Influence of turbulence model on prediction of Tesla turbine performance

Krzysztof Rusin[1] and Włodzimierz Wróblewski[2]*

[1] *Faculty of Energy and Environmental Engineering, Silesian University of Technology*
Konarskiego 18, 44-100 Gliwice, Poland
e-mail:krzysztof.rusin@polsl.pl

[2] *Faculty of Energy and Environmental Engineering, Silesian University of Technology*
Konarskiego 18, 44-100 Gliwice, Poland
e-mail: wlodzimierz.wroblewski@polsl.pl

Abstract

The article presents numerical analysis of Tesla turbine with the aid of Ansys products. Geometrical model was created using dimensions of existing turbine. In order to obtain results unaffected by discretization method mesh independence study was made. Steady state and transient numerical simulation was carried out for different types of turbulence model. Obtained results were compared to experimental data.

Keywords: Tesla turbine, boundary layer model, turbulence models, CFD

1. Introduction

Tesla turbine, also known as bladeless, was constructed for the first time in 1906 by Nikola Tesla, and patented in 1913. The most important part, which distinguishes Tesla turbine, is rotor. It consists of multiple thin discs mounted on a common shaft. The spacers are placed between the discs. Their aim is to hold constant distance during exploitation. Working fluid is delivered from one or multiple inlet nozzles tangentially to the surface of disc.

Principle of operation of the turbine is based on the adhesion and viscosity. Working fluid adheres to the disc surface, because the adhesion forces between fluid particles and disc particles are greater than cohesive forces of fluid particles. Energy transfer from fluid to disc is carried out by the means of momentum diffusion.

Important part of Tesla turbine is also supply system, which directs fluid jet in the area between the discs. Despite relatively simple construction, flow phenomena are quite complex [3]. They consist of fluid expansion both in the supply nozzles and in inner space of the turbine and interaction between fluid jet and discs rotating with high speed. Proper model of these phenomena allows determination of basic operation parameters and subsequently assessment of possibility of turbine construction improvement [4]. In this paper sensitivity of flow parameters and forces acting on discs to different types of turbulence model and inflation layer discretization is investigated. Two equation models of turbulence are analysed: k–ω SST (Shear Stress Transport), RNG (Renormalization Group) k–ε and SST–SAS (The Scale – Adaptive Simulation). A model k–ω SST combines advantages of k–ε and k–ω models. Phenomena occurring near the wall are modelled with use of k–ω, which smoothly switches into k – ε with increasing distance from the wall. This model is recommended for accurate boundary layer simulations [1]. Renormalization group methods rely on removing the smallest turbulence scales, so the remaining are resolvable with available computer capacities [5]. In the standard k–ε model turbulence kinetic energy is excessively produced near the stagnation point. Model RNG k–ε has modified epsilon equation which takes into account different scales of motion through changes to the production term. This modification leads to enhancement of turbulence kinetic energy dissipation and thereby solution improvement [2]. Model SST – SAS belongs to URANS (Unsteady Reynolds Averaged Navier – Stokes) class models. In this model von Karman length scale is introduced into the turbulence scale equation. It allows adjustment of the turbulence length scale to the local flow inhomogeneities. As a result SAS model behaves LES-like (Large Eddy Simulation) in unsteady regions and RANS-like in other areas [1].

Selected turbulence models are commonly used in solving engineering problems due to their relative good accuracy and low demand of computing capacities.

2. Numerical model

Numerical simulations were carried out with the aid of Ansys products: DesignModeler, Meshing, CFX. Geometry model is based on outside dimensions of existing turbine: diameter of 105 mm and axial length of 76 mm. Turbine rotor consists of 5 discs with a diameter of 73 mm and the distance between them 1,5 mm. Numerical model includes only half of the turbine, i.e. area of two discs, one wall of third disc, supply and outlet systems as well. Geometrical model with the markings is shown in fig. 1. Supply system consists of tubular collector (number 1 in fig.1), from which the medium is delivered in between rotor discs by means of two holes in collector wall (2). Area number 3 models cavity between supply system and edges of discs. Tip clearance between disc edges and turbine casing is also modelled (4). View A is presented without this space in order to clearly show the spaces between the discs. Gap between the discs, in which outflow from nozzles is carried out is marked as number 7. Between the disc spacers are placed (one of them marked as 6), which in this particular case are responsible not only for holding constant distance, but also due to their characteristic shape help to organize outflow. Outlet system (5) consists of outflow duct to ambient air and two choking chambers connected with each other by 4 cylindrical linkers.

*This research was supported in part by PLGrid Infrastructure and Statutory Research of Silesian University of Technology.

Domain of gap between the discs is stationary, but with rotating walls. Spacers domains are rotating and connected with stationary domains of gap between the discs with frozen rotor interface. The outflow is carried out into the ambient air.

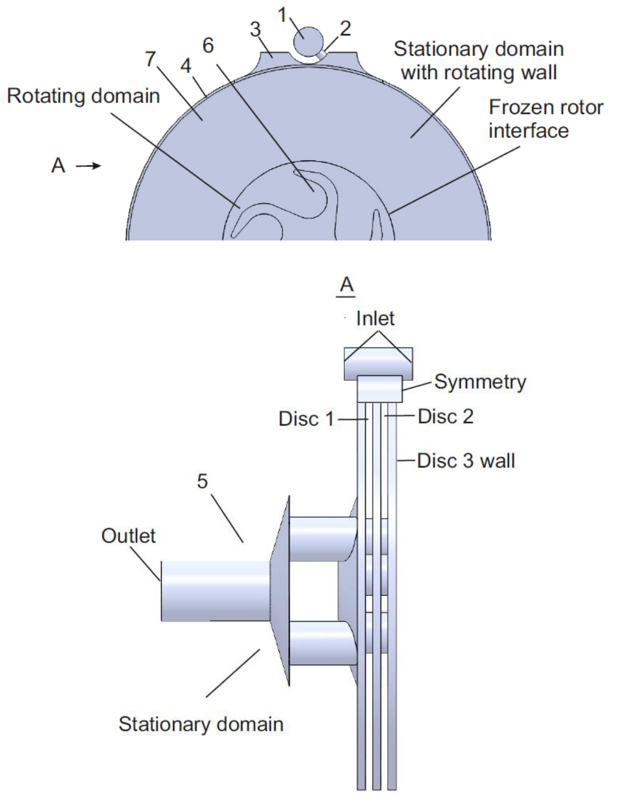

Figure 1. Calculation domain and most important boundary conditions

In order to obtain results unaffected by discretization method, mesh independence study was done. Due to necessity of precise modelling of inflation layer parameters, simulations for different y+ parameter were carried out. Fig. 2 presents discretization method in part of the turbine. The mesh is finer in area of the largest fluid gradients. Apart from inflation layers, the largest gradients occur in jet expansion area. Supply system domain was also finely meshed in order to get appropriate resolution of flow contraction phenomenon.

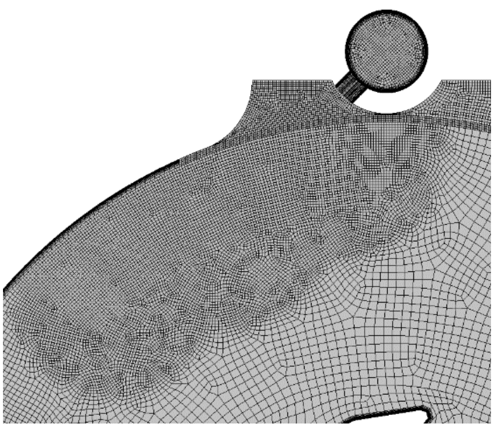

Figure 2. Discretization of the supply region

3. Results

Numerical analysis was carried out for air treated as ideal gas and inlet parameters: total pressure 3 bar and total temperature 303K. Rotating velocity of disc walls was equal to 25000 rev/min. Ambient pressure was set at the outlet. These basic parameters were based on results obtained from experimental data. Flow structure in Tesla turbine was analysed with the aid of different turbulence models. For SST model mesh providing independent result consisted of 4 263 000 nodes and y+ parameter in gap between the discs was below 1. Fig. 3 presents Mach number distribution. Maximum Mach number M = 1,33 occurs slightly after outlet from cylindrical conduit. At the inflow to the gap between the discs overexpansion and overcompression take place.

Comparative characteristics of power versus rotational velocity of each turbulence model and experimental data were also made. The differences between values of isentropic power obtained from models were within approximately 5%. In addition power obtained from numerical calculations was compared to available experimental data. The differences were within range 25%-30%.

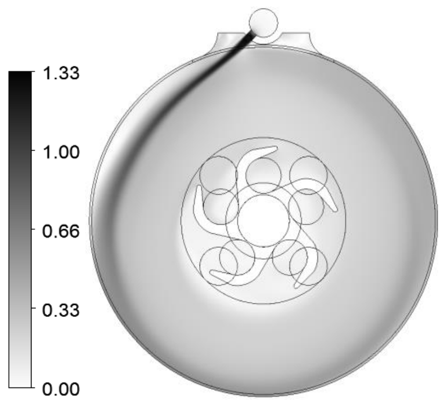

Figure 3. Mach number distribution

References

[1] Ansys CFX 17 Technical Documentation.

[2] Lee, S., Unsteady aerodynamic force prediction on a square cylinder using k–ε turbulence models, *Journal of Wind Engineering and Industrial Aerodynamics* 67&68, pp. 79-90, 1997.

[3] Neckel, A.L., Godinho, M., Influence of geometry on the efficiency of convergent – divergent nozzles applied to Tesla turbines, *Experimental Thermal and Fluid Science* 62, pp. 131-140, 2015.

[4] Tohidi Sardasht, M., Hosseini, R., Amani, E., An analysis of turbulence models for prediction of forced convection of air stream impingement on rotating disks at different angles, *International Journal of Thermal Scienses* 118, pp. 139-151, 2017.

[5] Yakhot, V., Orszag, S.A., Thangam, S., Gatski T.B., Speziale, C.G., Development of turbulence models for shear flows by a double expansion technique, *Physics of Fluids A: Fluid Dynamics* 4, pp. 1510-1520, 1992.