3D panel methods for turbomachinery design

Ing. Vít Štorch

Vedoucí práce: Prof. Ing. Jiří Nožička, CSc.

Abstrakt

Panelové metody umožňují rychlé řešení potenciálního proudění kolem jednoduchých i složitějších geometrií. Jsou vhodné pro návrhové výpočty a optimalizační algoritmy, kde je kladen důraz na rychlost řešiče a kde není dominantní vliv disipace, viskozity a turbulence.
Práce se zabývá možnostmi a omezeními současných panelových metod v aplikaci na točivé stroje a popisuje vývoj vlastního výpočetního modelu založeného na Hessově formulaci 3D panelové metody upravené pro případy vnějšího a vnitřního proudění v okolí rotorů točivých strojů.

Keywords

3D Panel method, potential flow, vortex element method, turbomachinery, flow field calculation, wake modeling

1. Introduction

The paper on calculation of potential flow about arbitrary bodies published by Hess and Smith in 1962 [1] is considered a pioneering work in the field of 3D panel methods, in fact it can be considered a pioneering work in the whole CFD field as well. Since then, the panel methods have evolved, although the main principle – covering the surfaces with singularity elements – has remained unchanged. In the last two decades the development efforts have moved from panel methods to more advanced and computationally demanding methods such as LES and RANS solvers. The 3D panel method remains the tool of choice for wind turbine and propeller simulations, with its strong point being the ability to rapidly simulate large and complex wakes in external flow domains. The advantage of short computation time also leads to the panel methods being used for full optimization or geometry initiation for optimization using more advanced methods. Another advantage of panel methods is discretization by surface elements only. This means much faster and simpler meshing.

The obvious shortcomings of panel methods are associated with the properties of potential inviscid flow. Panel methods do not model viscosity, compressibility and vorticity. The method can be corrected for compressible cases to some degree using Prandtl-Glauert transformations [2] and viscosity can be artificially accounted for by boundary layer simulation, however with mixed results. Panel methods therefore give best results for streamlined bodies in high Reynolds number flows with thin boundary layer attached all the way to the trailing edge. The examples of their use can range from simulating aircraft wing-fuselage interaction to studying wake behind submarine propeller.

A low order panel method based on the Hess formulation is being developed by the author. The goal is to develop a set of scripts and functions in MATLAB® environment that enable fast simulation of flow field around propellers, wind turbine blades and in turbomachines. The information provided by the panel method includes pressure profiles, blade loading, total shaft power estimation (losses not included), velocities throughout the domain and the shape of the wake. The results can be used either for performance estimation during preliminary design process or for use with optimization algorithms, where the routine is run repeatedly.

2. Panel methods

The properties of potential flow can be expressed by Laplace's elliptic differential equation:

$$\nabla^2\phi = \Delta\phi = \frac{\partial^2\phi}{\partial x^2} + \frac{\partial^2\phi}{\partial y^2} + \frac{\partial^2\phi}{\partial z^2} = 0 \tag{1}$$

Its desirable properties are linearity and independence of frame of reference. Since the Laplace's equation does not include pressure term, it must be expressed using the Bernoulli equation in the form [3]:

$$gz + \frac{p}{\varrho} + \frac{c^2}{2} + \frac{\partial\phi}{\partial t} = Const.(t) \tag{2}$$

Boundary conditions are specified at the body surface and at infinity. The perturbation caused by the body must vanish towards the infinity and the surface of the body must be impermeable to the fluid. Mathematically this translates to the velocity being equal to free stream velocity at infinity and surface velocity component normal to the surface being equal to zero:

$$\lim_{r\to\infty} \vec{c} = \vec{c}_\infty, \qquad \frac{\partial\phi}{\partial n} = \nabla\phi\cdot\vec{n} = 0 \tag{3}$$

Because Laplace's equation is linear, any superposition of solutions is also a solution. The idea of panel methods is the superposition of elementary solutions of Laplace's equation that together satisfy the boundary condition. The commonly used elementary solutions include free stream, source, potential vortex and dipole (doublet). Except the first mentioned all solutions have a singularity point. For this reason the elementary solutions are often called singularities. In panel methods the singularities are distributed continuously over each panel forming the body surface. Depending on the order of the panel method the distribution may be constant, linear or quadratic. The surface boundary condition is evaluated at colocation points usually placed in the middle of each panel resulting in one equation to be solved for each colocation point. The boundary condition at infinity is satisfied automatically by each singularity and therefore is satisfied by their linear combination.

The unknowns are the singularity strengths (i.e source strength, circulation etc.). The potential and velocity at a colocation point is the result of superposition of influences of all panels and the free stream. The influence of one panel on a collocation point is expressed by a product of singularity strength and a coefficient dependent solely on geometry. These coefficients are called influence coefficients.

3. Low order 3D panel method developed for turbomachinery design

Since there are several formulations of panel methods, a thorough research preceded the decision on exact type of the described panel method. The low order quadrilateral element Hess formulation was used. Unlike Morino formulation [4] based on potential with modified boundary conditions the Hess family of panel methods [5] is based on velocity formulation and direct implementation of BC. The influence coefficient of a j-th panel on an i-th colocation point has three components U_{ij}, V_{ij}, W_{ij}. Its direct meaning is the velocity vector induced by the panel at the collocation point by a unit "1" singularity strength. This concept has the advantage of having the velocity field readily available without having to calculate gradients of potential, the disadvantage being the use of three times as many influence coefficients, which has some impact on calculation time. The equation resulting from this formulation follows:

$$\{\sum_j([U_{ij},V_{ij},W_{ij}]\cdot ss_j)+\vec{c}_\infty\}\cdot\vec{n}_i=0 \tag{4}$$

Where ss_j is the singularity strength of j-th panel.

The developed panel method is low order, meaning constant distribution of singularities over each panel and flat panels without curvature. The results of panel methods developed by other authors suggest, that while higher-order panel methods give better precision with given number of panels, the computation time is affected heavily. For the same precision, more low order panels are needed, however the computation time for low order panel method will be generally still lower.

Another important aspect to consider is the type of singularity used. For lifting flow, either doublet or vortex filament panels are required to produce the circulation. For symmetric non-lifting flows, source panels are sufficient. It was proven by Hess [5] that a flat quadrilateral panel with constant doublet distribution over its surface is equivalent to a panel with vortex filaments along its edges (vortex ring panel).

Originally it was decided to use only vortex ring elements for the described panel method. Commercial programs mostly use the combination of source panels and doublets or vortex rings. While from theory vortex rings are sufficient, it was proven by author's trial and error that addition of source elements increases precision significantly, especially close to the trailing edges. The source strengths cannot be unknowns since the system of equations would be over-defined (twice the needed equations). The source strengths can be defined in such a way, that the source panel takes on most of the perturbation needed to satisfy the boundary condition. The solver PMARC [6] fixes the source strength σ of panels to:

$$\sigma=\frac{1}{4\pi}(c_n-\vec{n}\cdot\vec{c}_\infty) \tag{5}$$

Where c_n is the normal velocity at panel surface. For solid surface $c_n=0$, for inlet or outlet surfaces it is a prescribed value. The reason why this addition of source panels increases precision is probably the fact that circulation values of vortex ring panels are generally lower and with smaller gradients, which leads to better accuracy of calculation of self-induced influence.

The formulas for influence coefficients for constant source panels are quite complex and can be found in [5]. The vortex ring on the other hand consists of 4 vortex filaments, for which Biot-Savart law can be used [3]:

$$[U_{ij},V_{ij},W_{ij}]=\frac{\Gamma}{4\pi}\frac{\vec{r}_1\times\vec{r}_2}{|\vec{r}_1\times\vec{r}_2|^2}\vec{r}_0\left(\frac{\vec{r}_1}{|\vec{r}_1|}-\frac{\vec{r}_2}{|\vec{r}_2|}\right) \tag{6}$$

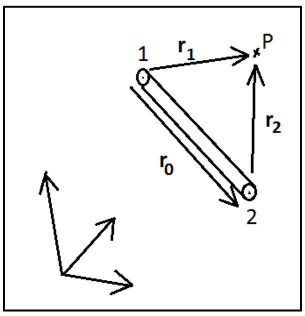

Fig.1 *– Vortex filament segment and its influence on point P.*

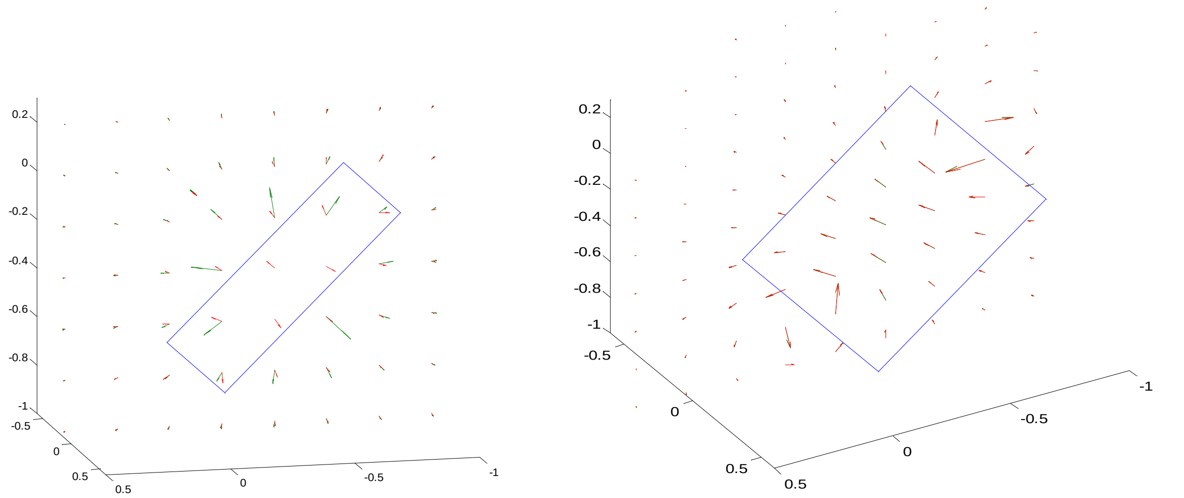

Fig.2 *– Flow field around panels. Left: Source panel. Green color represents velocity by point source in the middle of panel, red color represents velocity induced by continuous distribution of source across the panel. Right: Vortex ring panel and its induced velocity .*

3.1 Kutta condition and wake treatment

The problem of lift cannot be solved by potential flow alone. Viscosity prevents the flow reaching infinite velocity around trailing edge and separating on the upper surface of the airfoil. Kutta condition solves this problem by stating that the flow must separate smoothly at the downstream sharp edge – usually the trailing edge. There are many formulations of Kutta condition only some of which are suitable for panel methods. The formulation selected for described 3D panel method is zero trailing edge circulation. The trailing edge consists of two filaments, one belonging to the upper surface vortex panel, the other to the lower surface vortex panel. The circulation at the trailing edge is the difference of upper and lower panel circulation. This circulation can be canceled by addition of third panel – wake panel with the proper circulation:

$$\gamma_w = \gamma_1 - \gamma_2 \tag{7}$$

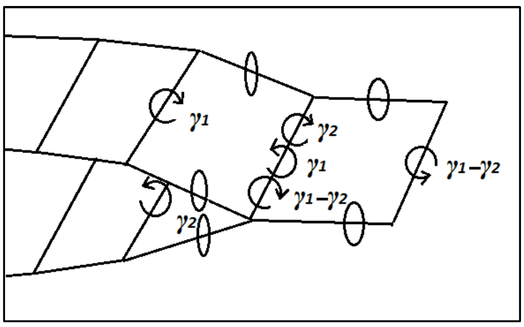

Fig.3 *– Kutta condition.*

The circulation generated on blade (wing) is shed into the wake. The wake geometry is unknown. There are generally two approaches to modeling wake. Its shape can be either fixed according to experience or best guess, or dynamically changing to form a force-free wake, which is always parallel to local velocity. For the described method, force-free wake alignment algorithm is proposed. The wake is generated with fixed geometry and then aligned iteratively with the local flow.

3.2. Practical considerations

When implementing the numeric schemes, there are many aspects that influence the precision of the results. If proper care is not taken when creating mesh, the results may become inaccurate or completely misleading. The panels should be with little skew and low aspect ratios. The density of panels should not change rapidly over the surface. If possible, the panels should form a structured grid with center points aligned parallel and perpendicular to the flow. For proper Kutta condition evaluation, the top and bottom trailing edge panels should be of the same size. In the case of internal flow the quality of mesh is even of greater importance. The panel leakage is an important issue when dealing with internal flow using panel methods. The influence coefficients must be calculated with full precision without approximations for internal flow. Far field condition which treats distant panels as point sources/doublets must not be used.

The modifications of the panel code to minimize negative effects degrading results of 3D panel methods applied on internal flow turbomachinery is currently one of the author’s key interest and are being researched.

4. Results

The method was first tested on a simple rectangular wing with NACA 4412 airfoil and aspect ratio equal to 4 and 10° angle of attack.

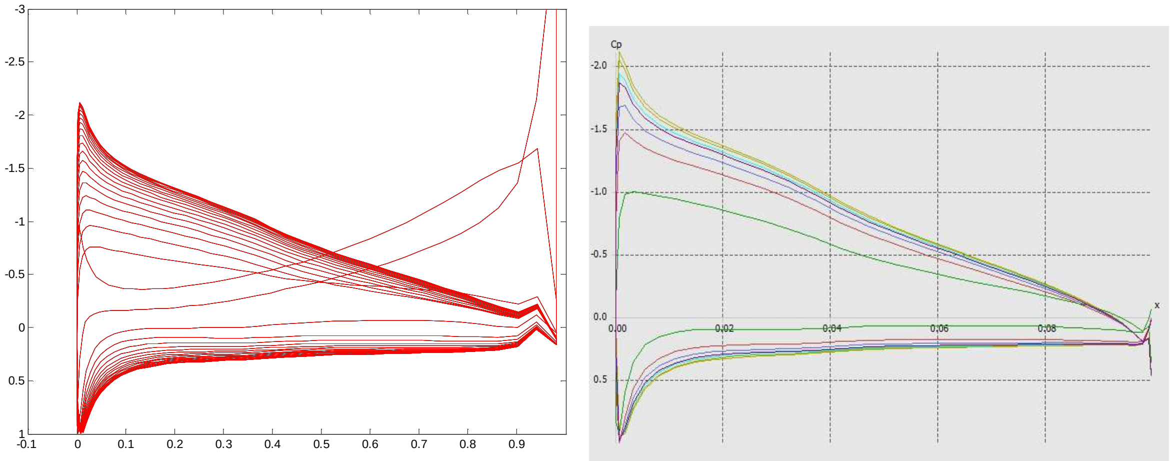

Fig.4 *– Pressure coefficient at different cross sections of NACA 4412 airfoil finite wing under 10° angle of attack. Left: Author’s panel code, Right: XFLR5 software*

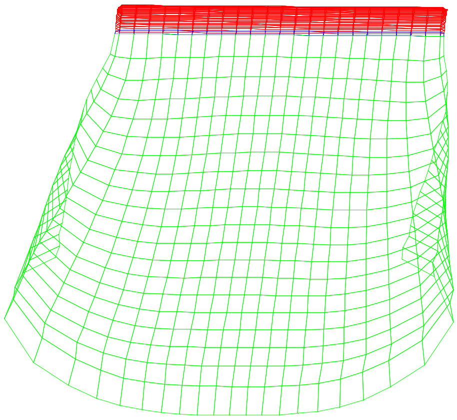

Fig.5 *– Wake formation after 20 iterations behind the NACA 4412 airfoil wing.*

The testing of the panel code continued with simulating the propeller designed and experimentally tested at the Department of Fluid Dynamics and thermodynamics, Czech technical university in Prague. The results showed good agreement regarding the thrust coefficient. The power coefficient was underestimated by the panel method in comparison to the experiment. This can be attributed to poor drag estimation which is common to all panel methods due to inviscid vorticity-free model.

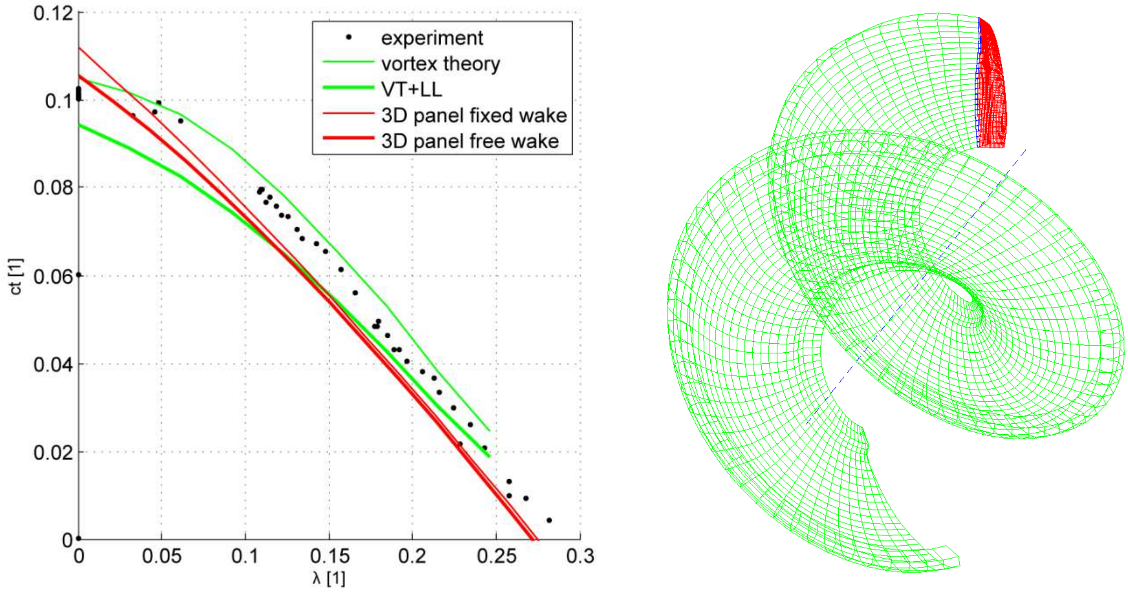

Fig.6 *– Left: Comparison of thrust according to simulation and experiment. Right: Wake formation behind single rotating blade*

Current efforts are focused on the third test case, a Kaplan turbine in a cylindrical pipe. The main issues with this model are panel leakage, wake-wall interactions and blade-wall intersection degrading precision.

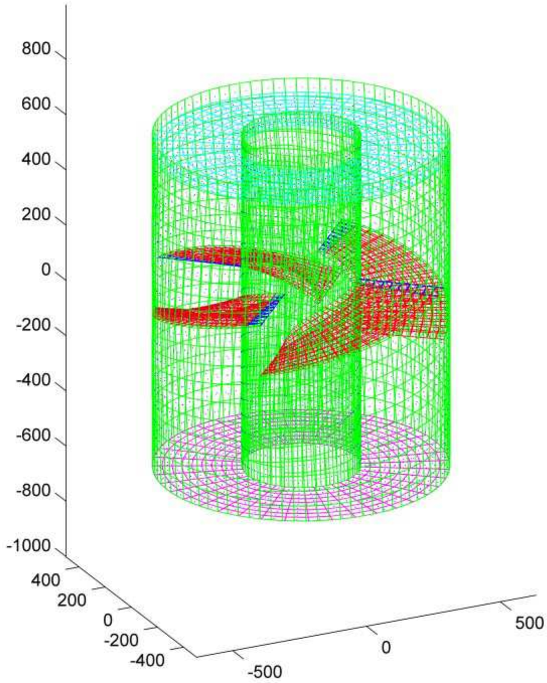

Fig.7 *– Mesh of the third test case – Kaplan turbine in an internal domain*

5. Conclusion

A 3D panel code has been developed for calculation of flow field around blades. The low order panel method based on Hess formulation uses combination of vortex ring and source panels with constant strength distributions. The method has been validated to give reliable results with standard external flow cases. Without much modifications, it can be used for evaluation of most geometries under steady flow conditions. Solving unsteady problems would require only minor changes.

The focus of current work is on increasing internal flow calculation precision. The effects of mesh density, inlet/outlet definition, free-stream velocity and wake are researched. There are not many available free or commercial panel codes optimized (or even able to handle) rotating blades in external flow. There are even less panel codes that solve internal flow problems. To the author's knowledge there is no 3D panel method available, that would solve turbines or ducted propellers. This is also one of the main motivations for the current work.

Future work will use the developed code for optimization. One such simple optimization algorithm based on random parameter modifications has been already tested on 2D airfoil panel code with good results.

List of symbols

c	velocity	(m/s)
g	gravitational acceleration	(m/s^2)
n	surface or panel normal	(m)
r	radiusvector	(m)
t	time	(s)
x,y,z	coordinates	(m)
γ	vortex ring circulation per unit filament length	(m/s)
ρ	density	($kg \cdot m^{-3}$)

σ	Source strength per unit area	(1/s)
ϕ	Potential	(m^2/s)

References

[1] A. Smith and J. Hess. Calculation of non-lifting potential flow about arbitrary three-dimensional bodies. Technical Report E.S. 40622, Douglas Aircraft Company, Inc., Long Beach, CA, 1962.

[2] H. Glauert, The Effect of Compressibility on the Lift of an Aerofoil. Proc. Roy. Soc. London. VOL. CXVIII, 1928, p. 113–119.

[3] J. Katz and A. Plotkin. Low-Speed Aerodynamics. Cambridge Aerospace Series. Cambridge University Press, 2001. ISBN 9780521665520.

[4] L. Morino and C. Kuot. Subsonic potential aerodynamics for complex configurations: A general theory. AIAA Journal, 12:191–197, 1974.

[5] J. Hess. Calculation of potential flow about arbitrary three dimensional lifting bodies. Technical Report MDC J5679-01, McDonnell Douglas Corporation, Long Beach, CA, 1972.

[6] D. Ashby. Potential flow theory and operation guide for the panel code PMARC_14. NASA, Ames Research Center, Moffet Field, CA, December 1999. TM-1999-209582.

Acknowledgements: This work was supported by the Grant Agency of the Czech Technical University in Prague, grant No. SGS14/057/OHK2/1T/12