Numerical Simulation of the Flow Field in a Friction-Type Turbine (Tesla Turbine)

Andrés Felipe Rey Ladino

Institute of Thermal Powerplants
Vienna University of Technology
Getreidemarkt 9/313, A-1060 Wien

School of Engineering,
National University of Colombia,
Calle 57c No. 40-51 Ap. 419, Bogotá, Colombia,

Tel.: ++57/1/3150057, Email: e0326542@student.tuwien.ac.at

INTRODUCTION

In the context of research project at Institute for Powerplants at the Vienna University of Technology in joint study with the National University of Colombia, the flow inside the Tesla turbomachine is investigated. The Tesla turbine, an unconventional turbomachinery that uses smooth disks instead of blades, is described principally by the loading coefficient curve, the efficiency and degree of reaction vs. the flow rate parameter. In order to describe its behaviour, the rotational speed is maintained constant and the flow rate is changed with the purpose of simulate a virtual brake. Since the flow is characterized in a transitional regime, both laminar and turbulent cases are simulated. The work presented represents an initial significant step towards the analysis of this type of flow using CFD tool. Starting from a simple axi-symmetric model of the flow between two co-rotating disks in two dimensions, the model is improved including the outlet of the turbine with the casing, and at the end a 3D simulation of a single disk is performed including the effects of the nozzles. A complete model of a Tesla turbine is restricted by computer resources. The simulations were carried out in winter and summer semester 2003-2004. The commercial computational fluid dynamics (CFD) code FLUENT as well as the grid generation software GAMBIT were used for the investigation. Both codes are available on the CFD-server COMPAQ SC45 (Fluid Dynamics and Finite Element Server) of the Computer Center of Vienna University of Technology.

PREVIOUS WORK

Starting from the patent of Nikola Tesla [1], (Figure 1), extensive analytical work were made in the 60´s, and 70´s. Most of literature consider the flow laminar and incompressible, but also some turbulent approach were made. The flow is characterized by the Reynolds number and the Mach number. It is found various regimes of flow and process as laminar, turbulent, forward transition and relaminarization from turbulent to laminar.

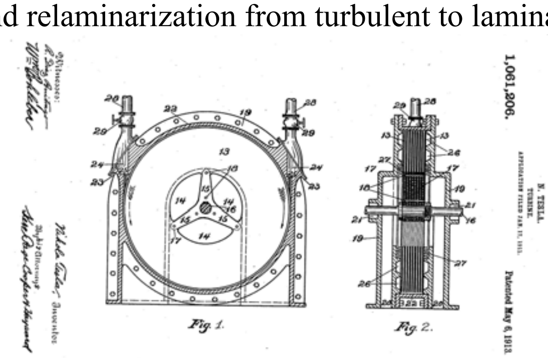


Figure 1.: American Patent No. 1,061,206 of Tesla turbine [1].

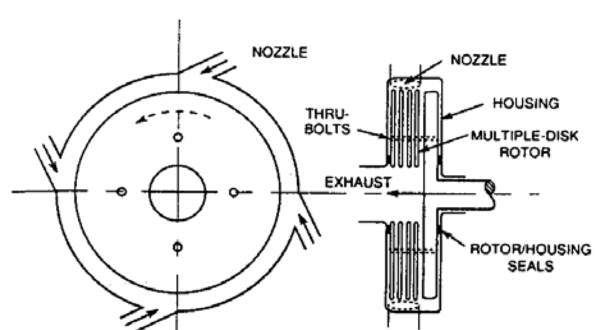


Figure 2.: Schematic diagram of Tesla turbine [2].

GEOMETRY AND MESHING

Following the experimental work and geometry used by Rice [2], three models with similar geometry are proposed and solved. The geometry is characterized by the ratio of radii: r_1/r_2=6,06 and the ratio of the gap to the outer radius: r_1/b=200 (these geometrical parameters are the same in the three models); the angle of the nozzle is β=20°. The flow is simulated for $\mathrm{Re}_b = \omega \cdot b^2 / \nu = 25.9$.

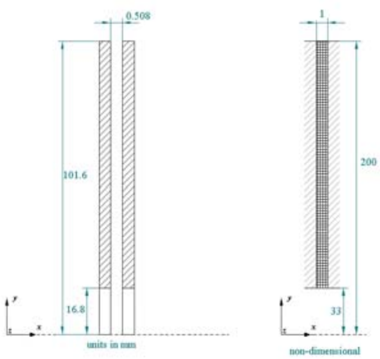


Figure 3.: Geometry for one gap 2D model.

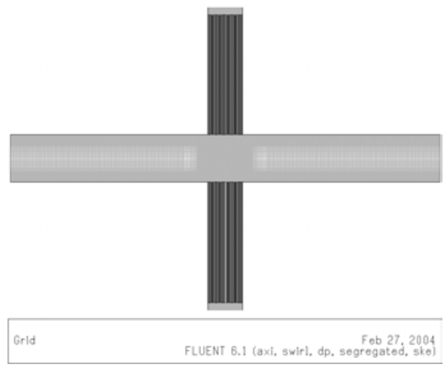


Figure 4.: 2D Model of the turbine with full peripheral admission.

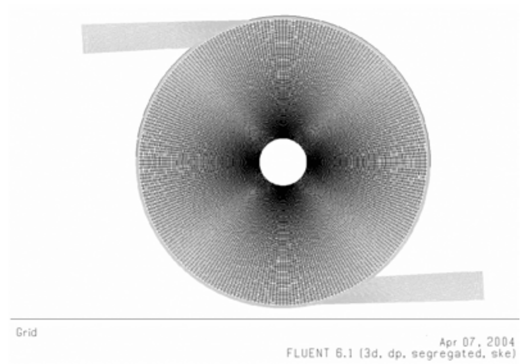


Figure 5.: Mesh for the 3D slide model.

DESCRIPTION OF THE NUMERICAL METHOD

Governing equations.

The Navier Stokes equations are solved for steady flow, incompressible, laminar and turbulent case, and 2D (Axisymmetric Swirl) and 3D approach. Fluent uses the Finite Volume Method and in addition the double precision characteristic was used. The following options were selected: Segregated and implicit solver, and for the discretization: PRESTO! (Pressure Staggering Option), SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) for Pressure-Velocity coupling, Second Order Upwind was selected for Momentum. For turbulent solution the standard k-ε model with near wall treatment is used. The y^+<5 relation near the wall must be accomplished.

Boundary Conditions.

Inlet was specified with a fixed velocity-inlet boundary. For the turbulence simulations was selected a value of 5% of turbulence intensity. The nondimensional turbulent kinetic energy is set to k*=0.00375 [-], and the turbulent dissipation rate is ε*=2,6953x10^{-4} [-].

Outlet condition was specified with a pressure-outlet boundary, with a gauge pressure value of 0 [-].

Disks and housing employ no slip condition rotating at 0.005 [-]. The walls are adiabatic.

Convergence and computed time.

The solution of the first model (Figure 3) take 4 minutes to converge with 6.800 quadrilateral cells, the second model (Figure 4) with 237,620 cells took 3h to reach a suitable solution and finally the 3D model (Figure 5) with 2,412,000 cells consume 14 days after 5,000 iterations.

RESULTS

First Model: Flow between disks. Figure 6 shows the velocity profile at three radial stations, inlet, middle and outlet. The strong acceleration between to frontier stations is noticed. Figure 7 presents one characteristic of the radial flow that appears only in the laminar case: the inflection of the profile in the middle of the radius. Figure 8 depicts the change of pressure (total, dynamic and static). As it can be seen, in the radial direction, the total pressure in the middle of the gap does not show variation, meaning that the flow in this plane do not perform work, but in other plane, at 5% of the gap from the wall, the variation is more appreciable.

Laminar solution

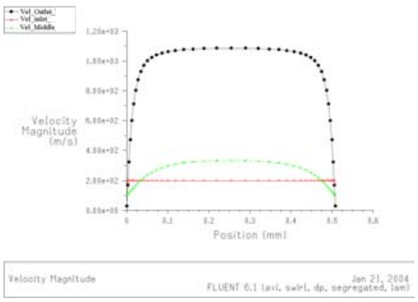


Figure 6.: Velocity magnitude, laminar solution.

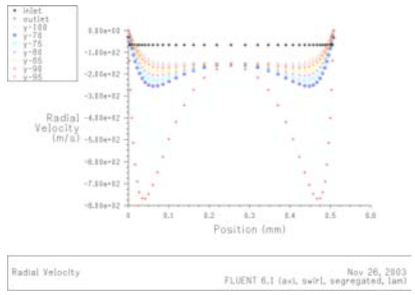


Figure 7.:Inflection of radial velocity for different radial stations.

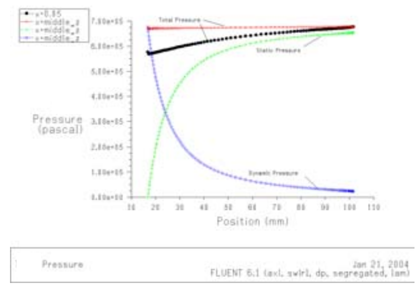


Figure 8.: Pressure distribution along radius, laminar solution.

Characteristic curves of the Tesla turbine are presented in Figure 9 and Figure 10.

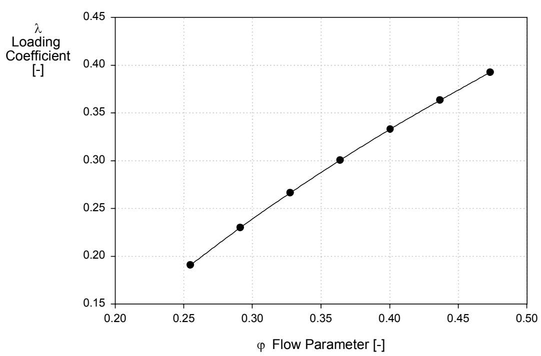


Figure 9.: Characteristic curve, loading coefficient, from laminar case.

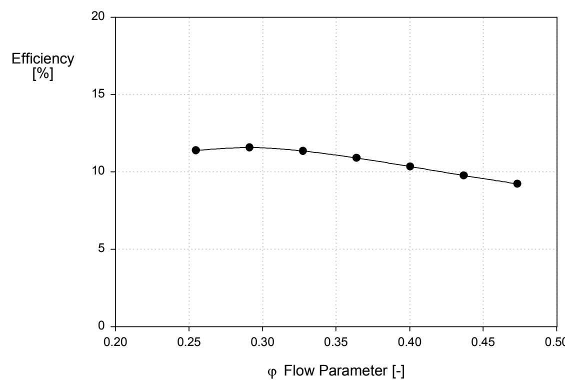


Figure 10.: Efficiency of an isolated disk, laminar, non-dimensional solution.

Turbulent solution

From turbulent case, the acceleration of the flow between inlet and outlet is lower (Figure 11) with a higher torque (Figure 14) and also higher efficiency; More energy is extracted with higher turbulence, the drag is higher. The total pressure at inlet is higher in comparison to the laminar case.

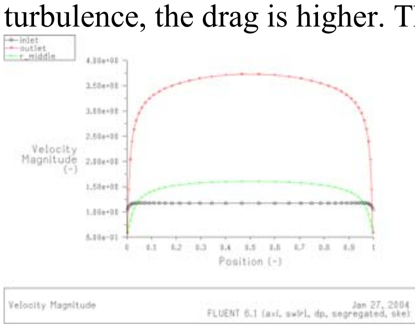


Figure 11.: Total velocity, turbulent case.

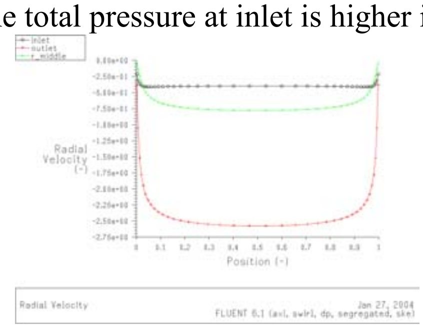


Figure 12.:Inflection of radial velocity for different radial stations.

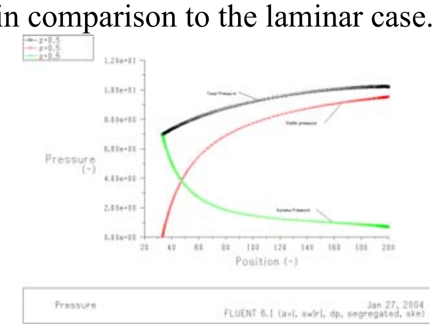
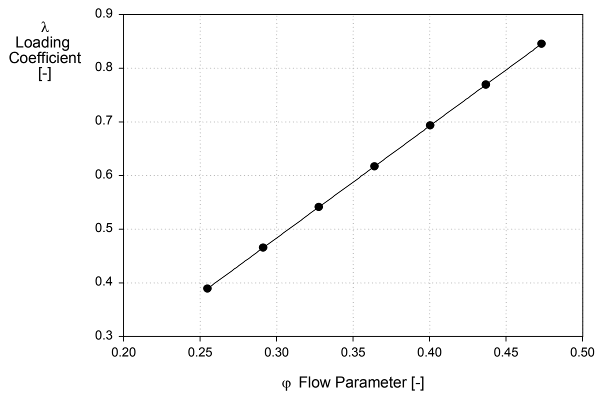


Figure 13.: Pressure distribution along radius, turbulent solution.



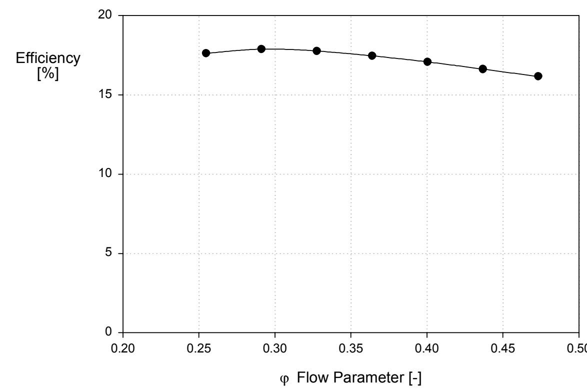
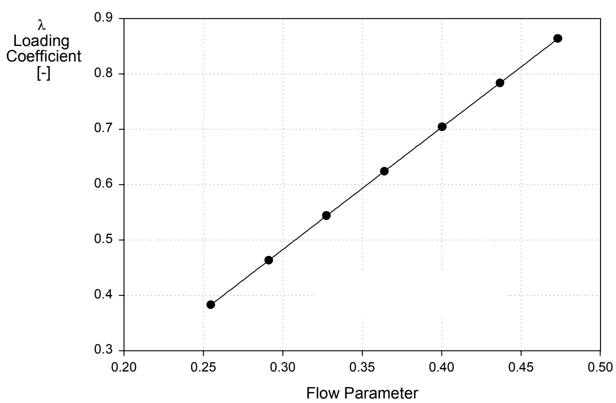


Figure 14.: Loading coefficient for turbulent solution, isolated disk.

Figure 15.: Efficiency for turbulent solution, isolated disk.

Rotor: This model includes the outlet of the turbine, Figure 17 presents the difference of computed efficiency of only the rotor and the rotor with the outlet, with a pipe length of 57 [-] from the rotor outlet. The losses due to the outlet are considerable due to change of area and direction.
The low efficiency value is also comparable to the recently experimental work of Schmidt [3].



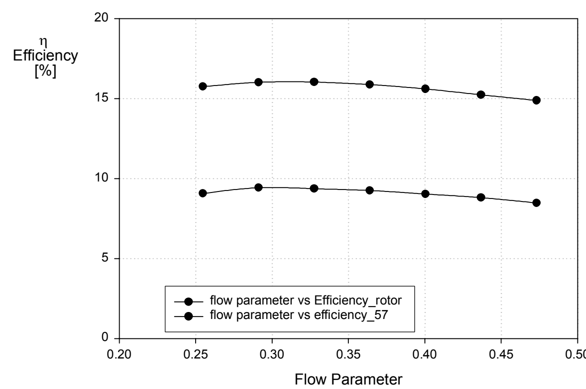
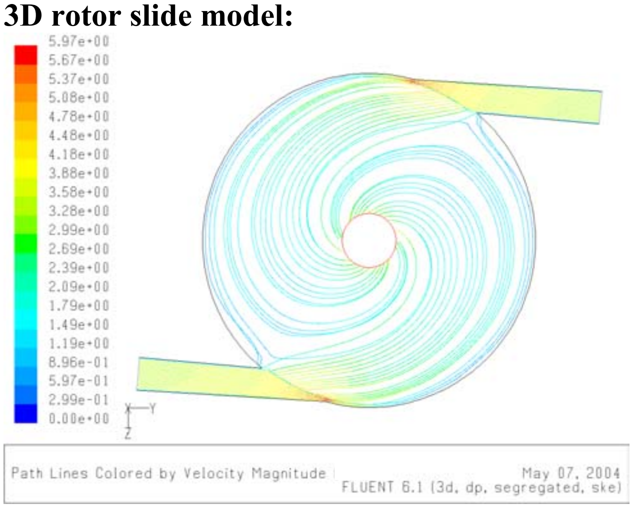


Figure 16.: Loading coefficient for turbulent solution, turbine.

Figure 17.: Efficiency for turbulent solution, turbine.

3D rotor slide model:



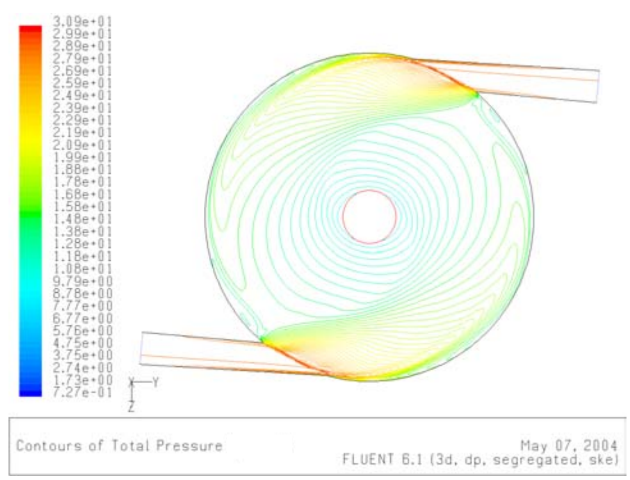


Figure 18.: Pathlines colored by velocity magnitude.

Figure 19.: Contours of total pressure at the middle of the gap.

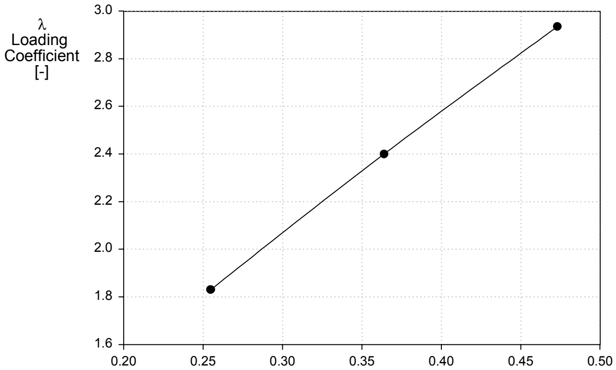


Figure 20.: Loading coefficient for turbulent solution, turbine.

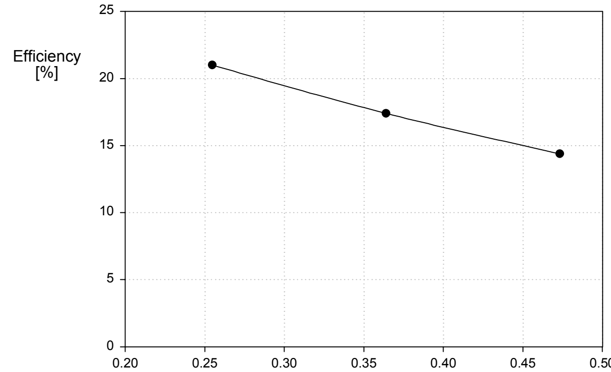


Figure 21.: Efficiency for turbulent solution, turbine.

The 3D model includes the effects due to the nozzle and the losses present in this zone are not very high when loading coefficient and efficiency are compared (Figure 20 and Figure 21).

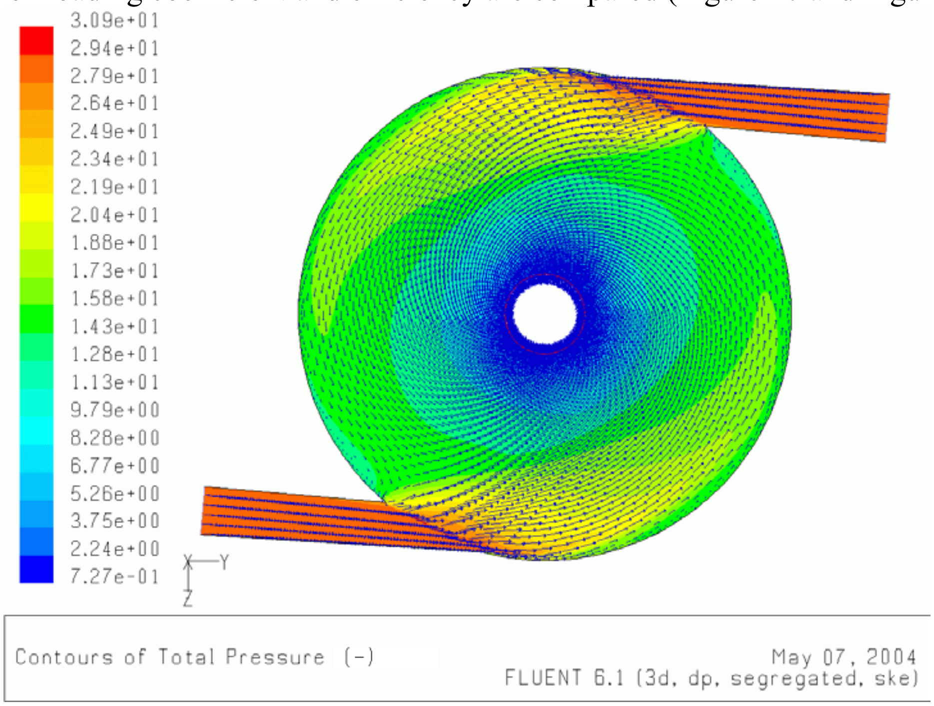


Figure 22.: Vectors of velocity magnitude and total pressure contours at the middle of the gap.

SUMMARY

The flow in the Tesla turbine was simulated with different geometrical models and laminar and turbulent approach. Since the flow itself is found in a transition regime with multiple processes - as transition, relaminarization, recirculation, between other phenomena- an exact simulation that full fills all the physical requirements is quite difficult to achieve with CFD tool. Nevertheless, both approximations are valid and they would show different features, characteristics and behaviour typical for each case and for the Tesla turbine. One of these differences is that the inflection point for laminar flow disappears in the turbulent case, for which the turbulent effects act as a mechanism of balanced in the axial direction. In contrast, experimental research cannot measured velocities profiles, only static pressure as it was reported by Adams [5], due to the fact that the gap between disks is very thin; CFD provide solution to this problem and furthermore to micro turbines. The flow hat also high swirling velocity components with significant gradients of acceleration, which makes the solution more difficulty and need extra iterations in order to achieve a good convergence solution. Convergence is an important issue in CFD because of the iterative nature of the solution, and it can give evidence of a well posed model indicating some physical facts. Extended details and results can be found in the diploma thesis of Rey A.F. [6].

The results show that multiple disk turbines are workable and feasible, in the engineering sense, but they present a low efficiency in both laminar and turbulent cases; lower for laminar case; the difference between these cases is significant. Moreover, the simulation of the gap and the results of 3D slide model show that the efficiency was not as high as it was suggest by experimental reports in literature that implying the low efficiency due to the presence of the nozzle. The efficiencies are similar in the three models. The machine is feasible for a low range of power where the efficiency of conventional turbomachinery is not very high and it can handle working fluid with particles, contaminants and high viscosity.

Some conclusions of each model are:

2D two disks model: For the laminar solution, the profile of the radial velocity component shows an inflection at about the middle radius; The flow presents strong acceleration especially at the outlet; The gap can be reduced.
2D turbine model: The effects of the walls on the overall performance are not much significant when the efficiency is compared with 2D two disks model; The losses due to the outlet are high, where the flow presents high swirls and change of area and direction; The leading edge and trailing edge of each disk can be improved.
3D turbine model: The high velocity of the flow increases the torque even though with the presence of zones of ventilation; The losses owed to the nozzles are not very high; The tangential component of velocity remains nearly constant from the outer to the inner region of the rotor, because of the high velocity after the nozzle.

Future Work

Some of the following topics would be interesting for further investigation: 1. Compressible analysis of the Tesla turbine; 2. Effects of the number of nozzles on performance of the turbine; 3.Optimization of the geometry, following analytical and the experimental work of Rice [2], and recently by Schmidt [3], the analytical work of Lawn and Rice [4] is useful for optimization; 5.Improvement of the numerical method in order to handle transitional flows as well as evaluation of other turbulence models to use in the simulations.

LITERATURE

1. Tesla, N.: “Turbine” United States Patent No. 1061206, May 6, 1913.
2. Rice, W.: “An Analytical and Experimental Investigation of Multiple-Disk Turbines”, Journal of Engineering for Power, Trans. ASME, series A, Vol. 87, No. 1 Jan. 1965, pp. 29-36.
3. Schmidt, D. D.: “Biomass Boundary Layer Turbine Power System”, California Energy Commission (CEC), EISG PROGRAM [online] Available from Internet: <URL: http://eisg.sdsu.edu/Fullsums/00-06.htm > <URL: http://eisg.sdsu.edu/shortsums/shortsum0006.htm>, 1991, California, USA.
4. Lawn, Jr. J., Rice, W.: “Calculated Design Data for the Multiple Disk Turbine using Incompressible fluid”, Journal of Fluids Engineering, Trans. ASME, Vol. 96 No. 3, September 1974, pp. 252-258.
5. Adams, R., Rice, W.: “Experimental Investigation of the Flow Between Corotating Disks”, Journal of Applied Mechanics, Vol. 37, Trans. ASME, Vol. 92 series E, No. 3, September 1970, pp. 844-849.
6. Rey A. F.: “Numerical Simulation of the Flow Field in a Friction-Type Turbine (Tesla Turbine)” Diploma thesis at the Institute of Thermal Powerplants, TUWien, Viena, Austria, July 2004.