CAV2021
11th International Symposium on Cavitation
May 10-13, 2021, Daejon, Korea

Preconditioning and dual-time stepping based approach for the simulation of cavitating flows with low nuclei concentration using a fully-compressible solver

Mrugank Bhatt [1] and Krishnan Mahesh[1]

[1]Department of Aerospace Engineering and Mechanics, University of Minnesota, Minneapolis, USA

Abstract: A numerical approach based on preconditioning and dual-time stepping (DTS) is proposed to simulate cavitating flows at low Mach numbers. The methodology is based on a fully-compressible homogeneous mixture model and finite rate mass transfer as discussed in [1]. The method has shown promising results for capturing the large-scale cavitation in the developed cavitation regimes[2,3]. In the present work, we discuss a numerical approach to enable calculations for wetted conditions, cavitation inception etc. These simulations typically requires low free-stream nuclei, and consequently the modeling of low Mach numbers in water. The key aspects of the numerical approach are: (i) preconditioning applied to the cavitating equations in a fully-compressible manner, (ii) the shock-capturing is modified based on All-speed Roe-scheme, and (iii) implementation in parallel and on unstructured grids that allow the simulation of complex problems. The methodology is demonstrated for the LES of flow over a propeller under wetted conditions. Overall, a significant saving in total run-time, accurate comparisons of mean and RMS pressure with the incompressible solver, and a good agreement of thrust/torque with the experiments are obtained.

Keywords: Preconditioning, Dual-time stepping, Compressible flows, Cavitation, Low Mach numbers.

1. Introduction

In many cavitation applications, it is essential to capture the the propagation of acoustic waves and strong shock waves and therefore, the compressibility of the medium. It is known that the speed of the sound of a pure water can drop by orders of magnitude (e.g. from 1480 m/s to 10 m/s) with the addition of a gaseous phase and/or phase change [4]. Numerical modeling of the compressibility in such flows poses a challenge due to the range of Mach numbers.

The most commonly used physical model to simulate cavitating flows is the homogeneous mixture model [5,6,7,8], where the mixture of water and vapor is treated as a single medium. The model is used with either pressure-based [5] or density-based [7] numerical solver; both of which have to address the range of Mach numbers in cavitating flows. For density-based solvers, such as considered in the current work, algorithms suffer from the accuracy and numerical stiffness due to nearly incompressible regions in water. Prior studies have used high free-stream nuclei (i.e. initial levels of vapor or non-condensable gases present in water) in water to mitigate very low Mach numbers in pure water [2,3,10]. This approach has been shown to work well for the developed cavitation regimes [2,3]. However, small-scale vapor regions in the incipient cavitation, cavitation inception and wetted conditions are sensitive to free-stream nuclei content [9,10,11]. Hence, another approach is to retain the low free-stream nuclei that are important for physical modeling and use preconditioning to address the numerical stiffness. Preconditioning has provided a powerful remedy for the accuracy and convergence of compressible solvers at low Mach numbers [12,13,14,15]. Many of these preconditioning formulations are applied to the multiphase pressure-based algorithms [14,15]. In the current work, we present a preconditioning and dual-time stepping framework for the density-based solver developed in [1].

* Corresponding Author: Krishnan Mahesh, kmahesh@umn.edu

2. Method

We use the homogeneous mixture model where the mixture of water and vapor is treated as a single compressible medium. The mixture is assumed to be in thermodynamic and mechanical equilibrium between the constituent phases. The governing equations are the compressible Navier-Stokes equations for the mixture quantities along with the transport equation for vapor mass fraction. In the vector form with the addition of dual-time derivative they are given as:

$$\mathrm{T_p}\frac{\partial Q}{\partial \tau}+\frac{\partial U}{\partial t}+\frac{\partial F_j}{\partial x_j}=S, \quad where,\ \mathrm{T_p}=\frac{\partial U}{\partial Q}|_p, \tag{1}$$

$$U=\begin{pmatrix}\rho\\ \rho u\\ \rho v\\ \rho w\\ \rho Y_v\end{pmatrix}, Q=\begin{pmatrix}p\\ u\\ v\\ w\\ Y_v\end{pmatrix}, F_j=\begin{pmatrix}\rho u_j\\ \rho u u_j+p\delta_{1j}+\sigma_{1j}\\ \rho v u_j+p\delta_{2j}+\sigma_{2j}\\ \rho w u_j+p\delta_{3j}+\sigma_{3j}\\ \rho u_j Y_v\end{pmatrix}, S=\begin{pmatrix}0\\ 0\\ 0\\ 0\\ S_e-S_c\end{pmatrix}, and\ \mathrm{T}_p=\begin{bmatrix}\rho_p{}' & 0 & 0 & 0 & \rho_{Y_v}\\ u\rho_p{}' & \rho & 0 & 0 & \rho_{Y_v}u\\ v\rho_p{}' & 0 & \rho & 0 & \rho_{Y_v}v\\ w\rho_p{}' & 0 & 0 & \rho & \rho_{Y_v}w\\ Y_v\rho_p{}' & 0 & 0 & 0 & \rho_{Y_v}Y_v+\rho\end{bmatrix}.$$

Here, U is conserved variable vector, Q is primitive variable vector, F is the sum of convective and viscous flux vector, S is the source term vector (S_e *and* S_c are evaporation and condensation source terms), and T_p is the time derivative preconditioning matrix. It is constructed by taking the Jacobian of U with Q, and preconditioning $\rho_p'=\frac{1}{U_\infty}$, where U_∞ is the free stream velocity and $\rho_p'=\frac{\partial p}{\partial \rho}|_T$ [14]. ρ is mixture density, p is mixture pressure, Y_v is vapor mass fraction, and σ_{ij} is the viscous stress tensor. Eigenvalues of the system are obtained by pre-multiplying T_p^{-1} with the convective flux Jacobian ($A=\frac{\partial F}{\partial U}$). The details are given in [16].

The governing equations are integrated using finite volume discretization. Time marching is performed using a DTS procedure (e.g.[17]), where the physical-time derivative is discretized using a second order backward differencing (BDF-2) and the pseudo-time derivative is discretized using the explicit Euler. The discretized form is given as:

$$\mathrm{T_p}\frac{Q^{m+1}-Q^m}{\Delta\tau}+\frac{3U^{n+1}-4U^n+U^{n-1}}{\Delta t}=R(U^m). \tag{2}$$

Here, `$m$' and `n' are used respectively to indicate current pseudo-time and physical time values. Note that the pseudo-time derivative term is converged to zero every physical time step so that the accuracy of the unsteady term is not affected. $R(U^m)$ contains specially discretized convective terms including shock capturing part, viscous terms, and source terms.

The shock capturing is modified based on All-speed Roe-type scheme [18] to obtain proper conditioning of eigenvalues also in the dissipative terms. The modified eigenvalues are given as:

$$\lambda=[\mathrm{V_n}, V_n, V_n, V_n-f(M)*c, V_n+f(M)*c], \quad where, \quad f(M)=\min(M^2,1), \quad satisfying,$$

$$\begin{aligned}&0<f(M)<1, \quad when \quad 0<M<1\\ &f(M)\to 1, \quad when \quad M\to 0\\ &f(M)=0, \quad when \quad M\geq 1.\end{aligned} \tag{3}$$

Here, $f(M)$ is chosen such that it satisfies the above mention properties to ensure proper conditioning of eigenvalues for all Mach numbers. This is applied to the characteristic-based filtering fluxes of the original scheme[1]. Please see [16] for more details.

* Corresponding Author: Krishnan Mahesh, kmahesh@umn.edu

3. Results

We simulate flow over a five-bladed marine propeller P4381 at the design advance ratio ($J = 0.89$), $Re = 894000$, and $\sigma = \infty$ (i.e. under wetted conditions). Here, $J = \frac{U_\infty}{nD}$, where `$n$' is the rotation rate and `D' is the propeller diameter. $\sigma = \frac{p - p_v}{0.5\,\rho U_\infty^2}$, where `$p_v$' is the vapor pressure. The original case is simulated in [10] without preconditioning using compressible LES. The case is termed as `Case-O'. The same case simulated at low Mach number is termed as `Case-OLM'. Main challenge for these simulations is to obtain an accurate solution in a reasonable amount of run-time. We simulate the case at low Mach number using the preconditioning, that is termed as `Case-PRLM'. The details of the computational domain, grid size and boundary conditions are given in [10]. Table 1 shows the run-time comparison for these cases. A significant savings from ~2 weeks per cycle to ~40 hours per cycle is obtained with preconditioning.

Table 1. Computational time comparison. Δt is the physical-time step, and Δτ is the pseudo-time step.

Cases simulated	Δt	$\Delta \tau$	Run-time/cycle
Case-OLM	2×10^{-6}	-	$\sim 2\ weeks$
Case-PRLM	5×10^{-3}	1×10^{-3}	$\sim 40\ hours$

We compare the solution accuracy of the Case-OLM and Case-PRLM with the benchmark incompressible solver at low Mach numbers. Figure 1(a) shows comparison of mean pressure contours in the *x-y* plane. The profiles of mean and RMS pressures extracted at various axial locations are plotted in figures 1(b) and 1(c) respectively. For the Case-OLM the horizontal stripes upstream of the propeller, the lack of visibility of the tip vortices, and the diffused regions of low pressure close to the shaft are all indicators of the deteriorated pressure field. This is due to the incorrect scaling of the pressure field for the compressible solver in the incompressible limit. With the use of preconditioning (Case-PRLM), the improvements on all the fronts are observed. This is evident visually from the comparison of the Case-PRLM to the incompressible MPCUGLES. The secondary vortices close to the shaft of the propeller are better captured in the Case-PRLM. A quantitative comparison of profiles in figures 1(b) and 1(c) also indicate that the Case-PRLM is in very good comparison with the incompressible result.

The propeller loads are compared in table 2. Non-dimensional thrust coefficient is given by $K_T = \frac{T}{\rho n^2 D^4}$, and non-dimensional torque coefficient is given by $K_Q = \frac{T}{\rho n^2 D^5}$. Improvement in K_T is observed for Case-OLM compared to Case-O. However, a significant simulation time of ~2 weeks per cycle is required. Also, 10% error in K_Q with experiments [19] is observed. With preconditioning (Case-PRLM), both K_T and K_Q are captured accurately. Also, this is achieved with significant time savings of ~40 hours per cycle as indicated in table 1.

The method is also assessed for cavitating vortex problem, wetted and cavitation inception conditions over a cylinder. More details are in [16].

* Corresponding Author: Krishnan Mahesh, kmahesh@umn.edu

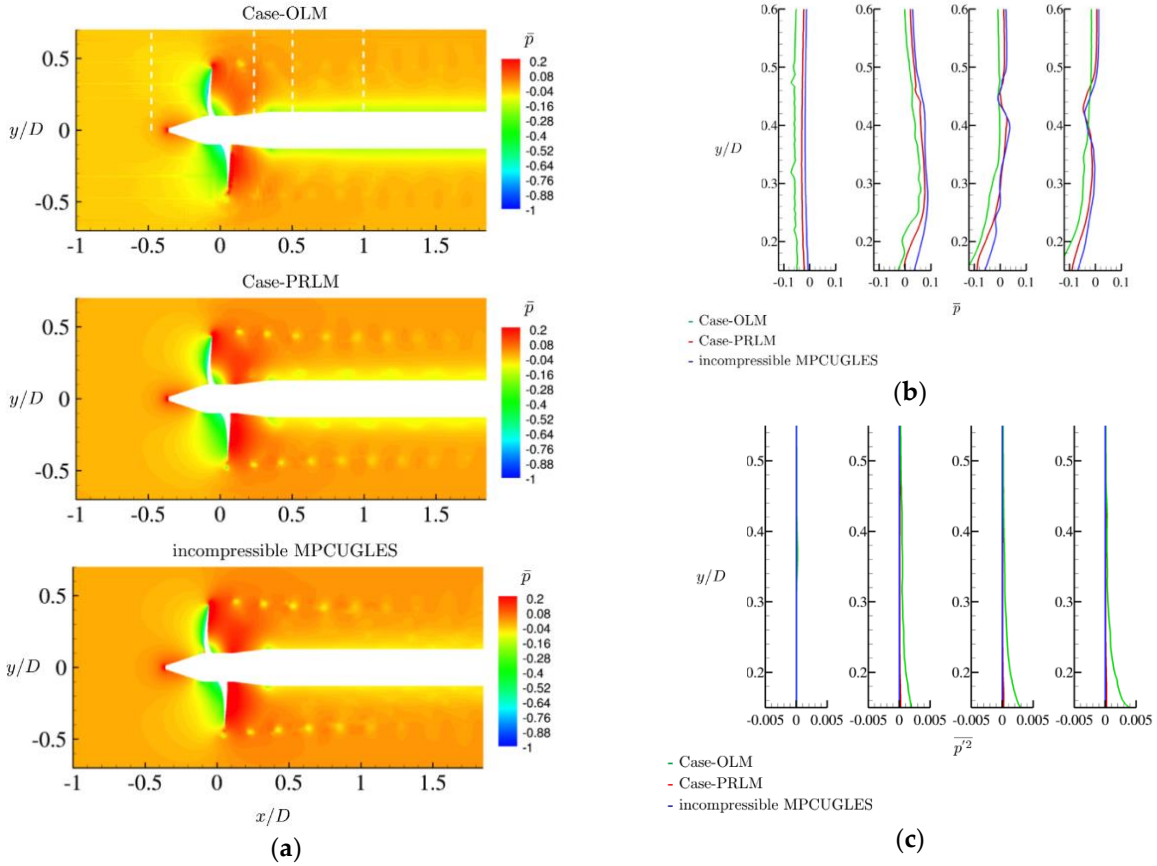

Figure 1. (**a**) Contours of the phase averaged pressure field along the x − y plane. (**b**) Profiles of mean pressure extracted along the y − axis at x = −0.5D, 0.25D, 0.5D and 1.0D. (**c**) Profiles of RMS pressure extracted along the y − axis at x = −0.5D, 0.25D, 0.5D and 1.0D.

Table 2. Comparison of propeller performance

Cases simulated	K_T	K_Q
Case-O	0.257	0.055
Case-OLM	0.226	0.050
Case-PRLM	0.227	0.0458
Boswell (1971)[19]	0.215	0.045

Acknowledgments: This work is supported by the United States Office of Naval Research under grant ONR N00014 - 17-1-2676 with Dr. Ki-Han Kim as the program manager. The computations were made possible through the computing resources provided by the High-Performance Computing Modernization Program (HPCMP), Texas Advance Computing Center (TACC), and the Minnesota Supercomputing Institute (MSI).

* Corresponding Author: Krishnan Mahesh, kmahesh@umn.edu

References

1. Gnanaskandan, A.; Mahesh, K. A numerical method to simulate turbulent cavitating flows. *International Journal of Multiphase Flow* **2015,** *volume 70*, pp. 22–34.
2. Bhatt, M.; Mahesh, K. Investigation of sheet to cloud transition due to the propagation of condensation fronts over a sharp wedge using large eddy simulations. *10th International Symposium on Cavitation*, Baltimore, USA **2018.**
3. Gnanaskandan, A.; Mahesh, K. Large eddy simulation of the transition from sheet to cloud cavitation over a wedge. *International Journal of Multiphase Flow* **2016,** *volume 83*, pp. 86–102.
4. Franc, J.P.; Michel, J.M. *Fundamentals of cavitation;* Publisher: Springer Science & Business Media, 2006, *Volume 76.*
5. Seo, J.H.; Lele, S. Numerical investigation of cloud cavitation and cavitation noise on a hydrofoil section. *7th International Symposium on Cavitation*, Ann Arbor, USA **2009.**
6. Schnerr, G.H.; Sezal, I.H.; Schmidt, S.J. Numerical investigation of three–dimensional cloud cavitation with special emphasis on collapse induced shock dynamics. *Physics of Fluids* **2008,** *volume 20*, pp. 1–9.
7. Singhal, A.K.; Athavale, M.M.; Li, H.; Jiang, Y. Mathematical basis and validation of the full cavitation model. *Journal of Fluids Engineering* **2002,** *volume 124*, pp. 617–624.
8. Kunz, R.F.; Boger, D.A.; Stinebring, D.R.; Chyczewski, T.S.; Lindau, J.W.; Gibeling, H.J.; Venkateswaran, S.; Govindan, T.R. A preconditioned Navier–Stokes method for two–phase flows with application to cavitation prediction. *Computers & Fluids* **2000**, *volume 29*, pp. 849–875.
9. Hsiao, C.T.; Chahine, G.L. Scaling of tip vortex cavitation inception noise with a bubble dynamics model accounting for nuclei size distribution. *Journal of Fluids Engineering* **2005,** *volume 127*.
10. Bhatt, M.; Mahesh, K. Investigation of propeller cavitation using compressible large eddy simulations. *Sixth International symposium on Marine Propulsors*, Rome, Italy **2019**.
11. Brandao, F.L.; Bhatt, M.; Mahesh, K. Numerical study of cavitating regimes in flow over a circular cylinder. *Journal of Fluid Mechanics* **2020**, *volume 885*, A19.
12. Weiss, J.M.; Smith, W.A. Preconditioning applied to variable and constant density flows. *Journal of Computational Physics* **1995**, pp. 2050–2057.
13. Turkel, E. Preconditioning techniques in computational fluid dynamics. *Annual Review of Fluid Mechanics* **1999,** *Volume 31*, pp. 385–416.
14. Lindau, J.; Kunz, R.; Venkateswaran, S.; Merkle, C. Development of a fully-compressible multi-phase Reynolds-averaged Navier-Stokes model. *15th AIAA Computational Fluid Dynamics Conference* **2001**, pp. 26-48.
15. Venkateswaran, S.; Lindau, J.W.; Kunz, R.F.; Merkle, C.L. Computation of multiphase mixture flows with compressibility effects. *Journal of Computational Physics* **2002,** *volume 180*, pp. 54–77.
16. Bhatt, M. LES of turbulent cavitating flows using the homogeneous mixture model. *PhD thesis* **2020**.
17. Hsu, J.; Jameson, A. An implicit-explicit hybrid scheme for calculating complex unsteady flows. *40th AIAA Aerospace Science Meeting & Exhibit* **2002**.
18. Li, X.S.; Gu, C. An all-speed Roe-type scheme and its asymptotic analysis of low Mach number behavior. *Journal of Computational Physics* **2008**, pp. 5144–5159.
19. Boswell, R.J. Design, cavitation performance, and open-water performance of a series of research skewed propellers. *Report : Naval ship research and development center*, Washington, D.C. **1971.**

* Corresponding Author: Krishnan Mahesh, kmahesh@umn.edu