PAMM · Proc. Appl. Math. Mech. **13**, 215 – 216 (2013) / **DOI** 10.1002/pamm.201310103

Simulation of a membrane flow interaction by an iteration based on a panel method

Herbert Steinrück[1,*] and **Markus Müllner**[1]

[1] Vienna University of Technology, Institute of Fluid Mechanics and Heat Transfer, Resselgasse 3, 1040 Vienna, Austria

A flow structure interaction of a membrane and a fluid is investigated. A conventional segregated numerical algorithm, where the membrane deformation and the flow dynamics are calculated alternately has to fail due to the artificial added mass instability. Thus, a new iteration scheme is proposed. In order to get a good prediction for the deformation of the membrane, the equations describing the membrane are coupled to a potential flow solver (panel method). Then a CFD solver can be used to determine the corrections of the flow field due viscosity and turbulence. An example is presented that this procedure seems to be numerically stable.

1 Introduction

The goal of the present study is the investigation of the working mechanism of a double channel membrane pump shown in figure 1. A channel of width $2h$ is divided in its midplane by a membrane of length l_m into two half channels. On the inflow side the membrane is excited periodically with frequency f and amplitude a by a movable flap of length l_f. Thus waves traveling along the membrane are initiated resulting in a net mass flow of the fluid with denstiy ρ and viscosity μ, [1].

In most possible applications the amplitudes of the membrane and the Reynolds numbers are large, such that a numerical simulation seems to be inevitable. However, it is well known that a fluid structure interaction of slender light structure with fluid is only possible using a monolithic approach, where in every time step the deformation of the membrane and the fluid flow is determined simultaneously. Segregated approaches, where the equations for the deformation and the fluid flow are solved alternatively, fail due to the so called “artificial added mass instability”, see [2].

The basic idea of the current approach is to use monolithic approach, but using a simpler flow model (potential flow for small amplitudes) and then iterate using a CFD solver to take large amplitudes, viscosity, turbulence and flow separation into account.

In section 2 we describe the potential flow in a 2D channel, in section 3 the coupling strategy is described and section 4 an example is shown.

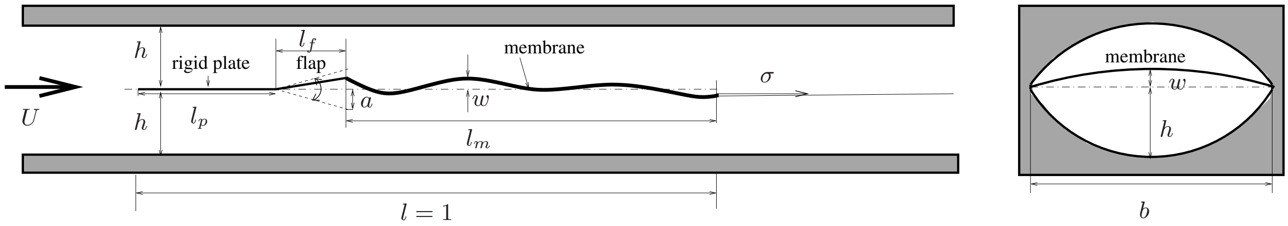


Fig. 1: The membrane in a channel

2 Potential flow theory

We consider a two dimensional inviscid potential flow in channel of width $2h$ with a movable membrane in near its midplane. The deformation of the membrane $w(x,t)$ is assumed to be small such that the kinematic boundary condition can be linearized. We describe the potential flow field by a vortex distribution $\gamma = \gamma(x,t)$ along the membrane in its rest position, the x-axis, see [3]. Of course a vortex distribution γ_w in the wake is also necessary. Let $v^{(P)}$ be the vertical velocity component induced by γ. Note $v^{(P)}$ has to be determined from the kinematic boundary condition taking the membrane motion and the wake induced flow field into account.

Then the pressure difference Δp across the membrane is given by [4].

$$\Delta p_P(x,t) = \rho_\infty \left(\int_0^x \gamma_t(\xi,t)\mathrm{d}\xi + U_\infty \gamma(x,t) \right), \qquad \gamma(x,t) = \frac{2}{\delta_h} \sqrt{\frac{\sinh \frac{\pi(l-x)}{\delta_h}}{\sinh \frac{\pi x}{\delta_h}}} \int_0^l \frac{v^{(P)}(\xi,0,t)}{\sinh \frac{\pi(\xi-x)}{\delta_h}} \sqrt{\frac{\sinh \frac{\pi\xi)}{\delta_h}}{\sinh \frac{\pi(l-\xi)}{\delta_h}}} \mathrm{d}\xi.$$

---

[*] Corresponding author E-mail: herbert.steinrueck@tuwien.ac.at, Phone: +00 43 1 5880132231, Fax: +00 43 1 5880132299

3 Membrane equation and coupling to CFD solver

A prerequisite is a CFD solver which allows for moving meshes and a mesh update during the iteration of a time step. We proceed as follows. From the given solution, including the membrane position $w(x, t - \delta t)$ at the previous time step, using the linearized potential flow coupled to the membrane equation an initial position of the membrane for the new time step $w^{(0)}(x, t)$ and an pressure difference due to the potential flow model $\Delta p_P^{(0)}$ is determined. Starting with this initial guess for the membrane position the pressure difference across the membrane $\Delta p_{\mathrm{CFD}}^{(0)}$ is calculated using the CFD solver. Then the position of the membrane is corrected by solving the membrane equation

$$\sigma \left(w_{xx}^{(n)} - \frac{w^{(n)}}{b^2} \right) = \Delta p_P^{(n)} + \left(\Delta p_{\mathrm{CFD}}^{(n-1)} - \Delta p_P^{(n-1)} \right) \tag{1}$$

simultaneously with the generalized Betz integral equation to obtain a new guess for the membrane position $w^{(n)}$ and vorticity distribution $\gamma^{(n)}$. The iteration is continued until convergence is obtained.

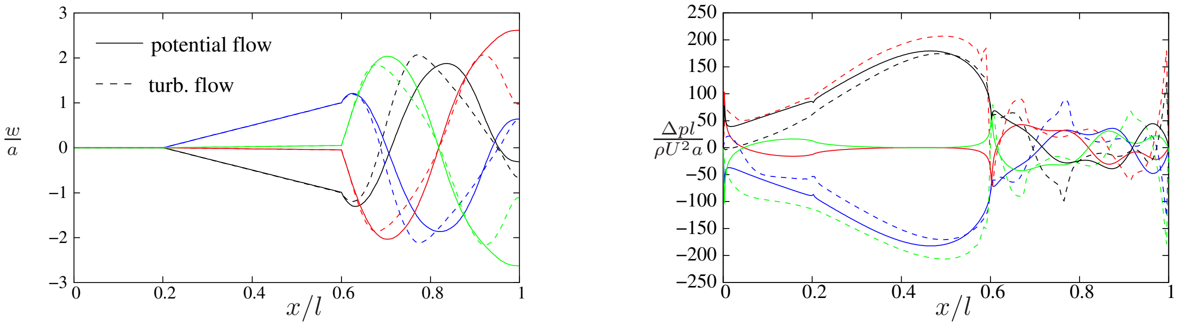
4 Results

The method has been tested for a membrane pump with the data given in table 1. Here σ is the tension of the membrane, f the frequency of the oscillation of the flap, ρ and μ the density and viscosity of the fluid, a the amplitude of the flap motion and U the mean velocity. The lengths are shown in figure 1. We compared the case of a turbulent flow with the potential flow with the corresponding dimensionless parameters. In figure 2a the deflection of the membrane is shown for different flap positions during one oscillation cycle. The shape of the membrane is near the flap look similar, but the behavior of the membrane near the trailing edge is different. The pressure difference of both cases is markedly different. However, we have doubts that the CFD solution is well resolved, since the prescribed shape of the structure (infinitely thin plate and sharp corners) challenges the CFD solver. The singularity near the leading is is not well resolved. The pressure field near trailing edge seem to violate the dynamic Kutta condition. Thus, a comparison with an improved CFD model is necessary.

flow	$Re = \frac{\rho u h}{\mu}$	a/h	fl/U	$A = \frac{\sigma h}{\rho u^2 l^2}$	b/l	l_f/l	l_M/l
turb	10^6	0.2	4	0.1	1	0.2	0.4
lin. pot.			4	0.1	1	0.2	0.4

Table 1: Data of test case

However, this preliminary study shows that the method converges and thus can be an alternative to a monolithic FSI-coupling approach.



a) b)

Fig. 2: Scaled membrane position (a) and pressure difference across the membrane (b) calculated with membrane membrane/potential flow interaction (solid line) and membrane turbulent flow interaction (dashed line)

References

[1] H. Steinrück, W. Zackl, M. Müllner, and H. Neth, PAMM **10**, 485–486 (2010).
[2] C. Förster, W. Wall, and E. Ramm, Computer Methods in Applied Mechanics and Engineering **196**, 1278–1293 (2007).
[3] J. Katz and A. Plotkin, Low Speed Aerodynamics (Cambridge University Press, Cambridge, 2001).
[4] M. Müllner and W. Schneider, Heat and Mass Trans. **46**, 1097–1110 (2010).