Chapter 1
Introduction

With the rapid development of electronic computers, numerical computation has become an important paradigm of scientific discovery as well as a powerful tool for engineering research. Solving complex problems in a computational fashion is more than applying theories, equations, and formulas. Computational methods, also called algorithms or schemes, have strong influences on the outcomes of computations.

The space–time conservation element and solution element (CESE) method is aimed to numerically solve equations of conservation laws in various physical systems. The major concern herein will be the computational fluid dynamics (CFD), which is a hot area in scientific and engineering researches. Nevertheless, it may help at the outset to recognize that the conservation laws in CFD problems have a lot in common with conservation laws in other fields, such as acoustics, solid dynamics, electromagnetics, and magnetohydrodynamics. The physics may be different, but the mathematics are similar. For instance, physics involving dynamical evolution of waves and discontinuities are usually modelled by time-dependent nonlinear hyperbolic partial differential equations (PDEs). Some CFD problems happen to be representative of such physical problems in a wider context.

1.1 Background

Most of the key achievements in conventional CFD have been incorporated into the computational procedure of the finite volume method (FVM). Indeed, the FVM is well established and widely used, and therefore it is sometimes regarded as a mature technique. There are two important steps in the FVM, namely the reconstruction and the evolution [1, 2]. In the reconstruction step, one needs to locally approximate the flow field with simple functions such as polynomials, and then interpolate the values of flow quantities at the cell interfaces by utilizing the cell-average values. In the evolution step, the numerical flux at each cell interface is determined by some kind

C.-Y. Wen et al., *Space–Time Conservation Element and Solution Element Method*, Engineering Applications of Computational Methods 13,
https://doi.org/10.1007/978-981-99-0876-9_1

of flux technique, and then the time integration of the conservation equations can be performed.

A major category of CFD problems face the challenges brought by (1) the coexistence of smooth regions and discontinuities in the flow (e.g., shock waves and material interfaces) and (2) the wide spectrum of wave numbers/frequencies in the flow (e.g., acoustic waves and turbulence). Due to these physical phenomena, the CFD solver is required to well handle the numerical dissipation, which leads to an endless debate in the CFD community. First, for the robustness in capturing shock waves, there must be a certain amount of numerical dissipation to suppress the spurious oscillations. However, a low numerical dissipation is very favourable for the accuracy and resolution of the viscous layers, contact surfaces, and small-scale structures in the flow, which would be smeared out by excessive dissipation.

To achieve the low but controllable numerical dissipation, the research works on FVM are mostly dedicated to two approaches: (1) developing high-order reconstruction techniques in conjunction with nonlinear limiters and (2) improving the flux solvers (e.g., approximate Riemann solvers). In both ways, numerous important advances have been obtained.

In spite of these successes in the FVM development, several shortcomings may arise. (1) A conventional FVM implementation highly relies on the selections of special techniques, such as the limiter and the Riemann solver. Actually, each special technique has its own pros and cons. There are many alternatives for each option, but none of them can be proven to be optimal and general. (2) The coupling of space and time is usually overlooked and may not be guaranteed in the numerical solution. (3) The multi-dimensionality can be questionable because most flux solvers are only based on one-dimensional physics. (4) The popular high-order FVM schemes are usually not compact, i.e., large stencils are used. Under this background, the CESE method was proposed to overcome the difficulties in the conventional FVM to some extent, and it became a new member in the family of FVM.

1.2 History of the CESE Method

The space–time conservation element and solution element method was initiated by Sin-Chung Chang of NASA Glenn Research Centre and his collaborators for aerodynamic computation in early 1990s [3, 4]. There were several motivations to propose the CESE method. First, they wanted to construct multidimensional and space–time unified CFD schemes, without using dimensional splitting or separated treatments of space discretization and time discretization. Second, they believed a CESE solver should be built from a non-dissipative core scheme so that numerical dissipation can be controlled effectively, dynamically, and even actively. Third, they attempted to avoid using the Riemann solver in the CESE method.

Around 1994, the CESE project was approved and supported by NASA Glenn Research Centre. The work of the research group showed that this time-accurate scheme possesses low numerical dissipation, which is valuable in CFD simulation.

For the first time, the transonic resonance in a convergent-divergent nozzle with frequency-staging effect was successfully simulated by the CESE scheme [5, 6]. In this case, the blind prediction by the CESE scheme matched the experimental data. Therefore, the CESE method is proved to be capable of simultaneously simulating multidimensional unsteady shock waves and acoustic waves with high accuracy.

Owing to its accuracy and robustness, the CESE method is employed in the CFD module of the simulation software LS-DYNA [7]. The Jacobs Technology Inc. (designer of the NASA's hypersonic flow test facilities) developed its own in-house CESE code called JUSTUS (Jacobs Unified Space–Time Unstructured Solver) for hypersonic flow simulations. Another in-house CESE code, called "ez4d" software, was developed by the NASA Langley Research Centre. This is a time-accurate 3D Navier–Stokes flow solver on unstructured meshes [8, 8].

1.3 Main Features of the CESE Method

The CESE method is a special finite-volume-type numerical method, with the aim of solving the governing equations of fluid dynamics and other conservation laws in various physical systems. The CESE method possesses many features such as the unified treatment for space and time, the fully discrete one-step explicit scheme, and the highly compact stencil. Without enlarging the stencil or adding stages of time integration, the CESE method can achieve arbitrary high-order accuracy for both space and time.

The essential ingredients in the CESE method include: (1) the spatial derivatives of physical variables are stored as independent unknowns, in addition to the physical variables themselves. In every time step, these derivatives are updated by a specially designed procedure. (2) a staggered mesh and a staggered time-marching strategy are employed. (3) the interior structure within each solution element is built with the Taylor expansion. (4) the time-marching approach is based on the Cauchy–Kowalewski procedure.

Through a combination of the above techniques, the CESE method has low dissipation and high compactness. When applied to the simulations of complex physical processes, the CESE method can catch shock waves, contact discontinuities, fine structures, and small disturbances, with high resolution and strong robustness. Therefore, the CESE method demonstrates good performances in the numerical simulations of wave-propagation problems (e.g., shock waves, acoustic waves, detonation waves, stress waves in solid, and the electromagnetic waves), interfacial instabilities, as well as the interactions of gaseous and liquid phases. In many research areas including high-speed aerodynamics, shock dynamics, detonation, aeroacoustics, and solid dynamics, the CESE method proves to be suitable and shows a good development prospect.

1.4 Outline of the Book

The remainder of this book is organized as follows. First of all, the non-dissipative core scheme of the CESE method is introduced in Chap. 2, with detailed descriptions of the basic concepts. Then, the practical shock-capturing CESE schemes with numerical dissipation including the classical a–α scheme, the Courant number insensitive scheme, and the recently proposed upwind CESE schemes are presented in Chap. 3. Furthermore, Chaps. 4 and 5 extend the CESE method to multidimensional and high-order versions. In Chap. 6, numerical properties of various CESE schemes are analysed, along with comparisons to other numerical schemes. Chapters 7–9 provide an overview of the applications of the CESE method, most of which are quite relevant to the aerospace engineering. Finally, a summary and an outlook of the CESE method are given in Chap. 10.

References

1. Toro, E. F. (2009). *Riemann solvers and numerical methods for fluid dynamics: A practical introduction.* Springer.
2. Kitamura, K. (2020). *Advancement of shock capturing computational fluid dynamics methods: numerical flux functions in finite volume method*. Springer Nature.
3. Chang, S. C. & To, W. M. (1991). *A new numerical framework for solving conservation laws: the method of space-time conservation element and solution element*. National Aeronautics and Space Administration.
4. Chang, S. C. (1995). The method of space-time conservation element and solution element—A new approach for solving the Navier-Stokes and Euler equations. *Journal of Computational Physics, 119*(2), 295–324.
5. Loh, C. Y., & Zaman, K. (2002). Numerical investigation of transonic resonance with a convergent-divergent nozzle. *AIAA Journal, 40*(12), 2393–2401.
6. Zaman, K., Dahl, M., Bencic, T., & Loh, C. (2002). Investigation of a 'transonic resonance' with convergent–divergent nozzles. *Journal of Fluid Mechanics, 463*, 313–343.
7. Cook, G., Zhang, Z. C. & Im, K. S. (2013). Applications of the CESE method in LS-DYNA. In *21st AIAA Computational Fluid Dynamics Conference.*
8. Chang, C. L. (2006). Time-accurate, unstructured mesh Navier-Stokes computations with the CESE method. In *42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*.
9. Chang, C. L. (2007). Three-dimensional Navier-Stokes calculations using the modified space-time CESE method. In: *43rd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*.