A high-order continuous Lagrange–Galerkin method for compressible flows

Manuel COLERA [*], Jaime CARPIO [†], Rodolfo BERMEJO [‡]

1 Introduction

We present a novel Lagrangian–Eulerian scheme for the resolution of two-dimensional compressible and inviscid flows. The scheme employs continuous space discretizations and implicit–explicit time marching schemes of arbitrary order. This is done with a view to consider, in the mid-term future, stiff viscous and reactive terms, for which implicit solvers and continuous discretizations may perform more efficiently [4].

In addition, the scheme considers high-order discretizations on unstructured triangular meshes, and preserves mass, momentum and total energy as long as some integrals in the formulation are computed exactly. The recent model proposed by Brenner [1] for viscous flows is employed to define the operators needed to stabilize the continuous Galerkin formulation.

2 Formulation

In this work, we focus on the Euler equations for compressible and inviscid flows,

$$\partial_t u_I + \partial_j \left(v_j u_I \right) = \partial_j \left(f_{Ij} \right), \tag{1}$$

with $\partial_t \equiv \partial/\partial t$, $\partial_j \equiv \partial/\partial x_j$, t the time, $\mathbf{x}$ the position vector, $\mathbf{u} = [\rho, m_1, m_2, \mathcal{E}]^T$ the vector of conservative variables, $\mathbf{f} = \mathbf{f}(\mathbf{u})$ the pressure flux matrix, $f_{1,j} = 0$, $f_{1+i,j} = -p\delta_{ij}$, $f_{4,j} = -pv_j$, ρ the density, $\mathbf{v}$ the velocity, $\mathbf{m} = \rho\mathbf{v}$ the momentum per unit of volume, $\mathcal{E} = p/(\gamma - 1) + \rho v_i v_i/2$ the total energy per unit of volume and γ the adiabatic constant of the gas. We adopt Einstein's summation convention, with uppercase and lowercase indices varying from 1 to 4 and from 1 to 2, respectively.

The so-called *weak Lagrange–Galerkin formulation* [2] associated with Eq. (1) is

$$\partial_t \int_{\Omega_f} u_I \psi \, \mathrm{d}\Omega = -\int_{\Omega_f} f_{Ij} \partial_j \psi \, \mathrm{d}\Omega + \oint_{\partial\Omega_f} f_{Ij} \psi n_j \, \mathrm{d}\sigma, \quad \forall \psi \in V, \tag{2}$$

where $\Omega_f(t) := \left\{ \mathbf{x}(t) \in \mathbb{R}^2 : \mathrm{d}\mathbf{x}/\mathrm{d}t = \mathbf{v}(\mathbf{x}(t), t) \right\}$ is a domain that moves with the fluid, $\mathbf{n}$ is the outward normal vector to the boundary $\partial\Omega_f$, and $V(t) := \left\{ \psi(\mathbf{x}, t) \in \mathcal{C}^0(\Omega_f(t)) : \partial_t \psi + v_j \partial_j \psi = 0 \right\}$ is the space of all continuous functions in $\Omega_f(t)$ whose values are constant along the trajectories of the particles.

At a given time t, $\Omega_f(t)$ is approximated by a polygonal domain $\Omega_h(t)$, which is partitioned into a triangular (curvilinear) finite element mesh $\mathbb{T}_h(t)$. The elements of the mesh are defined by the position of the nodes $\mathbf{x}_n(t)$ and by the standard isoparametric transformation. The finite element space $V_h(t)$ associated with $\mathbb{T}_h(t)$ is the direct sum of a *large-scale* space, $\overline{V}_h(t)$, and a *fine-scale* space, $V'_h(t)$, so that any function $\psi_h \in V_h$ is also expressed as the sum of a large-scale term, $\overline{\psi}_h$, and a fine-scale term, ψ'_h. That is,

$$\psi_h = \overline{\psi}_h + \psi'_h, \quad \overline{\psi}_h \in \overline{V}_h, \quad \psi'_h \in V'_h, \quad V_h = \overline{V}_h \oplus V'_h.$$

In this work, $\overline{V}_h$ is a standard (continuous) polynomial space of arbitrary order and V'_h is the corresponding bubble space [2]. Only the conservative variables in $\mathbf{u}$ are discretized; the rest are computed using appropriate relations whenever required.

To obtain a numerical solution $\mathbf{u}_h$ to Eq. (2), we successively integrate from t^n to t^{n+1} via an implicit–explicit Runge–Kutta (RK) method as follows. Let the superindex $[k]$ denote any variable evaluated at the kth stage of the RK method, $[s]$ denote the

[*] Departamento de Ingeniería Energética, Universidad Politécnica de Madrid. Email: m.colera@upm.es
[†] Departamento de Ingeniería Energética, Universidad Politécnica de Madrid. Email: jaime.carpio@upm.es
[‡] Departamento de Matemática Aplicada a la Ingeniería Industrial, Universidad Politécnica de Madrid. Email: rodolfo.bermejo@upm.es

final stage, $\widetilde{\Omega}_h$ be a reference (fixed) domain, $\widetilde{\mathbb{T}}_h$ be the associated (rectilinear) mesh and $\widetilde{V}_h$ be the corresponding finite element space. At the first stage, we set $\mathbf{u}_h^{[1]} = \mathbf{u}_h^n$, $\Omega_h^{[1]} = \widetilde{\Omega}_h$ and $\mathbb{T}_h^{[1]} = \widetilde{\mathbb{T}}_h$. For the next stages $k > 1$, we perform the following steps:

1. The mesh nodes are displaced by solving $\mathrm{d}\mathbf{x}_n/\mathrm{d}t = \mathbf{v}(\mathbf{x}_n(t), t)$ with the explicit part of the method. The displaced nodes define a curvilinear mesh $\mathbb{T}_h^{[k]}$.
2. The space-discretized version of Eq. (2) is solved with the explicit part of the method to obtain a non-stabilized solution $\mathbf{u}_*^{[k]}$. Note for that purpose that, if $\psi_h^{[k]}$ is the ith shape function of $V_h^{[k]}$, then $\psi_h^{[l]}$ is the ith shape function of $V_h^{[l]}$ [2].
3. The fine scale terms in the non-stabilized solution, $\mathbf{u}'_*$, are expected to be large at the discontinuities and small at the smooth regions. Thus, they are post-processed to define adequate discontinuity-capturing operators based on artificial viscosity. Subgrid stabilization is also employed [3]. Brenner's model for viscous flows [1] is considered in both cases and the resulting stabilizing term is denoted by $\mathbf{S}(\mathbf{u}_h^{[k]}, \psi_h^{[k]})$.
4. The implicit part of the method is employed to integrate the stabilized equation

$$\partial_t \int_{\Omega_h} u_{h_I} \psi_h \, \mathrm{d}\Omega = - \int_{\Omega_h} f_{h_{Ij}} \partial_j \psi_h \, \mathrm{d}\Omega + \oint_{\partial\Omega_h} f_{h_{Ij}} \psi_h n_j \, \mathrm{d}\sigma + S_I\left(\mathbf{u}_h, \psi_h\right), \quad \forall \psi_h \in V_h.$$

The resulting nonlinear system of equations is solved via Anderson's method [6].

The RK method provides a solution $\mathbf{u}_h^{[s]}$ defined over the domain $\Omega_h^{[s]}$. Then, we project this solution onto the reference (fixed) mesh, that is, $\mathbf{u}_h^{n+1}$ is the solution of

$$\int_{\widetilde{\Omega}_h} u_{h_I}^{n+1} \psi_h \, \mathrm{d}\Omega = \int_{\widetilde{\Omega}_h} u_{h_I}^{[s]} \psi_h \, \mathrm{d}\Omega, \quad \forall \psi_h \in \widetilde{V}_h.$$

The term in the right-hand side, which involves the product of functions defined piecewise in different meshes, is computed via high-order quadrature rules.

3 A numerical example

To check the accuracy of the method, we have solved the so-called shock–vortex interaction problem [5] (among others). The results for the density are shown in Fig. 1.

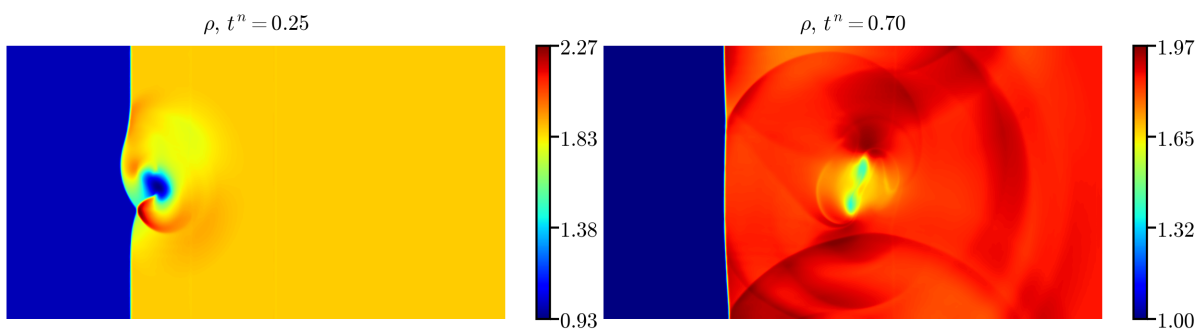

Figure 1: Numerical solution of the shock–vortex interaction problem for fifth-order elements and mesh size $h \simeq 0.01$.

Acknowledgements

This research has been partially funded by project PGC-2018-097565-BI00 from the "Ministerio de Ciencia, Innovación y Universidades" of Spain and the European Regional Development Fund, and by the FPU16/05509 grant to M. Colera from the "Ministerio de Ciencia, Innovación y Universidades" of Spain.

References

[1] Howard Brenner. Fluid mechanics revisited. *Physica A: Statistical Mechanics and its Applications*, 370(2):190–224, October 2006.

[2] Manuel Colera, Jaime Carpio, and Rodolfo Bermejo. A nearly-conservative high-order Lagrange–Galerkin method for the resolution of scalar convection-dominated equations in non-divergence-free velocity fields. *Computer Methods in Applied Mechanics and Engineering*, 372:113366, December 2020.

[3] Manuel Colera, Jaime Carpio, and Rodolfo Bermejo. A nearly-conservative, high-order, forward Lagrange–Galerkin method for the resolution of scalar hyperbolic conservation laws. *Computer Methods in Applied Mechanics and Engineering*, 376:113654, April 2021.

[4] P. Rajaraman, G. D. Vo, G. Hansen, and J. J. Heys. Comparison of continuous and discontinuous finite element methods for parabolic differential equations employing implicit time integration. *International Journal for Computational Methods in Engineering Science and Mechanics*, 18(2-3):182–190, May 2017.

[5] Audrey Rault, Guillaume Chiavassa, and Rosa Donat. Shock-Vortex Interactions at High Mach Numbers. *Journal of Scientific Computing*, 19(1):347–371, December 2003.

[6] Homer F. Walker and Peng Ni. Anderson Acceleration for Fixed-Point Iterations. *SIAM Journal on Numerical Analysis*, 49(4):1715–1735, January 2011.