THE ARCHIVE OF MECHANICAL ENGINEERING

Vol. LII 2005 Number 3

Key words: *dynamic flow, turbine stage, Lagrange method*

NIKOŁAJ UZUNOW[*)]

ONE-DIMENSIONAL NUMERICAL SIMULATION OF TRANSIENT FLOW IN A TURBINE STAGE

The investigation presented in this work concerned one-dimensional modelling of transient flow in an axial turbine stage. Because of the compound motion in the rotor-blade channels, the model of this flow path element was considered the most difficult to solve. The basic modelling approaches and the usefulness of the Euler and Lagrange methods for mathematical description are discussed. Relevant applied model based on the Lagrange method is described. Results of a successful numerical simulation of transient flow in a turbine stage are presented. This success opens the opportunity to build and solve one-dimensional models of dynamic flows in turbine cascades and entire flow paths.

NOMENCLATURE

Symbols:

A – cross-section, [m^2],
c – absolute velocity, [m/s],
e – specific internal energy, [J/kg],
G – mass flow rate, [kg/s],
h – specific enthalpy, [J/kg],
h_0 – specific stagnation enthalpy, [J/kg],
h_r – reference, constant value of the specific enthalpy, [J/kg],
k – isentropic exponent,
l – space co-ordinate along the mean flow line, [m],
$\dot{m}$ – unitary mass outflow through the model channel walls, [kg/(m · s)],

*) *Warsaw University of Technology, Faculty of Power and Aeronautical Engineering, Institute of Heat Engineering, 21/25 Nowowiejska St., 00-665 Warsaw, Poland; E-mail: uzunow@itc.pw.edu.pl*

p – static pressure, [Pa],
$\dot{q}$ – unitary heat outflow through the model channel walls, [J/(kg · s)],
t – time, [s],
u – peripheral speed, [m/s],
v – velocity, [m/s],
w – relative velocity, [m/s],
x – axially directed space co-ordinate, [m],
α – angle of deviation of the absolute velocity from the axial direction,
β – angle of deviation of the relative velocity from the axial direction,
λ – friction coefficient,
Π – total perimeter of the model channel, [m],
ρ – density, [kg/m3],
τ – tangential stress on the model channel walls, [Pa].

Subscripts:

1 – relates to inlet quantities,
2 – relates to outlet quantities,
c – relates to quantities, determined in absolute terms (inertial system),
w – relates to quantities, determined in relative terms (non-inertial system).

1. Introduction

The research on the performance of devices, machines, and installations under transient conditions is necessary for the purposes of direct security (emergency cases), design, selection of control systems, staff training, on-line simulation (e.g., building of turbine simulators), and better understanding of the relevant physical processes. The experimental methods are inapplicable to research on dynamic thermal-flow phenomena in real objects of the size and importance of utility power turbines. The adequate research tools are therefore reduced to the means of computational fluid dynamics (CFD). Landau and Lifschitz [1] described profoundly the relevant theoretical fundamentals, while Fletcher [2] and Hirsch [3] discussed the contemporary models being applied in CFD. The complexity and degree of correctness of entire turbine flow models and their mathematical description depend on the object flow path discretisation method. The continuous approach (one-, two-, or three-dimensional) is expected to provide a uniform dynamic model of the whole flow path, while the discrete approach requires distinction of static elements (e.g., turbine stages, valves) and dynamic elements (e.g., chamber-type volumes, pipelines). The essential assumptions of lumped parameter models of turbine flow paths could be found in the work by Badyda [4]. The selection of appropriate modelling approach depends mainly on the available computing power and time. This limitation practically reduces the applicability of three-dimensional (3-D) models to steady flows and a very small range of pulsating flows. On the other hand, lumped parameter models make obtaining calculation time shorter than real process time possible (e.g., they enable building of on-line simulators), but they are based on multiple simplifications and averages. Another serious limitation could be the unavailability of detailed design and dimension data, caused by corresponding patents, copyrights, mental property rights, trade secrets, etc. At the present time, the one-dimensional (1-D) model concept seems to be an adequate compromise between the advantages and disadvantages in applying discrete and 3-D models. However, no credible signal of success in this area was detected.

The investigation under consideration concerned 1-D modelling of transient flow in a turbine stage. Because of the compound motion in the rotor-blade channels, the model of this flow path element was considered the most difficult to solve. Results of a successful numerical simulation of such a flow are presented in this work, as well as the applied model and its mathematical description. This success opens the opportunity to solve 1-D models of turbine cascades and entire flow paths.

2. Modelling concept development

The essential idea of 1-D turbine stage model is to transform the real flow channel into a model channel, in which the space co-ordinate converges to the mean flow line. The parameter values are averaged for cross-sections perpendicular to this line. In this connection, two basic requirements need to be fulfilled: the accumulation volume of the model channel should be equal to the accumulation volume of the real channel; and analogical phenomena should take place in the corresponding channel sections (e.g., mass transfer, heat transfer, receipt of labour). The general concept scheme and axial turbine stage velocity triangles are shown in Fig. 1.

Detailed blade profile dimension data are necessary for the model channel development. In general, there are two types of blade profile formation: by circle lines (older); or by polynomial functions. In both cases, in order to obtain smooth parameter distribution, polynomial approximation of the cross-section and the deviation angle (of the mean flow line from the axial direction) as functions of the axially directed space co-ordinate is recommended. However, special attention should be paid to the accuracy of this approximation, as it is one of the decisive factors for the model correctness.

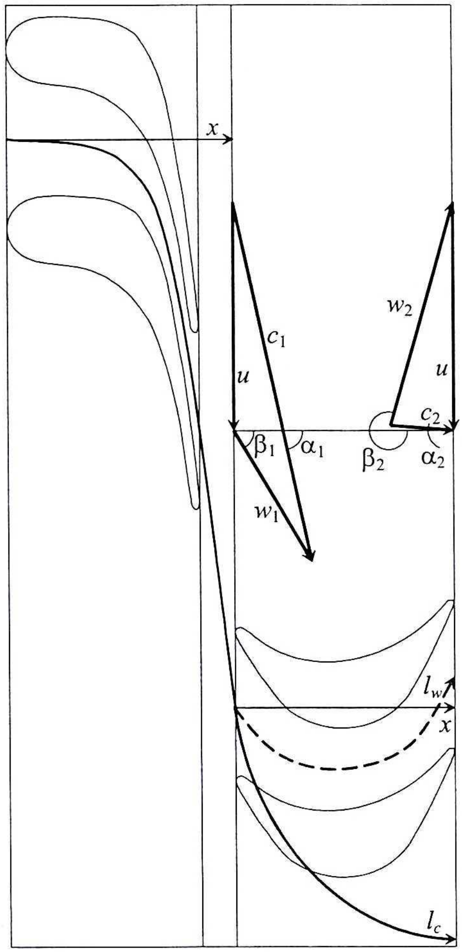

Fig. 1. General concept scheme of 1-D flow in an axial turbine stage and relevant velocity triangles. Symbols: u – peripheral speed; c – absolute velocity; w – relative velocity; x – space co-ordinate in axial direction; l – model space co-ordinate; α – deviation angle of the absolute velocity from the axial direction; β – deviation angle of the relative velocity from the axial direction. Subscripts: 1 – relates to inlet quantities; 2 – relates to outlet quantities; c – relates to absolute flow terms; w – relates to relative flow terms

The mathematical description of contemporary flow models is based on the mass, momentum and energy conservation laws. In accordance with the development presented by Szumowski, Selerowicz and Piechna [5], the following one-dimensional, conservative forms of the corresponding balance equations in Eulerian terms were received:

$$\frac{\partial}{\partial t}(\rho A) + \frac{\partial}{\partial l}(\rho v A) = -\dot{m} \quad \left[\frac{\text{kg}}{\text{m} \cdot \text{s}}\right]; \tag{1}$$

$$\frac{\partial}{\partial t}(\rho v A) + \frac{\partial}{\partial l}[A(p + \rho v^2)] = p\frac{\partial A}{\partial l} - \Pi\tau - \dot{m}v \quad \left[\frac{\mathrm{N}}{\mathrm{m}}\right]; \tag{2}$$

$$\frac{\partial}{\partial t}\left[\rho A\left(e + \frac{v^2}{2}\right)\right] + \frac{\partial}{\partial l}\left[\rho v A\left(h + \frac{v^2}{2}\right)\right] = -\rho A\dot{q} - \dot{m}\left(h + \frac{v^2}{2}\right) \quad \left[\frac{\mathrm{W}}{\mathrm{m}}\right]. \tag{3}$$

The particular mathematical approach depends on the assumed reference system and flow description method. An inertial reference system (i.e., connected to the stator) would condition an absolute velocity model, while a non-inertial system (i.e., connected to the rotor) would condition a relative velocity model. In both cases, the flow process could be described using either the Euler method (i.e., in reference to the channel) or the Lagrange method (i.e., in reference to the flow). Furthermore, Eqs. (2) and (3) could require make-up. For example, in terms of inertial system, the receipt of labour in the rotor-blade section should be additionally modelled. For this purpose, an artificial body force directed counter to the flow could be introduced and corresponding elements provided to the above-mentioned equations. This force could be defined in compliance with the 1-D theory of turbine stage, described by Puzyrewski [6]. Another example could be the eventual need to take the Coriolis force into consideration in case of a non-inertial system assumed.

The realised relevant, initial investigation showed that the application of absolute velocity model to the compound motion in the rotor-blade section causes fatal solution problems. The compound character of the motion introduces changeability of the channel geometry, as the absolute velocity and its deviation angle are interdependent. This could be expressed by the equations, resulting from the velocity triangles shown in Fig. 1:

$$c\cos\alpha = w\cos\beta; \tag{4}$$

$$c\sin\alpha = w\sin\beta + u. \tag{5}$$

After transformation of Eqs. (4) and (5), the above-mentioned interdependence receives the following form:

$$\sin(\alpha - \beta) = \frac{u}{c}\cos\beta. \tag{6}$$

The on-line determination of the channel geometry based on the solution of Eq. (6) results in fatal integration instability. The divergent character of the interdependence between the absolute velocity and the corresponding channel cross-section causes it. This determines the necessity to model the flow process in the rotor-blade section in relative terms.

Therefore, the model needed to be divided into two parts: an absolute part, which includes the stator-blade channel section and the space between the stator blades and the rotor blades, and a relative part, which includes the rotor-blade channel section. The transition from absolute to relative description terms was expected to be critical for the solution of this model. Initially, the Euler method of mathematical description was applied. The numerical simulation was based on the finite difference method and a direct approximation of the hyperbolic Eqs. (1)–(3) using the two-step Lax-Wendroff integration scheme. The relevant multiple simulation attempts failed. This was caused by the need to locate an integration grid node at the transition point. The inevitable determination of the flow velocity and cross-section at this and the neighbouring nodes in both absolute and relative terms on each time-step effected all the integration functions and introduced self-feeding numerical oscillations of rapidly increasing amplitude. In conclusion, because of the stationary approach and the compound character of the integration functions, the Euler method was found inadequate for the problem under consideration.

Summarising, a division of the model channel into a stator-blade part and a rotor-blade part, in which inertial and non-inertial reference systems are assumed, respectively, combined with use of the Lagrange method for the flow process mathematical description, remained the only potentially successful approach.

3. Applied model

Potter [7] described the Lagrange method intelligibly. It is based on a reference to the flow being described: the co-ordinate system points (integration grid nodes) are connected to flow elements and move with the flow velocity. The flow is divided into moving cells of constant mass (as mass is not transferred between the cells) and changeable length (changeable grid intervals). The velocity is being determined for the cell borders (i.e., for the primary grid nodes, or velocity grid nodes), while the density and static pressure are being determined as averages for the particular cells and assigned to their centres, thus forming a secondary grid. Second order of accuracy of the numerical integration scheme is obtained by alternate integration on the primary and secondary grid; related to the primary grid, the secondary grid is shifted by half of the integration time step.

The Lagrange method is particularly adequate to 1-D problems. It is simple and describes the flow physics directly. The relevant integration scheme is also simple and natural, and does not create or sustain numerical oscillations (unlike the Lax-Wendroff scheme, being the most often used in applications based on the Euler method). But, in this particular case, another crucial advantage appeared: because of the integration grid character, there was no grid node permanently located at the point of transition from absolute to relative velocity model, and they were the flow parameters integrated (and not functions of them); i.e., this source of numerical disturbance (oscillations) was eliminated.

Compared to the general case, described by Eqs. (1)–(3), two model simplifications were introduced. Firstly, the channel walls were assumed to be impermeable. And secondly, the heat transfer from the flow to the channel walls was assumed to be negligible. In addition, the turbine stage was assumed to be strictly axial, and the Coriolis force did not effect the flow process. The Lagrangian mathematical description is based on the introduction of a substantial derivative, defined as:

$$\frac{d}{dt} = \frac{\partial}{\partial t} + v\frac{\partial}{\partial l}. \tag{7}$$

After certain transformations, the flow parameter substantial derivatives were drawn from the balance Eqs. (1)–(3) in an explicit form:

– for the absolute part:

$$\frac{dc}{dt} = -\frac{1}{\rho}\left(\frac{\partial p}{\partial l_c} + \frac{\Pi_c}{A_c}\tau_c\right); \tag{8}$$

$$\frac{d\rho}{dt} = -\rho\left(\frac{\partial c}{\partial l_c} + \frac{c}{A_c}\frac{\partial A_c}{\partial l_c}\right); \tag{9}$$

$$\frac{dp}{dt} = k\frac{p}{\rho}\frac{d\rho}{dt} + (k-1)\,c\frac{\Pi_c}{A_c}\tau_c, \tag{10}$$

– and for the relative part:

$$\frac{dw}{dt} = -\frac{1}{\rho}\left(\frac{\partial p}{\partial l_w} + \frac{\Pi_w}{A_w}\tau_w\right); \tag{11}$$

$$\frac{d\rho}{dt} = -\rho\left(\frac{\partial w}{\partial l_w} + \frac{w}{A_w}\frac{\partial A_w}{\partial l_w}\right); \tag{12}$$

$$\frac{dp}{dt} = k\frac{p}{\rho}\frac{d\rho}{dt} + (k-1)\,w\frac{\Pi_w}{A_w}\tau_w. \tag{13}$$

The friction phenomena are complex and difficult to describe. The proper description of the turbulence phenomenon is the essential problem of building 3-D models of turbine stage flows. However, within discrete and 1-D dynamic models of whole turbine flow paths, it is the working fluid's mass and energy accumulation being essential, while the friction is of secondary importance. Hence, a simplified model of hydraulic losses was applied, based on a dependence received by Szumowski, Selerowicz and Piechna [5]. Its forms for the two respective model parts are:

$$\tau_c = \lambda\,\frac{\rho c^2}{8}; \tag{14}$$

$$\tau_w = \lambda\,\frac{\rho w^2}{8}. \tag{15}$$

In order to enable the numerical simulation, application of a simplified working medium model was needed. In general, the perfect gas equation is being used in such cases. For the convenience of the description and solution, the calorific form of the perfect gas equation was used:

$$h = h_r + \frac{k}{k-1}\frac{p}{\rho}. \tag{16}$$

The disturbances, introducing transient conditions to adiabatic turbine flow, could be divided into two general groups: disturbances, causing (or caused by) significant rotational speed changes; and disturbances causing negligible changes in the rotational speed. At this investigation stage, the applied model was designed to describe transient flows of the second group; i.e., a constant peripheral speed was assumed.

4. Simulation results

The numerical simulations of unsteady flow processes in a turbine stage were realised using a set of original, author's computer programmes. The relevant integration procedure was based on the finite difference method and a direct integration of Eqs. (8)–(13) using the integration scheme described in Ch. 3.

The first stage of the 13K215 steam turbine high-pressure cascade was selected to be the modelling object. The 13K215 turbine is the most widely applied machine in the Polish utility power industry, and its detailed design and dimensional data were available. As a result from the geometrical data processing, the real flow channel axial cross-section and the deviation angle of the mean flow line from the axial direction were received as functions of the axially directed space co-ordinate. This enabled not only the model channel development along the mean flow line, but also the exact determination of the particular grid node positions in reference to the turbine axis (the determination of the exact position is one of the additional problems in applications based on the Lagrange method).

The initial parameter distribution was assumed to correspond to the modelling object's rated, steady operating conditions. These conditions include a static pressure of 9.16 MPa and a density of 28 kg/m^3 at the inlet, a static pressure of 8.06 MPa at the outlet, and a mass flow rate of 168 kg/s. A test of the main numerical integration code's stability was carried out by its long-term run under unchanged boundary conditions.

A sudden, rapid change in the flow parameters at the modelled turbine stage inlet (inlet boundary conditions) was selected to be the disturbance, introducing transient flow conditions. Within 0.1 ms, the inlet pressure was reduced to 8.90 MPa (i.e., by 2.84%), and the inlet density was decreased to 27.4 kg/m^3 (i.e., by 2.14%). The flow stabilised itself under the new, off-design conditions within about 2 ms. The results from the numerical simulation of this transient process are shown in Figs. 2–10. They are organised in three groups, in which the distribution of normalised static pressure (Figs. 2–4), specific stagnation enthalpy (Figs. 5–7), and mass flow rate (Figs. 8–10) are shown. In the first figures in each group (Figs. 2, 5 and 8), the transient process in the disturbance introduction period is shown; i.e., during the first 0.1 ms (t = 0–0.1 ms). The second figures in each group (Figs. 3, 6 and 9) concern the flow parameter changes within the next 0.1 ms (t = 0.1–0.2 ms). In the last figures in each group (Figs. 4, 7 and 10), the process in the next 0.2 ms period is shown (t = 0.2–0.4 ms) together with the parameter distribution in the final steady state (t = 2 ms).

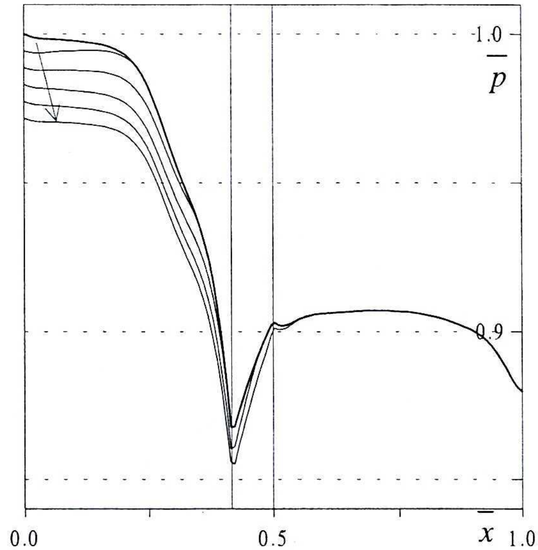

Fig. 2. Static pressure distribution at $t = 0$ (thicker line), $t = 0.02$ ms, $t = 0.04$ ms, $t = 0.06$ ms, $t = 0.08$ ms, and $t = 0.1$ ms. The arrow shows the development in time

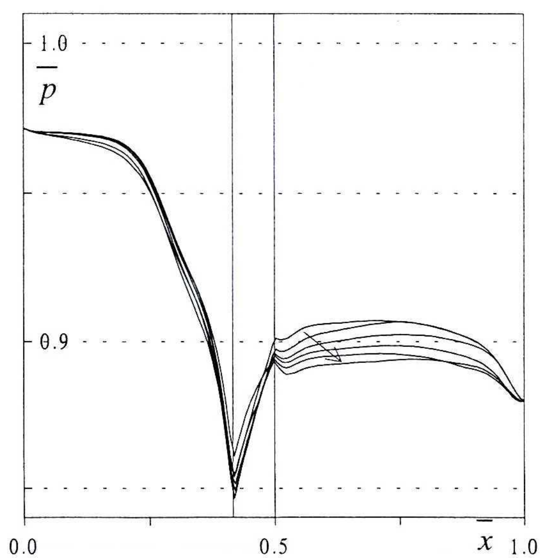

Fig. 3. Static pressure distribution at $t = 0.1$ ms, $t = 0.12$ ms, $t = 0.14$ ms, $t = 0.16$ ms, $t = 0.18$ ms, and $t = 0.2$ ms. The arrow shows the development in time

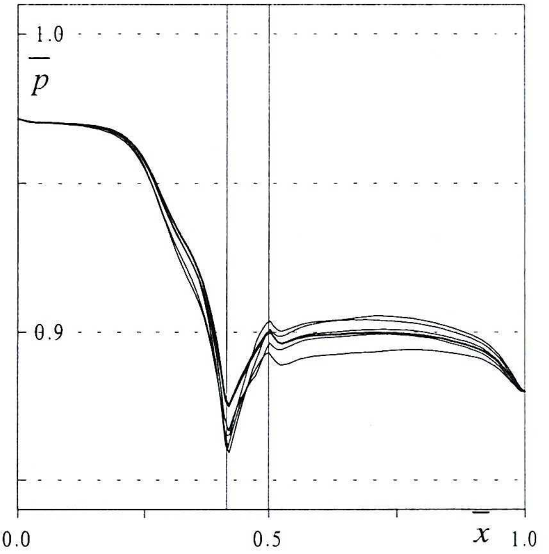

Fig. 4. Static pressure distribution at $t = 0.2$ ms, $t = 0.25$ ms, $t = 0.3$ ms, $t = 0.35$ ms, $t = 0.4$ ms, and $t = 2$ ms (thicker line)

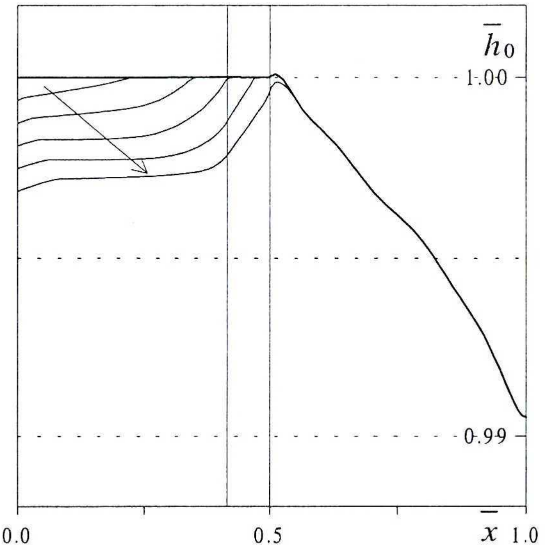

Fig. 5. Specific stagnation enthalpy distribution at $t = 0$ (thicker line), $t = 0.02$ ms, $t = 0.04$ ms, $t = 0.06$ ms, $t = 0.08$ ms, and $t = 0.1$ ms. The arrow shows the development in time

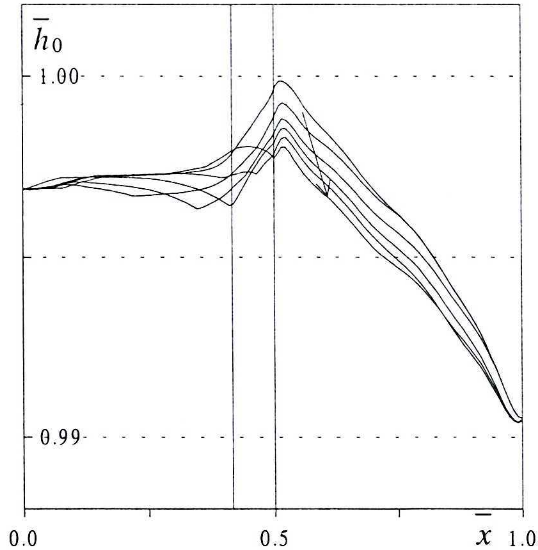

Fig. 6. Specific stagnation enthalpy distribution at $t = 0.1$ ms, $t = 0.12$ ms, $t = 0.14$ ms, $t = 0.16$ ms, $t = 0.18$ ms, and $t = 0.2$ ms. The arrow shows the development in time

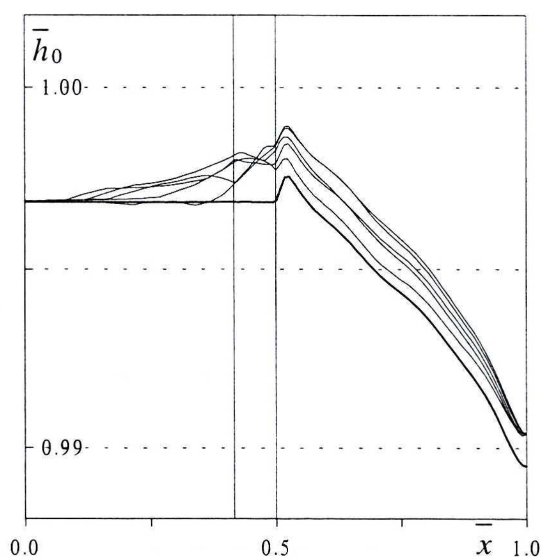

Fig. 7. Specific stagnation enthalpy distribution at $t = 0.2$ ms, $t = 0.25$ ms, $t = 0.3$ ms, $t = 0.35$ ms, $t = 0.4$ ms, and $t = 2$ ms (thicker line)

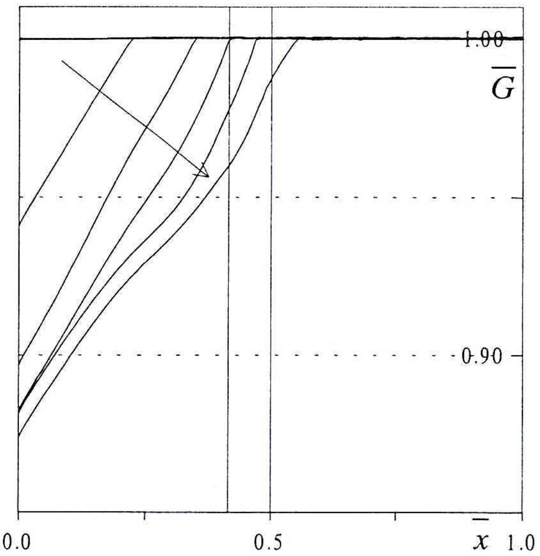

Fig. 8. Mass flow rate distribution at $t = 0$ (thicker line), $t = 0.02$ ms, $t = 0.04$ ms, $t = 0.06$ ms, $t = 0.08$ ms, and $t = 0.1$ ms. The arrow shows the development in time

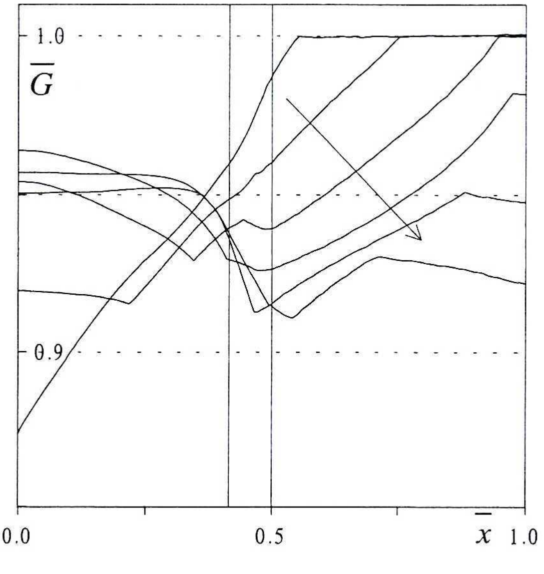

Fig. 9. Mass flow rate distribution at $t = 0.1$ ms, $t = 0.12$ ms, $t = 0.14$ ms, $t = 0.16$ ms, $t = 0.18$ ms, and $t = 0.2$ ms. The arrow shows the development in time

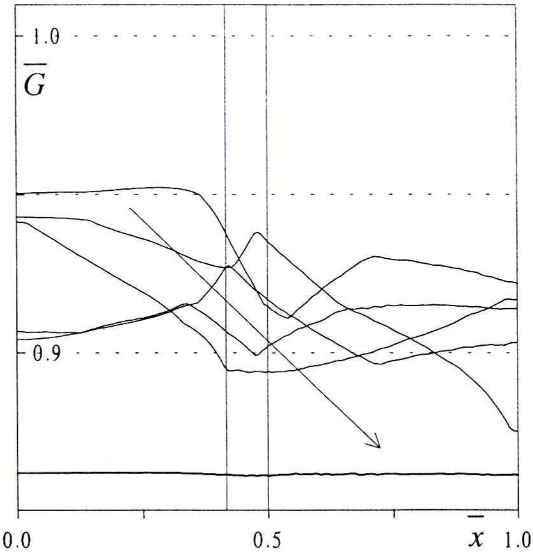

Fig. 10. Mass flow rate distribution at $t = 0.2$ ms, $t = 0.25$ ms, $t = 0.3$ ms, $t = 0.35$ ms, $t = 0.4$ ms, and $t = 2$ ms (thicker line). The arrow shows the development in time

The calculation of the specific stagnation enthalpy, defined as:

$$h_0 = h + \frac{c^2}{2}, \tag{17}$$

required determination of the absolute velocity in the relative model part. For this purpose, Eqs. (4) and (5) were transformed to:

$$\tan\alpha = \tan\beta + \frac{u}{w\cos\beta}, \tag{18}$$

and thus the absolute velocity deviation angle was determined. Next, the absolute velocity was calculated from appropriately transformed Eq. (4). The resulting decrement of the specific stagnation enthalpy is identical to the labour, received in the rotor-blade section.

5. Conclusions

The performed investigation results in several general conclusions. First of all, it proved that successful one-dimensional numerical simulation of transient flow in a turbine stage is possible. The basic modelling approach requirements seem to be established: the flow process in the compound-motion section needs to be modelled in relative terms, and the Lagrange method of mathematical description needs to be applied. Such an approach provides simplicity and stability of the numerical solution procedure.

The successful solution of the presented single turbine stage model opens the opportunity to build one-dimensional dynamic models of turbine cascades. Such models will be more accurate than the lumped parameter or empiric models being widely applied nowadays. Another direction of further research activities should be the enhancement of the model applicability. This would include adaptation to changeable rotational speed problems and introduction of heat transfer and mass transfer through the channel walls. The latter is essential for the modelling of extractions in steam turbine flow paths or of surface-cooled gas turbine stages.

The latest achievements in the flow parameter measurement techniques, applied by Matsunuma and Yasukata [8], make eventual model verification possible; i.e., they give the opportunity to confront numerical simulation results with laboratory measurement data. Such a confrontation would contribute to the development of distributed parameter modelling in particular, and of applied computational fluid dynamics in general. In this connection, the presented successful one-dimensional numerical simulation of transient flow in a turbine stage seems to gain additional importance.

Manuscript received by the Editorial Board on March 22, 2005;
final version, June 3, 2005.

REFERENCE

[1] Landau L. D., and Lifschitz E. M.: Hydrodynamics (Gidrodinamika). Nauka, Moscow, 1986 (in Russian).

[2] Fletcher C. A.: Computational Techniques for Fluid Dynamics, Vol. I, II. Springer Verlag, Berlin, 1988.

[3] Hirsch C.: Numerical Computation of Internal and External Flows, Vol. I, II. J. Wiley & Sons, New York, 1989.

[4] Badyda K.: Problems of Mathematical Modelling of Power Installations (Zagadnienia modelowania matematycznego instalacji energetycznych). Warsaw University of Technology Publishers (OWPW), Warsaw, 2001 (in Polish).

[5] Szumowski A., Selerowicz W., and Piechna J.: Gas Dynamics (Dynamika gazów). Warsaw University of Technology Publishers (WPW), Warsaw, 1988 (in Polish).

[6] Puzyrewski R.: Fundamentals of Theory of Turbomachinery in One-Dimensional Terms (Podstawy teorii maszyn wirnikowych w ujęciu jednowymiarowym), Ossolineum, Wrocław, 1992 (in Polish).

[7] Potter D.: Computational Physics. J. Wiley & Sons, New York, 1973.

[8] Matsunuma T., and Yasukata T.: LDV Measurements of Unsteady Midspan Flow in a Turbine Rotor at Low Reynolds Number. IGTI, GT2003-38468, 2003.

Jednowymiarowa symulacja numeryczna nieustalonego przepływu przez stopień turbinowy

S t r e s z c z e n i e

Przedstawiono badania nad jednowymiarowym modelowaniem nieustalonych przepływów przez osiowy stopień turbinowy. Z powodu złożonego ruchu w kanałach wirujących, model tego elementu układu przepływowego turbiny uznano za najtrudniejszy do rozwiązania. Omówiono podstawowe podejścia przy modelowaniu oraz przydatność metod opisu matematycznego Eulera i Lagrange'a. Opisano odpowiedni model stosowany, oparty na metodzie Lagrange'a. Przedstawiono wyniki udanej symulacji numerycznej nieustalonego przepływu przez stopień turbinowy. Sukces ten stwarza możliwość budowy i rozwiązania jednowymiarowych modeli kaskad turbinowych i całkowitych układów przepływowych turbin do badania zachodzących w nich procesów nieustalonych.