Budapest University of Technology and Economics (BUTE)
Faculty of Transportation Engineering
Department of Aircraft and Ships

NUMERICAL METHODS AND APPLICATIONS FOR FLOW CALCULATIONS IN TURBO AND FLUID MACHINES

Summary of Ph.D. Thesis

Árpád VERESS

Supervisor: Dr. Imre SÁNTA associate professor
BUTE
Department of Aircraft and Ships

BUTE Department of
Aircraft and Ships

Budapest, 2004

Introduction

Nonlinear partial differential equations such as continuum-mechanics based Euler and Navier-Stokes (NS) equations[1] have no general closed form solution as yet. However, in consideration of the high level evolution of computer technology and expectations arisen from the industry, numerical methods for fluid flow (CFD (Computational Fluid Dynamics)) have come into prominence to solve fluid dynamic problems in engineering sciences.

Today's intensively developing CFD methods such as Direct Numerical Simulation (DNS) [6,8,10,12] and Large Eddy Simulation (LES) [11,3,17,10] require a huge amount of computational time and memory comparing to conventional high level flow modelling (e.g. turbulence modelling for Reynolds Averaged Navier-Stokes equations {RANS}), so their widespread industrial applications are going to be realised only sometime in the future.

Nowadays, in the practical-industrial application point of view, the best flow simulation is based on RANS. The application of different turbulence models (e.g. Baldvin-Lomax, Spalart-Almaras, k-ϵ [14 and others]) can be described by using a huge amount of empirical and experimental data, which may cause considerable differences between the results of different methods and reality. This problem has more emphasis in the case of transitional flow.

Assuming a frictionless and ideal flow, the NS equations – neglecting viscous and heat conducting terms – can be reduced to the Euler equations, which are the highest level approximation of inviscid flow. As is known, the set of Euler equations allows discontinuous solutions in certain cases, namely vortex sheets, contact discontinuities or shock waves. Particularly, the Euler equations are valid for the modelling of compressible high Reynolds number flows outside of the boundary layer without separation. As most of the industrial process as well as most of the flow situations encountered in nature, are dominated by convective effects, and therefore characterized by high Reynolds number, they are well approximated by the Euler equations. In case of incompressible flows, the Euler equations also represent a higher level of approximations compared to the potential flow

[1] In modern fluid dynamics literature, the NS or Euler equations refer to the complete system of equations, including both continuity and energy conservation laws.

models because of the resolving ability of contact discontinuities and vortex sheets.

On the other hand the number of computations is less, so computational time is shorter in the case of inviscid flow modelling – with no additional equations related to the turbulence modelling – than in case of viscous flow models. By the way, most 3D flows – with the appropriate consideration of symmetry – can be simplified into a 2D process. In case of Euler based approximation, because of the absence of a boundary layer modelling and so the unresolvability of secondary flows for instance, the difference between the results of 3 and 2D applications are less than in the case of viscous flow simulations.

Take all the above into consideration – for time reduction in the preliminary design procedures – the approximated results of the highest level of 2D inviscid flow simulation may be well compensated by less computational time.

Goals and Summary of Thesis

In this work a couple of new design procedures are developed in the field of turbo and fluid machinery using 2D Euler based own codes, commercial software and numerical optimization tools developed at von Kármán Institute for Fluid Dynamics (VKI). New design methodologies are demonstrated through the development of a certain machine, but the governing principles and design steps are valid for all machines of similar types. Besides the development of the new design procedures, particular care has been taken of the compactness and further improvability (3D, time dependent, viscous flow, etc.) of own codes. The thesis can be divided into two main parts. The first one deals with compressible and second one with incompressible flow modelling. The structure of both parts is similar. After rigorous mathematical deduction and description of the numerical methods, the next step is validation. Finally, by the application of own codes and other CFD techniques in the design of a new product, a conclusion has been made concerning the correctness of the new design procedures.

In case of compressible flow, a convergence accelerator characteristic type boundary condition (BC) and for rotating machines applications, a special inlet boundary condition are established. A 2D cell centered finite volume method with a computationally effective artificial dissipation method,

originally developed by Jameson [9], and Roe approximated Riemann solver [7] are reconstructed for the adaptation of the BCs. In addition a 2D Van Leer flux splitting upwind method [14] is also developed for the validation of own codes. The application of the special inlet boundary condition for rotating machines is tested on a 2D DCA (Double Circular Arc) cascade, which is made by the projection of the outer axis-symmetric stream surface of the transonic axial rotor into 2D.

With the use of modern commercial CFD codes (CFX-Tascflow) and an inverse design program developed at VKI [2], a new design strategy is established for deswirl vane design in the return flow channel for multistage centrifugal compressors. As a first step a constant blade loading condition has been made and an analytical procedure is developed to determine the blade camber line. As a result of the use of the Euler based VKI inverse design program and the introduction of negative lean, significant improvements can be observed in the design specification namely the loss coefficient and pressure recovery factor, which parameters were determined by using CFX-Tascflow NS solver.

In order to be able to use the same scheme for compressible and incompressible flow modelling, Chorin's pseudo-compressibility method [1] is developed for the incompressible flow by means of 2D finite volume discretization based on the Euler equations.

Beside the code, a soft solid wall convergence accelerator technique and static pressure special iterative inlet boundary condition are also developed. By using the own code explained in this paragraph, a new design strategy is developed for the optimization of the return type jet pumps. It has been verified by numerical simulations that the chamfered throat, the flow driver lug and a special variation of the pump geometry determine a certain jet pump configuration, which belongs to the maximum flow transportation (Fig. 5.).

List of Thesis

Numerical Modelling of Compressible Flow

Numerical methods are redeveloped and validated for 2D inviscid compressible flow modelling based on the Euler equations, which are in conservation form [4]:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}=0\ ;$$
$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u^2+p)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}=0\ ;$$
$$\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho uv)}{\partial x}+\frac{\partial(\rho v^2+p)}{\partial y}=0\ ;$$
$$\frac{\partial(\rho E)}{\partial t}+\frac{\partial(\rho u h_0)}{\partial x}+\frac{\partial(\rho v h_0)}{\partial y}=0\ ;$$

1. A Riemann problem approximated by Roe [7] is reconstructed for 2D compressible inviscid flow computation (subchapter 2.2.1.2/b) to adapt the boundary condition described in 1.1. (*Note: Mulder limiter is used at the second order spatial extension to decrease unwanted oscillation near the shock wave [5].*)

   1.1. In order to decrease computational time, a new, characteristic type inlet boundary condition is developed using a nonliner function of cell face normal Mach number to determine numerical boundary conditions (subchapter 2.4.3). (*Note: the flow parameters are transformed from the Cartesian space to the another one, which is defined by the unit vector pointing outward from the cell face and its normal direction. At the inlet boundary, keeping the Riemann invariant constant, the next nonlinear equation can be deduced for the unknown boundary Mach number:*

$$a\left(M_B^{-2}+\frac{\gamma-1}{2}\right)^{-\frac{1}{2}}-b\left(1+\frac{\gamma-1}{2}M_B^2\right)^{-\frac{1}{2}}-k=0\ ;$$

   The Newton-Raphson method is used for calculating M_B. The intensity of reflected disturbances is decreased by using BC, which led to a faster convergence.)

2. An Euler based 2D inviscid flow solver is developed using artificial dissipation (subchapter 2.2.1.1) to model the compressible and incompressible flow in the same scheme, to perform transonic flow simulation by means of the cheapest way (simple programming, low computational time, and storage with appropriate accuracy) and to apply new boundary condition described in 2.1. *(Note: the 4th order Runge-*

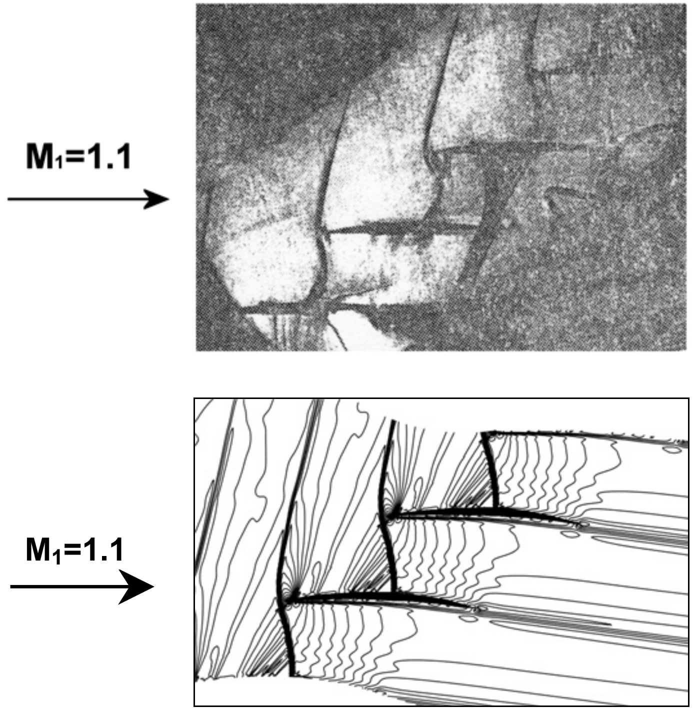

Figure 1. Shock wave pattern in DCA cascade (above: Schlieren photograph [13], below: computation with iso-Mach lines)

Kutta method is used for time stepping and Jameson artificial viscosity [9] is adopted for the stability of the code.)

   2.1. A simplified inlet boundary condition is developed for modelling 2D transonic axial compressor rotor flow and setting up the performance map. This procedure takes the number of revolutions and the outlet pressure of the compressor into consideration, by which the relation between the relative and absolute parameters can be determined (subchapter 2.6). (*Note: Generally, in case of rotational motion, at the subsonic inlet three of the physical: T_0, p_0, α and numerical: p boundary conditions are not independent, so the pressure rise over the rotor and the number of revolutions determine the massflow and the inlet flow angle.*

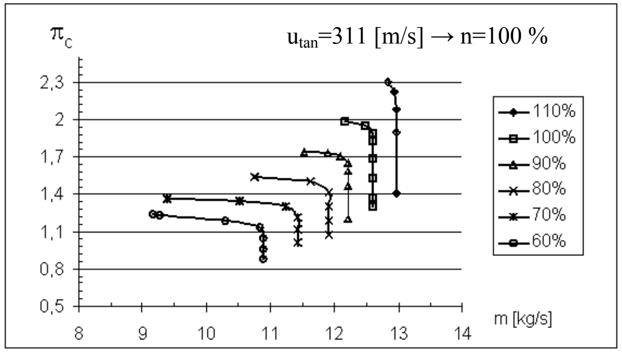

Figure 2. 2D compressor rotor performance map

The validation of this procedure is performed by a 2D DCA ((Double Circular Arc) cascade, which is placed in transonic flow (Fig. 1.).
2D DCA compressor performance map is determined for the application of the BC (Fig. 2.). The 2D cascade is generated by projection of the outer stream surface of the rotor into the plane.

3. A new design procedure is developed for deswirl vane design in the return flow channel for multistage centrifugal compressors (Fig. 3.) (subchapter 2.7) using modern commercial CFD codes (CFX-Tascflow) and the inverse design programs developed at VKI [2]:

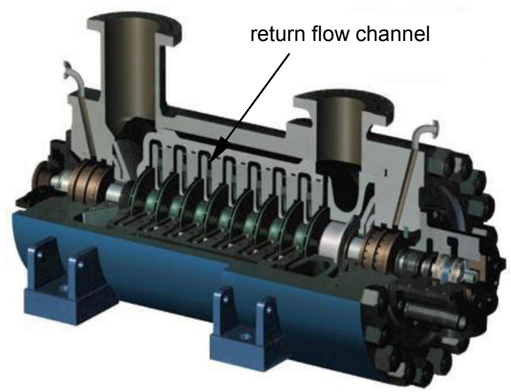

Figure 3. A-C multistage centrifugal compressor unit

3.1. A zero circulation and constant blade loading (CBL) assumption has been made and an analytical 3D blade design procedure is developed to determine initial blade geometry (subchapter 2.7.3). (*Note: the zero circulation was imposed on closed contours, defined by part of*

Design	**Without extension**	**CBL + extension**	**CBL + inverse design**
P_2^o [Pa]	299699,1	299526,6	299696,6
P_2^s [Pa]	159038,5	182298,2	174936,4
P_3^o [Pa]	236665,3	261108,8	264954,2
P_3^s [Pa]	225215,1	258238,	258414,2
$\boldsymbol{\omega}$	**0,448**	**0,327**	**0,278**
$\boldsymbol{C_p}$	**0,470**	**0,647**	**0,669**
$\dot{m}$ [kg/s]	4,68	4,5	4,64

Table 1. Comparison of overall performances

the suction and pressure side of two adjacent vanes and two lines at constant radius, providing the following relation between vane loading and the change in flow angle β_{bl}:

$$W_{ps} - W_{ss} = \cos\beta_{bl}\left(\frac{2\pi}{z} - \frac{\delta_{th}}{R\ \cos(\beta_{bl})}\right)\frac{d}{dm}\left(W_m R\ tg\left(\beta_{fl}\right)\right) = C\ ;$$

The blade angle β_{bl} is then calculated by numerical integration from the leading and to the trailing edge. Blade thickness distribution is constructed by the abscissa of an ellipse in the first 75 % of blade camber while linear blade thickness distribution is used from the thickest part of the blade to the given trailing edge thickness.)

3.2. According to viscous numerical fluid dynamics investigations, the blade extension over the return bend has an expected effect to decrease the probability of separation near to the shroud section inlet of the return bend, hereby improving design specification (loss coefficient and pressure recovery factor) (Table 1.):

$$\omega = \frac{\overline{p}_{in}^{\,to} - \overline{p}_{out}^{\,to}}{\overline{p}_{in}^{\,to} - \overline{p}_{in}^{\,st}}\ ;\ C_p = \frac{\overline{p}_{out}^{\,st} - \overline{p}_{in}^{\,st}}{\overline{p}_{in}^{\,to} - \overline{p}_{in}^{\,st}}\ ;$$

(*Note: all 3D viscous numerical simulation was performed by using CFX-TASCflow commercial CFD software*)

3.3. As a part of modern optimization strategy, an Euler based inverse design program – developed at VKI [2] – is used to further improve the loss coefficient and the pressure recovery factor (ω, C_p) (subchapter 2.7.4). (*Note: The inverse design program is an iterative procedure in which an initial geometry is modified until the desired/prescribed velocity or pressure distribution on the blade contour is obtained.*)

	No lean	**Negative lean**	**Positive lean**
ω	**0.278**	**0.263**	**0.280**
C_p	**0.669**	**0.681**	**0.638**

Table 2. Impact of lean on return vane performance

3.4. Finally, the effect of lean is investigated resulting in improved design specifications in case of negative lean (Table 2.) (subchapter 2.7.6).

Numerical Modelling of Incompressible Flows

A numerical method is developed and validated for 2D inviscid incompressible flow modelling based on Euler equations, which are in conservation form [4]:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0\,;$$

$$\frac{\partial u}{\partial t}+\frac{\partial\left(u^2+p/\rho\right)}{\partial x}+\frac{\partial(uv)}{\partial y}=0\,;$$

$$\frac{\partial v}{\partial t}+\frac{\partial(uv)}{\partial x}+\frac{\partial\left(v^2+p/\rho\right)}{\partial y}=0\,;$$

4. For the compressible and incompressible flow modelling by the same numerical procedure and for the adaptation of boundary condition described in 4.1. and 4.2. Chorin's pseudo-compressibility method [1] is developed for 2D inviscid and incompressible flow computation based on the Euler equations (subchapter 3.1). In the cell centered finite volume method Jameson artificial dissipation [9] guarantees numerical stability. (*Note: the 4th order Runge-Kutta method is used for time stepping*)

4.1. In order to model the inflow driven by surrounding static pressure, a mass flow and prescribed static pressure based feedback is developed for the homogeneous and constant inlet cross section boundary condition (subchapter 3.1.1).

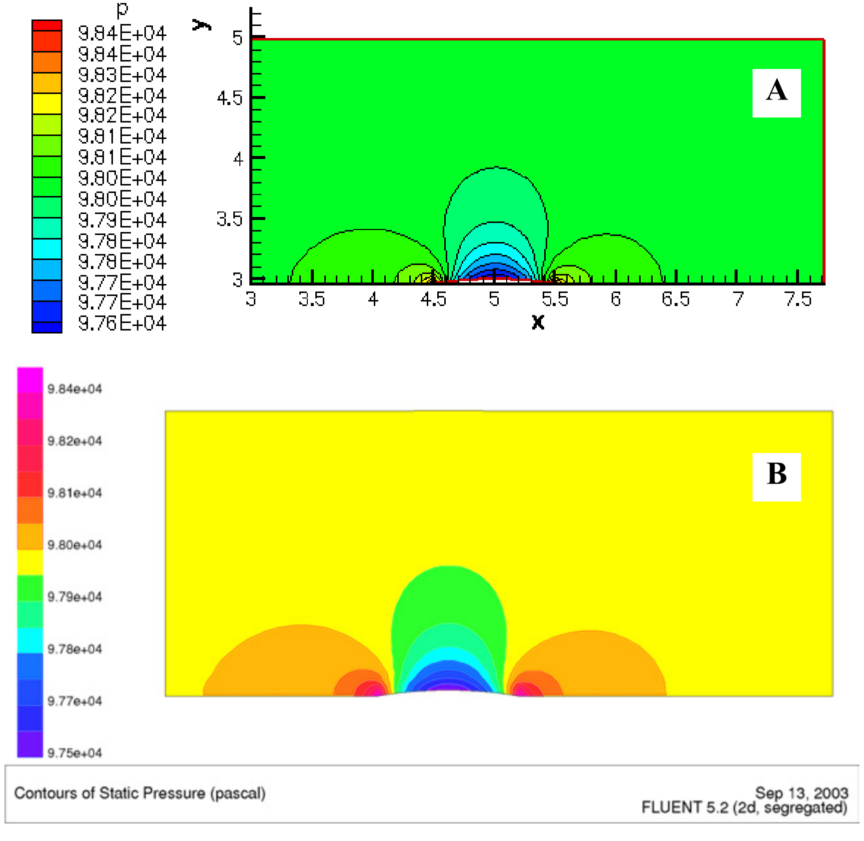

Figure 4. Static pressure iso-lines (A: result of own code, B: result of FLUENT)

4.2. A new soft solid wall technique is developed for the scheme explaining in 4. to make the convergence faster towards the steady state solution (3.2. subchapter).

(*Note: The numerical method and its boundary conditions was validated by Fluent commercial CFD software (Fig. 4.)*).

5. A new design strategy is developed for the optimization of the return flow jet pumps using own code described in 4. (subchapter 3.3):

   5.1. Chamfered throat at the inlet of jet diffuser helps to avoid the separation bubble, which evolves at the outer part of the return bend in the jet pumps (Fig. 5.).

   5.2. Flow transportation is also improved by using a relatively small flow driver lug located at the orifice outlet of jet pumps (Fig. 5.).

   5.3. Keeping the smallest modification concerning the production cost it has been verified that three main geometrical elements: inlet width,

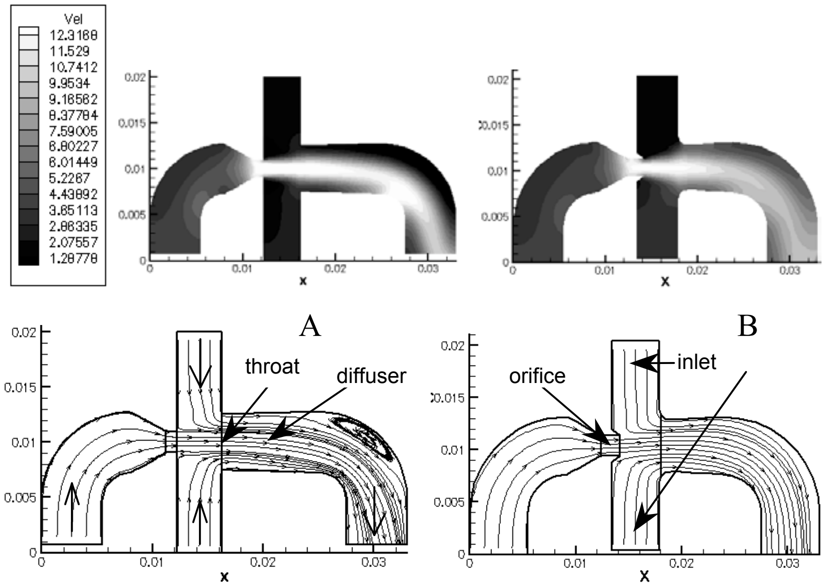

Figure 5. Velocity distributions [m/s] (over) and stream traces (below) in the 1.9 mm orifice diameter jet pump; A: initial geometry, B: optimized geometry

diffuser inlet diameter and diffuser length have a certain combination of its sizes for the flow transportation to be maximized (Fig. 5.) [15].

References

[1] **Chorin, A. J.**: A Numerical Method for Solving Incompressible Viscous Flow Problems. *Journal of Computational Physics, 2, 12-26.*, 1967.

[2] **Demeulenaere, A.**: Conception et development d'une methode inverse pour la gmeration d'aubes de turbomachines, *Ph.D Thesis at VKI*, 1997.

[3] **Germano M., Piomelli U., Moin P. and Cabot W. H.**: A Dynamic Subgrid-Scale Eddy Viscosity Model. *Phys. Fluids, A3 (7), pp. 1760-1765.* 1991.

[4] **Hoffmann K. A., Chiang S. T. L. Siddiqui M. S. and Papadakis M.**: Fundamental Equations of Fluid Mechanics. *Engineering Education System, ISBN 0-9623731-9-2*, 1996.

[5] **Mulder W. A. and Van Leer B.**: Implicit Upwind Methods for the Euler Equations. *AIAA 6th Computational Fluid dynamics Conference, pp. 303-310, AIAA paper 83-1930*, 1983.

[6] **Orszag S. A. and Patterson G.S.**: Numerical Simulation of Three-Dimensional Homogeneous Isotropic Turbulence. *Phys. Rev. Lett., 28:76-79,* 1972.

[7] **Roe, P. L.**: Approximate Riemann Solvers, Parameter Vectors, and Difference Schemes. *Journal of Computational Physics, Vol. 43 pp. 357-372.*, 1981.

[8] **Rogallo R. S.**: Numerical Experiments in Homogeneous Turbulence. *NASA TM 81315,* 1981.

[9] **Schmidt, W., Jameson, A.**: Recent Developments in Finite-Volume Time-Dependent Techniques for Two and Three Dimensional Transonic Flows. *Lecture Series at VKI: Computational Fluid Dynamics*, 1982.

[10] **Shen L. and yue D. K. P.**: Large-Eddy Simulation of Free Surface Turbulence. *J. Fluid Mech., 440, pp. 75-116*, 2001.

[11] **Smagorinsky J.**: General Circulation Experiments With the Primitive Equations. *Mon. Weather Rev., 93, pp. 99-164*, 1963.

[12] **Spalart P. R.**: Direct Numerical Simulation of a Turbulent Boundary layer up to $R_\theta = 1410$. *J. Fluid Mech., 187:61-98,* 1988.

[13] **Starken, H.**: Untersuchung der Strömung in ebenen Überschallverzögerungsgittern, *DLR-Forschungsbericht 71-99*., 1971.
[14] **Van Leer B.**: Flux Vector Splitting for the Euler Equations. *8th International Conference on Numerical Methods in Fluid Dynamics, Berlin Springer Verlag*, 1982.
[15] **Veress, Á., Bányai, T., Berke, P. and Bitvai, I.**: Numerical Simulation and Semi-Optimisation on 1.9 Orifice Fuel Jet-Pump. *BME, DAS-VISTEON Project Report 2003*, 23 January, 2003.
[16] **Wilcox D. C.**: Turbulence Modelling for CFD. *DCW Industries ISBN 0963605151*, 1998.
[17] **Zang Y., Street R. L., and Koseff J. R.**: A Dynamic Mixed Subgrid-Scale Model and Its Application to Turbulent Recirculating Flows. *Phys. Fluids A5 (12), pp. 3186-3196*, 1993.

Major Publications in the Subject

1) Veress, Á. – Braembussche, R.: Inverse Design of a Return Channel for a Multistage Centrifugal Compressor: **ASME Journal of Fluids Engineering**, Log Number 6069BS.
2) Veress, Á. – Sánta, I.: A 2D Mathematical Model on Transonic Axial Compressor Rotor Flow: Budapest University of Technology and Economics, **Periodica Polytechnica Transportation Engineering**, 2002 30/1-2, pp. 53-68.
3) Veress, Á.: Computational Investigation on Deswirl Vanes for Multistage Centrifugal Compressors, **Periodica Polytechnica Transportation Engineering** 2003 31/1-2, pp. 53-78.
4) Veress, Á.: Incompressible Flow Solver by Means of Pseudo-Compressibility Method, **Periodica Polytechnica Transportation Engineering** (www.pp.bme.hu)
5) Veress, Á.: Simplified Theory on Mathematical Model of Compressor Characteristic Working over Sound Speed, **22nd International Congress of Aeronautical Sciences** conference, Harrogate, Great Britain, ISBN 0 9533991 2 5, number 7.7.5, Optimage Ltd. 2000.
6) Veress, Á. - Sánta, I.: High Resolution Euler Solver for 2D Transonic Flow: **8th Mini Conferences on Vehicle System Dynamics, Identifications and Anomalies VSDIA'2002** conference**,** Budapest, 2002.
7) Veress, Á. – Braembussche, R.: New Approach to Radial Compressor Return Channel Design, **Conference on Modelling Fluid Flow (CMFF'03)**, The 12th International Conference on Fluid Flow Technologies, Budapest, Hungary, 2003.
8) Veress, Á.: Rumsey's Approximated Riemann Solver for 2D Transonic Axial Compressor Rotor Performance Map**, Conference on Modelling Fluid Flow (CMFF'03),** The 12th International Conference on Fluid Flow Technologies, Budapest, Hungary, 2003.

Industrial Reports

1) Veress, Á. – Bányai, T. – Bitvai, I. – Berke, P.: Kutatási Jelentés: Numerical Simulation and Semi-Optimisation on 1.9 Orifice Fuel Jet-Pump, **VISTEON Hungary Kft**., 8000 Székesfehérvár, Aszalvölgyi út 9-11. Tel.: (06-22) 530-122. 2003.
2) Veress, Á. – Bitvai, I. – Nagy, L. – Korody, E.: Kutatási Jelentés: Jaguar X400 Fuel Tank Internal Fuel Delivery System and Suction Type Jet Pump Analysis and Developments – Project, **VISTEON Hungary Kft**., 8000 Székesfehérvár, Aszalvölgyi út 9-11. Tel.: (06-22) 530-122. 2003.

Budapest, 06-01-2004 …..……………………..
Árpád VERESS