Silesian University of Technology

EXTENDED ABSTRACT OF PH.D. THESIS

Assessment of the possibility of improving the momentum transfer in the flow between rotating discs

Mohammadsadegh Pahlavanzadeh

Supervisor: Prof. Włodzimierz Wróblewski, DSc (Eng), PhD

Co-supervisor: Dr Krzysztof Rusin, PhD (Eng)

Scientific discipline: Environmental Engineering, Mining and Energy

Gliwice, Poland, 2025

List of publications

The thesis consists of five papers, listed below and organized into Chapters 2–4. In addition, roughness modeling using the Porous Medium Layer (PML) model on a Tesla turbine operating with real gas was conducted, and this work is presented in Chapter 5. The full texts of these papers are provided in the appendices. The papers are referred to by Roman numerals throughout the thesis.

Paper I: **Pahlavanzadeh, M.**, Rusin, K., Wróblewski, W., (2023). Evaluation of dynamic correction of turbulence wall boundary conditions to simulate roughness effect in minichannel with rotating walls. International Journal of Numerical Methods for Heat & Fluid Flow. https://doi.org/10.1108/HFF-03-2023-0160.

Paper II: **Pahlavanzadeh, M.**, Rulik, K., Wróblewski, W., Rusin, K., (2024). Application of roughness models to stationary and rotating minichannel flows. International Journal of Numerical Methods for Heat & Fluid Flow. https://doi.org/10.1108/HFF-05-2024-0379.

Paper III: **Pahlavanzadeh, M.**, Wróblewski, W., Rusin, K., (2024). On the Flow in the Gap between Co-rotating Disks of Tesla Turbine with Different Supply Configurations: A Numerical Study. Energies. https://doi.org/10.3390/EN17174472.

Paper IV: **Pahlavanzadeh, M.**, Wróblewski, W., Rusin, K., (2025). Evaluation of nozzle configuration impact on flow structures and performance in Tesla turbine. International Journal of Heat and Mass Transfer. https://doi.org/10.1016/J.IJHEATMASSTRANSFER.2025.126900.

Paper V: **Pahlavanzadeh, M.**, Rusin, K., Wróblewski, W., Rulik, S., (2025). Roughness effects on flow in Tesla turbine with parametric adjustment of porous layer model. Physics of Fluids. https://doi.org/10.1063/5.0247548/3329214.

Chapter 1: Introduction

The Tesla turbine is a bladeless turbomachine that relies on viscous shear forces rather than momentum transfer from blades, with its performance determined by design, operating conditions, and component interactions. It comprises three main parts—the supply apparatus, rotor, and outlet system—and requires narrow disk spacing and minimal velocity differences to sustain laminar flow, which limits mass flow rate, velocity gradients, and overall power output. While adding multiple disks or surface modifications like micro-grooving can enhance momentum diffusion, they increase complexity or disrupt laminar flow. Its main limitation is low efficiency, largely due to the supply apparatus, where small nozzles and boundary-layer effects cause significant viscous losses, further compounded by flow disturbances at the rotor's disk edges. To comprehensively evaluate the performance and optimization of the Tesla turbine, researchers have adopted both experimental and numerical approaches.

Experimental studies on the Tesla turbine have been essential in evaluating its practical performance, validating analytical models, and understanding the influence of design and operating parameters.

Early experimental studies on the Tesla turbine focused on evaluating efficiency and mechanical performance under varying conditions, examining parameters like inter-disk spacing, disk count, inlet pressure, nozzle design, and working fluid. Owing to its simple construction, the turbine was also explored for applications in micro-scale energy systems, waste heat recovery, and low-temperature power generation. More recent experiments have shifted toward performance enhancement through innovative design features, optimized flow distribution, and modifications such as surface roughness, flow control devices, and advanced nozzle configurations. Improved instrumentation has further enabled precise measurements of key flow variables, providing deeper insights into losses and internal flow behavior.

Numerical studies on the Tesla turbine provide a computational framework to analyze its complex viscous and boundary layer–driven flow, unlike traditional blade-based aerodynamics. While early approaches relied on analytical models, advances in computational fluid dynamics (CFD) have enabled detailed simulations of turbulent flow between rotating disks, allowing researchers to predict performance, identify loss mechanisms, and optimize geometries while reducing experimental costs. A major challenge lies in realistically modeling fluid delivery through nozzles, where jet–boundary layer interactions near the disk surfaces create highly transient and complex flow behavior that requires in-depth analysis.

Surface roughness has a major impact on Tesla turbine performance because viscous forces dominate the boundary layer flow between rotating disks, making efficiency highly dependent on wall shear stress and near-surface momentum transfer. However, existing roughness modeling methods are not well-suited for the turbine's unique flow conditions, where small inter-disk gaps, opposing boundary layer interactions, rotational body forces, and streamline curvature complicate turbulence predictions. These challenges often cause numerical results to diverge from experimental data, as accurately simulating such small domains requires fine meshes and high computational effort. Among turbulence models, the $k - \omega$ Shear Stress Transport ($k - \omega\ SST$) approach has shown the best suitability for these conditions. Recent studies highlight that rotor disk roughness is a critical parameter for enhancing efficiency and power output, underscoring the need for detailed analysis of flow behavior in co-rotating disk systems.

Scope of thesis

This Ph.D. research aims to assess the possibility of improving the momentum transfer in the flow between rotating disks, with particular emphasis on Tesla turbines. In the present investigation, the effects of disk roughness, nozzle, and supply configuration on the turbine's efficiency are evaluated.

This thesis aims to answer the following main research questions through a three-step approach, presented in Chapters 2 to 5:

- What numerical approach provides the most accurate simulation of flow between co-rotating disks, with a gap size similar to that used in a Tesla turbine, considering turbulence models, mesh quality, and roughness implementation?
- How does surface roughness influence wall shear stress and flow characteristics in channel flow and in the gap between co-rotating disks?
- How does a nozzle jet affect the flow characteristics between co-rotating disks compared to a uniform inlet flow?
- How do the number of nozzles and their configurations affect the flow aerodynamics, efficiency, and output power of a Tesla turbine?
- How do the $k - \omega\ SST$ turbulent model and Larg Eddy Simulation (LES) approaches compare in predicting the internal flow behaviour and performance of Tesla turbines?
- How can the effect of real surface roughness be more accurately represented in CFD simulations, particularly in the context of Tesla turbine modeling?
- How does the surface roughness affect the performance of the Tesla turbine operating on selected Organic Rankine Cycle (ORC) fluids?

Chapter 2: Investigation of flow characteristics in minichannel with stationary and rotating walls – Papers I and II

The scope of the investigation

This chapter (Papers I and II) addresses the challenge of modeling surface roughness effects in narrow flow domains, specifically the submillimeter gaps between co-rotating disks in Tesla turbines, where geometry-driven boundary layer interactions strongly affect turbulence and momentum transfer. To capture these complex dynamics, the study evaluates turbulence models and two roughness modeling strategies: velocity profile shifts in wall functions and Aupoix method, which modifies turbulence quantities near walls. Their accuracy and limitations are tested across rough flat plates, stationary and rotating minichannels, and Tesla-like disk gaps, with validation against Direct Numerical Simulation (DNS) data, literature experiments, and in-house measurements. The analysis focused on key parameters, including gap size, roughness height/shape, laminar–turbulent transitions, wall shear stress, eddy viscosity, and the influence of disk rotation on boundary layer interaction. The results provide a simplified but representative framework for understanding flow behavior between co-rotating Tesla turbine disks.

Flow between co-rotating disks

Preliminary research on simpler geometries presented in papers I, and II indicates that when the domain scale and flow cross-section are reduced, generating a compatible mesh for the $k - \varepsilon$ turbulence model with the first layer located in the log-law region of the flow regime results in only a few cells in the normal direction to the wall, which is inappropriate. Moreover, the $k - \omega\ SST$ model, with a denser mesh in regions experiencing intense changes in eddy viscosity and a coarser mesh in farther areas, performs well due to its ability to blend between $k - \omega$ and the $k - \varepsilon$ model. The study further demonstrates that the Aupoix method artificially modifies eddy viscosity on the wall surface based on roughness parameters, eliminating the need to capture roughness in the first layer of the generated mesh. Implementing this method with the $k - \omega\ SST$ model yields good results in both stationery and rotating minichannels.

The flow within the gap between two co-rotating disks is a fundamental configuration that has been extensively analyzed to understand flow structures and disk-flow interactions, both of which are essential phenomena for Tesla turbine optimum efficiency. Gap size, rotational speed, and roughness play crucial roles in shaping the flow structure, and boundary layer between the disks, as extensively studied in Papers I and II. In the case of the flow through the mini gap between co-rotating disks, the velocity profile is influenced by the interaction between developing boundary layers from the parallel co-rotating disks. Roughness height, as a key parameter in boundary layer development, leads to a downward shift in the velocity profile. Flow through the co-rotating disks is influenced by boundary layer formation on both parallel surfaces, and the interaction of these layers prevents the velocity profile from fully developing. This explains the observed deviation between the results of the smooth-wall case and predictions based on laminar theory and the log-law velocity distribution. Effective parameters influencing power generation and flow characteristics between simple co-rotating disks with uniform inlet flow were discussed in Papers I and II.

Achievements

The effectiveness of the $k - \omega\ SST$ turbulence model combined with the Aupoix roughness method was validated on a minichannel and subsequently applied to simulate the flow between co-rotating disks. Despite the known limitations of the Aupoix method, this approach demonstrated reliable performance across all studied cases. It optimized computational costs while maintaining accuracy, even in challenging near-wall regions, without the need for ultra-fine meshes.

Research on minichannels demonstrates that roughness elements constrict the flow cross section, leading to discrepancies between experimental results and theoretical predictions.

A reduction in gap size from $15\ mm$ to $0.75\ mm$ demonstrated an increased interaction between the developing boundary layers from the co-rotating disks, as well as a greater discrepancy between the observed velocity profile and theoretical estimations.

A change in rotational speed affects power generation, with a maximum value of $7\ W$ observed for a gap size of $0.75\ mm$ and a roughness height equal to 3.5% of the gap size.

An increase in roughness height in the studied case (with a 0.75 mm gap) led to an efficiency improvement from 30% to 36%, considering four roughness heights equal to 3.5%, 7%, 10%, and 15% of the gap size.

Chapter 3: Investigation of flow characteristics between co-rotating disks of Tesla turbine – Papers III, and IV

The scope of the investigation

General design parameters, e.g., number of nozzles, and nozzle configuration pose a challenge in order to achieve the full potential of Tesla turbine which are studied in this chapter of thesis.

Nozzle Configuration and Supply System Design: The study investigated momentum diffusion and kinetic energy transfer within the narrow, submillimeter-scale gap between co-rotating disks, with particular attention to the complex flow behavior resulting from rotational effects and turbulence. It examined nozzle configurations, comparing one-to-one setups, where each nozzle feeds a single gap, to one-to-many setups, where each nozzle supplies multiple gaps. Additionally, it analyzed two supply arrangements: N6, a six-nozzle system, and N40, a forty-nozzle system, focusing on their effects on turbine efficiency, fluctuation behavior, and wall shear stress distribution.

Turbulence modeling approaches: This section compared two turbulence modeling approaches: the $k-\omega\ SST$ model, valued for its computational efficiency and suitability for engineering applications, and LES employing the Smagorinsky subgrid-scale (SGS) model, which offers high-fidelity flow characterization by explicitly resolving larger turbulent structures. The evaluation assessed how each model captures essential features such as flow structures, transient fluctuations, shear stress distribution, and overall turbine efficiency, highlighting their respective strengths and limitations in simulating complex turbulent flows.

Tesla turbine with different supply configurations

Two different variants of the supply system are considered with six (N6) and forty (N40) nozzles having one-to-one nozzle arrangement with the individual nozzle for each gap. To minimize computational effort, a reduced calculation domain is considered. The flow in each domain, consisting of one inlet nozzle and a periodic segment of one gap between the disks, is examined. LES with the Smagorinsky SGS model is used to verify the results of the $k-\omega\ SST$ turbulence model in the N6-case study. Analyzing the results indicates that the $k-\omega\ SST$ model provides valuable insights with appropriate accuracy. From the comparison between the $k-\omega\ SST$ turbulence model and LES simulation, it was observed that although the $k-\omega\ SST$ model slightly overestimates the general parameters and damps fluctuations, it still provides valuable insights for assessing flow structures. The verified turbulence model is used to simulate N40 for evaluation of flow structure, flow parameters, and their impact on the system's efficiency. Using a high number of nozzle jets results in a noticeable interaction of the nozzle jet, manifesting as fluctuations in parameters near the outer edge of the disks. Moreover, it increases the mass flow rate, resulting in nearly four times more power generation. However, this also causes the efficiency of the system to drop by almost 16 percent point. The comparison of N6 and N40 nozzle configuration is fully described in Paper III.

Tesla turbine with different nozzle configurations

This section of the study focused on two Tesla turbine configurations with six nozzles: one featuring a one-to-one nozzle arrangement with the individual nozzle for each gap (referred to as N6O-O) and the other adopting a one-to-many nozzle design where the nozzle provides fluid to all gaps (referred to as N6O-M). In N6O-O, the simulation accounted for the plenum chamber's thickness being equivalent to the gap size. In N6O-M, aside from the gap size, half of the disk's size was factored into the thickness of the supply chamber. The one-to-one configuration allowed for direct passage of the nozzle jet through the gap, while the one-to-many setup incurred losses from the interaction between the inlet jet and the disk tips. Each case exhibited symmetrical behavior every 60 degrees in the investigated geometry then a reduced calculation domain is considered. LES employing the Smagorinsky SGS model is used for flow simulation, enabling a comparison of flow structures, fluctuations, parameters, and their impact on system efficiency. The assessment of the mesh quality for LES simulation is presented in Paper IV.

Parametric analysis of the observed results demonstrates that the higher mass flow rate and the convergent-divergent formation of flow in the area close to the inlet nozzle cause higher velocity and lower pressure in this region in the N6O-M case. Moreover, In the N6O-M case, Reynolds stress magnitude increases as they approach the mid-gap. In contrast, in the N6O-O case, where wall shear stress is the only source of fluctuations, the maximum values are observable near the disk surfaces and decrease as they approach the mid-gap.

Achievements

The $k - \omega\ SST$ model, while slightly overestimating torque and wall shear stress (~4.25%), provides a fast and sufficiently accurate prediction for general flow behavior at a much lower computational cost.

The LES model, though more computationally intensive, captures transient fluctuations with greater fidelity, particularly in regions near the outer edge of the disks, where vortices are generated and momentum transfer is most significant.

Nozzle count and configuration significantly influence flow behavior. A higher number of nozzles increases mass flow rate and power output (up to 4 times) but leads to complex jet interactions and a drop in system efficiency of about 16 percent point.

Comparison of N6O-M and N6O-O nozzle designs shows that N6O-O is 17.8 percent point more efficient with lower mass flow rate and power generation, and it demonstrates slower flow, lower turbulence, and a more stable wall shear stress distribution. Additionally, in the N6O-M case, the convergent–divergent flow structure near the inlet increases velocity, reduces pressure, and shifts the region of maximum wall shear stress toward the inner disk diameter. It produces stronger jet-disk interactions, higher Reynolds stresses and turbulence, especially near mid-gap, unlike the N6O-O case, where viscous effects damp fluctuations and remain concentrated near the walls.

Inlet jet–disk tip interactions in N6O-M generate significantly higher fluctuations than in N6O–O, where fluctuations originate mainly from inlet jet–boundary layer interaction.

Chapter 4: Simulation of roughness using Porous Medium Layer (PML) – Paper V

The scope of the investigation

This PhD research explores an alternative approach to modeling surface roughness in CFD by replacing the conventional equivalent sand grain roughness method with a Porous Medium Layer (PML) model. The PML technique adjusts porous medium parameters to replicate roughness effects on fluid flow without relying on empirical sand-grain correlations. The model is validated through experimental pressure drop measurements in a minichannel and further tested against the Aupoix roughness model in Tesla turbine simulations. Using the $k - \omega$ SST turbulence model, cross-verified with LES results for smooth walls, the study conducts a parametric analysis of different porous layer thicknesses to assess their impact on turbine efficiency, flow behavior, and momentum transfer, demonstrating the PML model's potential for more accurate roughness representation in turbine flow studies.

Tesla turbine simulation with roughness model

The performance of the PML roughness model was initially tested on a minichannel by numerical and experimental study of a minichannel characterized by a width (z) of 50 mm, a length (x) of 150 mm, and a height (y) of 0.75 mm, at varying flow velocities. A developed inlet flow and ambient pressure at the outlet were considered. The side walls were smooth, while the top and bottom walls were rough.

The simulations incorporated roughness using both the Aupoix method and drag correction via a PML model for a roughness height of 60 μm. The thickness of the porous layer was set equal to the highest roughness peak, while permeability was adjusted to modify drag generation within the minichannel. To ensure that the drag in the minichannel matched the theoretical predictions for a roughness height of 60 μm, the permeability was set to $K = 23\ nm^2$. In this investigation, porosity changes from zero to one, assumed to be a second-order function of the thickness of the porous zone, with the derivation of zero at the interface between the porous and flow zones. The CFD simulation results were validated against experimental investigations.

In the next phase, the PML model was implemented in the simulation of the Tesla turbine featuring six nozzles (N6O-M). Fig. 1 represents a schematic of real roughness, the equivalent sand grain roughness used in the Aupoix method, and the porous layer in the PML method.

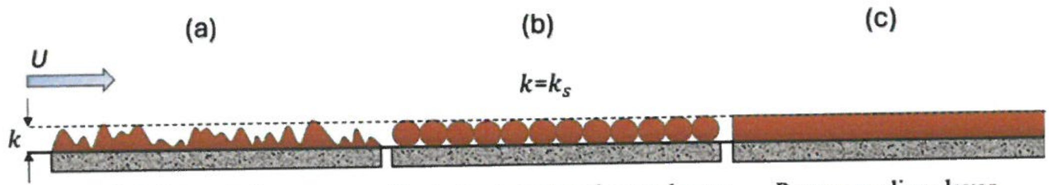

Figure 1. Schematics of (a) real random roughness, (b) equivalent sand-grain roughness, and (c) porous layer with the same thickness as the highest roughness peak.

In this section, the performance of the adjusted parameters of the PML roughness method is evaluated in Tesla turbine simulations with a 0.75 mm gap size. In the model, the porous medium layer is added on both sides of the disk surfaces. The global parameters obtained from these simulations are then compared with those obtained from the Aupoix method. A porous medium layer with the same modifications as those tested in the minichannel was applied to the disk surfaces of the Tesla turbine. Three heights of roughness were considered in this study: 30 μm (N6-S-R30), 60 μm (N6-S-R60), and 120 μm (N6-S-R120). The turbine general parameters for the rough and smooth cases were compared. As the level of roughness on the surface of the co-rotating disks increases, it leads to greater momentum transfer between the operating fluid and the disks resulting in a faster drop in the energy level of the flow. The tangential component of velocity, responsible for torque generation, also decreases faster with increased wall shear stress due to higher roughness height, thus improving momentum transfer. This occurs because the rough surfaces reduce the boundary layer thickness, causing enhanced interaction between the fluid and the rotating surfaces leading to more effective drag force generation, which increases torque and power output.

Achievements

The development and validation of a PML model, based on experimental data from a minichannel matching the Tesla turbine's gap height, shows that this method can accurately capture real roughness geometries in simulations without fine near-wall meshing, thereby lowering computational costs while preserving accuracy.

Comparison with the Aupoix roughness method illustrates that PML yields compatible trends with only minor discrepancies in the general parameters.

The flow separation because of inlet jet to the gap creates a converging–diverging pattern of inlet flow at this area. As the height of the roughness increases, this phenomenon is dampened by viscous forces, leading to a smoother transfer of operating flow from the inlet to the outlet.

Increasing the roughness height in the PML model accelerated the decay of kinetic energy in the flow, leading to improved system efficiency compared to the Aupoix model. The two models showed a maximum efficiency difference of 2.7 percentage points at a roughness height of 120 μm.

Analyzing the results from the smooth and two rough cases (30 μm, and 60 μm) at different n reveals that roughness has the greatest impact on improving turbine efficiency when the rotational speed is in the range of 17000–26000. The efficiency improvement observed with the implementation of a 60 μm porous layer compared to the smooth case was 4.61 percent point at 22000 (rev/min).

Chapter 5: Roughness modeling by Porous Medium Layer model in Tesla turbine working on ORC fluids

The scope of the investigation

Tesla turbine shows significant potential for application in ORC systems. In this chapter, a comprehensive analysis is performed to simulate the effects of surface roughness on the flow within the gap between the co-rotating disks of a Tesla turbine, using two low-boiling media, R1234yf and n-hexane as the working fluids. Flow simulations are conducted with the $k-\omega\ SST$ turbulence model to assess how surface roughness influences momentum transfer, key flow parameters, and overall turbine efficiency. Additionally, the investigation compares flow characteristics for the two fluids within the Tesla turbine, demonstrating that the PML model effectively captures roughness effects and improves turbine efficiency. Comparative analysis between smooth and rough surface cases is conducted to quantify efficiency improvements and understand fluid-specific behavior.

Tesla turbine simulation with roughness

The performance of the adjusted parameters of the PML roughness method was validated by experimental data of minichannel in Chapter 4. In this section, this method is tested in Tesla turbine (N6O-M) simulations working with real gases. Since the operating fluid considered in the current study is a real gas, the gap size is six times smaller than the case in chapter 4. However, considering six times smaller PML thickness cannot demonstrate the impact of surface roughness on flow characteristics and turbine efficiency, then a thicker PML layer of 28.8 μm is used in both cases.

The boundary conditions applied in the CFD simulations were selected to ensure that the working fluid remained in the superheated vapor region, thereby enabling accurate modeling of single-phase flow within the Tesla turbine. R1234yf shows peak efficiency at lower speeds (~5000 rpm), with rough surfaces yielding a maximum of ~27%, compared to ~18% for smooth, representing a 50% improvement. On the other hand, n-hexane performs best at higher speeds (~12000 rpm), where roughness increases peak efficiency from ~8% to ~15%, an 87.5% gain. The efficiency curves for all cases follow a parabolic trend, peaking before declining. These results demonstrate that surface roughness enhances efficiency for both fluids. R1234yf is more suitable for low to moderate speeds, while n-hexane excels at higher speeds. The analysis highlights the critical role of both the selection of working fluids and surface condition in maximizing Tesla turbine efficiency.

Achievements

The validated model was applied to simulate an expansion of two ORC fluids—R1234yf and n-hexane—in the Tesla turbine, demonstrating the adaptability of the method for fluids with real gas properties and a wide range of operating speeds.

Surface roughness can significantly improve turbine efficiency. For R1234yf, efficiency increased from ~18% to ~27% at 5000 rpm, while for n-hexane, it rose from ~8% to ~15% at 12000 rpm.

Flow analysis revealed that R1234yf provided a more uniform and stable velocity field, contributing to better energy extraction at lower speeds. In contrast, n-hexane generated a stronger but less efficient flow due to high dissipation and reduced residence time.

The roughness model helped mitigate certain factors that limited the system's performance with n-hexane, leading to improved flow uniformity and enhanced angular momentum transfer.

Conclusions

Key Findings of the Thesis are listed in 6 categories as follows:

1- Turbulence modeling:

- The $k - \omega$ SST model provided reliable predictions of flow behavior and shear stress in the co-rotating disk configurations and Tesla turbine simulations, particularly for benchmark validation cases and roughness modeling, while offering a good balance between accuracy and computational efficiency.
- LES offers superior resolution of transient and complex flow phenomena, especially near the outer disk edges where vorticity and fluctuations are dominant, making it ideal for high-fidelity simulations.

2- Roughness modeling approaches:
- There is no universally accepted method to determine the equivalent sand-grain roughness, and conventional roughness models (e.g., Aupoix) fail to accurately represent the influence of actual roughness geometries on flow behavior, especially in flows with small cross-sectional dimensions.
- The Aupoix roughness model produced results consistent with the validation case of the minichannel; however, it still has limitations, such as its simplified representation of roughness and its limited applicability to small or confined flows.
- The PML model emerged as a robust and flexible alternative for modeling rough surfaces. It was validated against experimental data and performed well, matching theoretical friction distributions.

3- Impact of roughness on flow and efficiency:
- Roughness enhances momentum transfer by promoting earlier transition to turbulence and increasing k near disk surfaces.
- Higher roughness leads to more uniform velocity profiles, greater wall shear stress, particularly in the outer disk region, and significantly improved power output and efficiency.
- In optimal operational ranges (e.g., $17000 - 26000$ rpm with air), efficiency improvements of up to 4.61 percent point were observed due to roughness height of 60 μm.

4- Effect of design parameters:
- Increasing nozzle count raises mass flow rate and power output but can also increase turbulence and reduce overall efficiency.
- Nozzle configuration (e.g., one-to-one vs. one-to-many) significantly affects flow behavior:
- One-to-many configurations induce stronger velocity fluctuations and higher wall shear stress.
- Jet interactions with disk tips intensify turbulence and affect Reynolds' stress and energy dissipation.

5- Working fluid compatibility:
- The performance of the Tesla turbine is dependent on matching the working fluid with the design.
- R1234yf supported more stable flow and efficient energy extraction at lower n.
- n-hexane induced faster radial flow, and more energy dissipation, still benefited from roughness-enhanced momentum transfer.

6- General contribution and optimization strategy:
- The study confirms that accurate CFD modeling, particularly turbulence and roughness modeling, is critical to optimizing Tesla turbine design.
- Roughness engineering plays a pivotal role in controlling boundary layer behavior and improving energy conversion efficiency.

The PML model holds strong potential for future refinement by incorporating real roughness geometry into its parameter definitions.