NUMERICAL SIMULATION OF THE FLUID FLOW BETWEEN BLADES AND AROUND THE TURBINE BLADE

Vesna Antoska, Ryszard Chodkiewicz and Bozin Donevski
Faculty of Technology and Technical Sciences, University St. Kliment Ohridski, Bitola, Macedonia;
Institute of Turbomachinery, Technical University of Lodz, Poland;
Faculty of Technical Science, University St. Kliment Ohridski, Bitola, Macedonia

Introduction

Numerical prediction methods become an essential tool for the design and analysis of turbomachinery components over the last two decades. The continuous increase in turbine inlet pressure and temperature, and modification of the turbomachinery parts, definitely require reliable and accurate predictions of the main flow dynamics characteristics and the heat loads imposed to the blades.

The interest of the present research is to contribute to the development of both numerical method and modified of upgrading mathematical model of a flow in the turbine cascade. The objective is to classify the influenced factors which affect the efficiency of the work of the turbine stage at defined thermodynamics properties of the flow. Quite a number of contributions on this topic have been presented in the open literature. The originality of the present work can be found in the application of different turbulence model. The Reynolds-averaged Navies-Stokes equations are solved with an implicit time-marching finite volume method around two-dimensional blade cascade, using CFD commercial computer code CFX-Tasc Flow. The computations were made for two cases. In the case 1, the flow around the turbine cascade was simulated. The authors present some conclusions how to change the blade profile to get better flow conditions around the blade. In the second part of analysis the results of computations of the flow in the whole turbine stage with the leaning stator blades are presented. The numerical computations of this study were conducted at the Institute of Turbomachinery, Lodz Technical University in Lodz, Poland.

Problem formulation

Physical model

The physical model of turbine stage schematically is presented on Figure 1. The geometric parameters necessary to specify the problem are the blade coordinates, the length of the arc of admission, the length of the distirbunce pitch, and the axial spacing between the nozzle block and the leading edge of the blade. The flow dynamic parameters necessary to specify the problem of the physical model are the following: inlet flow angle, total pressure, total temperature, and the back pressures downstream of the blades.

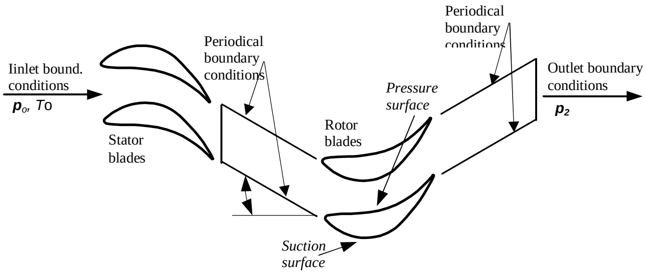

Fig. 1. Phusical model of the turbine stage

Governing equations

The equations used to model the flow in this research are the Reynolds-averaged and compressible continuity equation, momentum equation, and energy equation. By assuming that the fluid is a perfect gas the pressure p. The shear stresses, τ_{ij}, are divided into a laminar τ_{ij}^{lam} and turbulent part. The turbulence model used in this work really on the Boussineseq's approximation that the principal axes of the turbulent stress tensor are coincident with those of the mean strain-rate tensor. This is conversation approach in which a particular region of space is chosen, and then description of fluid flow is done in terms of the assumption in the control volume and flux through the control volume of appropriate quantities.In turbulent flows, the value of scalar variables fluctuates, and the instantaneous value of any scalar may be expressed as the sum of a mean and fluctuating component. To solve the turbulent flows, the original conservation equations must be time-averaged. Substituting the time-average quantities into the original, unsteady Navies-Stokes equations results in Reynolds Average Navies-Stokes (RANS) equations given below:

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x_i}\left[\bar{\rho}\tilde{u}_i\right] = 0$$

$$\frac{\partial}{\partial t}(\bar{\rho}\tilde{u}_i) + \frac{\partial}{\partial x_j}\left[\bar{\rho}\tilde{u}_i\tilde{u}_j + \bar{p}\delta_{ij} + \overline{\rho u_i'' u_j''} - \bar{\tau}_{ji}\right] = 0$$

$$\frac{\partial}{\partial t}(\bar{\rho}\tilde{e}_0) + \frac{\partial}{\partial x_j}\left[\bar{\rho}\tilde{u}_j\tilde{e}_0 + \tilde{u}_j\bar{p} + \overline{u_j'' p} + \overline{\rho u_j'' e_0} + \overline{q_j} - \overline{u_i\tau_{ij}}\right] = 0$$

the density averaged total energy $\tilde{e}_0$ is given by:

$\tilde{e}_0 \equiv \tilde{e} + \frac{\tilde{u}_k \tilde{u}_k}{2} + k$, where κ is the turbulent kinetic energy, defined by: $k \overset{def}{=} \frac{\widetilde{u_k'' u_k''}}{2}$

Equations above contains terms that cannot be expressed as functions of the mean flow variables. These terms must be related to known quantities using a turbulence model before a closed solution of the above equation system becomes possible *(CFX-TaskFlow, 2000, Theory Documentation)*. To solve the RANS equations, κ *-ω SST (Shear Stress Transport)* turbulence model was applied. This is two-equation turbulence model, with κ-representing the turbulence kinetic energy and the frequency $\omega = \varepsilon / \kappa$, where ε is the rate of dissipation of κ.

ε – dissipation of turbulent kinetic energy, u_{ij} – Cartesian velocity components

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}\left[\rho u_j k - \left(\mu + \frac{\mu_t}{\tilde{\sigma}_k}\right)\frac{\partial k}{\partial x_j}\right] = P - \beta' \omega k$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_j}\left[\rho u_j \omega - \left(\mu + \frac{\mu_t}{\tilde{\sigma}_\omega}\right)\frac{\partial \omega}{\partial x_j}\right] = \alpha \frac{\omega}{k} P - \beta \omega^2 + (1 - F)\frac{2}{\sigma_\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}$$

The eddy viscosity μ_t is related to κ and ω by: $\mu_t = c_\mu \frac{\rho k}{\omega}$ and $P = \tau_{ij}^{turb} \frac{\partial u_i}{\partial x_j}$ and the distribution of κ and ω is calculated from two model transport equations with the original set of constants (Wilcox, 1988).

Numerical methods

As a solver in this work CFX-TaskFlow is used. Code for solving equations for ideal gas used conservative forms of Navies Stokes Equations and k-ω SST turbulence model. A part of solver strategy used in TaskFlow is multigrid methods. The particular variant used in TaskFlow is based on conservation principles already implicit in the Finite Volume discretization and is called Additive Correction Multigrid. The linearized discrete algebraic equations that arise from most finite volume methods are sufficiently diagonally dominant to permit solution by simple relaxation methods, such as Gauss Seidel.

Comuptational grid

The computational grid is generated with a multi-block grid generator based on transfinite interpolation. The computational grid is structured of 6 blocks including with additional blocks at inlet and outlet of the computational domain. This grid has a relatively complex topology which gives it a high quality and avoids strongly skewed cells. The *O-mesh* around the blade allows for a good resolution of the leading and trailing edges. Total cells is 60 000 hexahedral elements of structured grid used to model the computational domain.

Boundary conditions

The computations were carried out by means of the CFX-TASCflow code. The flow was treated as compressible, and steam as an ideal gas. The k-ω SST turbulent model was employed. The turbine rotated with standard 50 Hz frequency (3000 rpm). The boundary conditions are:

Total presure constant over the inlet, P_e Total= 107 116 Pa; Inlet Mass Flow $\dot{m}$ =3.4 kg/s; Absolute total temp. T= 315 K ; Static pressure constant over the outlet, P= 101 657 Pa; Adiabatic wall, Outlet.

Result and discussion

From the numerical calculations one can obtain the following results:

- Visualization of the total pressure in characteristic sections, for the frozen rotor
- Visualization of the speed in characteristic sections

The total pressure distribution allows one to observe the losses in the stage. The places with lower values of the total pressure inform us about the higher losses in those places. The distribution of the total pressure in a control section downstream the stator blade row is presented. According to the predictions, the higher losses exist only in boundary regions, especially close to the hub in a plane downstream the stator.

the distribution of the total pressure along the meridional direction is presented. It is visible that the majority of the losses is generated at the outlet where the velocities are high and secondary flows developed.

The distribution of the static pressure around the blade in a 50% spa. That diagram points out to some possibilities of the profile refining. At the inlet, there are no significant pressure differences between the pressure and suction side of the blade. The suction side of the blade can be changed to get a more significant pressure difference between two sides of blade, however these changes should not cause the separation because they result in a higher loss level.

References

1. Baldwin S.B. and Lomax H. 1978. *Thin layer approximation and algebraic model for separated turbulent flows.* AIAA Paper 78-257.
2. Borzecki T., *Numeryczna analiza wplywu lopatek szablokszaltnych na spawnosc grupy stopni turbiny osiowej*. Institut Maszyn Przeplywowych Politechniki Lodzkej, Zespol Turbin i Sprezarek.
3. CFX-TaskFlow, 2000. *Computational Fluid Dynamics Software-Premier, Theory Documentation*, AEA Technology. Advanced Scientific Computing Ltd. Waterloo, Ontario, Canada.
4. Wilcox D.C., 1988. *Reassessment of the Scale-Determining Equation for Advanced Turbulence Models.* AIAA Journal, Vol.26.No.11. pp.1299-1310.