



The Simulation of Convective Heat Transfer in a Tesla Turbine Gap

Roberto Lisker^{a*}, Udo Hellwig^b

^a*Roberto Lisker M.Eng. Technical University of Applied Sciences Wildau, 15745 Wildau, Germany*

^b*Udo Hellwig Prof. Dr.-Ing. ERK Eckrohrkessel GmbH, Am Treptower Park 28a 12435 Berlin, Germany*

^a*Email: roberto.lisker@th-wildau.de*

^c*Email: uhellwig@eckrohrkessel.com*

Abstract

Due to its unique design and large rotor surface, the Tesla turbine is predestined for heat transfer processes. To calculate the heat transfer, a simple steady state model based on the Blasius boundary layer analogy was developed and compared to a numerical simulation. It could have been shown, that the model fits the numerical simulation well for the surface heat flux and the Nusselt number. For the heat transfer coefficient, the simplified model needs a refinement due to a different calculation approach between the numerical simulation and the analytic solution. Furthermore, it could be shown, that the simplified model can only be applied if the turbine inlet velocity is in the range of the disk velocity. Otherwise a flow channel develops which is not covered by the simplified model.

Keywords: Tesla turbine; Friction turbine; heat transfer; rotating flows; simulation.

1. Introduction

Invented by Nicola Tesla in 1909 [1], the turbine was never a commercial success and is regarded as a “lost invention” today. Although the fail as a commercial product, the Tesla turbine has stayed an object of academic research throughout the last decades due to its simplicity, low production costs and unique flow field. Since 2008

* Corresponding author.

the Tesla turbine is a constant research topic at the Technical University of applied Sciences Wildau with the aim to develop a high efficient Tesla turbine for small scale application [2, 3]. The Turbine consists of several disks, which are plane parallel arranged on a shaft. The gap between these disks is depending on the working fluid which enters the gap at the outer perimeter with a high tangential velocity. Then the fluid continues its way in a spiral pathway to the exit, located at the centre of the disks. Due to frictional forces, caused by the viscosity of the working fluid, the rotor starts spinning and torque is produced. In figure 1 left, a Tesla turbine model is displayed. The unconventional build-up of the turbine has a lot of advantages. It is resistant against particles enclosed in the working fluid stream if the particle diameter is smaller than the gap width. The turbine can easily be adjusted to different working fluids and conditions by simply changing the gap width. On the downside of this turbine type are the rising tangential and radial velocities towards the outlet, caused by the continuous expansion of the working fluid in the gap and the decreasing of the cross section whilst increasing the specific volume of the working fluid (continuity). Especially the increase in relative tangential velocity accounts for high energetic losses and leads to a significant turbine efficiency decrease [2, 4]. In figure 1 right, the streamlines of the flow field inside a turbine gap rotating at 3000 rpm is displayed. The inlet velocity is 60 m s^{-1} and accelerates up to 134 m s^{-1} at the outlet.

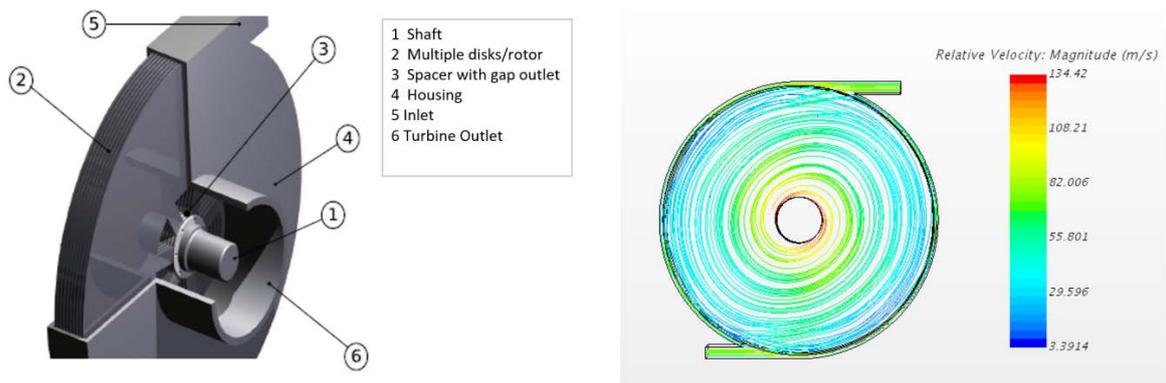


Figure 1: left; two-plane- section of a Tesla turbine model; right: Streamline plot of the flow field inside of a Tesla turbine, the rising relative velocities towards the outlet in the centre can be seen.

Due to the high surface area in contact with the working fluid and the ability to handle particle laden flows, the turbine is predestined for the use as heat exchanger and the utilization as condensation turbine. Within the Tesla turbine gap all three kinds of heat transfer can apply. Conduction through the disk wall and the cooling agent, forced convection from the working fluid to the disk and radiation at higher temperatures. In this article, the heat from the working fluid to the disk will be transferred by forced convection, therefore it is essential to know the velocity field. A detailed description of the flow field has been given by Guha, Lisker and Rice [5–7].

The idea of using this turbine as a heat exchanger is rather old. A first patent was filled by the Oklejas in 1974 where they described a regeneration turbine with cooled disks [8]. A subsequent patent by the Oklejas in 1975 described an air-cooled disk for cooling combustion gases [9]. The next patent about a cooled disks rotor came from Prueitt in 2007 who patented an isothermal power system. The basic principle of the cycle is the operation

of the turbine in an isothermal state. Thus, the cooling agent evaporates inside the disks, resulting in a uniform disk temperature [10]. The latest patent regarding a cooled turbine rotor is the idea of Cochran to use heat pipes to transport heat off the working fluid to the cooling agent flowing through the centre of the shaft [11]. Although patents do exist, there is little scientific debate on heat transfer in friction turbines. The first attempt to describe the heat transfer in a friction turbine was carried out by Matveev. He used a numerical scheme to determine the velocity field, the heat transfer and the heat flux on the disc surface. Unfortunately, he did not publish the heat transfer results [12]. Furthermore, the heat transfer processes in a friction turbine gap has been carried out by Felsch and Piesche for a compressible flow. They concluded that due to the large shear force at the outlet, dissipation dominates and the working fluid temperature rises [13].

2. Basics of heat transfer in a Tesla turbine gap

To apply heat transfer in a friction turbine, a cooling system must be integrated into the disks. Here, several options are available. The first is to design the disk with an integrated friction pump, so the entire inner disk surface is covered with the cooling agent. A second option would be the integration of cooling channels. In both cases, the cooling agent flows from the centre to the outer perimeter of the disk due to the centrifugal force. The advantage is an increased pressure of the cooling agent which results in an increase of heat capacity. The downside is that the energy for the compression must come from the working fluid and is an additional energy loss. In a Tesla turbine, and especially in a Tesla turbine suited for heat transfer, forced convection and conduction prevails. The heat from the fluid to the solid disk is transferred by forced convection. It is transferred to or from the disk wall due to the relative velocity of the fluid to the disk wall. Here, the relative velocity in tangential direction is the predominant parameter since it is larger than the radial velocity. If the velocity field and the relative tangential velocity is known, the heat transfer coefficient, the Nusselt numbers and the surface heat flux can be determined. To model the heat transfer, a flow over a flat plate is considered to better represent the physical system of a friction turbine. The model was first presented by Blasius and is based on Prandtl's boundary layer theory. He solved the energy equation by assuming, that the temperature and the velocity profile are similar [14].

In a Tesla turbine, the fluid enters the gap at the outer perimeter of the disk and after an initial phase, a velocity and thermal boundary layer develops along the overflow length due to the no-slip condition and a condition without sudden temperature changes.

Within the thermal boundary layer, the heat exchange between the solid disk and the fluid takes place. In figure 2 the flow over a flat plate is shown, with the gas velocity c_{gas} and the temperature T_{gas} . Due to the flow, a velocity boundary layer develops, in which the shear forces are transferred to the disk wall.

Parallel to the velocity boundary layer, a thermal boundary layer is generated, in which the heat transfer takes place. The Blasius theory can be applied to the friction turbine, if the gap width is so thick, that a velocity and thermal boundary layer develops on the disk surface, and enough space is left to establish free flow. Both profiles (temperature and velocity) are assumed to be parabolic.

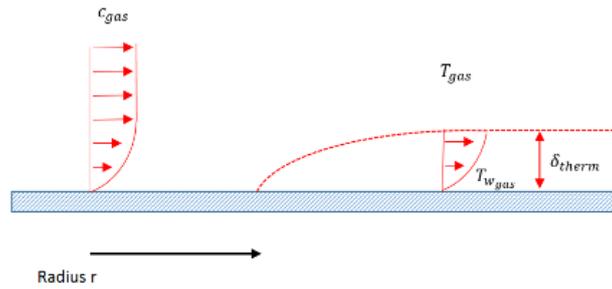


Figure 2: flow past a plate with velocity c_{gas} and the developing thermal boundary layer δ_{therm} .

In general, the transferred heat can be described through equation 1

$$\frac{Q}{A} = q = \alpha (T_f - T_D) \quad (1)$$

where Q is the transferred heat, A is the surface area of the disk, T_f is the fluid temperature and T_D is the disk surface temperature. Since the area and the temperatures can be easily determined, the unknown variable in equation 1 is the heat transfer coefficient α . It describes how much heat is transferred across the surface area. To determine the heat transfer coefficient, the boundary layer equations must be solved. For a fully developed steady, laminar flow of a compressible fluid a solution was presented by Schlichting. The temperature profile for a flow over a flat plate with a constant wall temperature, can be determined by equation 2[15].

$$\frac{T - T_{aw}}{T_{aw}} = \frac{\kappa - 1}{2} Ma^2 \left[1 - \left(\frac{c_v(y)}{c_{v,max}} \right)^2 \right] \quad (2)$$

In this equation, κ is the isentropic coefficient of the fluid, T_{aw} is the adiabatic wall temperature, c_v is the velocity in the boundary layer, $c_{v,max}$ the maximum velocity at the boundary layer boarder and Ma is the Mach number. In a system with an adiabatic disk wall and an increasing relative tangential velocity, the temperatures at the disk wall rises due to friction losses. The resulting temperature is the adiabatic wall temperature T_{aw} . It is the temperature due to irreversible dissipation/friction and can be determined through equation 3.

$$\frac{T_{aw}}{T_{ff}} = 1 + \Lambda \frac{\kappa - 1}{2} Ma^2 \quad (3)$$

Here, the recovery factor Λ is introduced. This factor is the ratio of the irreversible loss to the reversible loss and is proportional to the Prandtl number. For Prandtl numbers in the range of $0.6 < Pr < 15$ the recovery factor is $\Lambda \approx Pr^{1/2}$ [16]. Furthermore, T_{ff} is the free flow temperature of the working fluid at the boarder of the boundary layer. For low Mach numbers the dissipation loss can be neglected. The temperature in the boundary layer can exceed the free flow temperature T_{ff} . Therefore, the dissipation generated at the outlet of a Tesla turbine, results in an increase in the working fluid temperature, causing efficiency losses due to lower fluid viscosities and higher outlet temperatures. One option to prevent this is to cool the disk in such a way that the generated supplementary heat is removed from the fluid flow. Ladino has investigated the impact of the Mach number on the flow and pressure fields in a Tesla turbine gap and states that the flow can be regarded as

incompressible up Mach 0.4. Even at higher Mach numbers, up to Mach 0.6, the error is small compared to equation 2 [17, p. 37]. If the temperature distribution at the wall and the velocity profile are known, the surface heat flux at the disk wall can be determined using the following equation.

$$q_w(r) = \frac{\lambda_w(T_0 - T_w)}{c_v(r)} \left(\frac{\partial c_v}{\partial y} \right)_w \quad (4)$$

where λ_w is the thermal conductivity of the wall, $T_0 - T_w$ is the temperature difference between the wall and the free stream and $\frac{\partial c_v}{\partial y}$ is the wall shear gradient, determined at the disk wall. A higher shear gradient results in an increase of the surface heat flux. Thus, the highest heat transfer can be expected at the in- and outlet region since the shear stress gradient is high in these areas. Especially when the fluid enters the gap between the disk a high shear force is wanted to generate a high drag force which results in high torque. It is convenient to introduce a dimensionless figure to allow an easier comparison to other heat transfer systems (heat exchangers etc.). Here, the Nusselt number is introduced, which is the ratio of heat convection to heat conduction and is a function of the disk radius.

$$Nu = \frac{q_w(r) r}{\lambda_w(T_w - T_0)} \quad (5)$$

For the calculation of the heat transfer coefficient α , equation 4 will be used.

$$\alpha = \frac{q_w(r)}{(T_w - T_0)} = \frac{Nu \lambda_w}{r} \quad (6)$$

In general, the heat transfer coefficient depends on the surface heat flux and the temperature difference between the free flow and the disk wall. About the turbine design, it can be said that the highest heat transfer coefficients are achieved in the in- and outlet region and thus most heat is transferrable in that area.

The presented model has some restrictions. Since the model is based on the Blasius similarity, it is only valid for a laminar flow, which should be ensured in a Tesla turbine gap, otherwise the drag applied to the disk is low and the resulting torque too. A turbulent flow field cannot be mapped by the presented model. The impact of the higher molecular movement and the different velocity profile are not predictable by the Blasius equation. Concerning the flow field, a spiralling path as described by Tesla in his patent should be achieved [1]. Here the fluid has a high residence time inside of the gap and thus, the heat transfer is maximised. This is achieved if the radial velocity at the gap inlet is low and the tangential velocity just over the tangential disk velocity.

3. Computational model

For the verification of the above described model, various computational fluid dynamics simulations have been carried out and compared to the model. The simulations have been carried out with version 11.04.012 of the commercial software Star CCM+. The dimensions and operation conditions of the simulation model are taken from an existing friction turbine researched at the Technical University of Applied Sciences in Wildau and are presented in table 2. The performance of this turbine was measured for saturated and superheated steam and the turbine efficiency could be determined at 0.3 for saturated and 0.7 for superheated steam [18, 19]. Figure 3

shows a two-plane section of a 3D model and the turbine prototype.

Table 1: dimensions of the turbine prototype and operation conditions

Parameter	Dimension
Disk diameter	0.175 m
No. of disks	7
Gap width	0.005 m
Design parameters:	
Inlet pressure	1 MPa
Inlet temperature	350 °C
Outlet temperature	100 °C
Max. mass flow rate	60 kg h ⁻¹

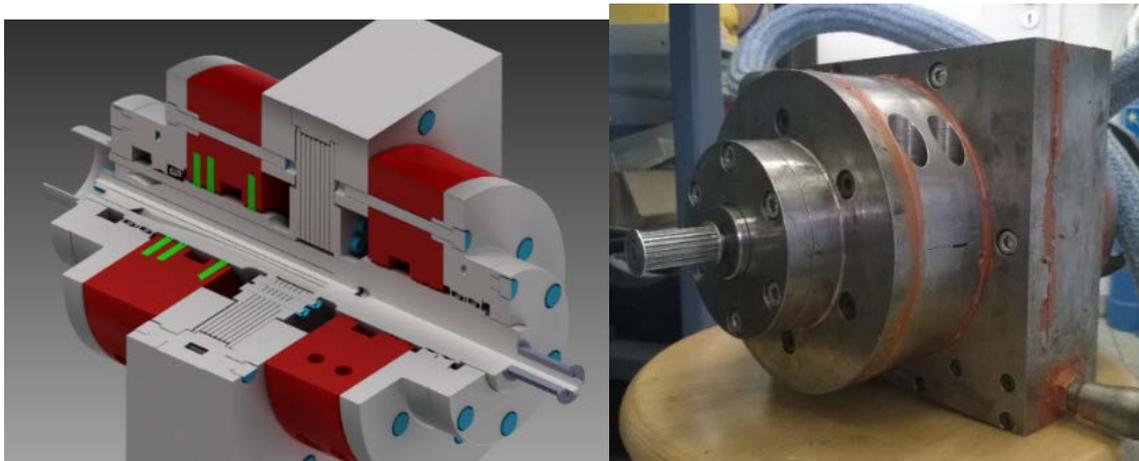


Figure 3: experimental friction turbine studied at TH- Wildau; left: a cross-section of the 3D model; right: the 2.2 kW friction turbine prototype [20]

The prototype shown in figure 3 is the basis for the simulation model, since the gathered results will be implemented in the design of a new turbine prototype with cooled disks and similar dimensions. In the numerical simulation, only one gap will be simulated, since the results can be extrapolated for further gaps. The first step is to define the geometry and provide it with a suitable mesh, where the algebraic equations (Navier-Stokes Equations) can be solved. The solution process is a numerical approximation based on the finite volume method. The following figure shows the basic model geometry and the mesh. The model is divided into two parts. A static part (stator), with the inlet and the turbine housing (figure 4, blue coloured) and a rotating part (rotor), which is the gap with the disk walls (figure 4 grey).

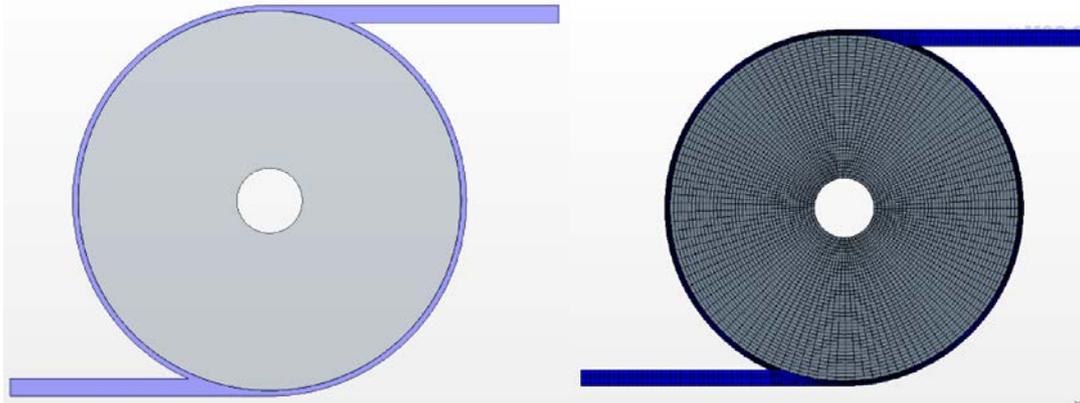


Figure 4: Basic simulation model built in Star CCM+; left: the geometry of the turbine model with its two inlets and the outlet in the centre; Right: the mesh of the turbine gap

The mesh consists of approximately 1.5 Million cells and the predominant mesh type is a hexahedron element due to the benefits in accuracy and calculation time. A mesh sensibility study on the above-mentioned geometry has been done by Lisker, Meller, Stein and Witt. The findings are that for the given geometry the cell number should be in the range of 1.2 -1.6 Million cells. A lower cell density leads to a non-sufficient resolution, whereas a higher cell density especially in the rotor part leads to bad skewness angles which influence the result and lead to a simulation crash [3, 21, 22]. The following table shows the models used for the heat transfer simulation. Detailed descriptions of these models are not possible, within the scope of this article. The model details not explained in the following are described by Meller, Stein and the Star CCM+ user guide [21, 23, 24].

Table 2: used models in the computational simulation

Settings	CFD Model
Material	gas
Gradient metrics	gradients
Real gas equation of state	IAPWS-IF97 (steam)
Viscous regime	laminar
Equation of state	real gas
Time	steady
Space	three dimensional
Flow	coupled flow
Energy	coupled fluid enthalpy
Wall treatment	all y+ wall treatment
	exact wall

The heat transfer simulation is carried out with a disk rotation rate of 3000 rpm and inlet velocities of 60, 90 and

120 m s⁻¹, which is two, three and four times the outer disk velocity. The working fluid is steam with an inlet temperature of 700 K and the wall temperature is set to a constant 400 K to obtain a sufficient temperature difference between the fluid and the wall and to avoid condensation. The turbine outlet pressure was set to normal pressure (101325 Pa). For the comparison of the heat transfer values to the simplified model, it is essential to understand how the simulation software calculates the Nusselt number and the heat transfer coefficient. The Nusselt number is calculated like equation 5 [24]:

$$Nu = \frac{\alpha r}{\lambda} \quad (7)$$

The heat transfer coefficient α is calculated according to the following equation

$$\alpha = \frac{\rho_f(y) c_{p,f}(y) u^*}{t^+(y^+(y))} \quad (8)$$

where ρ_f is the density and $c_{p,f}$ is the specific heat capacity of the fluid [24]. Furthermore, u^* is a reference velocity calculated with the standard wall function. In the denominator t^+ is a dimensionless temperature and y^+ a dimensionless wall function, defined as follows

$$y^+ = \frac{u^* y}{\nu_f} \quad (9)$$

Here, y is the normal distance of the near wall cell which is the distance from the wall to the centre of the adjacent cell. In general, if y^+ is below 30, the viscous boundary layer can be considered laminar. Therefore, a low y^+ is desired in the simulation. Star CCM+ internally uses the above equations to calculate the heat transfer in laminar and turbulent flows and thus, it cannot be changed [24].

4. Results

In the following the results of the comparison are presented. For the analytic model steam was used as working fluid and the steam properties are calculated according to IAPWS 97.

In figure 5 the surface heat flux q along the disk wall is shown from the analytic model and the simulation. It can be seen, that it reaches its highest values at the outer radius of the disk.

The reason is the combination of the high relative tangential velocity (accounts for a high wall shear gradient) and the high temperature difference of 300 K between the disk and the fluid at the gap inlet.

For higher inlet velocity cases (3000 rpm 90, 120 m s⁻¹), a rise in the surface heat flux along the radius can be seen in the outlet area. The values of the surface heat flux at the inlet of the gap in the simulation are lower than in the model.

This is due to the uneven velocity distribution along the outer perimeter in the simulation and the point where the data is obtained. At this point the tangential velocity was lower than the inlet one. In figure 5 a scalar plot of the wall heat flux for all three simulations is shown. The red dashed line represents the line where the data for

the comparison are obtained.

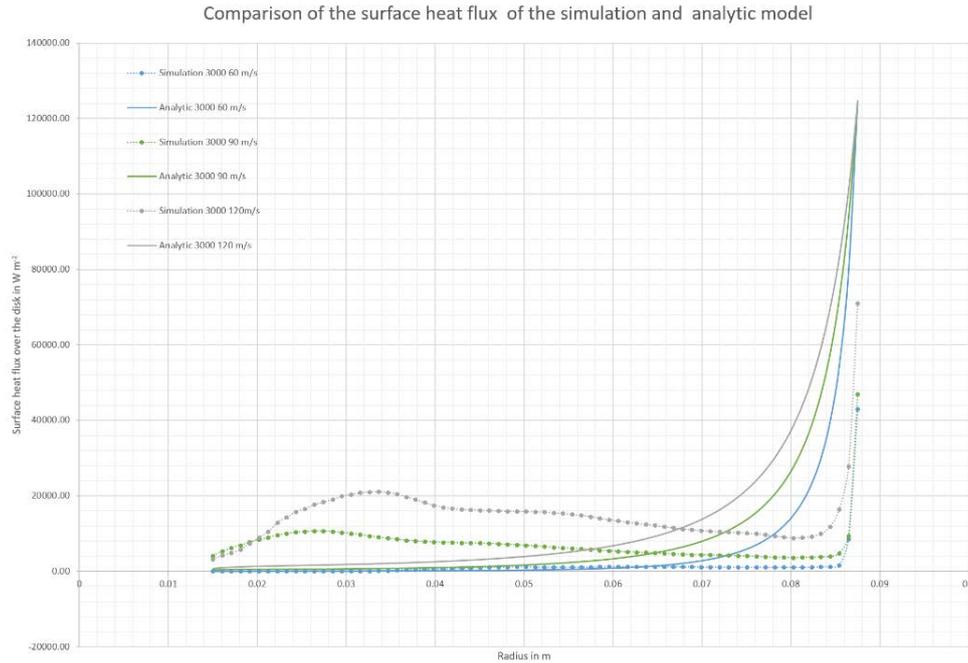


Figure 5: comparison of the simulation’s and model’s wall heat flux for 3 different tangential inlet velocities at a constant rotation rate of 3000 rpm. The dotted lines represent the numerical data and the straight lines the analytic model.

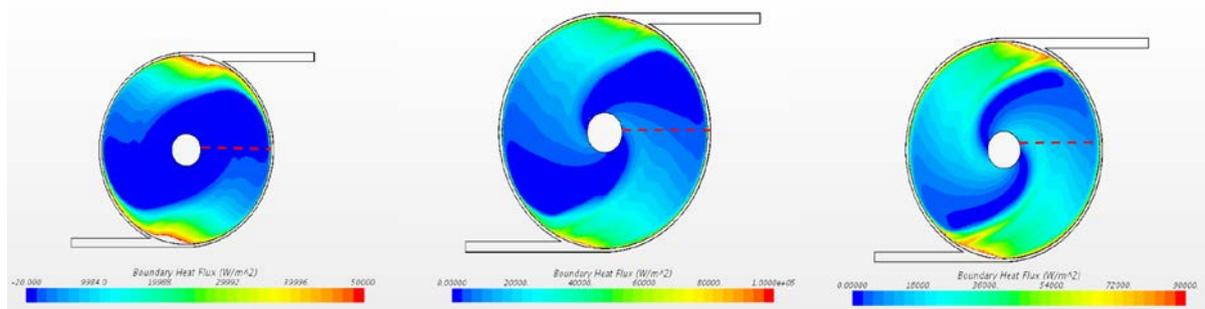


Figure 6: plot of the surface heat flux from the simulation; left: 3000 rpm and 60 m/s inlet velocity; centre: 3000 rpm and 90 m/s inlet velocity; right: 3000 rpm and 120 m/s inlet velocity; the red dashed line represents the line where the simulation’s data was obtained.

A good agreement between the simulation and the analytic model shows the 60 m s⁻¹ simulation case. Despite the inlet differences the analytic model shows a good correlation to the simulation.

The proposed increase in the surface heat flux towards the outlet cannot be shown, due to the high surface heat flux at the inlet. The working fluid cools down fast and at a third of the radius the wall temperature is reached. From this point onwards, no significant temperature difference between the fluid flow and the wall is present. In the simulation cases with 90 and 120 m s⁻¹, the heat flux rises again at a radius of 0.08m, so that a temperature difference must be present. The cause of this is a flow channel which develops if the inlet tangential velocity is

significantly higher than the disk velocity. Within this flow channel, the velocities as well as the mass and heat flow are concentrated and significantly higher than in other regions of the gap. The analytic model is not able to predict the development of a flow channel and thus, it fails to predict the rise of the surface heat flux towards the outlet. In figure 7, a scalar plot of the temperature distribution from the simulation is shown. As expected from the surface heat flux, the temperature at the outlet rises due to the high velocities caused by the flow channel.

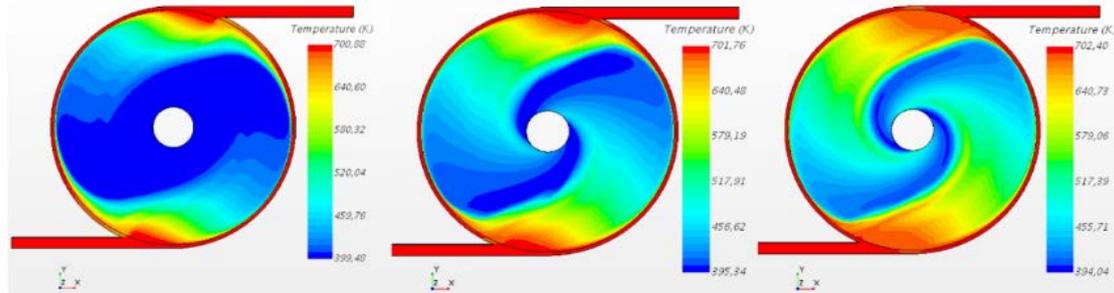


Figure 7: Scalar plot of the temperature field from the simulation; left: 3000 rpm and 60 m/s inlet velocity; centre: 3000 rpm and 90 m/s inlet velocity; right: 3000 rpm and 120 m/s inlet velocity

In figure 8 the distribution of the Nusselt number is shown. The Nusselt number of the simulation at the inlet of the gap is high, and strongly decreases within the first millimetres. This is due to an initial turbulence occurring at the outer edge of the disk and the high tangential velocities in this region. Here, the flow is not fully developed and turbulent so that the heat transfer increases due to higher momentum transport. Even though the calculation model in the simulation has been set to laminar, the programme automatically switches in this area to a turbulent flow regime, to provide a convenient solution. The analytic model assumes a laminar flow throughout the whole gap and fails to predict the high Nusselt numbers at the inlet. Again, despite of the inlet region, the 60 m s^{-1} simulation shows a good agreement with the analytic model (blue straight line).

The distribution of the wall heat transfer coefficient along the radius is displayed in figure 9. It can be seen, that the values strongly differ and the model fails to predict the heat transfer coefficient. The cause is the different approach to the heat transfer coefficient calculation. The analytic model uses the maximum free flow velocity and temperature to determine the surface heat flux and wall heat transfer coefficient.

The simulation uses the velocity and temperature values adjacent to the disk wall. Here, the relative velocity as well as the temperatures strongly differ. The high initial heat transfer coefficient in the simulation is caused again by the turbulences occurring at the outer edge of the disk and the high tangential velocity. Interestingly, the simplified model predicts a positive heat transfer coefficient at the outlet for the 60 m s^{-1} case.

An explanation would be, that due to the expansion of the working fluid and the high velocity the pressure drop is so large that the temperature of the fluid is lower than the disk temperature and heat is transferred from the disk to the fluid. The influence of dissipation can be neglected in this case, since the Mach numbers at the outlet are in such range that dissipation does not play a role.

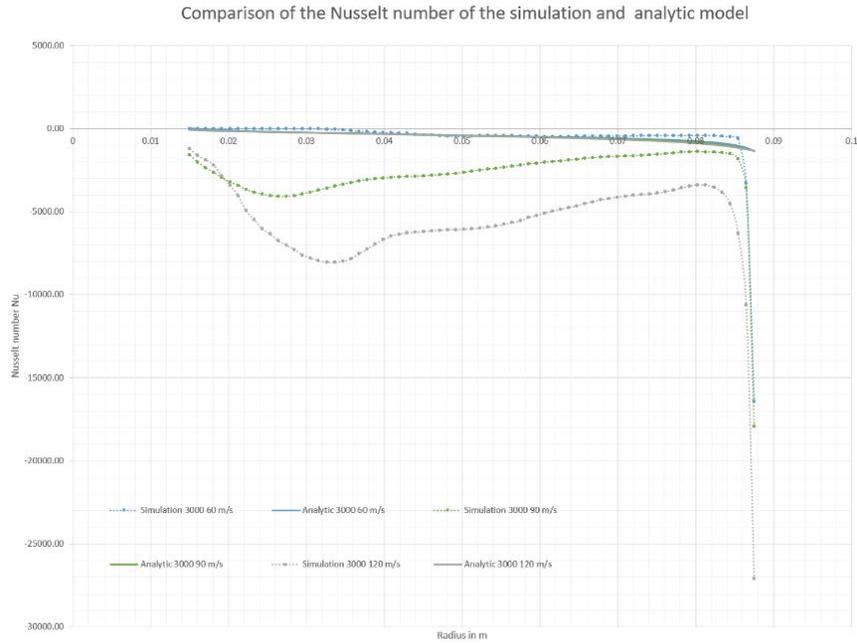


Figure 8: comparison of the simulation's and the model's Nusselt number

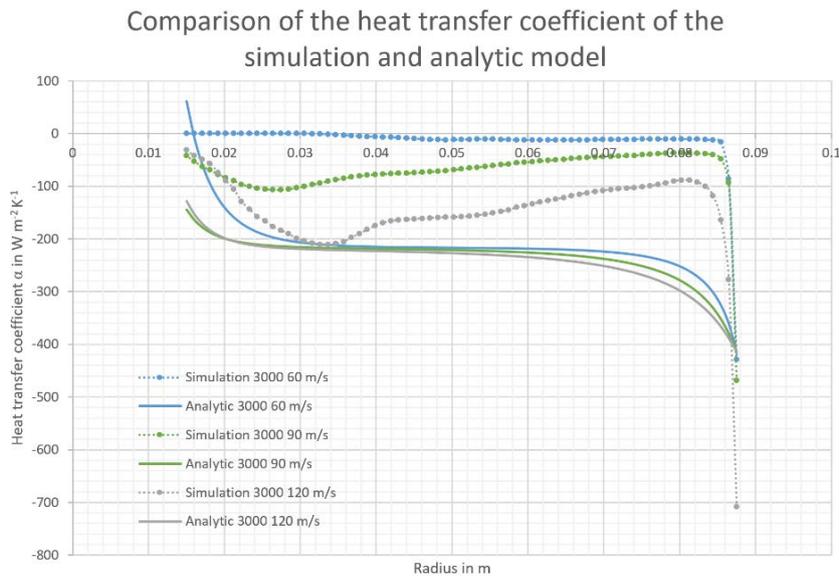


Figure 9: comparison of the simulation's and model's wall heat transfer coefficient

5. Discussion

Comparing the results of the numerical simulations to the simplified model, it can be seen, that the analytic model cannot be used for the development of Tesla turbines with heat transfer, if the tangential inlet velocity is significantly higher than the tangential disk velocity. The reason for this is the development of a flow channel, in which most of the mass and heat is transferred. Since the main flow concentrates in the flow channel, most of the heat will be transferred in this area. Figure 10, shows a combined plot of the surface heat flux and the

streamlines within the gap. It can be seen, that the absolute velocity and the surface heat flux are high within the flow channel whereas other areas are not affected. The downside of the formation of a flow channel is the reduction of the heat transfer area and residence time of the fluid inside the gap. Thus, the Tesla turbine cannot unfold its full potential in heat transfer and torque production.

The determination whether a flow channel develops or not is complicated. An approach could be the determination of the fluid particle path line. Therefore, the residence time of a fluid particle must be known. A first attempt to compute the fluid path lines has been done by Guha and Sengupta. They used a finite difference scheme to derive the fluid path lines [5].

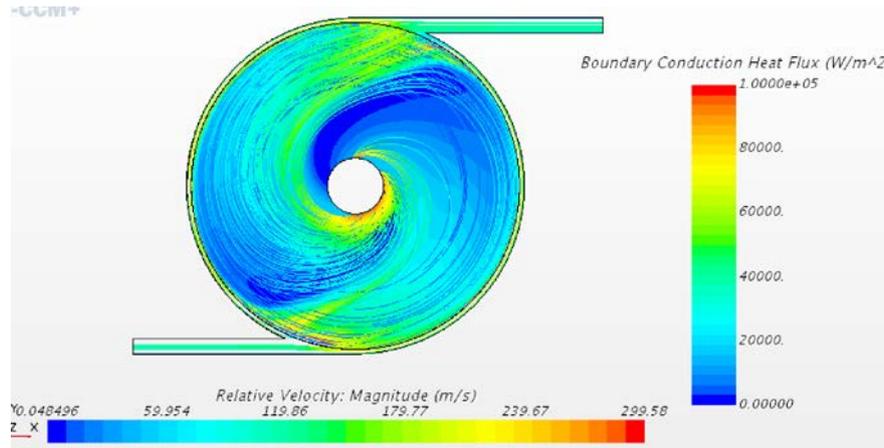


Figure 10: Combined plot of the streamlines and the surface heat flux; the flow channel with high velocity areas can be seen.

To overcome the development of a flow channel, the rotation rate could be increased until the tangential inlet velocity is in the range where the model is valid.

A downside of the simplified model is the prediction of the heat transfer coefficient α . To improve the prediction of the heat transfer coefficient, the relative velocity used for the determination of the surface heat flux should be changed to a relative velocity next to the wall. With the given velocity profile, it is possible to determine the relative velocity close to the disk wall and thus, a more accurate way to predict the heat transfer coefficient is achieved.

6. Conclusion

Concluding from the cases presented above, the analytic model fits the simulation well if the tangential velocity does not exceed the disk velocity by far. The reason for the large deviation of cases with high tangential inlet velocities is the development of a flow channel where the mass flow and the heat transfer concentrates. The simplified model, conforms well to the values of the 3000 rpm, 60 m s^{-1} simulation. Here, the tangential velocity is twice the disk velocity and the mass flow rate is low. The presented model can be used for the design of a Tesla turbine with cooled disks, the model can be used to determine the surface heat flux and the Nusselt number. It is suitable for the calculation of the heat transfer in the gap of a Tesla turbine, if the tangential velocity at the inlet is bigger than the tangential disk velocity and lower than twice the tangential velocity of the

disk. The calculated heat transfer coefficients should be used with care since the model overpredicts these coefficients. For the determination of the heat transfer coefficient, the model must be adapted to a near wall velocity to accurately predict the heat transfer coefficients. With the gained insight, it is possible to develop Tesla turbines which work as heat exchanger. Thus, a cooling system must be integrated into the disk which can provide a constant wall temperature and temperature difference. The application of such turbine can be a high temperature turbine for hydrogen combustion as well as low temperature applications such that condensation inside of the Tesla turbine gap is possible. The later would further increase the turbine efficiency.

References

- [1] N. Tesla, "Turbine", Patent, USA 106120606.05.
- [2] R. Lisker, "Numerical investigation of a Tesla Turbine," Diploma Thesis, Technical University of Applied Sciences Wildau, Wildau, 2009.
- [3] R. Meller, "Fluid dynamic investigations of the flow inside of a Tesla turbine," Bachelor Thesis, Technical University of Applied Sciences Wildau, Wildau, 2013.
- [4] B. König, "Development of a friction turbine (Tesla turbine)," Diploma Thesis, Institut für Strömungslehrer und Wärmeübertragung, Johannes Kepler University of Linz, Linz, 2011.
- [5] A. Guha and S. Sengupta, "The fluid dynamics of the rotating flow in a Tesla disc turbine," *European Journal of Mechanics - B/Fluids*, vol. 37, pp. 112–123, 2013.
- [6] R. Lisker, "The Design of a Tesla Turbine based Organic Rankine Cycle for Waste Heat Recovery," Master Thesis, Technical University of Applied Sciences Wildau, Wildau, 2013.
- [7] W. Rice, "An Analytical and Experimental Investigation of Multiple-Disk Turbines," *J. Eng. Power*, vol. 87, no. 1, p. 29, 1965.
- [8] E. J. Oklejas and R. A. Oklejas, "Gas regeneration tesla-type turbine", USA, August 19, 1975.
- [9] R. A. Oklejas and E. Oklejas, "Tesla-type turbine with alternating spaces on the rotor of cooling air and combustion gases", USA US 05/584,361 December 28, 1976.
- [10] M. L. Prueitt, "Isothermal power system", USA US 12/245,224 October 22, 2009.
- [11] W. W. Cochran, "Disk turbine using heat pipes", USA US 14/180,213 September 18, 2014.
- [12] Matveev.Y.Y., "Calculation of laminar flow of a viscous fluid between rotating disks," *Mekhanika Zhidkosti i Gaza*, no. 1, pp. 63–67, 1982.
- [13] M. Piesche and U. Müller, "About the Heat Transfer in the Gap of a Friction Pump," *Acta Mechanica*,

vol. 35-35, no. 3-4, pp. 173–186, 1980.

- [14] F. Bošnjaković and K. F. Knoche, *Technical Thermodynamic Part II*, 6th ed. Heidelberg: Steinkopff, 1997.
- [15] H. Schlichting, *Boundary-Layer Theory*, 9th ed. Berlin, Heidelberg: Springer Berlin Heidelberg, 2016.
- [16] M. Jischa, *Convective Momentum-, Heat- and Species xchange*. Wiesbaden, s.l.: Vieweg+Teubner Verlag, 1982.
- [17] A. F. R. Ladino, “Numerical Simulation of the Flow Field in a Friction-Type Turbine (Tesla Turbine),” Diploma Thesis, Institute of Thermal Powerplants, Technical university of Vienna, Vienna, 2004.
- [18] M. Bolle and M. Schmidt, “The Operation of a Friction Turbine under Parameter Variation and the Influence of the Processes od Condensation, Evaporation and Superheating,” Bachelor Thesis, Technical University of Applied Sciences Wildau, Wildau, 2014.
- [19] C. Schulze Zurmussen, “Experimental Investigation of the Operation Characteristics of a Friction Turbine with superheated steam,” Bachelor Thesis, Technical University of Applied Sciences Wildau, Wildau, 2016.
- [20] R. Lisker, U. Hellwig, and F. X. Wildenauer, “Development of a 2.2 kW Tesla Turbine,” *Wissenschaftliche Beiträge der TH Wildau*, vol. 19, no. 1, pp. 67–72, 2015.
- [21] C. Stein, “Simulation of a liquid film in the gap of a friction turbine,” Bachelor Thesis, Technical University of Applied Sciences Wildau, Wildau, 2015.
- [22] B. Witt, “Optimisation of the position of an inlet nozzle of a Tesla Turbine,” Bachelor Thesis, Technical University of Applied Sciences Wildau, Wildau, 2015.
- [23] R. Meller, “Numerical Simulation of the gap of a Friction Turbine,” Master Thesis, Beuth Hochschule für Technik Berlin University of Applied Sciences, Berlin, 2016.
- [24] CD Adapco, *Star CCM+ User guide*. 2016