Calculation Process Development for Optimizing Geometry at Separation Flow Regime

ÁRPÁD VERESS
Department of Aircraft and Ships, Budapest University of Technology and Economics, HUNGARY

LÁSZLÓ PALKOVICS
Knorr Bremse R&D Centre Budapest, HUNGARY

A publikáció célja egy olyan számítási eljárás kidolgozásának és tesztelésének bemutatása, amely segítségével optimalizálható a leválásos áramlásban elhelyezkedő szilárd falak geometriai kialakítása. A módszer egy, a súrlódásmentes áramlás modellezésére és tervezésére alkalmas inverz eszköz kidolgozásán alapul, amely egy nulla gradiensű (belső áramlásra) és egy optimális fali nyomáseloszlást (külső, profil körüli áramlásra) alkalmaz bemeneti feltételként. Ez utóbbi esetben a nemlineáris és korlátos optimalizációs feladat megoldására az SQP (Sequential Quadratic Programming) eljárás szolgál, kiegészítve Stratford leválást becslő módszerével. A kétdimenziós numerikus áramlástani eljárás alapegyenletei az Euler egyenletek, amelyek diszkretizációjára egy cella központú véges térfogat módszer és Roe által közelített Riemann megoldó került alkalmazásra. Az optimalizációs módszer tesztelése belső és külső (profil körüli) áramlás figyelembevételével történt. Belső áramlás esetén nulla gradiensű nyomáseloszlás és külső áramlások esetén SQP alapú optimalizált nyomáseloszlás lett előírva az inverz tervező eszköz bemeneti feltételeként.

The goal of the present publication is to introduce a new surface-morphing technique which implies CFD (Computational Fluid Dynamics) and evolution-strategy-based inverse design method for the optimization of geometry at separation flow conditions in internal and external flows. The calculation process is based on an inviscid inverse design method. In order to have an optimum pressure distribution for the inverse design method, zero gradient and optimized pressure distributions are generated. A non-linear constrained SQP (Sequential Quadratic Programming) method coupled with Stratford's limiting flow theory is used in the second case. As a framework, a two-dimensional flow solver is developed to solve Euler equations numerically. They are discretized by a cell centred finite volume method with a Roe's approximated Riemann solver. The optimization process is tested over the internal and external flow applications. Zero pressure gradient flow conditions are imposed in case of internal flow and the SQP-based optimization is used for testing external flow application.

1 INTRODUCTION

The shape, thickness, curvature and diffusivity of different aerodynamic geometries or pipe systems as intake and exhaust manifolds, which are generally found in vehicle systems, have a strong influence not only on drag, loss, supply parameters and so the performance, but also on fuel consumption and emission. The pressure and velocity variation over the specific geometries are controlled by the solid walls, and these surfaces are not necessarily aligned with the flow. Several stagnation points, separations, chocked flow conditions can be evolved in the complex geometry. Hence, a surface-morphing method is going to be developed, tested and presented herein to improve design specifications by means of approaching the optimal wall shapes belonging to the previously imposed, favourable graduated pressure distribution.

Today, the optimization methods and especially the possibilities of its coupling with flow solvers are under intensive research due to the cost, capacity and time reduction of design and development processes. There are several direct methods which have already been implemented in different applications. They typically utilize some sort of search techniques (gradient-based optimizer), stochastic-based algorithms (e.g. evolutionary strategies, genetic algorithms), artificial neural networks or some other optimization methods. These procedures can be computationally expensive because several flow solutions have to be completed to determine the direction of deepest descent, fitness of individuals in the population, etc. in order to determine the shape changes. Furthermore, the required number of flow solutions increases dramatically with the number of design variables. Some interesting applications of CFD with different optimization methods are found in [1].

In case of a specific set of the inverse design-type methods [2 and 3], the geometry modification is based on the prescribed set of the pre-defined variables at the wall by simple, fast and robust algorithms, which makes them especially attractive amongst other optimization techniques. The wall modification can be completed within much less flow solutions for inverse design techniques than for direct optimization methods. Thus, the inverse design methods typically are much more computationally efficient and they are very innovative to be used in practice. The main drawback of inverse design methods is that the optimum pressure or velocity distributions are not available. Also, one cannot guarantee that an arbitrarily prescribed pressure/velocity distribution will provide mechanically correct airfoils, for example without negative thickness or opened geometry. Hence, the main goal of the present project, beside the testing of the capability of the inverse design method, is to develop a computational process, in which a specific optimization procedure results in target pressure distribution for the inverse design method and it provides the geometry belonging to target pressure distribution.

2 NUMERICAL METHOD

Due to the high-speed aeronautical applications with the assumption of no separation, the conservative form of the unsteady 2D compressible Euler equations has been used as a governing equations for flow modelling, which are given by (1) in Cartesian coordinate system in (x,y), where $x, y \in R$ and $t \in R^+$.

$$\frac{\partial U}{\partial t}+\frac{\partial F(U)}{\partial x}+\frac{\partial G(U)}{\partial y}=0 \tag{1}$$

Finite volume discretization method has been applied in the present report, by which the system of eq. (1) is integrated over a control volume Ω bounded by interface Γ and applying the Gauss divergence theorem gives,

$$\frac{\partial}{\partial t}\iint_{\Omega} U d\Omega + \int_{\Gamma} \bar{H}\bar{n} d\Gamma = 0\,, \tag{2}$$

where $\bar{n} = (n_x, n_y)$ is the local outward pointing unit normal vector, $\bar{H} = F\bar{e}_x + G\bar{e}_y$ and $\bar{H}\bar{n}$ is given by (3),

$$H_n = \bar{H}\bar{n} = \begin{pmatrix} \rho V_n \\ \rho u V_n + p n_x \\ \rho v V_n + p n_y \\ \rho V_n H \end{pmatrix}, \quad U = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{pmatrix}, \quad F(U) = \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho u v \\ \rho u H \end{pmatrix}$$

$$\text{and } G(U) = \begin{pmatrix} \rho v \\ \rho v u \\ \rho v^2 + p \\ \rho v H \end{pmatrix}. \tag{3}$$

In order to pass from continuous to a discrete form, a choice about the type of representation of the solution vector over the finite volume has to be made. Replacing the second integral by a summation over the number of faces N_f of the chosen control volume $\Omega_{i,j}$, eq. (2) can be written in the following semi-discrete form for the point i, j,

$$\frac{\partial}{\partial t} U_{i,j} = -\frac{1}{\Omega_{i,j}} \sum_{k=1}^{N_f} [H_n]_{i,j,k} \Gamma_{i,j,k} = \Re_{i,j}^n \tag{4}$$

where $[H_n]_{i,j,k}$ is the total inviscid flux normal to the cell interface with the length of $\Gamma_{i,j,k}$ cell boundary exchanged between points i, j and k. $\Re_{i,j}^n$ represents the residual, the scalar value of the line integral in (2). In case of upwind differencing schemes, the quantity $[H_n]_{i,j,k}$ is characterized by a flux function $\tilde{H}_n$ which takes into account the sign of the Jacobian matrices. Then, the $\tilde{H}_n(U^L, U^R)$ can be evaluated by a linear wave decomposition, where a unique average state denoted by a hat, originally developed by Roe, of the left and right states exist [4],

$$\tilde{H}_n(U^L, U^R) = \frac{1}{2}\left\{H_n(U^L) + H_n(U^R) - \left|\hat{D}_n(U^L, U^R)\right|(U^R - U^L)\right. \tag{5}$$

The method of Roe is highly non-dissipative and closely linked to the concept of characteristic transport. It is one of the most powerful linear Riemann solvers due to the excellent discontinuity-capturing property including shear waves. However, the entropy condition is not always satisfied. Hence, the method of Yee [5] has been used for entropy correction. The MUSCL (Monotone Upstream Schemes for Conservation Laws) approach is implemented for higher order spatial extension and Mulder limiter for monotonicity preserving [5]. For the minimum computational storage and the large stability range with the optimal choice of its parameters, the 4th order Runge-Kutta method is used to solve the time derivatives of the conservative variables. The coefficients of the scheme are derived to maximize the CFL number. Local time stepping has been used to optimize the time step behind the stability criterion.

3 BOUNDARY CONDITIONS

Concerning the boundary conditions, the theory of characteristics is used to determine the number of the physical and the numerical boundary conditions. In case of subsonic inlet condition there are three ingoing and one outgoing waves. Hence, 3 parameters; total pressure p^{to}, total temperature T^{o} and flow angle α are imposed as a physical boundary condition. Converting the characteristic problem into the cell normal and tangential direction, the $dn/dt=Vn-c$ curve has a negative slope, hence the fourth compatibility equation belongs to

$$\begin{pmatrix} \partial W_n^{(1)} \\ \partial W_n^{(2)} \\ \partial W_n^{(3)} \\ \partial W_n^{(4)} \end{pmatrix} = \begin{pmatrix} \partial p - c^2 \partial \rho = 0, \text{ on the curve } dn/dt = V_n \\ \partial V_s = 0, \quad \text{on the curve } dn/dt = V_n \\ \partial p + \rho c \partial V_n = 0, \text{on the curve } dn/dt = V_n + c \\ \partial p - \rho c \partial V_n = 0, \text{on the curve } dn/dt = V_n - c \end{pmatrix} \tag{6}$$

$\partial W_n^{(4)}$ in (6) is considered as a numerical boundary condition in its discretized form [6]. The combination of the energy and adiabatic Poisson sate equation is used to derive the second equations for missing variables. The Newton-Raphson method has been implemented to solve the system of two mentioned equations for pressure and normal velocity at the new time level. The static temperature and the components of the velocity vector are recovered by using ideal gas law and inlet flow angle, while the tangential velocity component is kept to be constant.

The characteristic method has also been applied at the subsonic outlet to determine the number and values of the unknown variables. In this case, there are three outgoing and one ingoing characteristics, hence the equations belonging to first three variable; $\partial W_n^{(1)}$ $\partial W_n^{(2)}$ and $\partial W_n^{(3)}$ in (6) are used in their discretized form. The static pressure at new time level $n+1$ is given by a physical boundary condition, hence the system of equations can be directly solved for density, normal velocity and tangential velocity at next time level. The static temperature is calculated by ideal gas law also.

The solid wall boundary conditions are considered as an outlet with the restriction of normal velocity is set to be zero across the wall. Hence, the equation belonging to $\partial W_n^{(1)}$ $\partial W_n^{(2)}$ and $\partial W_n^{(3)}$ in (6) are used to determine missing variables at the wall with $(V_n^{n+1} = 0)$. The static temperature is calculated by ideal gas law also.

Meanwhile the expected pressure distribution is imposed at the solid wall boundary in the inverse mode of the solver, the opening boundary is used instead of solid wall to control the local flow direction determined by the pressure difference between the boundary and computational domain. The main outcome of the present mode is to have velocity profile over the geometry, by which the wall is going to be changed in the wall modification module.

The method of characteristics is called upon also for determining unknown parameters at opening boundary. The curve $dn/dt=Vn-c$ is always the case as an outgoing characteristic, hence compatibility equation belonging to $\partial W_n^{(3)}$ in (6) is always considered. Two additional characteristics are considered, if the flow is outcoming ($V_n^{n+1} > 0$); $\partial W_n^{(1)}$ and $\partial W_n^{(2)}$ in (6). In case of incoming flow, the total pressure and total temperature are supposed to be constant over the entire flow field due to the adiabatic flow assumption, and these parameters are used to determine the static temperature and the magnitude of the velocity at the next time level. The density and tangential velocity are easily recovered by ideal gas law and Pythagoras rule, if the direction of tangential velocity is the same as in the previous time step.

The opening boundary condition is also used also at the remaining (bounding) domains (between the inlet and outlet BC's) far enough from the profile for example in case of external flows.

The validation of the described method is found in [7].

4 WALL MODIFICATION ALGORITHM

While the incoming and outcoming velocity distribution is given at the solid wall, based on the inverse mode of the analysis, the last step of the iterative design cycle is the modification of the geometry. The new position of the solid boundary coordinates are calculated by setting the wall parallel to the local velocity vector,

$$\Delta y_i(x_i) = \sum_{k=Le}^{i}\left(\frac{v_k}{u_k}\Delta x_k\right), \qquad (7)$$

where u and v are the Cartesian component of the velocity vector. The wall modification starts from the leading edge or inlet stagnation point till the trailing edge or the outlet stagnation point and completed in vertical directions [6].

5 APPLICATIONS OF THE INVERSE DESIGN-BASED OPTIMIZATION

It has been pointed out in the 1st chapter that the inverse design methods require optimal pressure or velocity distributions to determine the belonging geometry. Two different approaches are described. The simplest one is the zero pressure gradient for the internal flows. The second one is more complex, it is based on Stratford's experimental investigation [8] on separation prediction and SQP nonlinear constraint optimization algorithm, which can be applied for external flows like flow over a wing profile to maximise lift force.

Sinusoidal bump in channel test case has been used for testing the inverse design optimization method for internal flows. The rectangular computational domain is bounded by inlet, outlet and two opposed walls. A sinusoidal bump has been found at the first third part of the lower wall. The boundary conditions are the following; total inlet pressure: $p_{tot,in}$=110729 [Pa]; total inlet temperature: $T_{tot,i}n$=293.15 [K]; static outlet pressure: $p_{stat,out}$=101325 [Pa]. 100×40 rectangular mesh has been used. Zero pressure gradient is imposed as target distribution for the inverse method.

The test of inverse design method is completed successfully within 3 inverse cycles. The pressure distributions along the lower wall surface are shown in **Fig. 1.** The "target" (required) and "results after 3 inverse steps" pressure distributions cover each other, the differences are negligible in the distribution of pressure and geometry.

The maximisation of the lift force can be a goal function of the optimization in case of external flows like flow over a wing profile. The pressure distribution should be as low as possible over the solid surface of the suction side at given operational conditions.

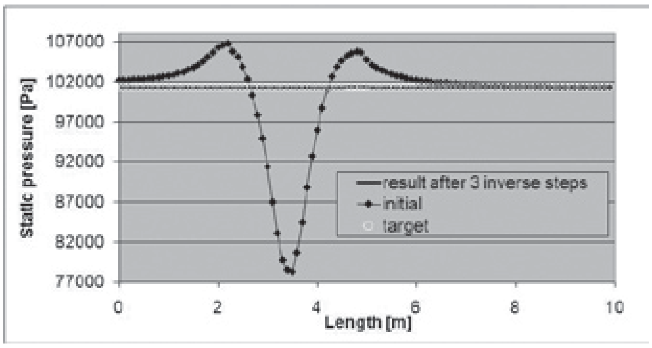

Figure 1: Pressure distribution along the lower solid wall of the flow channel for "initial", "target" and "result after 3 inverse steps" computational analysis and design.

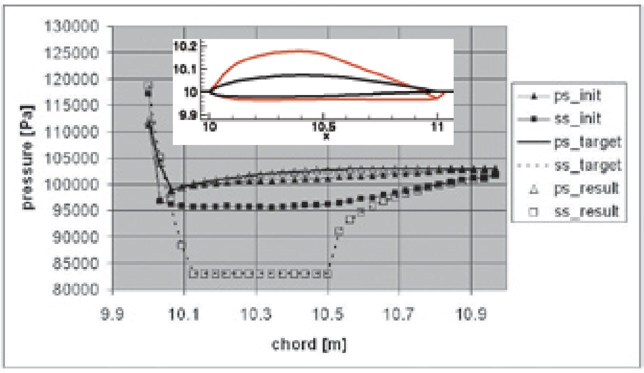

Figure 2: Pressure distribution of the initial (init), optimum (target) and result (of the inverse design based optimization procedure) cases (ss: suction side, ps: pressure side) with the geometry of dark (black) profile: original and light (red) profile: optimized versions

However, the adverse pressure gradient must present after the location of the maximum velocity and minimum pressure in order to recover downstream conditions. Stratford's flow limiting theory [8] is used coupling with the SQP nonlinear constraint optimization to provide such a pressure distribution, which gives the maximum lift force close to the separation. The results are shown in **Fig. 2.** The optimized geometry has improved design and off-design specifications.

6 CONCLUSION

A new computational procedure has been proposed for determining the optimal wall geometry at adverse pressure gradient flow conditions. The flow solver is based on the Euler equations, which are discretized by Roe's approximated Riemann method with MUSCL approach and Mulder limiter. The boundary conditions are based on the theory of characteristics. The optimal pressure distribution is determined by internal and external flows separately. A rectangular channel flow is used with a sinusoidal bump inside of it for internal flow and zero pressure gradient is imposed as optimal pressure distribution. An aerodynamic profile has been used for the external flow testing of the optimization method. In this case, the goal function of the SQP non-linear constraint optimization is the maximum area of the closed surface bounded by the suction and pressure side pressure distributions in the function of the chord length, meanwhile Stratford's limiting flow theory is used to evaluate pressure in each point of the suction side providing maximum flow deceleration close to the separation. Inverse design methods are finally successfully applied in both test cases to recover the geometry belonging to the required and previously imposed pressure distributions.

ACKNOWLEDGEMENTS

This work has been supported by the Hungarian National Fund for Science and Research (OTKA) under the fund No. F 67555.

REFERENCES

[1] D. Thévenin, G. Janiga: Optimization and Computational Fluid Dynamics, Hardcover, ISBN 978-3-540-72152-9, Springer-Verlag Berlin Heidelberg, 2008, Berlin

[2] A. Demeulenaere: An Euler/Navier-Stokes Inverse Method for Compressor and Turbine Blade Design, Von Kármán Institute for Fluid Dynamics, Inverse Design and Optimization Methods, Lecture Series (1997-05), 1-45

[3] L. De Vito, R. Van den Braembussche, H. Deconinck: A Novel Two-dimensional Viscous Inverse Design Method for Turbomachinery Blading, International Gas Turbine and Aeroengine Congress and Exhibition, Amsterdam, PAYS-BAS (03/06/2002) 2003, vol. 125, n2, pp. 310-316

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

[5] H. C. Yee: A class of high-resolution explicit and implicit shock-capturing methods, VKI lecture series 1989-04, March 6-10, 1989; NASA TM-101088, Feb. 1989

[6] O. Leonard, R. Van den Braembussche: Subsonic and Transonic Cascade Design, AGARD-VKI Special Course on Inverse Methods in Airfoil Design for Aeronautical and Turbomachinery Applications, May, 14-18, 1990

[7] Á. Veress, T. Gallina, J. Rohács: Fast and Robust Inverse Design Method for Internal and Cascade Flows, International Review of Aerospace Engineering, February issue, http://www.praiseworthyprize.com/irease.htm, 2010

[8] B. S. Stratford: The Prediction of Separation of the Turbulent Boundary Layer, Journal of Fluid Mechanics, Vol. 5. pp 1-16, 1959

AUTHOR DATA

Árpád Veress: Department of Aircraft and Ships, BME, Budapest, 1111 Sztoczek u. 6. J. ép. 426. Hungary. E-mail: averess@rht.bme.hu

László Palkovics: Knorr-Bremse R&D Centre Budapest, 1119, Major u. 69., Hungary. E-mail: laszlo.palkovics@knorr-bremse.com