CFD ANALYSIS OF A TESLA TURBOEXPANDER USING SINGLE PHASE STEAM

Konstantinos C. Ntatsis¹, Anastasia N. Chatziangelidou¹, Theofilos G. Efstathiadis¹, Vasilis G. Gkoutzamanis¹, Paolo Silvestri² and Anestis I. Kalfas¹

¹Aristotle University of Thessaloniki
Laboratory of Fluid Mechanics and Turbomachinery, Department of Mechanical Engineering
ntatsisk@auth.gr, anchatzian@meng.auth.gr, theofil@auth.gr, vgkoutzam@meng.auth.gr, akalfas@auth.gr
GR-54124, Thessaloniki, Greece

²University of Genoa
Thermochemical Power Group, Department of Mechanical Engineering, Energy, Management and Transports
p.silvestri@unige.it
IT-16145, Genova, Italy

ABSTRACT
The present paper investigates the performance of a Tesla disk turbine used as a turboexpander for small waste heat recovery applications. The geometry of the stator is slightly involuted and the admission of the fluid takes place through a two-convergent-nozzle configuration with supersonic flow conditions close to the nozzle outlet. Three cases with varying disk tip clearance are simulated for the entire operating range. The preliminary results suggest that a small decrease in the tip clearance can lead to a considerable increase in the performance of the turbine up to 57%. Specific adverse flow characteristics are analyzed: i) The circumferential extend of the supersonic region inside the stator, ii) flow asymmetry in the axis perpendicular to the flow and iii) flow reversal inside the rotor due to the local over-expansion close to the nozzles. The current simplified investigation is directed towards the improvement of Tesla turbines performance operating in the two-phase regime and under local supersonic conditions.

INTRODUCTION
The Tesla turbine owes its name to its creator Nikola Tesla who first patented the turbine in 1913 (Tesla, 1913). The turbine rotor consists of multiple flat concentric disks that are fixed on a shaft and rotate together. The flow admission takes place through one or more nozzles and fluid enters the rotor almost tangentially. The fluid moves spirally through the tight gap between the disks transferring energy and forcing the disks to rotate before exiting through the outlet located at the center of the rotor. Other names for the Tesla turbine is "viscous turbine" or "boundary layer turbine", as the sole mechanism of energy transfer between the fluid and the rotor disks is the viscous force.

The main reason that the Tesla turbines are not commercially available for industrial applications is their low efficiency. Most of the efficiencies experimentally derived are in the region of 25% (Lemma, et al., 2008), (Rice, 1965). Despite, its low efficiency the Tesla turbine offers many advantages such as the simple design, the low cost of production and maintenance as well as the ability to operate with various working fluids or even two-phase flows. (Guha & Smiley, 2010). In addition, such turbines tend to become competitive compared to the conventional bladed turbines as the size decreases (Epstein, 2004).

Considering the above, a suitable application of the Tesla turbine is the waste heat recovery as often done using Organic Rankine Cycles. (Manfrida, et al., 2017) proposed an improved modular design for the power range of 500W to 5kW and developed an analytical model for two working fluids. (Song, et al., 2017) assessed the performance of a Tesla turbine in the kW range for various organic fluids, by using a 0D thermodynamic model. Small power generation applications may require the Tesla turbine to work with a two phase fluid or a particle laden flow (Neckel & Godinho, 2015). Such turbines are supposed to have the characteristic of “self-cleaning” by forcing heavier droplets or particles to their periphery (Sengupta & Guha, 2012).

A parameter that often influences shrouded turbomachinery performance is rotor tip clearance, which is the size of the gap between the rotor and the shroud or casing.
Its effect on the performance of a Tesla turbine has been investigated by (Sengupta & Guha, 2018). They varied the tip clearance in a simplified 8-nozzle configuration by changing the outer diameter of the casing while keeping the mass-flow and the rotational speed of the rotor fixed. The flow was steady, laminar and subsonic. The results showed that the efficiency has a maximum value as the tip clearance changes from finite to zero.

The current investigation varies the tip clearance by changing the radius of the rotor while keeping the radius of the casing fixed. The insights of such a study can lead to improved designs for Tesla turbine applications with imposed restrictions in the maximum allowed size of the turbine. In addition, it adds to the research of Tesla turbines with turbulent and locally supersonic flows, a requirement more and more common as the scales decrease and the required pressure drops increase.

In this paper, the flow field inside a simplified Tesla turbine is simulated for three disk radii in the entire operating range. The stator geometry has a slight involute shape. Considerations for the mesh development are presented and specific flow characteristics, especially regarding the supersonic admission, are discussed. However, this work shall be viewed as a preliminary study to a more elaborated two-phase investigation to follow.

**METHODOLOGY**

**Experimental Rig Design**

The fluid control volume can be seen on Figure 1 (left). The stator casing has a slightly involuted shape with the largest and smallest radius exactly after and before the nozzle outlet, respectively (Figure 1B). The nozzle and the outer region of the volute constitute the stator while the center circular part of the volute is the rotor. The nozzle and the outer dimension of the volute are fixed. The rotor radius is varied to allow the investigation of the effect of different tip clearances. Three rotor disk radii are studied: \( r_d = 98.5 \text{mm} \), \( r_d = 99 \text{mm} \) and \( r_d = 99.5 \text{mm} \) resulting in minimum tip clearance of 1.5mm, 1mm and 0.5mm, respectively (minimum since the casing has a spiral shape). These three distinct cases simulated are referred as rotor with “\( r_d = 98.5 \text{mm} \)”, “\( r_d = 99 \text{mm} \)” and “\( r_d = 99.5 \text{mm} \)” for the rest of the manuscript.

In Figure 1, the fluid enters through a circular inlet on the top of the domain with diameter \( D_{in} = 24 \text{mm} \). The fluid flow turns 90° inside the nozzle and exits the nozzle through a rectangular outlet with area \( A_{out} = 1 \text{mm}^2 \). The area ratio is \( A_{in}/A_{out} = 452 \) with a total to static pressure ratio \( p_{in}/p_{out} = 4 \), therefore a choked flow is expected. The rotor outlet is a circumferential surface at the center of the rotor with \( r_o = 17.5 \text{mm} \). The inter-disk spacing is 1mm. The summary of geometric and flow details can be seen in Table 1.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Casing radius</td>
<td>( r_c )</td>
<td>100 to 101 mm</td>
</tr>
<tr>
<td>Minimum tip clearance</td>
<td>( c )</td>
<td>1.5 / 1 / 0.5 mm</td>
</tr>
<tr>
<td>Disk radius</td>
<td>( r_d )</td>
<td>98.5 / 99 / 99.5 mm</td>
</tr>
<tr>
<td>Inter-disk gap</td>
<td>( b )</td>
<td>1 mm</td>
</tr>
<tr>
<td>Outlet radius</td>
<td>( r_o )</td>
<td>17.5 mm</td>
</tr>
<tr>
<td>Pressure ratio</td>
<td>( p_{in}/p_{out} )</td>
<td>4</td>
</tr>
<tr>
<td>Nozzle inlet area</td>
<td>( A_1 )</td>
<td>424 mm²</td>
</tr>
<tr>
<td>Nozzle outlet area</td>
<td>( A_2 )</td>
<td>1 mm²</td>
</tr>
<tr>
<td>Total inlet temperature</td>
<td>( T_{out} )</td>
<td>393 K</td>
</tr>
<tr>
<td>Mass flow per nozzle</td>
<td>( \dot{m} )</td>
<td>6.1 g/s</td>
</tr>
<tr>
<td>Number of nozzles</td>
<td>-</td>
<td>2</td>
</tr>
<tr>
<td>Revolution per minute</td>
<td>-</td>
<td>0 to 45k RPM</td>
</tr>
</tbody>
</table>

**Computational Grid**

Previous researchers that simulated flows inside a Tesla rotor (Guha & Sengupta, 2014) or inside a rotor combined with a simple nozzle (Lampart & Jedrzejewski., 2011), developed a structured grid consisting of hexahedral elements with a high aspect ratio, without compromising the accuracy of the simulations. However, the more complex nozzle inlet of the current geometry and the involuted stator geometry do not permit the generation of a structured mesh in this study and, thus, an unstructured mesh is used. Apart from the different mesh resolutions, the grid generation does not distinguish

![Figure 1 Schematic of the simulated geometry](image)
between the stator and the rotor and, thus, no mesh interface exists between the two, in contrast to meshes used in simulations of bladed turbomachinery. The distinction between the stationary and the rotating walls occurs by setting the appropriate rotational velocity for the wall boundary condition; zero for the stationary walls and finite for the rotating ones.

The tight inter-disk spacing necessary for an efficient Tesla turbine combined with the relatively larger rotor radii lead to computational domains with a "high-aspect ratio" (z direction much smaller than the radial and tangential directions). In addition, there is the requirement of fine resolution in direction perpendicular to the direction of the flow (z-axis) in order to capture accurately the velocity profiles, allowing the adequate prediction of the torque output. Thus, the mesh generation requires a high total cell count especially when unstructured meshes are concerned, as in the current study.

The computational grid is generated in the commercial pre-processing software BETA CAE ANSA v18.1.0. The surface mesh is unstructured and consists of a mix of mainly quadrilateral elements and fewer triangular elements only to allow gradual coarsening of the mesh. The mesh is refined close to nozzle outlet and progressively coarsened towards both the nozzle inlet and rotor outlet. Based on the surface mesh, prism layers are generated with 8 basic layers and 8 outer layers, as shown in Figure 2 at the bottom. The first element height was 0.001 mm, chosen to account for the target y⁺ required by the selected turbulence model. The growth ratio for each basic layer is 1.3. The growth rate of the outer layers is determined by specifying the ratio of length to thickness of the last outer layer, which is chosen equal to 0.6. Extra effort is put to improve mesh quality, in particular cell skewness and non-orthogonality since, in the current setup, those attributes influence solution convergence and resolve issues with reversed flow at the cell faces of the outlet. The total cell count is approximately 8 million.

**Figure 2** A section of the nozzle mesh (top) and of the rotor mesh (bottom) showing the prism layers

### Computational Setup

The Reynolds-Averaged Navier Stokes equations are solved using the finite volume discretization of the domain in the commercial software ANSYS FLUENT 18.0. The energy equation is enabled to allow the simulation of heat transfer inside the flow and the influence of gravity is neglected. The working fluid is steam as a single phase. The viscosity and thermal conductivity of the fluid are constant and their values equal to the reference state. The specific heat is linearly interpolated for different local temperatures as well as extrapolated at temperature and pressure values where phase change is naturally expected. The ideal gas equation of state is used to model the density changes of the fluid allowing the simulation of the compressibility effects.

The Shear Stress Transport k-ω (SST k-ω) turbulence model is used to model the turbulence of the flow and the maximum y⁺ is 2.1. The SST k-ω model is currently the industry standard for turbulence modeling. Although, it was initially developed for external aerodynamic flows, its validity has also been also demonstrated for internal flows (Besagni, et al., 2015). In addition, a curvature correction is included to the turbulence model to account for the high steering of the flow streamlines (Ansyl, 2013). The flow is steady and locally supersonic. The solver is pressure-based with SIMPLE algorithm for pressure-velocity coupling. The discretization of the equations is second order for the pressure, momentum, density and energy and first order for the quantities of turbulence.

Boundary conditions are total pressure 4 bar and total temperature 393K at inlet and static pressure 1 bar at outlet. It should be noted that, under such boundary conditions, the turbine is expected to operate in the two-phase regime. However, in the current preliminary study only the vapor phase of the steam is considered. The stator walls are assigned with the no-slip condition. The rotor walls are also assigned with the no-slip condition and a fixed rotational speed based on the desired RPM value. All walls are considered adiabatic. Due to the chocked nozzle and the fixed pressure inlet, the mass flow is the same for all cases and equal to \( \dot{m} = 6.1 \, g/s \) with small deviations in the second decimal point. However, as these deviations are in the order of magnitude of the continuity residuals, they are not considered.

The initialization used for all the operating points is the Full-Multigrid (FMG) initialization offered by the software (Ansyl, 2013). A number of coarser meshes are automatically created based on the user created mesh. The flow is initialized and solved first in the coarsest mesh until certain residual values are achieved and then interpolated as a starting flow field for the immediate next finer mesh. This process is repeated until the original mesh is initialized. In the current study the solution is considered converged when the following criteria are met:

- Continuity residual < 10⁻⁴
- Momentum residuals < 10⁻⁵
- Energy residual < 10⁻⁶
- Constant value of torque output on both disks
- Constant value of mass flow
**Simulation Matrix**

The flow is modelled for the entire RPM range of the turbine from 0 to 45k RPM with an increment of 5k RPM as well as for the three geometries, resulting in 30 cases in total. The specific rotational velocity is set on the disks and the output torque is monitored. As long as the torque is positive the flow rotates the disks and the results are considered physically acceptable. The maximum RPM values of the turbine are specified as the point where the torque on the turbine disks becomes zero. Each case is computed until convergence with approximately 7000 iterations to be sufficient (lower RPM operating points required fewer iterations). The current study involves computationally demanding parallel computations considering that each case required approximately i) 28 hours in 24 physical cores of Intel Xeon Gold 5118 with clock frequency 2.3 GHz base and 3.2 GHz turbo or ii) 63 hours in 32 physical cores of AMD Opteron(TM) 6274 with clock frequency 2.2 GHz base to 2.5 GHz turbo.

**Grid Independence**

As the mesh is considered already refined compared to the respective meshes in literature, the mesh sensitivity is assessed using a single finer mesh for the case of rotor with \( r_a = 99.5 \text{mm} \). The finer mesh is generated with the same technique as the original mesh. The differences are: i) The height of first elements is changed to \( 8 \times 10^{-4} \text{mm} \) ii) The rotor surface mesh is more refined to maintain the desired mesh quality. The total cell count of the finer mesh is approximately 15 million. A quantitative comparison between the two meshes is shown in Table 2 for various RPM. The torque values suggest that the coarse mesh can be regarded in agreement with the finer mesh up to 25k RPM with a difference less than 5%. As the rotational speed further increases, the results appear mesh dependent. An additional observation for rotational speeds over 25k RPM is that the solution of the finer mesh also manifests great oscillations of the torque values. This observation indicates that the flow is possibly dominated by transient phenomena at higher RPM and a transient solver should be considered.

**RESULTS**

**Comparing Initialization**

The speed of convergence in number of iterations is compared among three different initialization techniques: i) "FMG" initialization, as described in the previous section ii) "Lower RPM initialization", the converged flow field of the first lower RPM operating point is used as the initial solution eg. the current rotational speed is set to 25k RPM and the converged solution of the 20K RPM case is used as initial solution iii) "Higher RPM initialization" where the converged solution of the first higher RPM operation point is used for the current e.g. the 30k RPM data used for the 25k RPM case.

Torque evolution with the solution iterations for both the bottom and the top disk are shown in Figure 3. The region of interest is close to the solution convergence and, thus, only the torque values over 3000 iterations are contained. The plot shows that, for the current case, there is no considerable difference in the number of iterations necessary for a converged solution, as all three simulations require approximately 5000 iterations until convergence.

![Figure 3 Torque value convergence for various initializations - case \( r_a = 99\text{mm} \) at 25k RPM](image)

The "Lower RPM" and "Higher RPM" initializations seem to agree on the final torque values. A notable feature is the initial (before convergence) under-prediction and over-prediction of the torque for the "Lower RPM" and "Higher RPM" techniques, respectively, due to the different initial velocity fields assumed. Thus, the solution curves for "Lower RPM" initially ascend until they, eventually, stabilize while the "Higher RPM" curves descend.

"FMG" initialization can, potentially, be a valuable tool for Tesla turbine simulations as it does not require any prior solution data when modelling the flow at a particular operating point. In that direction, the current computational comparison shows that a solution initialized with the "FMG" technique does not suffer from lower speed of convergence compared to the other methods. However, it is apparent that "FMG" initialization results in an oscillatory behaviour with periodic low and high peaks of the torque on both disks. These results in addition to the observation that a uniform initial flow field leads to divergence, underline the high sensitivity of the solution to the initial conditions.

<table>
<thead>
<tr>
<th>RPM</th>
<th>10k</th>
<th>20k</th>
<th>25k</th>
<th>30k</th>
<th>40k</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse Mesh</td>
<td>15.410</td>
<td>12.770</td>
<td>11.133</td>
<td>9.398</td>
<td>1.685</td>
</tr>
<tr>
<td>Fine Mesh</td>
<td>15.432</td>
<td>12.431</td>
<td>12.549</td>
<td>10.418</td>
<td>2.397</td>
</tr>
<tr>
<td>Difference</td>
<td>-0.1%</td>
<td>2.6%</td>
<td>-12.7%</td>
<td>-10.8%</td>
<td>-42.2%</td>
</tr>
</tbody>
</table>

Table 2 Mesh independence study based on output torque \([10^{-3} \text{Nm}]\)
Effect of the Tip Clearance on the Turbine Performance

The results of varying the disk tip clearance are discussed in the current subsection. Figure 4 shows the output torque for the three tip clearances at all rotational speed values. Laminar flow theory predicts a linear drop in torque for a specific turbine that the linear trend is valid up to 0.7 bar pressure drop (Lemma et al., 2008). Previous work on turbulent flows inside Tesla turbines has shown a non-linear relationship with RPM (Qi et al., 2018). However, the onset of torque non-linearity with RPM in the current study coincides with the mesh independence of the solution, as explained in the “Grid Independence” section. It is speculated that the issue can be resolved by employing a transient solver for the rotational speeds over 25k RPM.

![Figure 4 Effect of the tip clearance on the output torque](image)

The results suggest that as the rotor radius increases or, respectively, the tip clearance decreases, the torque output increases dramatically. These findings are in accordance with the results of (Guha & Sengupta, 2014), where it was analyzed that the torque output is proportional to the rotor radius. Hence, a small increase in the radius between the cases can have a major effect on the torque generated. In addition, due to the differences in torque output, the calculated limit for the rotational speed is different for each case and increases as the rotor radius increases.

The power curves, calculated by the relation

\[ \dot{W} = T \cdot \omega. \]  

where \( T \) is the sum of the torques in both disks [Nm] and \( \omega \) is the rotational speed of the disks [rad/sec], are shown in Figure 5. The curves exhibit an inverted bucket shape which is typical for Tesla turbines (Sengupta & Guha, 2012). As summarized in Table 3, the maximum power is 46.74W for the rotor radius \( r_d = 98.5 \) mm case at 25k RPM, 58.74W for \( r_d = 99 \) mm at 30k RPM and 73.40W for \( r_d = 99.5 \) mm at 30k RPM. Decreasing tip clearance by increasing rotor radius from 98.5mm to 99.5mm can increase the calculated power output by 57%. As it has already been discussed, the results are considered only qualitative of the expected turbine behaviour, since the mesh independence has not been achieved for the entire operating range. In addition, the assumption of a single phase fluid when a two-phase flow is expected, render the current work a preliminary study.

![Figure 5 Effect of the tip clearance on the power and efficiency](image)

The isentropic efficiency is defined as the ratio of the power delivered by the shaft (\( \dot{W} \)) to the power of the respective isentropic process (\( \dot{W}_s \)):

\[ \eta = \frac{\dot{W}}{\dot{W}_s} = \frac{T \omega}{m c_p T_o(1 - (p_3/p_{o,1})^{(\gamma-1)/\gamma})}. \]  

where \( m \) is the mass flow rate of both nozzles, \( c_p \) is the specific heat under constant pressure, \( T_o \) is the total temperature at the inlet, \( p_3 \) is the static pressure at the outlet, \( p_{o,1} \) is the total pressure at the inlet, and \( \gamma \) is the specific heat ratio. The efficiency is also plotted in Figure 5. The denominator is the same for all the cases since the mass flow rate does not change between the cases (chocked nozzles) and the pressures and temperature involved in the calculation are specified as boundary conditions. Thus, the efficiency has the exact same shape as the power but with different magnitude. The maximum value is 21.32% for the rotor with \( r_d = 99.5 \) mm at 30k RPM rotational speed. However, the maximum value lies inside the mesh dependent region.

Table 3 Summary of the maximum power output

<table>
<thead>
<tr>
<th>Case, ( r_d )</th>
<th>98.5mm</th>
<th>99mm</th>
<th>99.5mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max. Power</td>
<td>46.74W</td>
<td>58.74W</td>
<td>74.40W</td>
</tr>
<tr>
<td>Relative Increase</td>
<td>-</td>
<td>26.67%</td>
<td>57.04%</td>
</tr>
</tbody>
</table>

Extend of the Supersonic Region

The azimuthal extend of the supersonic region is calculated using the angle difference between the start and the end of the region of the flow where the Mach number is greater than one. The results are plotted against the rotational speed for each of the three simulated cases in Figure 6. For reference, this supersonic region covers approximately 35° of the circumference for the 0 RPM and it is observed that it gets shorter as the RPM increases. However, for all the cases this region of high momentum fluid is inside the stator indicating...
considerable losses due to the unexploited high relative velocities. When increasing the rotor radius and the rotor disks start to increasingly reside in the aforementioned region, it allows the turbine to extract more energy from the fluid. Hence, this results in an increase in the torque and power output, as discussed in “Effect of the Tip Clearance on the Turbine Performance” subsection.

To further support the statement, the maximum Mach number has been plotted for all cases over the entire RPM range. As seen in Figure 7, the maximum Mach number drops with increasing RPM. In addition, the case with the smallest tip clearance corresponds to the lower overall Mach numbers (blue line). The effect of a tighter tip clearance is to reduce maximum available Mach number of the flow indicating a greater use of the available momentum of the flow.

Figure 8 Asymmetric behavior in radial velocity contour close to the nozzle outlet, case \( r_d = 99\text{mm} \) at 25k RPM

However, the only geometrical source of asymmetry in the z-direction is the turbine inlet. In order to assess its influence, the velocity profiles along the z-axis are plotted exactly at the nozzle outlet and appear with yellow color in Figure 8. The line furthest away from the rotor outlet is Line 1 and the line closest to the rotor outlet is Line 9. The velocities are normalized by dividing with the maximum velocity of Line 9. However, the velocity profiles plotted over these lines do not show any particular strong asymmetry (Figure 9). Thus, it is speculated that small velocity deviations along the z-axis grow in magnitude as the flow starts rotating inside the rotor resulting in this asymmetric behavior.

Figure 9 Velocity profiles in the z-direction exactly at the nozzle throat

Overexpansion & Reversed Flow

Figure 10 shows the radial velocity contour at the midplane between the two disks. Radial velocity is expected to be negative inside the rotor since the fluid moves ideally only towards the outlet losing energy in the process. In the current simulations, a region of positive \( U_r \) appears for all the RPM close to the nozzle outlet (Figure 10A). As the RPM increase this region occupies a greater part of the flow but with reduced...
magnitude (Figure 10B). Eventually, the region is separated into a smaller one close to the nozzle outlet and a larger one covering a great part of the plane (Figure 10C).

This behaviour can be attributed to the fact that the flow exactly downstream the nozzle outlet is highly overexpanded with negative gauge pressures (Figure 11). This region of low pressure has an adverse effect on the performance of the turbine as it creates a part of the flow inside the rotor with the tendency to move away from the outlet (positive radial velocity).

**CONCLUSIONS**

The current CFD investigation assesses the influence of the rotor tip clearance in a Tesla turbine with two convergent nozzles, an involuted stator and a simplified outlet. Certain flow characteristics are discussed such as the extend of the supersonic region and its impact on the performance, the asymmetric velocity field in the axis perpendicular to the flow and the evolution of a region with positive radial velocities across the RPM range. The current study is preliminary since it simulates an expected two-phase flow with single phase steam as well as due to the mesh dependency for higher rotational speeds (over 25k RPM). Despite these limitations several initial observations can be made. In particular:

1. The demanding parallel computations are performed to calculate the turbine performance for three tip clearances for the entire operating RPM range. The trend is clear with an increase in torque output as the rotor radius increases. For rotational speeds over 25k RPM the mesh dependency is accompanied with greater torque oscillations in the case of fine mesh. This observation suggests that a transient solver shall be considered for the higher rotational speeds.

2. The area increase at the outlet of the convergent nozzle makes the flow locally supersonic, with Mach numbers up to 2 depending on the RPM. The circumferential extend of this supersonic region has been calculated, which ranges from 35° for the 0 RPM down to 3° for the 35k RPM. Considering that this region of high momentum fluid is rubbed against the stator generating losses, an enhancement of the turbine performance could be achieved by minimizing the stator area. As the radius of the rotor increases, the rotor has greater access to this high momentum region. The result is an increased rotor performance and a reduced maximum Mach number of the flow.

3. The existence of the asymmetric behavior of the velocity distribution along the axial direction (z-axis) is observed. The radial velocity manifests positive and negative values of the same order of magnitude along the same axis, at the azimuthal region behind the nozzle outlet. As the only source of geometrical asymmetry, the influence of the nozzle inlet to this specific characteristic is assessed. However, the velocity profiles at the nozzle outlet show no strong asymmetry in the z-axis and it is speculated that smaller velocity disturbances are propagated and amplified as the flow rotates inside the rotor.

4. The sudden acceleration of the fluid at the nozzle outlet results in local negative gauge pressures. Due to the pressure difference, positive radial velocities are observed inside the rotor and close to the nozzle outlet. For low RPM this region is spatially confined and has high velocity magnitudes but with increasing RPM it expands covering a large part of the rotor. At 35k RPM this region is separated into two parts: a small one close to the nozzle outlet and a larger one covering almost half of the circumference of the rotor.

5. The rotational speed for maximum power and efficiency is 30k RPM. The case with the highest rotor radius (smallest tip clearance) delivers the maximum power of 73.40W and maximum isentropic efficiency of 21.32%. Decreasing the disk tip clearance by only 1mm leads to a 57% increase in the performance of the turbine.

A more general conclusion of this study is that alternative nozzle geometries such as convergent-divergent should be investigated for Tesla turbines operating under flow conditions with expected supersonic conditions.
NOMENCLATURE

- $r_c$: Rotor disk radius [mm]
- $r_d$: Casing radius [mm]
- $r_o$: Rotor outlet radius [mm]
- $b$: Inter-disk spacing [mm]
- $y^*$: Dimensionless wall distance [-]
- $m$: Mass flow rate [g/s]
- $T_{o,1}$: Total inlet temperature [K]
- $p_{o,1}$: Total inlet pressure [bar]
- $p_3$: Static outlet pressure [bar]
- $W$: Power on shaft [W]
- $W_s$: Isentropic power [W]
- $T$: Torque on disks [Nm]
- $\omega$: Rotational speed [rad/sec]
- $\eta$: Isentropic efficiency [-]
- $\gamma$: Specific heat ratio
- $c_p$: Specific heat at constant pressure [J/kg K]
- $U_r$: Radial velocity [m/s]

ACKNOWLEDGEMENTS

This project has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No 764706, PUMP-HEAT.

Additionally, the authors would like to thank BETA CAE SYSTEMS S.A for their invaluable overall support.

REFERENCES