PROCEEDINGS OF ECOS 2023 - THE 36TH INTERNATIONAL CONFERENCE ON
EFFICIENCY, COST, OPTIMIZATION, SIMULATION AND ENVIRONMENTAL IMPACT OF ENERGY SYSTEMS
25-30 JUNE, 2023, LAS PALMAS DE GRAN CANARIA, SPAIN

Assessment of turbulent parameters modification to model roughness in the flow between rotating disks of Tesla turbine

Mohammadsadegh Pahlavanzadeh[a], Krzysztof Rusin[b] and Włodzimierz Wróblewski[c]

[a] Silesian University of Technology, Department of Power Engineering and Turbomachinery, Gliwice, Poland, Mohammadsadegh.pahlavanzadeh@polsl.pl, CA
[b] Silesian University of Technology, Department of Power Engineering and Turbomachinery, Gliwice, Poland, Krzysztof.Rusin@polsl.pl
[c] Silesian University of Technology, Department of Power Engineering and Turbomachinery, Gliwice, Poland, Wlodzimierz.Wroblewski@polsl.pl

Abstract:

Momentum diffusion and the transfer of kinetic energy from flow to a rotor have always been of great importance in turbomachinery. The performance of a bladeless Tesla turbine depends on wall shear stress generation on parallel surfaces of rotational disks. Considering the small gap between the co-rotating disks of the Tesla turbine, micro-scale size, variation of flow cross-section and moving walls, make flow analysis a challenge in such a domain. The additional difficulty comes from the impact of roughness, which can enhance the momentum diffusion and alter the velocity profile. Lack of validation data is another challenge in this way. To deal with these issues, a set of preliminary studies are considered herein. The investigation started with flow over a rough flat plate and continued with the flow in minichannel with rough surfaces and ends on the flow through the narrow gap between co-rotating disks. The numerical simulation is done with the $k-\omega\, SST$ turbulent model and the Aupoix's method of roughness, and velocity profile formation is analysed in every case. The Aupoix's method implements the roughness in the simulation by modification of turbulent parameters on the wall based on roughness features. The effect of the roughness constant in the model which is the function of roughness distribution on the surface on the performance of the method is tested.

Keywords:

Eddy viscosity; turbulent parameters; roughness; momentum transfer

1. Introduction

In fluid dynamics, roughness is an important factor that affects the flow characteristics in channels. In minichannel flow, the presence of roughness elements can significantly alter the flow behavior. The implementation of roughness in numerical simulations can be done using various approaches, including methods based on experimental data or analytical solutions. Understanding the effects of roughness on flow characteristics is essential for the design and optimization of microfluidic devices, as well as for the development of accurate predictive models for fluid flow in mini channels.

Performance of friction turbomachines is based on momentum transfer between operating flow and rotating disks utilizing momentum diffusion [1]. Walls as the main source of turbulence and the formation of a boundary layer highly affect the performance of such systems [2]. Flow properties, domain size and surface roughness are the essential quantities needed to be considered in the analysis of flow behaviour in the Tesla turbine [3].

Flow behaviour and boundary layer formation close to smooth and rough surfaces is an interesting field of research which has been studied extensively[4]. There are two main approaches proposed to account for the roughness. The first approach is based on the logarithmic law of the wall. The dimensionless profile is shifted downwardly according to roughness parameters. This approach, however, requires the adaptation of the wall function. The second approach modifies the turbulence parameters on the wall in order to artificially increase wall shear stress. Both of these methods have advantages and disadvantages. The first method requires a coarse grid to adapt the wall function, which can be an issue in highly confined internal flows but is generally desirable in external flows. The second method necessitates fine mesh, what results substantially increases the computational cost.

https://doi.org/10.52202/069564-0048

The relations between flow parameters and roughness have been the subject of intense investigations since the beginning of the 20th century. Nikuradse [5] studied the wall law in rough pipes and presented a function for dimensionless velocity profiles for smooth and rough wall cases. He also investigated the downward shift of the velocity profile due to roughness and presented this shift as a function of the height of roughness.

$$U^+ = \frac{1}{\kappa}\ln(y^+) + C - \Delta U^+ \tag{1}$$

$$U^+ = \frac{U}{u_\tau} \tag{2}$$

$$y^+ = \frac{u_\tau y}{\nu} \tag{3}$$

Colebrook [6] investigated the turbulent flow in pipes with particular reference to the transition region between the smooth and rough pipe laws. In Grigson's study [7], the performance of finishes representing hull surfaces in trials condition is investigated. The friction coefficient is determined at full scale for many examples. The accuracy of the measurements used to get the actual data on the velocity loss functions of the surfaces is also investigated and the roughness function is presented below:

$$\Delta U^+ = \frac{1}{\kappa}\ln\left(1 + \frac{k_s^+}{\exp(3.25\kappa)}\right) \tag{4}$$

$$k_s^+ = \frac{k_s u_\tau}{\nu} \tag{5}$$

Where k_s stands for sand-grain roughness. There are different correlations proposed for calculations of equivalent sand-grain roughness. Stimpson *et al.*[8] presented the following equation to calculate this parameter:

$$\frac{k_s}{D_h} = 18\left(\frac{R_a}{D_h}\right) - 0.05 \tag{6}$$

Where R_a is the arithmetic mean deviation of the roughness profile and D_h is the hydraulic diameter.

When the utilized mesh is fine with the first layer located in a laminar or transient part of the regime, the roughness function may cause negative values of U^+, therefore, using wall functions, it is better to place the first grid element in the log-law area. Chedevergne and Aupoix [9] developed a wall function to complement the RANS turbulent model involving roughness correction. In another investigation, a new model is proposed to calculate the roughness effect in flow behavior in the channel considering the mixing length [10]. Aupoix [11] introduced a new way of implementing the roughness in $k - \omega$ model for flow simulation by modification of turbulent parameters on the wall. In the current study, the impact of the roughness of the rotating discs is investigated utilising Aupoix's method. A systematic investigation including flow over a rough flat plate, flow through rough minichannel and flow through the gap between rough co-rotating disks is studied. Flow behaviour, velocity profile development, and momentum diffusion in the domain affected by roughness are studied. Table.1 represents a literature review regarding preliminary and main aim of proposed study.

Table 1. literature review of preliminary studies and flow through the gap between co-rotating disks

Authors	year	Object of study	description
Hosni et al. [12]	1993	Flow over rough flat plate	Flow over a flat plate with hemispheric roughness elements is studied
Yuan and Piomelli [13]	2014	LES simulation of channel flow	Turbulent open-channel flows are simulated utilizing Large-Eddy Simulation to determine the roughness function and the equivalent sand-grain roughness height, over sand-grain roughness and different types of realistic roughness.
Romanin and Carey [14]	2011	Flow between co-rotating disks	Integral perturbation solution of the momentum transport and energy conversion equations in microchannels
Rusin et al. [15]	2021	Optimization of efficiency of Tesla turbine	Efficiency based numerical optimization of a Tesla turbine considering different effective parameters

 https://doi.org/10.52202/069564-0048

Considering the accuracy of numerical simulation, when the flow cross section comes to micro scale, it needs more investigation and variable flow cross section like what happens in flow through the gap between co-rotating disks makes it even more challenging.

2. Mathematical model

The main aim of this research is to evaluate the impact of roughness on the flow phenomena in the mini gap between co-rotating disks. To approach this issue, standard three-dimensional (3D) cases are tested utilizing $k-\omega\ SST$ as the turbulent model and Aupoix [11] roughness method.

A finite volume method is employed to solve discretized Reynolds-averaged Navier Stokes equations for the compressible fluid, and the governing equations are continuity, momentum, and energy conservation in the form of:

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_j}(\rho U_j)=0 \tag{7}$$

$$\frac{\partial(\rho U_j)}{\partial t}+\frac{\partial}{\partial x_j}(\rho U_i U_j)=-\frac{\partial p}{\partial x_i}+\frac{\partial \tau_{ij}}{\partial x_j}+\rho f_i \tag{8}$$

$$\frac{\partial\left(\rho\left(e+\frac{1}{2}U_iU_i\right)\right)}{\partial t}+\frac{\partial}{\partial x_j}\left(\rho U_j\left(e+\frac{1}{2}U_iU_i\right)\right) =-\frac{\partial}{\partial x_j}(pU_j)+\frac{\partial}{\partial x_j}(\tau_{ij}U_i)-\frac{\partial}{\partial x_j}(q_j)+\rho f_i U_i \tag{9}$$

The Aupoix method modifies turbulent parameters on the wall resulting in artificial changes in the Eddy Viscosity, depending on the dimensionless sand grain roughness:

$$k_w^+=\max\,(0;k_0^+) \tag{10}$$

$$k_0^+=\frac{1}{\sqrt{\beta^*}}tanh\left[\left(\frac{ln\frac{k_s^+}{30}}{ln10}+1-tanh\frac{k_s^+}{125}\right)tanh\frac{k_s^+}{125}\right] \tag{11}$$

$$\omega_w^+=\frac{300}{{k_s^+}^2}\left(tanh\frac{15}{4k_s^+}\right)^{-1}+\frac{191}{k_s^+}\left(1-exp\left(-\frac{k_s^+}{250}\right)\right) \tag{12}$$

where k_w^+ and ω_w^+ are the dimensionless modified turbulent kinetic energy and specific dissipation rate.

3. Study cases

To approach the main purpose of this investigation, preliminary study cases are chosen to test the performance of the employed turbulent model and roughness approach.

3.1. Flow over a rough flat plate

The presented research starts with the simple case of flow over a flat, rough plate. Hosni et al. [12] studied the flow over a flat plate with hemispheric roughness elements with the diameter of hemispheres equal to D = $1.27mm$. A free stream inlet velocity with $U=58.2\ m/s$ to a computational domain with 5m length and 2m height is simulated. Boundary conditions in this simulation are pressure far field and symmetry for top and side surfaces, and no-slip wall for the bottom surface where the roughness effects are present. Utilizing $k-\omega$ model and the Aupoix method, it is recommended to employ a fine mesh with the y^+ lower than one, therefore, a fine mesh with 400 nodes along the length, 140 nodes along the height and 4 layers in the width direction is used. Results of $k-\omega$ model with the implementation of the Aupoix method on $X=1.68m$ are compared to experimental data of Hosni in Figure 1.

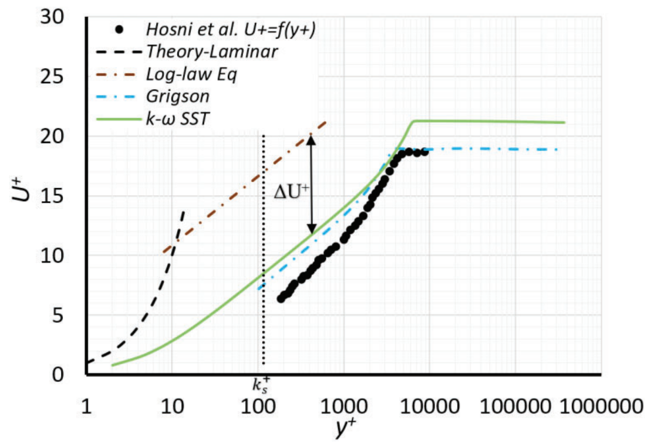

Figure 1. numerical simulation of flow over a flat plate compared to experimental investigation of Hosni et al.

Considering the height of the roughness and obtained friction velocity, k_s^+ is calculated using Eq. (5). Downward shift of the velocity profile is calculated employing Eq. (4). Figure demonstrates good performance of $k\text{-}\omega\, SST$ turbulent model with roughness simulation utilizing the Aupoix method. In the transient and turbulent part of the profile, there is a good agreement between the results of numerical simulation, the theory and experimental data. Moreover, the roughness constant as a function of roughness distribution on the wall is tested in this part of the study. The simulation is done for two values for roughness constant equal to 0.5 and 1, which corresponds to uniform and non-uniform roughness distributions, respectively. The results are presented in Figure 2 for friction coefficient at different distances from the leading edge described by the dimensionless value of $Re_X = \frac{\rho U_\infty X}{\mu}$.

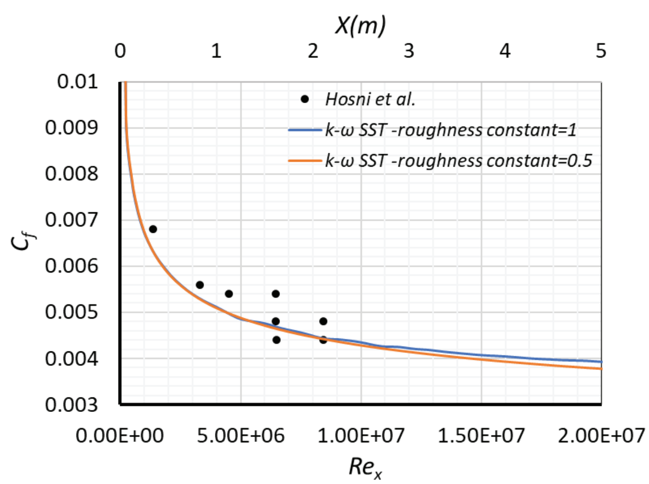

Figure 2. Flow over rough flat plate considering roughness constant equal to 0.5 and 1

Far from the leading edge of the plate where the boundary layer is well developed, the roughness constant equal to 1 contributed to the increment of shear stresses. However, an increase in roughness constant does not reveal a substantial effect on wall shear stress, which means that the uniformity of the distribution is not the most important factor influencing the shear stress.

3.2. Flow in a minichannel

The next step in the preliminary study was the investigation of the flow in minichanel. Yuan and Piomelli [13] simulated a channel flow in the domain with dimensions of 6h×1h×3h in length, width and depth, respectively. They utilized Direct Numerical Simulations (DNS) in their investigation. Roughness height in their simulation was equal to 7% of the channel height, which corresponds to $k_s^+ = 72$. To obtain a well-

https://doi.org/10.52202/069564-0048

developed flow in the whole domain, periodic boundary condition in the front and back faces of the domain was used. The results in that study were shown for the friction Reynolds number (Re_τ) equal to 1000, therefore, the boundary conditions employed herein had to reproduce the same range of Re_τ. Friction Reynolds number was calculated using Eq. (13) where the hydraulic diameter D_h is calculated based on h.

$$Re_\tau = \frac{\rho u_\tau D_h}{\mu} \tag{13}$$

Figure 3 illustrates the schematic of the computational domain of this case study. The numerical simulation reflected the dimensions of a Tesla turbine gap size; hence, the dimensions of the computational domain were 3×0.5×1.5 mm with 35μm height of the roughness. A fine mesh with one million nodes is utilized in the computational domain. The working medium is water. The top and side surfaces are symmetric boundaries, the bottom surface is a no-slip, rough wall. The flow is due to the constant pressure drop between the inlet and outlet. In order to achieve this, velocity components and turbulent parameters are copied from the outlet to the inlet repeatedly during the simulation, until the steady-state solution is reached.

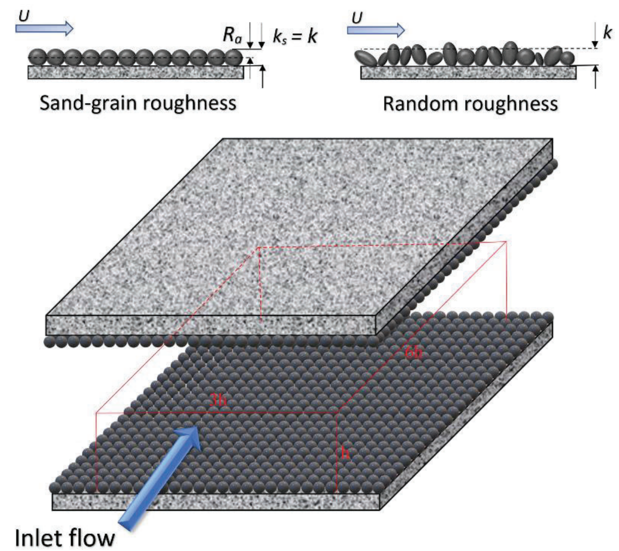

Figure 3. Schematic of Yuan and Piomelli [13] *DNS simulation*

Figure 4 represents the velocity profile of flow in the channel with mentioned dimensions compared to DNS simulation and theoretical lines of smooth and rough cases.

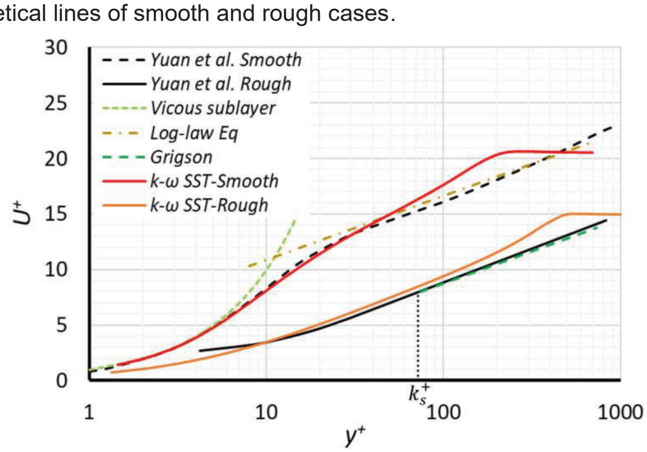

Figure 4. Comparison of the computed dimensionless velocity profile with DNS simulation and theoretical estimations

The results are in good agreement with both DNS simulation and theoretical estimations in the smooth case. The results in the rough case are under the effect of the interaction of boundary layers of parallel surfaces. In smooth case, results are in good agreement with both theoretical lines and DNS simulation in laminar and turbulent regions of the boundary layer. Although the obtained Re_τ in the rough case is equal to 1000, still, the downward shift of the velocity profile is less than the estimated value. This phenomenon demonstrates the effect of channel size on developing boundary layers and velocity profile formation in the computational domain.

4. Flow through the gap between corotating disks

In the main part of the study, the inward flow between co-rotating disks is studied. The performance of the employed turbulent model and roughness method is analysed. Considering the high level of complexity, i.e., the micro-scale size of the gap size, variable flow cross-section, and rotation of walls, the flow analysis is challenging.

To investigate the roughness effects, the simulation is done for a 0.75mm gap between co-rotating disks with 0.1m outer and 0.04m inner diameter. The outer and inner diameters are considered inlet and outlet boundary conditions, respectively. For simplicity reasons, only half of the gap is simulated. One wall of the domain is a rotating disk and on the other one is a symmetry boundary condition. In the computational domain, a fine mesh with ~387 000 nodes and a maximum value of y^+ less than one is used. The study is done for five different roughness heights equal to 2%, 3.5%, 5%, 7% and 10% of the gap size. All results are referenced to the smooth case. Figure 5 demonstrates the viscosity ratio in the middle of the gap from the inlet to the outlet. The viscosity ratio as an indicator of turbulence level is considered to demonstrate the effect of roughness on the development of the turbulent flow.

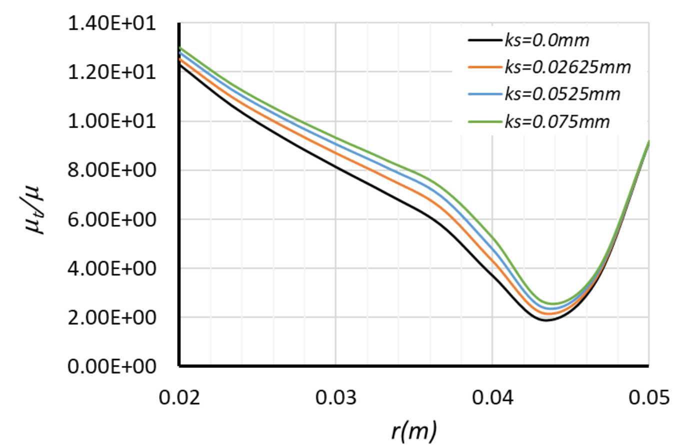

Figure 5. Roughness effect on viscosity ratio along the radius and in the middle of the gap for roughness height equal to 3.5%, 7% and 10% of gap size.

The increase in height of the roughness causes a rise in the viscosity ratio. In the area close to the inlet, the increase in viscosity ratio compared to the smooth case is insignificant and the development of boundary layer and roughness effect on momentum diffusion is more visible in the middle of the channel. An increase in viscosity ratio represents transient from laminar to turbulent boundary layer. In the region close to the inlet, turbulent inlet flow changes to a laminar regime and moving from inlet to outlet, the development of boundary layers under the impact of roughness changes it to a fully turbulent regime.

Also, the effect of roughness height on the downward shift of the velocity profile is depicted in Figure 6.

https://doi.org/10.52202/069564-0048

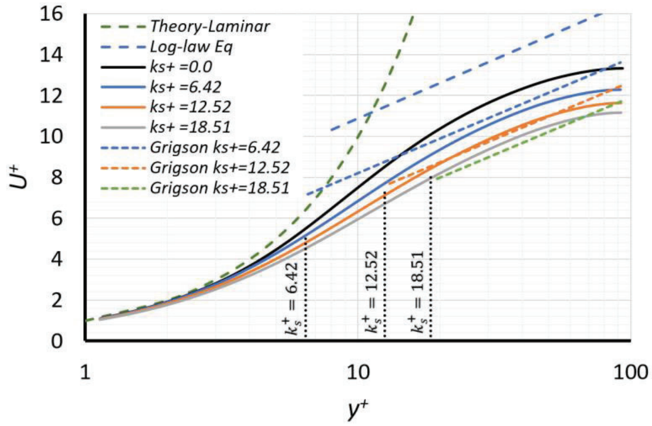

Figure 6. Velocity profile of flow in the middle of disk for smooth and three different roughness compared to theorical lines.

The increase in height of the roughness causes a downward shift of the velocity profile and a rise in the viscosity ratio. The interaction of developing boundary layers from parallel co-rotating disks affects the velocity profile and shifts it down. In Figure 6, different roughness heights are presented as dimensionless values. There is no flow in roughness elements, so the velocity profiles start from the top of the sand-grain roughness.

A rise in roughness height causes an increase in turbulent kinetic energy in the region close to the wall. Farther from the wall and close to the symmetric boundary, the interaction of boundary layers is expected. It can be seen in Figure 7, which shows the turbulent kinetic energy in the middle of the domain for the cases with different roughness heights compared to the smooth case.

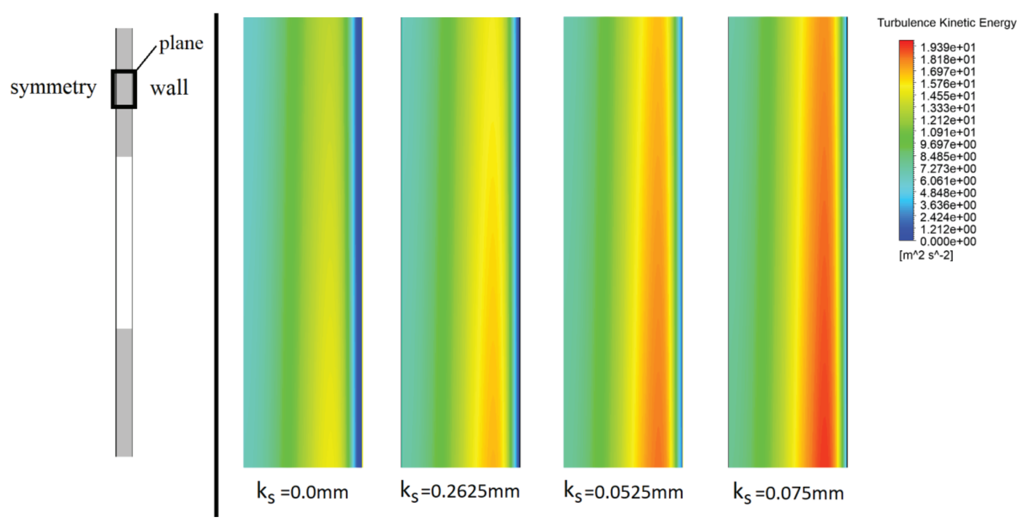

Figure 7. turbulent kinetic energy in the middle of the domain.

The higher roughness intensifies velocity fluctuations close to the wall. An increase in height of the roughness rises the momentum diffusion in the near-wall region. In the vicinity of a symmetric boundary, this parameter is affected by the interaction of boundary layers growing from the other surface of a disk.

In Figure 8, velocity components of flow in the middle of the computational domain in a radius equal to 0.035m for five heights of sand-grain roughness (k_s) is plotted and the results are compared to a smooth case. Although tangential velocity shows a typical downward shift of profile according to implemented roughness, the plot represents an increase in radial velocity magnitude in the area close to the wall, but this value decreases with the distance from the wall. Moreover, the smooth case demonstrates the most changes in radial velocity in the studied location.

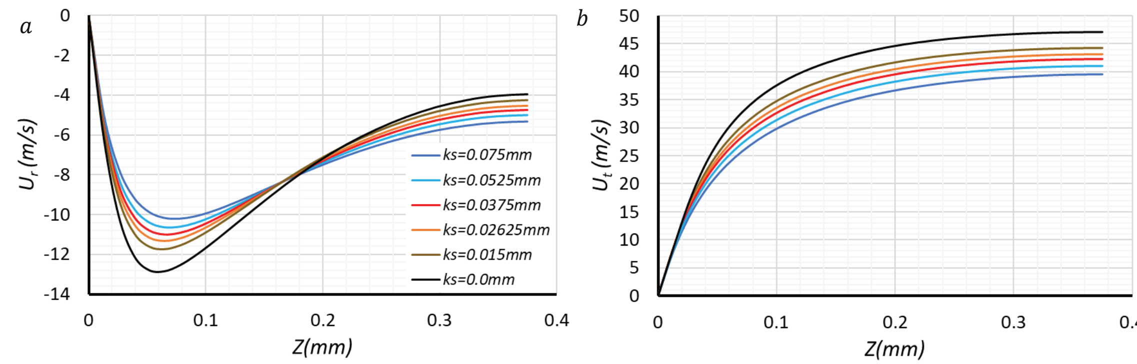

Figure 8. Velocity components in the middle of the disk for different roughness parameters (a) radial velocity (b) tangential velocity.

Roughness height shifts both components of the velocity profile down. An increase in roughness height causes a downward shift of velocity profile and a faster approach to a turbulent regime. considering turbulent kinetic energy, viscosity ratio and radial velocity distribution in the area close to the wall, an increase in height of the roughness rises the momentum diffusion in this area. Accordingly, by the increase in roughness height, wall shear stresses and generated torque of the rotor are raised. The radial velocity profile has a maximum value close to the wall. This behaviour is visible in both smooth and rough cases and causes the discrepancy between the resolved velocity profile and the theoretical lines.

5. Conclusion

The present research work studies the flow through the gap between co-rotating disks with a smooth and rough surface and investigates the roughness effect on flow parameters utilizing Computational Fluid Dynamics software. The main purpose of this study is to simulate flow in the narrow gap between co-rotating disks, like in the Tesla turbine, and to approach this aim, a preliminary study on two standard cases is done.

The formation of velocity profile in the gap between corotating disks is under the effect of variable flow cross-section, moving walls, and small size of the gap. The effect of mentioned parameters causes an increment of radial velocity close to the wall, which decreases in the near symmetry region. This influence is also visible in turbulent kinetic energy and viscosity ratio which represent more momentum diffusion close to the wall surface than close to symmetry. Roughness affects the boundary layer formation and shifts the velocity profile downward. Considering the flow in the gap between co-rotating disks, by the increase in height of the roughness the viscosity ratio increases, and the transition from laminar to turbulent regime occurs in the area closer to the wall. Moreover, roughness affects the turbulent kinetic energy and momentum diffusion, and an increase in roughness height causes more obtainable values of these two parameters close to the wall. Close to the symmetry in the middle of the gap, the flow character is under the effect of the interaction of boundary layers and this phenomenon impacts the flow behaviour in the middle of the gap.

Acknowledgement

The presented research was conducted within the UMO-2019/35/B/ST8/01871 research project financed by the Polish National Science Centre and internal project 08/050/BKM_22/0271 financed by the Silesian University of Technology.

Nomenclature

c	Smooth-wall interception
D_h	Hydraulic diameter, m
e	Inner energy, J
h	Half of channel height, m
k	Turbulent kinetic energy, J/kg
k_s	Equivalent sand-grain roughness, m
U^+	Dimensionless mean velocity (related to wall shear stress)
U_∞	Free stream velocity, m/s
u_τ	Skin friction velocity, m/s
ΔU^+	Roughness function
y^+	Dimensionless normal-wall distance (related to wall shear stress)

Greek letters

https://doi.org/10.52202/069564-0048

k_s^+	Roughness Reynolds number	κ	*Von Karman constant*
k_w^+	Dimensionless Turbulent kinetic energy close to the wall	ν	Kinematic viscosity, m^2/s
p	Pressure, Pa	ρ	Density, kg/m^3
R_a	Arithmetic mean deviation, m	τ	Tangential stress, Pa
Re	Reynolds number	μ	Dynamic viscosity, $Pa.s$
Re_τ	Friction Reynolds number	ω	Specific dissipation rate, $1/s$
k	Roughness height, m	ω_w^+	Specific dissipation rate close to the wall
T	Temperature, K	*Subscripts and superscripts*	
t	Time step, s	*r*	*radial*
U	Mean velocity, m/s	*t*	*tangential*

References:

[1] Nikola Tesla U.S. Patent 1,061,206 - Turbine n.d. https://teslauniverse.com/nikola-tesla/patents/us-patent-1061206-turbine (accessed October 26, 2022).

[2] Farzaneh-Gord M, Pahlevan-Zadeh MS, Ebrahimi-Moghadam A, Rastgar S. Measurement of methane emission into environment during natural gas purging process. Environmental Pollution 2018;242:2014–26. https://doi.org/10.1016/J.ENVPOL.2018.07.027.

[3] Rusin K, Wróblewski W, Rulik S. The evaluation of numerical methods for determining the efficiency of Tesla turbine operation. Journal of Mechanical Science and Technology 2018 32:12 2018;32:5711–21. https://doi.org/10.1007/S12206-018-1118-4.

[4] Bezaatpour J, Ghiasirad H, Bezaatpour M, Ghaebi H. Towards optimal design of photovoltaic/thermal facades: Module-based assessment of thermo-electrical performance, exergy efficiency and wind loads. Appl Energy 2022;325:119785. https://doi.org/10.1016/J.APENERGY.2022.119785.

[5] Nikuradse J. Laws of Flow in Rough Pipes Nikuradse. 1933.

[6] Colebrook CF. Turbulent Flow in Pipes, with particular reference t o the Transition Region between the Smooth and Rough Pipe Laws. Journal of the Institution of Civil Engineers 1939;11:133–56. https://doi.org/10.1680/ijoti.1939.13150.

[7] Grigson C. Drag Losses of New Ships Caused by Hull Finish. Journal of Ship Research 1992;36:182–96. https://doi.org/10.5957/JSR.1992.36.2.182.

[8] Stimpson CK, Snyder JC, Thole KA, Mongillo D. Scaling roughness effects on pressure loss and heat transfer of additively manufactured channels. J Turbomach 2017;139. https://doi.org/10.1115/1.4034555/378795.

[9] Chedevergne F, Aupoix B. Accounting for wall roughness effects in turbulence models : a wall function approach 2017. https://doi.org/10.13009/EUCASS2017-372.

[10] Chedevergne F, Forooghi P. On the importance of the drag coefficient modelling in the double averaged Navier-Stokes equations for prediction of the roughness effects. Https://DoiOrg/101080/1468524820201817465 2020;21:463–82. https://doi.org/10.1080/14685248.2020.1817465.

[11] Aupoix B. Roughness corrections for the k-ω shear stress transport model: Status and proposals. Journal of Fluids Engineering, Transactions of the ASME 2015;137. https://doi.org/10.1115/1.4028122/374661.

[12] Hosni MH, Coleman HW, Garner JW, Taylor RP. Roughness element shape effects on heat transfer and skin friction in rough-wall turbulent boundary layers. Int J Heat Mass Transf 1993;36:147–53. https://doi.org/10.1016/0017-9310(93)80074-5.

[13] Yuan J, Piomelli U. Estimation and prediction of the roughness function on realistic surfaces. Http://DxDoiOrg/101080/146852482014907904 2014;15:350–65. https://doi.org/10.1080/14685248.2014.907904.

[14] Romanin VD, Carey VP. An integral perturbation model of flow and momentum transport in rotating microchannels with smooth or microstructured wall surfaces. Physics of Fluids 2011;23:82003. https://doi.org/10.1063/1.3624599/378002.

[15] Rusin K, Wróblewski W, Rulik S. Efficiency based optimization of a Tesla turbine. Energy 2021;236:121448. https://doi.org/10.1016/J.ENERGY.2021.121448.

 https://doi.org/10.52202/069564-0048