Recent Developments and Applications of Computational Fluid Dynamics

Ong Kian Chuan

Ong Kian Chuan, *B.Eng (Mechanical) (University of Nottingham) is currently a postgraduate research student at the University of Nottingham with research interest is on computational fluid dynamics (CFD). He looks into new numerical solution algorithms for fluid flow with multidisciplinary applications as well as turbulence modelling, and aims to tackle these challenges by the terms efficiency, robustness, and accuracy.*

Professor Andy Chan

Professor Andy Chan, *PhD (Univ. of Hong Kong), currently is the Dean of the Faculty of Engineering, University of Nottingham Malaysia Campus (UNMC) and the director of the Asia Aerospace City Research and Technology Centre (AAC) which focus on research on aerospace manufacturing technologies. Prior to joining UNMC, he was a scientist in the National Center for Atmospheric Research (NCAR), USA.*

Computational Fluid Dynamics (CFD) is the application of the numerical method to solve fluid flow problems. With the development of accurate and robust numerical algorithms, CFD has now matured to where it used as a key tool for a broad range of applications such as aerospace, automotive and various engineering design process.

As CFD becomes increasingly routine, it is even more prudent that attention is focused on developing a method with robustness, accuracy and generality and which must be able to compute stable and accurate solutions under various flow conditions. In addition, as the regime of application is extended, e.g. to distinct fluid flow regimes as a function of Mach number or to different sets of conservation laws, robustness and accuracy should be maintained (1). The Mach number represents the ratio of the local flows speed and the local speed of sound.

In many applications, Mach number varies throughout the flow, for instance, a re-entering space shuttle (Figure 1), where low-Mach viscous boundary layers are embedded in a hypersonic flow, and transonic flow over a RAE2822 airfoil (Figure 2), where a supersonic region with shockwave is presented within a subsonic/transonic flow.

For such an application, Mach-uniform algorithm is important which is a unified numerical formulation for fluid flow computations at all speeds, i.e. arbitrary Mach number regimes.

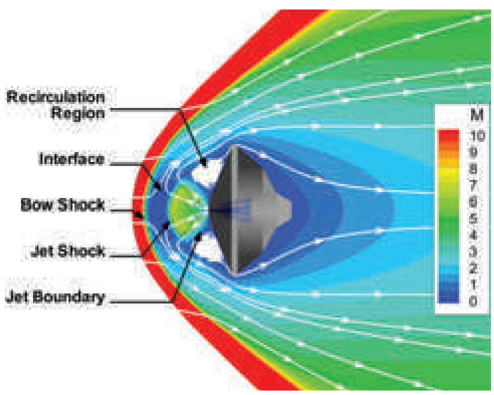

Figure 1: Re-entering space shuttle.

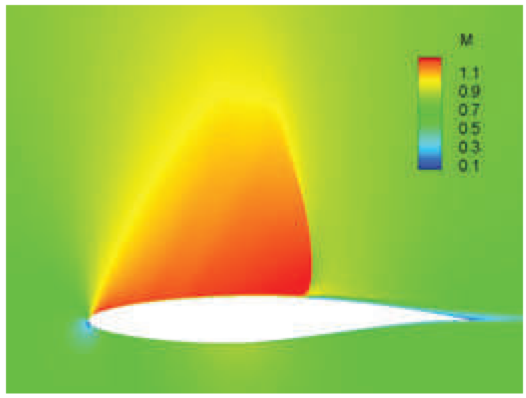

Figure 2: Transonic flow over a RAE2822 airfoil.

Historically, most of the existing techniques are developed specifically for either compressible fluid flow or incompressible fluid flow regimes. These are beset by substantial barriers when one applies a scheme of one regime to a problem of another regime. Continual efforts have been carried out to develop Mach-uniform methods by bridging the gap between the two distinct methods, namely density-based algorithm and pressure-based algorithm, to enhance them to compute fluid flows at arbitrary Mach number regimes.

The density-based algorithms were initially developed for high-Mach number applications. The conservation of mass acts as an equation for density whereas pressure is computed from the energy equation and equation of state. They are very effective for high-Mach number fluid flow, but stability and robustness significantly deteriorate when solving low-Mach number fluid flow.

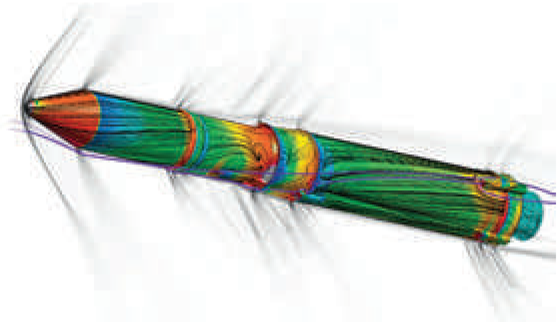

Figure 3: Computation of flow stream-line around a rocket.

Conventionally, they are applied to low-Mach number regime through preconditioning (2). In the more recent decades, all-speeds versions of Advection Upstream Splitting Method (AUSM) have been proposed (1). The all-speeds AUSM and the variant AUSM-family schemes are simple, accurate, robust and possess superior shock-capturing properties (3). These methods do not involve sophisticated differentiations, such as Jacobian matrix, in the evaluation of numerical fluxes and so, are readily extendible to a general equation of state, to thermal non-equilibrium flows or to turbulence model equations (4). Hence, all-speeds AUSM-family schemes are very promising for the computation of fluid flows at arbitrary Mach number regimes. Figure 3 shows an example of the application of AUSM-family schemes in the computation of flow topology around a rocket.

Contrary to density-based algorithms, pressure-based algorithms were originally proposed to solve incompressible fluid flow. The first pressure-based algorithm proposed for all-speeds fluid flow was based on a semi-implicit formulation that extended from Marker-And-Cell (MAC) method to Implicit-Continuous-Fluid-Eulerian (ICE) method for solving transient fluid flow problems at all-speeds (5).

Subsequently, some all-speed pressure-based algorithms have been developed. The feasibility of pressure-based algorithm is based on the fact that pressure variation remains finite, spanning arbitrary Mach number regimes.

All-speeds pressure-based algorithms generally suffer from numerical instability in the computation of compressible flow due to the hyperbolic nature of governing equations and the lack of the shock-capturing capability (6).

Recently, the pressure-based algorithm was combined with AUSM-family schemes for fluid flows computations at arbitrary Mach numbers. The AUSM-family schemes fit perfectly into the pressure-based algorithm due to the separate treatment of convective and acoustic part (pressure) (7). The advantages of the hybrid combination are that the shock-capturing properties at high-Mach number regime are greatly improved, and ad hoc modifications are not needed at low-Mach number regimes (8). This algorithm is successfully extended to solve the magnetohydrodynamics at all-speeds (9), as shown in Figure 4, and it is readily applied to various application of fluid flows at arbitrary Mach number regimes, with complex flow topologies such as shock wave/ boundary layer interactions. ■

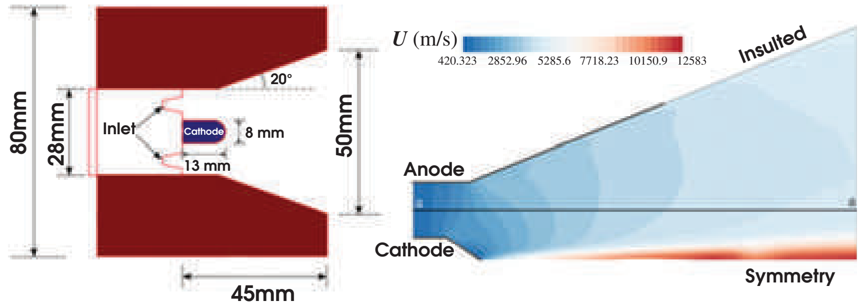

Figure 4: Magnetoplasmadynamic (MPD) thruster.

REFERENCES

[1] M.S. Liou. A sequel to AUSM, part II: AUSM-up for all speeds. Journal of Computational Physics, 214(1):137–170, 2006.

[2] E. Turkel. Preconditioning techniques in computational fluid dynamics. Annual Review of Fluid Mechanics, 31(1):385–416, 1999.

[3] K. Kitamura and E. Shima. Towards shock-stable and accurate hypersonic heating computations: A new pressure flux for AUSM-family schemes. Journal of Computational Physics, 245(0):62–83, 2013.

[4] M.S. Liou and C.J. Steffen Jr. A new flux splitting scheme. Journal of Computational Physics, 107(1):23–39, 1993.

[5] F.H. Harlow and A.A. Amsden. A numerical fluid dynamics calculation method for all flow speeds. Journal of Computational Physics, 8(2):197–213, 1971.

[6] K.C. Karki and S.V. Patankar. Pressure based calculation procedure for viscous flows at all speeds in arbitrary configurations. AIAA Journal, 27(9):1167–1174, 1989.

[7] K. Nerinckx, J. Vierendeels, and E. Dick. A Mach-uniform pressure-correction algorithm with AUSM flux definitions. International Journal of Numerical Methods for Heat & Fluid Flow, 16(6):718–739, 2006.

[8] C.M. Xisto, J.C. Páscoa, P.J. Oliveira, and D.A. Nicolini. A hybrid pressure-density-based algorithm for the Euler equations at all Mach number regimes. International Journal for Numerical Methods in Fluids, 70(8):961-976, 2012.

[9] C.M. Xisto, J.C. Páscoa, and P.J. Oliveira. A pressure-based method with AUSM-type fluxes for MHD flows at arbitrary Mach numbers. International Journal for Numerical Methods in Fluids, 72(11):1165-1182, 2013.