RATIONALITY OF SOLVING 3D CIRCULATION PROBLEMS EXCLUSIVELY WITH THE USE OF THE NAVIER-STOKES EQUATION (REYNOLDS EQUATION)

TADEUSZ KORONOWICZ

Institute of Fluid Flow Machinery,
Polish Academy of Sciences,
Fiszera 14, 80-952 Gdansk, Poland
ttk@imp.gda.pl

(Received 3 August 2001; revised manuscript received 20 November 2001)

Abstract: The paper presents problems connected with solving 3D circulation problems. The existence of vortex singularities in the flow behind a body is a characteristic feature of such problems. The vortices influence the velocity and pressure fields in the vicinity of a body. A proposition of rational merging the methods based on the N-S equation together with vortex methods (based on the vorticity equation) is discussed. Such a connection enables accurate and efficient determination of vortex singularities in the flow behind a body.

Keywords: 3D circulation problems, vortex theory, Navier-Stokes equation

1. Introduction

A free foil, whose mean lines are characterised with nonzero camber, placed in uniform flux in infinite space and fixed with a certain angle of attack may be considered as the simplest case of a 3D circulation problem. A lift force L is induced on the foil (Figure 1).

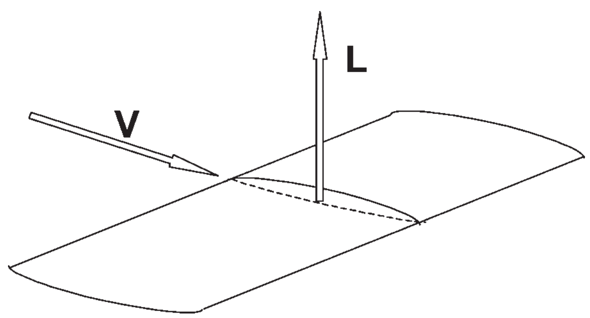

Figure 1. A lift force induced on the foil

Airplane wing, screw propeller blade, axial-flow pump blade, water or steam turbine blades, compressor blade, fan blade and guiding vane, *etc.* serve as the examples.

Circulation around the foil (guiding vane) section, as well as variation of circulation along the foil span are the most remarkable features of such flows (3D flows).

The characteristic feature of such a flow is that a vortex singularity appears behind a foil (blade). Within the classical approach the singularity is modeled as a non-deformable vortex layer [1–3]. In reality such a layer deforms, winds up, creates concentrated structures and dissipates simultaneously (Figure 2), [2, 4, 5].

Within the simplest, classical approach the presence of free vortices influences the velocity and pressure fields on the foil (blade) surface significantly. The following example may serve as the confirmation of the above. The lift coefficient for rectangular foil, characterised by aspect ratio $\lambda = 3$ and positioned with 3° angle of attack calculated neglecting free vortices equals $C_L = 0.63$. Pressure coefficient distributions along particular foil sections are presented in Figures 3a and 3b by solid lines.

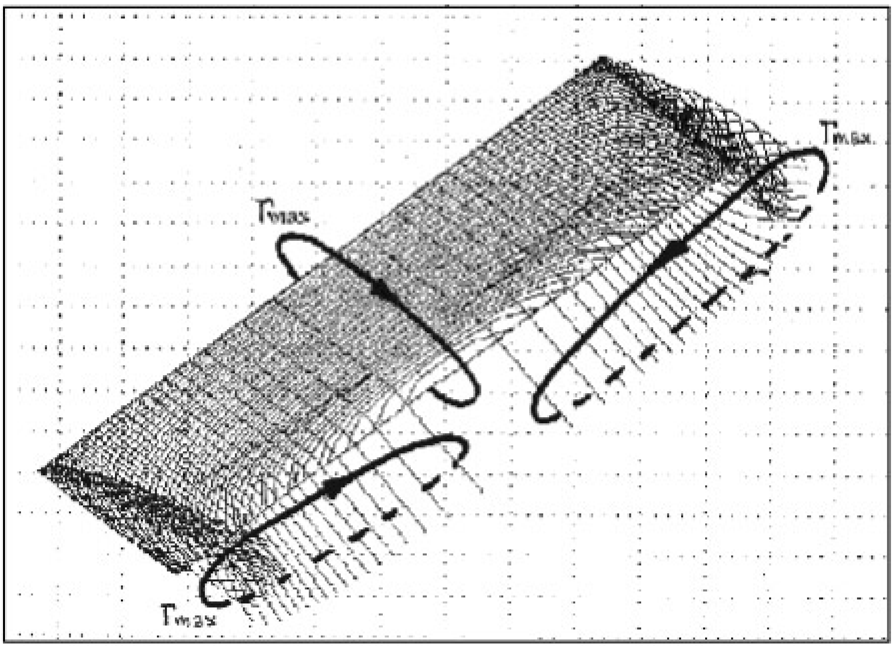

Figure 2. Vortex lines around the foil in real flow

The calculations carried out for the same foil and free vortices taken into account (as in the classical approach) provide the value $C_L \cong 0.50$. Velocity distributions corresponding to identical foil sections are marked in Figures 3a and 3b by dashed lines. The differences are distinct.

Enabling the separation of the vortex layer from the blade surface (double layer lifting surface model) brings the model closer to reality [6]. A vortex "pocket" appears close to the blade tip. The flow around the sections in the blade tip vicinity is significantly influenced by deforming vortex structures. It is not only the matter

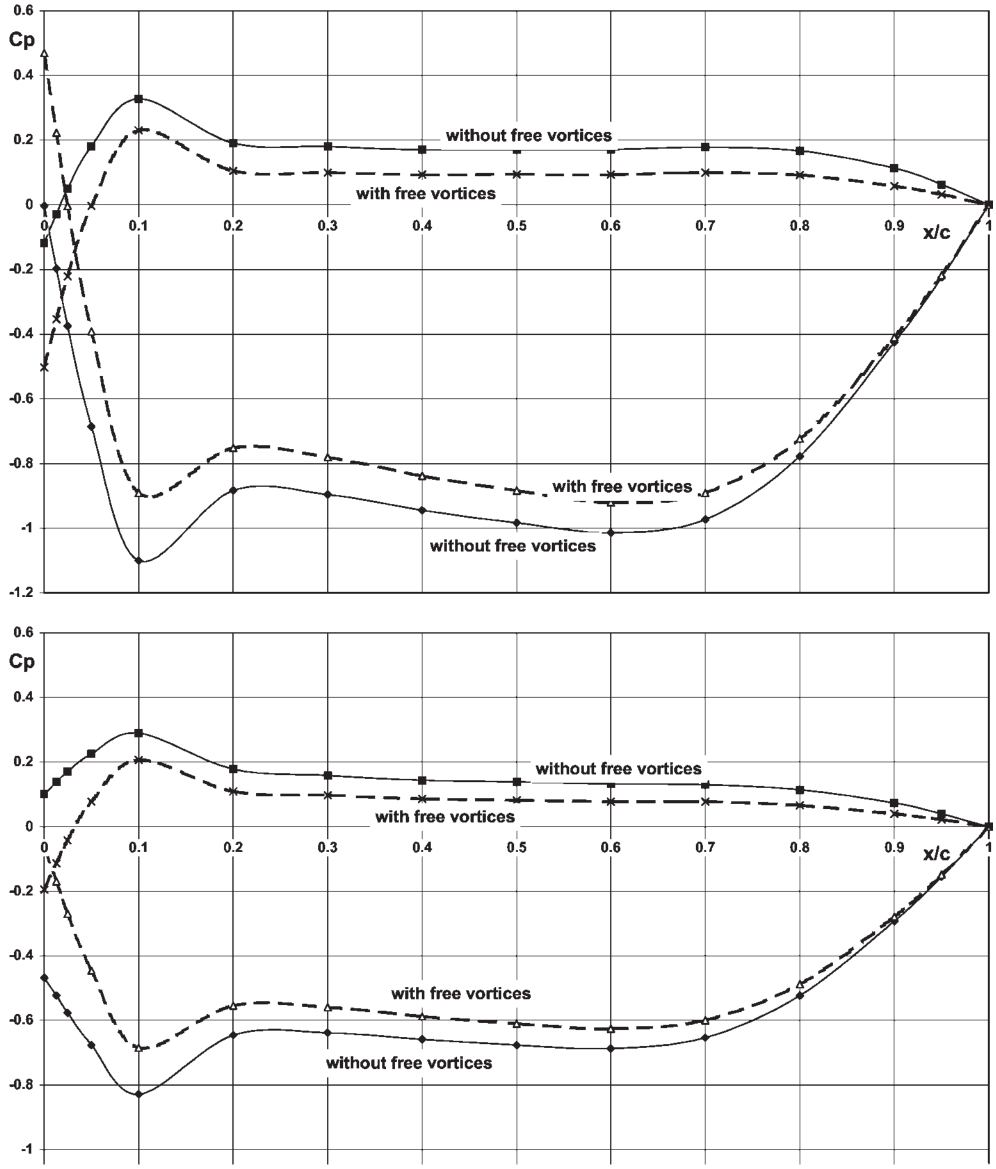

Figure 3. Pressure coefficient distributions along foil sections with or without free vortices at mid-span and 40% from the tip

of the calculation model. The experimental investigation of vorticity behind a blade indicates the appearance of such regions containing strong vorticity. The results of measurements behind the blade tip are presented in Figure 4. The measurements were carried out at the Institute of Fluid Flow Machinery (Gdansk, Poland) in the cavitation tunnel with the use of a rotameter. The rotameter's vane diameter was equal to 3mm. Within the section distanced $x = 0.145$mm from the rectangular blade (breadth $c = 0.12$m, length $L = 0.26$m) axis, the vorticity field was obtained. The maximum vorticity value was equal to $\omega = 50$ 1/sec.

The theorem concerning vortex flow stability (the second Helmholtz law) indicates the strict interdependence between the vorticity attached to the blade and vorticity shedding from the blade [1, 4]. Each variation of the circulation around a foil section manifests in the form of free vortices. The maximum circulation around a foil (blade) equals the circulation around the set of trailing edge free vortices (Figure 2). The flows around tips and between the pressure and suction side close to the fluid flow machinery blade root are called the secondary flows and correspond to the same physical mechanism.

It is not enough to know the vorticity field $\vec{\Omega}(\vec{r})$ behind a body. This field induces the velocity field in front, as well as in the vicinity of the considered body (foil, blade). As it is presented in Figure 3, the differences are substantial, even when classical approach concerning bounded vorticity is applied. As far as screw propeller theory is considered the capability of calculation of the velocity field due to free vortices determines the accuracy of pressure field prediction on the propeller blade. Consequently it determines also the reliability of cavitation and acoustic pressure field predictions.

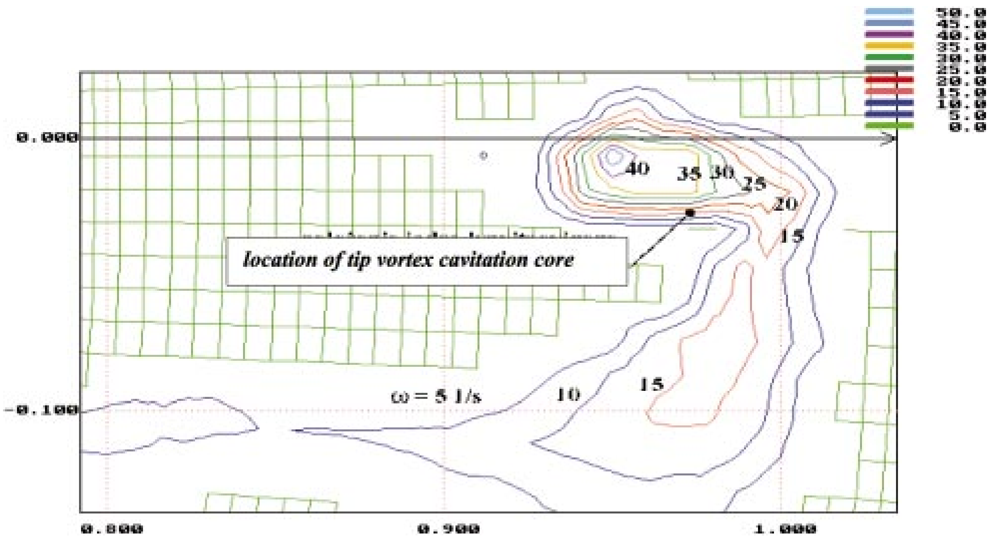

Figure 4. The results of measurements of vorticity (Ω [1/sec]) behind the blade tip

2. Solution of 3D problems with circulation

In order to obtain accurate results of calculations based on the N-S equation, boundary conditions have to be precisely defined. The presence of free vorticity set behind a body determines velocity and pressure fields in the sections behind it. For example, such a set of free vortices, for which the field of $\vec{\Omega}(x,y)$ in a certain section is presented in Figure 4 alternates significantly the surrounding pressure and velocity fields. This field varies with the distance from the body, because the set of free vortices undergoes certain deformation and simultaneously dissipation. The specification of boundary conditions at the presence of free vortices basing only on solutions of the N-S equations causes difficulties and is considerably time-consuming. The solution can be obtained only by way of successive approximations when the vorticity field $\vec{\Omega}(\vec{r})$ is properly specified (the theorem concerning stability of rotational motion has to be taken into account, *i.e.*: in incompressible, inviscid fluid without walls and bodies flowed around the vorticity cannot rise and the decay of vorticity is realized by the vortices cascade. The possibility of obtaining the correct solution applying the methods based exclusively on the Navier-Stokes equations or their modified form – the Reynolds equations cannot be negated. The attention should be focused on rationality of such methods. In the case of 3D circulation problems it can be more reasonable to merge together the methods based on the N-S equations and vortex methods. Namely for the 3D problems with circulation vortex methods may serve as the first approximation. Further calculations can be carried out basing on the N-S equations, taking advantage of the already determined set of vortices.

Vortex methods – prevailing during design calculations and determination of hydrodynamic, cavitation or acoustic characteristics of screw propellers are considerably effective and accurate. The accurate design of screw propeller can be carried out on a PC in seconds and the analysis of propeller operation in a non-uniform velocity field can be done in few minutes. More complicated calculations concerning tip vortices and associated with them hydroacoustic pressure usually take less than twenty minutes.

3. Theoretical background of vortex method

The vortex methods in general are based on vorticity equation, which is a form of the N-S equation, subjected to the rotation operator. The vorticity equation for viscous, incompressible liquid, including the body force potential gains the following form [1, 2, 4, 3]:

$$\frac{\partial\vec{\Omega}}{\partial t}+\mathrm{rot}(\vec{\Omega}\times\vec{V})=\frac{\partial\vec{\Omega}}{\partial t}+(\vec{V}\cdot\nabla)\vec{\Omega}-(\vec{\Omega}\cdot\nabla)\vec{V}=\nu\Delta\vec{\Omega}, \qquad (1)$$

where $\vec{\Omega}=\mathrm{rot}\vec{V}$. The considered domain is the vorticity field $\vec{\Omega}(\vec{r},t)$.

The molecular vorticity diffusion term $\nu\Delta\vec{\Omega}$ (vorticity dissipation) has to be taken into account in the above equation only within the regions of high vorticity concentration [7], for example where free vortices form concentrated structures. When such a concentration does not appear, this term can be neglected and the third Helmholtz equation can be obtained:

$$\frac{d\vec{\Omega}}{dt}-(\vec{\Omega}\cdot\nabla)\vec{V}=0. \qquad (2)$$

Combining the above equation with the Stokes equation

$$\oint_l \vec{V}d\vec{l}=\int_S \vec{n}\cdot\mathrm{rot}\vec{V}dS \qquad (3)$$

enables evaluation of the series of theorems concerning velocity field – Helmholtz and Kelvin. Those equations create the base for development of vortex models.

In general the vortex model consists in determination of the distribution of vorticity in the liquid, which represents the foils (blades) and vortex wake behind them. Such a time- and space-dependent distribution serves as the starting point for determination of the induced velocity field.

If at any arbitrary moment $t=t_1$, a spatial distribution of vorticity $\vec{\Omega}(\vec{r}_1 t_1)$ in a liquid volume is known, then the induced velocity field can be determined by way of applying vector analysis laws. The following formula can be obtained:

$$\vec{V}=\frac{1}{2\pi}\mathrm{rot}\int_{\tau}\frac{\vec{\Omega}(\vec{r})}{r}d\tau. \tag{4}$$

When the vorticity distribution is given in the form of vortex filaments distribution:

$$\vec{\Omega}(\vec{r})d\tau=\Gamma d\vec{l}, \tag{5}$$

then the Biot-Savart law (6) can be derived from Equation (4). This law is widely applied in vortex methods:

$$\vec{V}=\frac{\Gamma}{4\pi}\int_{L}\frac{d\vec{l}\times\vec{r}}{r^3}. \tag{6}$$

According to circumstances vortex models are more or less complicated. Within the classical screw propeller theory approach (single layer lifting surface) the model consists of the set of bounded vortices distributed over the surface created by chords or mean lines of blade sections and the set of helicoidal free vortices. More complex approach (double layer lifting surface model) presumes that vortices are distributed both on pressure and on suction sides of a blade. The system of free vortices undergoes deformation as well as free vortices can detach from the blade surface.

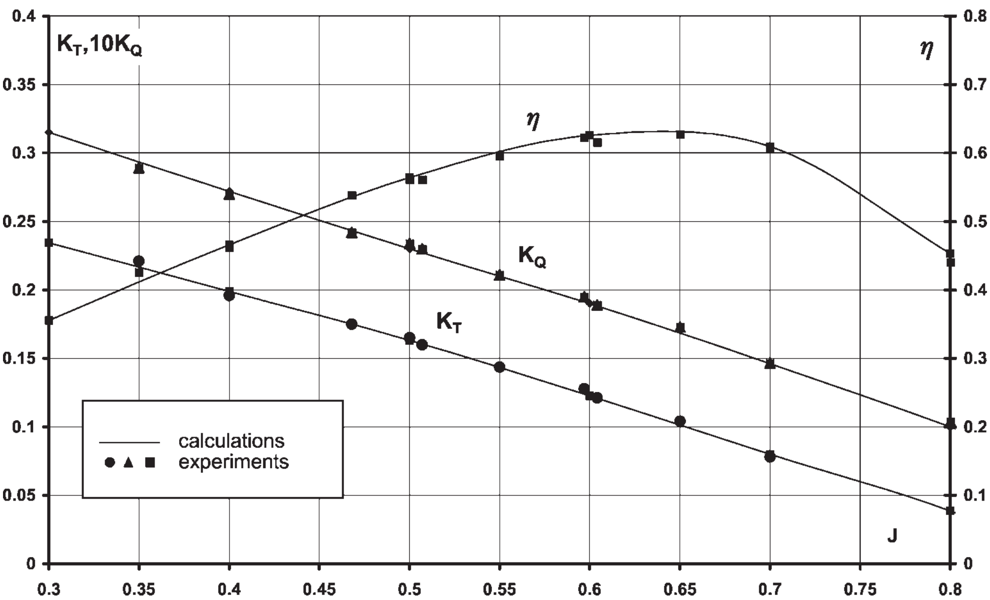

Figure 5. Hydrodynamic characteristics of screw propeller K_T – thrust coefficient; K_Q – torque coefficient; η – efficiency

The considered flow around propeller blades or elements of fluid flow machines is characterized by high Reynolds numbers. The boundary layer at the design operation point is thin and there is no viscous flow separation. In such a case, modeling of the boundary layer with the thin vortex surface does not introduce substantial errors during calculation of the velocity field (*e.g.* of normal velocities on blades, when Neumann's boundary condition is applied). Therefore vortex methods provide valuable results concerning screw propellers. The accuracy of design calculations is verified by way of experimental model testing (more than ten such tests are carried out annually in Poland). The error concerning global thrust or torque is usually less than 1% (Figure 5). It confirms the practical efficiency of vortex methods.

Therefore it seems reasonable that, when it is necessary to determine velocity field within the boundary layer on a foil (blade) accurately and solve a 3D circulation problem, the application of the N-S equations methods can be preceded by the use of vortex method. This first step of velocity field analysis serves as the rational first approximation. Further calculations, with already evaluated induced velocities taken into account can be carried out on the basies of N-S methods.

4. Concluding remarks

In order to solve 3D circulation problems it seems advantageous to merge together the two following methods:

- based on the N-S equation (operating within the velocity field domain),
- based on the vorticity equation (operating within the vorticity field domain).

Such a symbiosis is much more desirable in the case of 3D circulation design problems. It is possible to determine the blade shape (including sections geometry) providing the appropriate pressure distribution with the use of vortex methods exclusively. Viscosity can be taken into consideration in the further step of calculations based on the N-S equations. The system of free vortices can be obtained during the initial, preliminary step of calculations.

Therefore it is impossible to agree with categorical statements that the methods applying vortex equation can be successfully replaced by the methods based exclusively on the N-S equations. Such efforts are irrational.

It should be emphasized that viscous drag of propeller blades at the design operation point usually equals to 2%−4% of the total induced lift force. Therefore the main effort is directed towards accurate determination of thrust. For this purpose vortex methods serve as excellent tools. Viscous drag can be determined with the use of approximate methods based on empirical knowledge or applying integral methods. Long-term practice indicates that the application of numerical algorithms based on vortex models enables determination of screw propeller performance with error less than 1%. Moreover such a phenomenon as cavitation can be predicted with high accuracy.

I am convinced that the calculation methods elaborated for screw propellers can be successfully applied to other fluid flow machines. Whenever the more exact knowledge about velocity field is necessary (within the boundary layer) it is possible to continue calculations with the use of N-S equations methods.

References

[1] Durand W F 1963 *Aerodynamics Theory*, Dover Publ.

[2] Koczin N J, Kibel J A and Rożе N W 1956 *Theoretical Hydromechanics*, Moscow (in Russian)
[3] Prosnak W J 1971 *Fluid Mechanics*, PWN Warsaw (in Polish)
[4] Potter D 1971 *Computational Physics*, PWN Warsaw (in Polish)
[5] Twaites B 1960 *Incompressible Aerodynamics*, Oxford
[6] Koronowicz T, Tuszkowska T and Wilczyński L 2000 *TASK Quarterly* **4** (2) 203
[7] Batchelor G K 1970 *An Introduction to Fluid Dynamics*, Cambridge Univ. Press