a steam turbine

Chapter 2: Experimental Study

The second chapter of this thesis presents a comprehensive experimental investigation of wet steam flow phenomena in a high-fidelity test facility. The experiments were carried out in a steam tunnel (Fig. 2.1) located in the Machine Hall of the Silesian University of Technology. This facility uses superheated steam to replicate the thermodynamic and flow conditions typically found in the low-pressure (LP) stages of steam turbines, allowing for precise control of steam parameters and detailed measurements relevant to real-world turbine applications.



1 – control valve; 2 – by-pass; 3 – stop gate valve; 4 – stop gate valve at by-pass; 5 – inlet nozzle; 6 – test section; 7 – outlet elbow; 8 – water injector; 9 – pipe; 10 – safety valve; 11 – condenser; 12 – suction line; 13 – throttle valve; 14 –desuperheater; 15 – condensate tank; 16 – control system of the condensate level; 17 – condensate pump; 18 – discharge line; 19 – stop valve; 20 – water injector pump; 21 – cooling water pump; 22, 23– condensate pump

Fig. 2.1. Steam tunnel facility

The main objective of the experimental campaign was to characterize the wet steam behavior under controlled conditions and provide accurate datasets for validating computational models. A variety of measurement techniques were employed, with a particular emphasis on the Light Extinction Method (LEM), which was used to evaluate droplet size distributions in wet steam. Additional techniques, including static pressure measurements and Schlieren flow visualization, provided further insights into the flow structure and condensation behavior in turbine-representative geometries.

Prior to using the LEM system for real-flow measurements, extensive calibration was conducted to ensure its accuracy and reliability. Calibration included:

- Controlled tests with known particle diameters in a calibration chamber,
- Cross-validation with a laser diffraction system (Sprytec),
- Benchmarking the LEM's calculation procedure using identical datasets, and

- Comparative measurements under real flow conditions using both LEM and laser diffraction methods.

The results demonstrated that LEM is a suitable technique for droplet sizing in condensing steam flows, with sufficient precision to serve as a validation tool for numerical modeling efforts.

To capture a broad range of flow behaviors, three different geometries, including a Laval nozzle (Fig. 2.2), linear stator (Fig. 2.3) and rotor cascades (Fig. 2.4), were examined. These geometries were selected to represent key components of a real steam turbine, each offering unique flow characteristics critical for validating CFD models. All setups were equipped with dense pressure measurement points and LEM probes placed strategically along the flow paths to capture critical changes in pressure and droplet formation.

These experimental insights will be crucial for guiding the simulations discussed in later chapters. The consistency and quality of the measurements across different configurations ensure that the data can serve as a reliable benchmark for model validation. Furthermore, the adaptability of the experimental setup and its ability to replicate realistic turbine conditions highlight its value not only for the current study but also for future investigations into wet steam behavior in turbomachinery applications.

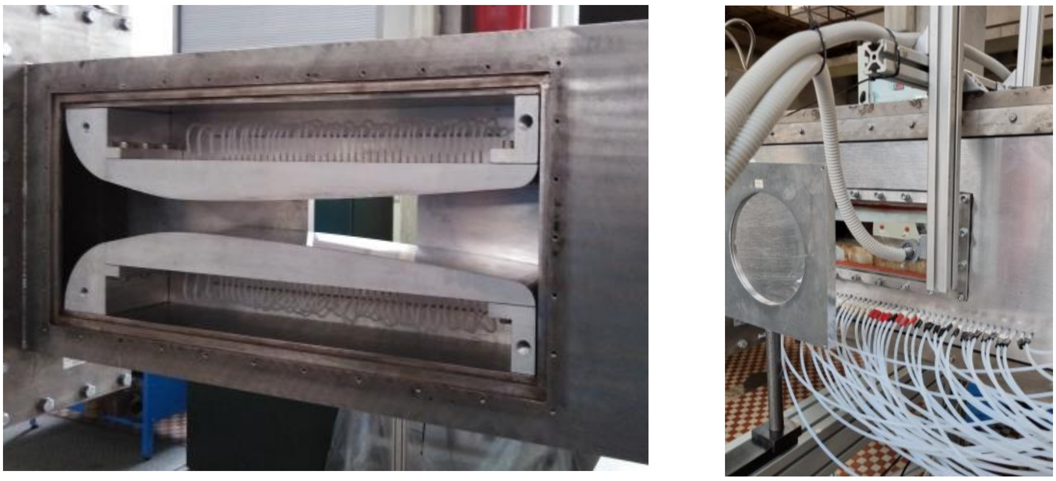


Fig. 2.2. View of IWSEP nozzle in the test section

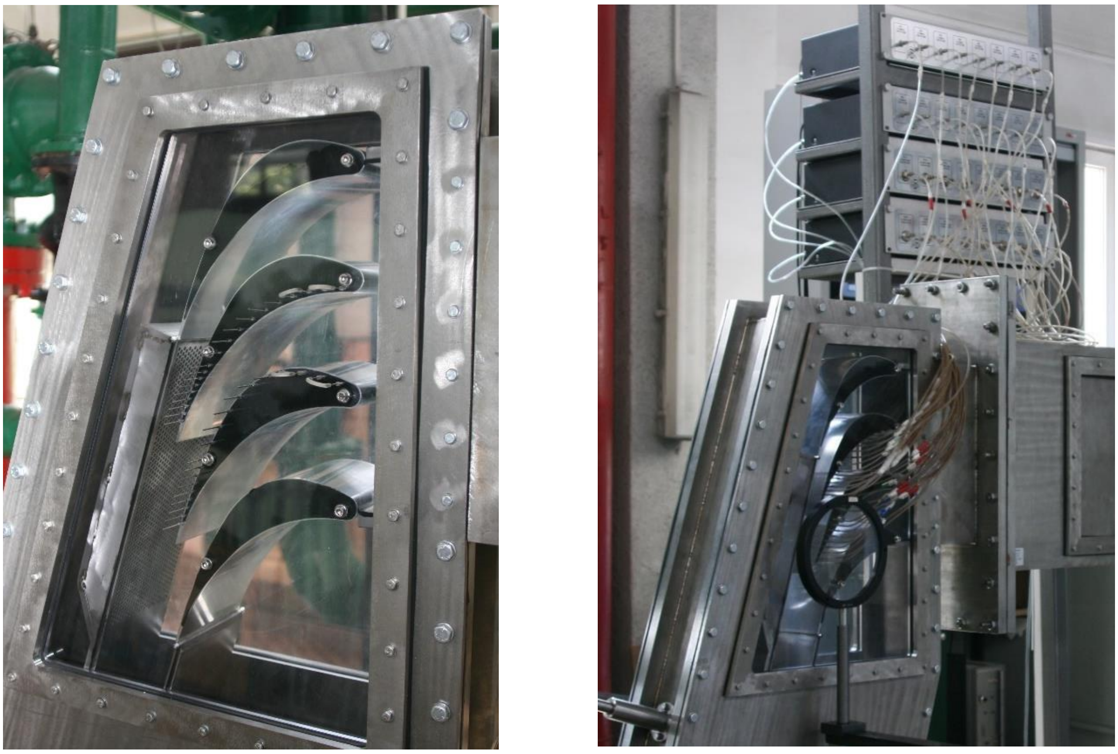


Fig. 2.3. Stator blade cascade in the SUT test rig and the measuring equipment.

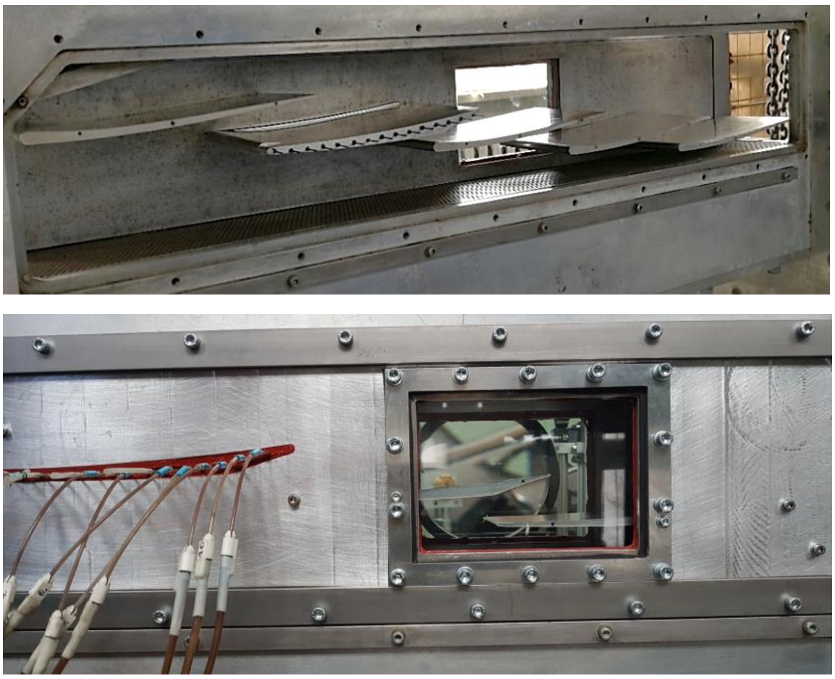


Fig. 2.4. Rotor blade cascade in the SUT test rig and the measuring equipment.

Chapter 3: Searching for the Proper CFD Simulation Tools

This chapter focuses on the evaluation and selection of numerical tools for simulating wet steam flows, addressing one of the key challenges in this field: the lack of a universally optimal computational method. Despite numerous prior studies, discrepancies in results are still common—mainly due to differences in condensation modeling, implementation strategies, and solver characteristics. Therefore, selecting an appropriate CFD tool and condensation model based on the specific case requirements is crucial for accurate prediction and analysis.

The objective of this chapter is to test and compare various numerical configurations using commercial CFD software—ANSYS Fluent and ANSYS CFX—by applying them to two geometries: a converging-diverging nozzle (IWSEP) and a linear rotor blade cascade, both previously introduced in Chapter 2. These geometries represent relevant conditions observed in the final stages of low-pressure steam turbines, where wet steam formation significantly affects performance and efficiency.

To evaluate the performance of various modeling strategies, seven simulation cases were developed (Table 3.1), differing mainly in the choice of condensation model and equation of state (EOS). These were grouped as follows:

- Cases 1 & 2: Used Fluent's built-in Wet Steam Model with two different EOS settings.
- Cases 3, 4 & 5: Implemented in-house User-Defined Functions (UDFs) within Fluent, with different condensation models.
- Cases 6 & 7: Conducted using ANSYS CFX, with different condensation models.

Table 3.1. Different cases used in the current study to model wet steam flows

Case	Droplet growth model	Thermodynamic properties	Abbreviation
1	Young	Vukalovich EOS	Yo-Vu
2	Young	Young EOS	Yo-Yo
3	Gyarmathy	NIST real gas models	Gy-NIST
4	Fuch-Sutugin	NIST real gas models	FS-NIST
5	Young	NIST real gas models	Yo-NIST
6	Gyarmathy	IAPWS-IF97	Gy-IAPWS
7	Young	IAPWS-IF97	Yo-IAPWS

The simulations employed Reynolds-Averaged Navier–Stokes (RANS) equations with a k-ω SST turbulence model. And the following assumptions were applied:

- The flow is steady, compressible, viscous, and turbulent.
- It is treated as two-phase, with steam as the continuous phase and liquid droplets as the dispersed phase.
- No relative velocity between phases is considered due to high-speed flow and the low droplet volume fraction.
- Heat exchange with surroundings and droplet-droplet interactions are neglected.
- Condensation is assumed to be homogeneous, and droplets are spherical with mean representative radii used in the growth models.

The computational domains were built to replicate the experimental conditions as closely as possible. Meshes were generated for both the IWSEP nozzle and the rotor cascade, and boundary conditions were aligned with those applied in the experimental setup.

The goal was to analyze and validate the pressure distributions and condensation behavior produced by each simulation case. The results were benchmarked against experimental data obtained at Silesian University of Technology to assess model accuracy and predictive capability. And the following key conclusions were drawn:

- Cases 3, 4, and 6 provided the best agreement with experimental pressure distributions, confirming their suitability for modeling wet steam flows.
- Cases 1, 2, and 7 significantly overpredicted or underpredicted performance metrics such as isentropic efficiency, indicating that these configurations are not reliable for accurate simulations in wet steam conditions.
- Gyarmathy and Fuchs–Sutugin droplet growth models (used in Cases 3, 4, and 6) consistently offering more realistic and stable predictions than the Young model (used in Cases 1, 2, 5, and 7).

Based on the outcomes, Cases 3 (Gy-NIST), 4 (FS-NIST), and 6 (Gy-IAPWS) are recommended for future simulations in the project. They offer the best compromise between accuracy and stability, making them appropriate candidates for continued use in modeling condensation phenomena within steam turbines.

Chapter 4: 3D Simulation of Wet Steam Flows in a Low Pressure Condensing Steam Turbine

Understanding and accurately simulating wet steam flows in steam turbines is essential due to their direct impact on performance and efficiency, especially in the low-pressure (LP) stages where condensation occurs. The presence of liquid droplets contributes to energy losses, blade erosion, and reduced output, making this phenomenon a critical consideration in the design and optimization of modern turbines.

Despite its importance, the accurate numerical modeling of wet steam in three-dimensional (3D) turbine geometries remains limited, largely due to the complexity of blade passages, strong flow unsteadiness, and the challenge of resolving intricate geometrical features. This chapter addresses these limitations by employing validated CFD techniques to simulate and evaluate wet steam behavior in the real 3D geometry of a 200 MW LP steam turbine. The simulations not only aim to predict the flow field accurately but also quantify the resulting losses and their distribution throughout the turbine stage.

The numerical strategy follows the validated Case 6 from Chapter 3, implemented in ANSYS CFX using the Gyarmathy droplet growth model and IAPWS equation of state. This model setup was selected due to its demonstrated reliability in two-phase flow predictions and its compatibility with complex turbomachinery geometries. The simulation encompasses the actual 3D geometry (Fig. 4.1) of the LP turbine's final stage, initially modeled in isolation and subsequently extended to include the Baumann rotor. This dual-configuration setup enables the assessment of both localized and cumulative effects of wet steam formation. The aim is to simulate real-world scenarios as closely as possible, particularly under wet steam conditions.

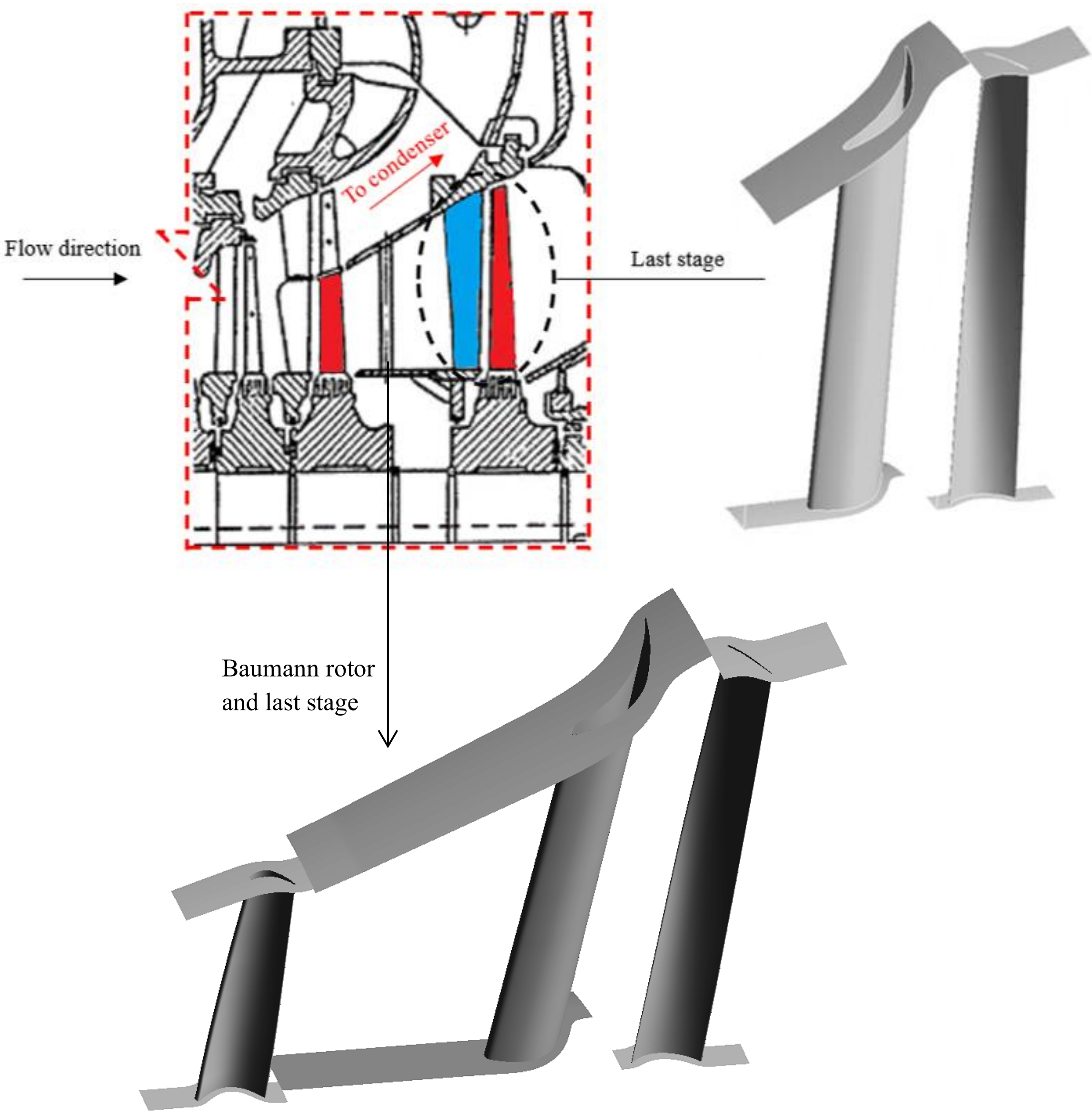


Fig. 4.1. Schematic of low-pressure part of steam turbine

The results reveal several important phenomena associated with condensation in LP turbine flows:

- In the last stage alone, condensation initiates near the stator trailing edge (hub region) and in the rotor suction side (shroud region). These zones correspond with the highest entropy rise, indicating significant irreversible losses. Specifically, entropy production is greater in the stator, with 10% of stage losses occurring there, compared to only 1% in the rotor, resulting in an overall stage efficiency of 89%.

- When comparing the results with adiabatic (non-condensing) flow, condensation is shown to:
    - Shift shock waves, pushing them downstream near the stator hub and upstream on the rotor suction side.
    - Increase shock intensity, due to rapid pressure and temperature changes during condensation.
    - Lower Mach numbers and decrease local velocities in high wetness regions.
    - Alter outlet flow angles, specifically reducing the rotor's exit angle.
    - Reduce overall turbine performance, with a calculated 1.27% drop in efficiency and a 15.04% reduction in output power under condensing conditions.
- In the extended model with the Baumann rotor, condensation was allowed throughout the entire LP section. This led to secondary condensation at the stator trailing edge, further increasing entropy generation and overall losses.

Ultimately, this chapter reinforces the crucial role of accurate 3D CFD simulations in evaluating wet steam flow behavior in steam turbines. Furthermore, the ability to simulate and evaluate losses due to wet steam formation provides a strong foundation for developing improved blade designs, optimizing turbine stages, and implementing loss-reduction strategies that can lead to meaningful performance enhancements in modern steam power systems.

Chapter 5: Assessing Stage Performance Sensitivity to Minor Variations in Stator Blade Configurations

This chapter explores the sensitivity of a steam turbine's final stage performance to small geometric modifications of stator blades. Building upon earlier findings that identified the stator as a dominant source of stage losses, this analysis focuses on how localized changes to stator blade design influence overall stage efficiency, energy losses, and output power in the context of wet steam flow. The three-dimensional (3D) geometry of the final stage of a 200 MW condensing steam turbine serves as the baseline. High-fidelity CFD simulations of wet steam, performed using consistent settings and models from earlier chapters, are employed to evaluate both two-dimensional (2D) and 3D alterations to the stator blade.

2D alterations to the stator blade are evaluated, including:

- Twist variation along blade span (st4–st8)
- Changes in blade count and pitch (st48 and st52)
- Alternative blade profiles (e.g., Bakhtar and White)
- Rotation angle modifications (±2°)

Results show that even minor modifications significantly affect critical stage parameters such as velocity distribution, mass flow rate, wetness, and output power. For instance, altering the blade rotation by just 2° (st-2deg and st+2deg) leads to contrasting outcomes. While the st-2deg variant increases both stage efficiency and output power, it also results in higher wetness. In contrast, st+2deg is detrimental across all evaluated metrics. Blade pitch variations also influence performance: fewer blades (larger pitch) may reduce blockage and improve flow uniformity, but can also increase losses due to flow separation. The choice of blade profile is likewise impactful. The Bakhtar blade consistently outperforms the baseline design in terms of efficiency, though care must be taken due to associated increases in wetness.

Modern stator blade designs often include 3D features such as axial sweep and circumferential lean, which are not present in traditional designs. Four modified configurations are evaluated (Fig. 5.1):

- Case AS: Axial sweep
- Case CL1: Circumferential lean in the direction of rotor rotation
- Case CL2: Circumferential lean against rotor rotation
- Case ASCL: Combined axial sweep and circumferential lean

The results show that circumferential lean in the direction of rotor rotation (CL1) provides the best improvement in efficiency, aligning flow better with the rotor inlet and minimizing secondary losses. On the other hand, leaning against the direction of rotation (CL2) causes performance deterioration.

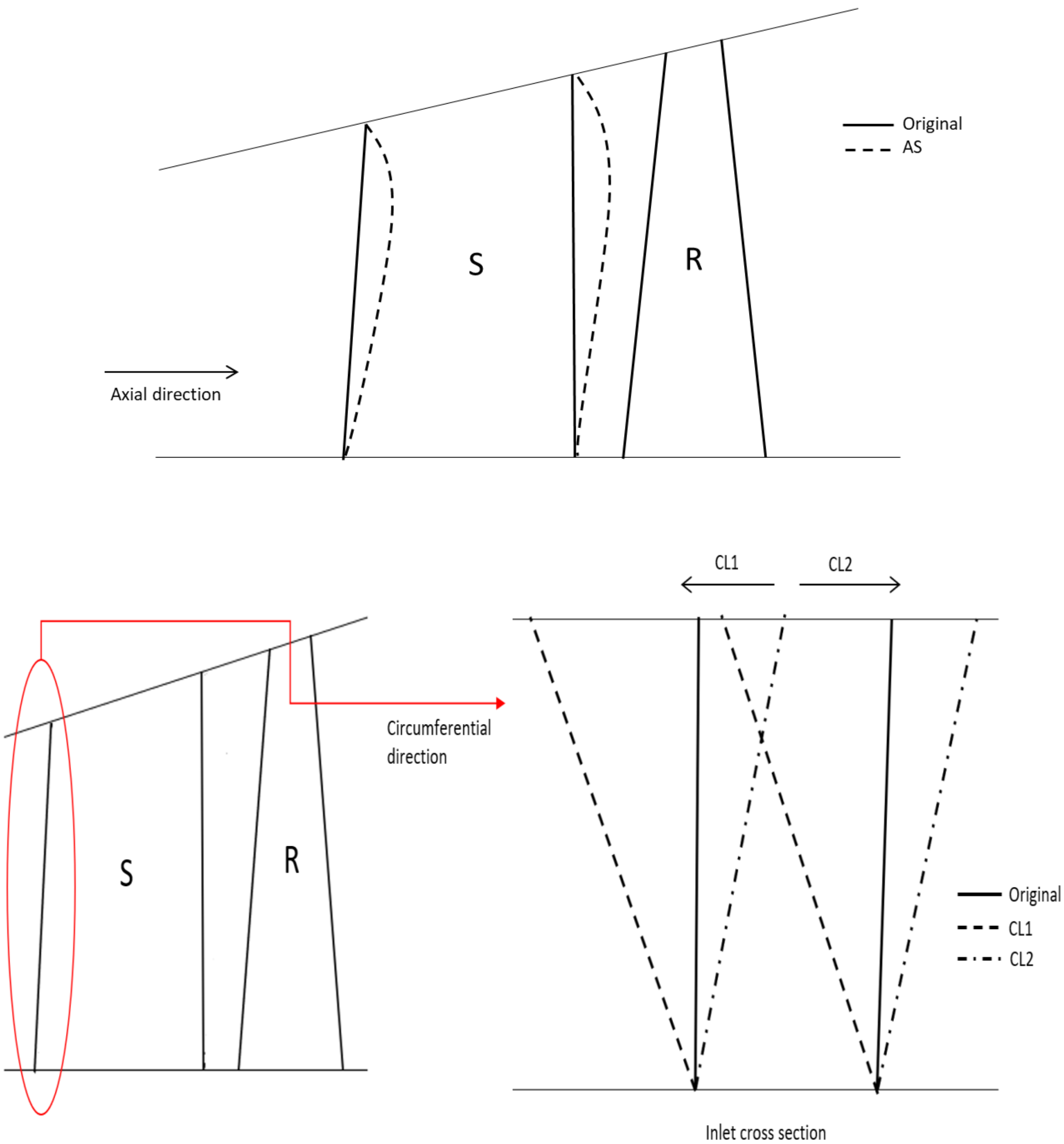


Fig. 5.1. Axial sweep (Top) and circumferential lean (Bottom) of the stator blade

Ultimately, the following key insights can be concluded:

- Flow near the hub is more sensitive to geometrical changes due to local velocity and pressure gradients. Optimization efforts should pay particular attention to this region.
- The Bakhtar profile and the st-2deg rotation configuration provide the highest efficiency gains.

- Among the 3D modifications, the circumferential lean in the direction of rotor rotation (CL1) improves stage efficiency, while the lean in the opposite direction (CL2) results in a decline in performance.
- The Bakhtar profile with CL1 lean, referred to as the Bakhtar-CL1 case, shows the most promising results, achieving a 3.2% increase in stage efficiency while maintaining acceptable flow and wetness characteristics (Fig. 5.2 and Fig. 5.3).

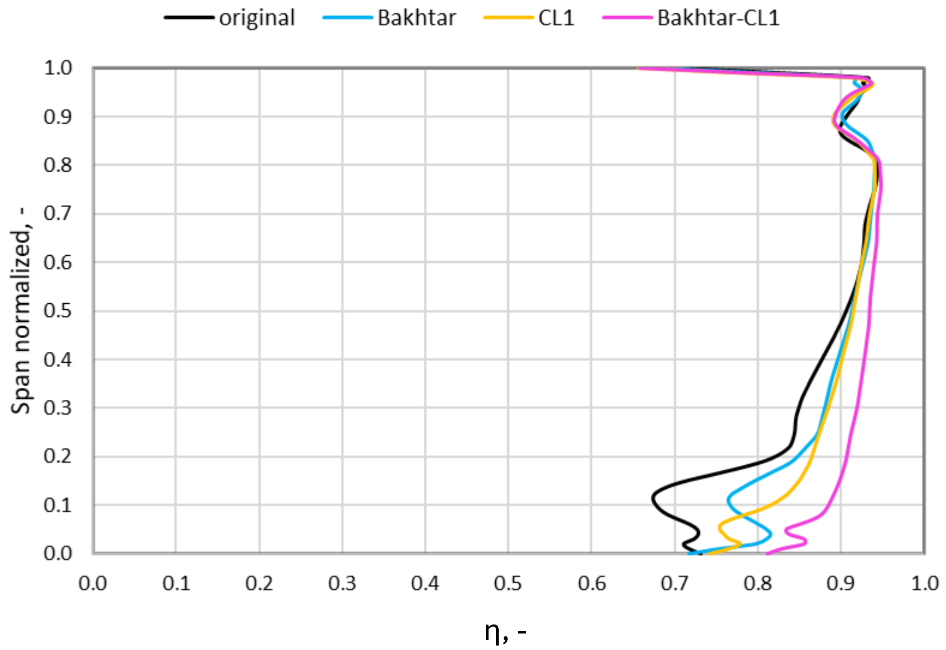


Fig. 5.2. Stage efficiency distribution along the span for the cases

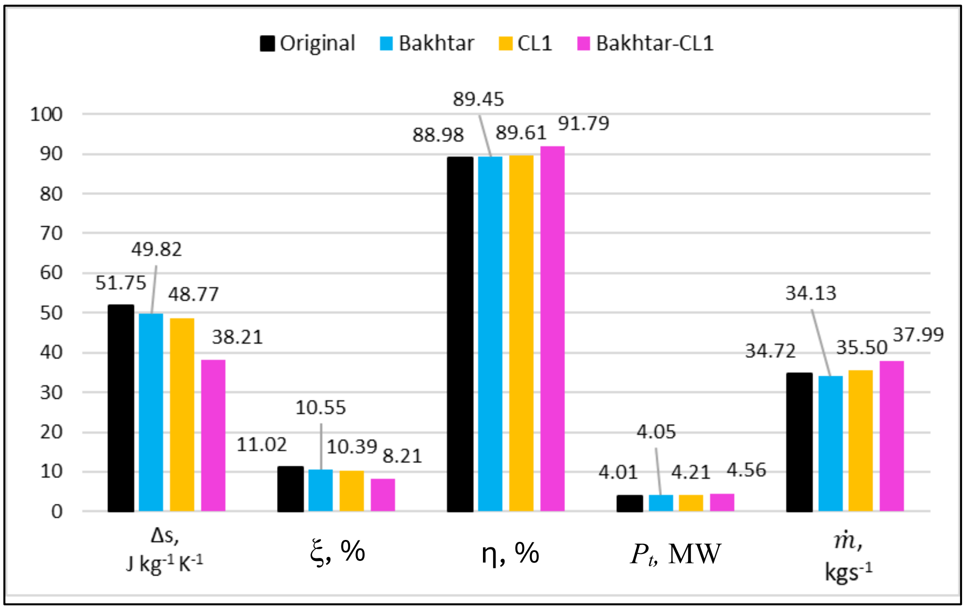


Fig. 5.3. The average values of entropy generation, entropy loss coefficient, isentropic efficiency, output power, and mass flow rate for the cases