Abstract

Momentum diffusion and kinetic energy transfer remain fundamental challenges in turbomachinery, particularly for bladeless configurations such as the Tesla turbine. This radial-flow machine exhibits significant potential for applications in Organic Rankine Cycles (ORC) and Combined Heat and Power (CHP) systems, as well as micro-power generation, waste heat recovery, geothermal energy conversion, and small-scale propulsion systems. However, analyzing the flow inside the narrow gap between the co-rotating disks of the turbine is complex due to the sub-millimeter length scales, the variable cross-sectional geometry, the interaction of rotational body forces with turbulence, the influence of the configurations of the inlet nozzles, and the moving wall boundaries. Surface roughness further complicates the flow field, and the turbine's global design parameters, such as the supply and nozzle configurations, the number and dimensions of co-rotating disks, must be carefully optimized to achieve maximum efficiency.

This research begins with a systematic assessment of turbulence and roughness modeling approaches to simulate flow within narrow gaps formed by rough co-rotating disks. Two roughness models are examined: one that applies a downward shift to the dimensionless velocity profile (modification of the wall function) and another that modifies the parameters of near-wall turbulence according to the Aupoix method. These models are validated across progressively complex geometries—starting with flow over rough flat plates, followed by flow through stationary and rotating minichannels. The $k - \omega$ Shear Stress Transport (SST) turbulence model combined with Aupoix's wall correction is identified as the most reliable and accurate approach, particularly for domains requiring fine near-wall resolution (low y^+). Parametric studies demonstrate how variations in minichannel height, mass flow rate, working fluid, and roughness height influence the velocity profile and wall shear stresses.

Subsequent investigations focus on the flow phenomena in the Tesla turbine. Two supply nozzle configurations, one with six nozzles (N6) and another with forty nozzles (N40), were studied to assess the influence on flow characteristics and turbine performance. To reduce computational cost, each configuration is simulated using a domain with a representative periodic nozzle sector and a periodic diskgap sector. Large Eddy Simulation (LES) with the Smagorinsky subgrid-scale (SGS) model is employed for the N6 case and compared to results from the $k - \omega\ SST$ model. The $k - \omega\ SST$ model demonstrates adequate accuracy for engineering analysis, despite slight overprediction of flow parameters. The N40 configuration reveals increased mass flow and power generation, but a notable reduction in efficiency due to intensified inlet jet interactions and high turbulence level near the disk tips.

Inlet jet configurations are further evaluated by comparing one-to-many and one-to-one nozzle configurations. LES simulations indicate that the one-to-many configuration causes higher flow fluctuations and decreased efficiency as a result of inlet jet/disk tip interactions. The one-to-one setup, while still affected by turbulence induced by rotating walls, yields a more uniform energy transfer. These findings emphasize the critical role of supply nozzle design in reducing losses and improving turbine performance.

Since direct modeling of real surface roughness through equivalent sand-grain proves insufficient due to the lack of general correlation, a Porous Medium Layer (PML) approach is introduced. The PML method uses a porous zone on the disk surface with tunable porosity and permeability to mimic roughness-induced resistance. The investigation began with the validation of the PML model against in-house experimental tests performed for water flow in a minichannel. The parameters of the PML model are adjusted to achieve the same pressure drop as observed in the experiments. Once validated, it is applied to rotating disk systems. The performance of the PML model is compared with the Aupoix roughness model, considering different heights of roughness.

In the final stage of the study, a comprehensive analysis is conducted 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-point working fluids: R1234yf and n-hexane. The previously validated PML model is employed to replicate the impact of realistic surface roughness on the flow characteristics of the Tesla turbine operating with real gases.

The turbulence closure used in this study was the $k - \omega\, SST$ model. The results demonstrated that the PML roughness model effectively captures the influence of surface roughness on flow characteristics, resulting in an accurate simulation of the Tesla turbine's performance in the tested cases. Furthermore, the analysis of two real working fluids—R1234yf and n-hexane—indicates that the PML model is applicable for simulating Tesla turbines operating with real gases and that fluid properties significantly influence flow throughout the expansion process.