Available online at www.sciencedirect.com
ScienceDirect
Comput. Methods Appl. Mech. Engrg. 317 (2017) 128–152
Computer methods in applied mechanics and engineering
www.elsevier.com/locate/cma

Numerical investigation of a viscous regularization of the Euler equations by entropy viscosity

Murtazo Nazarov[a,*], Aurélien Larcher[b]

[a] *Division of Scientific Computing, Department of Information Technology, Uppsala University, Sweden*
[b] *ERCIM ABCDE Fellow, Department of Mathematical Sciences, NTNU, Trondeim, Norway*

Received 9 March 2016; received in revised form 2 December 2016; accepted 7 December 2016
Available online 19 December 2016

Abstract

The Navier–Stokes viscous fluxes are a well-known viscous regularization of the Euler equations. However, since these fluxes do not add any viscosity to the mass equation, the positivity of density is violated. This paper investigates a new class of viscous regularization of the Euler equations, which was recently proposed by Guermond & Popov (2014). In contrast to the Navier–Stokes fluxes, the new regularization adds a viscous term to the mass equation. Since non-physical viscous terms are used, it is important to show that the exact solution's properties, such as the location of shocks, contact and rarefaction waves are not violated. The present study concerns a careful numerical investigation of the new viscous regularization in a number of well-known 1D and 2D benchmark problems. Also, a direct numerical comparison with respect to the physical Navier–Stokes regularization is shown. The numerical tests show that the entropy viscosity method can achieve high order accuracy for any polynomial degrees. Detailed algorithms for the implementation of a slip wall boundary condition are presented in a weak and a strong form.

Keywords: Entropy viscosity; Conservation laws; Compressible flow; Euler equations; Finite elements; Nonlinear stabilization

1. Introduction

One of the big challenges in Computational Fluid Dynamics is to design and implement efficient high-order accurate numerical methods for approximating nonlinear hyperbolic systems of conservation laws. Since high order discretizations produce spurious oscillations in shock regions, nonlinear stabilization techniques are needed to avoid or control these oscillations. There exists a large class of methods addressing the question of instabilities of high-order methods and most of them rely on flux/slope limiters. The design of limiters is not an easy task, especially in the case of unstructured meshes. One of the traditional ways of solving the entropy solution of conservation laws consists in adding an artificial viscosity term: a mesh dependent parabolic operator. This approach has been introduced by Von Neumann and Richtmyer in 50s for the numerical approximation of conservation laws. The classical artificial viscosity term is constructed to be proportional to the gradient of the solution, which loses regularity near shock

* Corresponding author.
E-mail address: murtazo.nazarov@it.uu.se (M. Nazarov).

http://dx.doi.org/10.1016/j.cma.2016.12.010
0045-7825/

and sharp discontinuities. To overcome this issue, a residual based stabilization term is used in Streamline Upwind Petrov–Galerkin schemes (SUPG) by Hughes and Mallet [1]. Later, Johnson et al. added an isotropic residual based artificial viscosity to SUPG [2] to obtain the convergence to the unique entropy solution. Recently, it has been shown in [3] that Streamline Diffusion terms are not necessary for the method to converge. The nonlinear isotropic viscosity is the main ingredient for the L^∞-bound and the convergence. Dropping Streamline Diffusion terms, simplifies the scheme significantly and there will be no demand to keep the method implicit, see e.g., [4] and [5] where the residual based artificial viscosity method is successfully applied for systems of compressible Euler equations using explicit Runge–Kutta methods.

In practice the nonlinear artificial viscosity is constructed locally in each cell, which may introduce nonphysical oscillations in the solution. The work by Barter and Darmofal [6] and Reisner et al. [7] constructs a smoother variation of artificial viscosity by solving additional scalar reaction–diffusion equation. This approach removes small nonphysical oscillations introduced by non-smooth viscosity and does not pollute the solution in the downstream region.

The so-called *entropy viscosity* method was first introduced by Guermond et al. [8,9], where the diffusive coefficient is constructed using the entropy residual. The key feature of the entropy viscosity method is that the amount of artificial viscosity it introduces, is determined by the size of the entropy residual. Scalar conservation equations have infinitely many entropy pairs and all physical systems have at least one entropy function satisfying an auxiliary entropy equation/inequality. The entropy "equation" is an equation only in the regions where the solution is smooth and it becomes an inequality in the shock regions. This inequality acts as a selection principle for choosing the physically relevant solution. The amount of violation of the entropy equation is called entropy production or entropy residual. The main idea is that there is a large entropy production in strong shocks (it can be proved in simple cases that the entropy production is a Dirac measure supported in shocks) which then activates the entropy viscosity wherever a shock or a discontinuity is traced. In this paper, the entropy viscosity is constructed on the finite element nodes rather than on the cells. This definition of the viscosity turns out to be more accurate especially for higher polynomial spaces.

Physics-based viscous fluxes or Navier–Stokes viscous terms are widely used in the literature in numerical approximations of compressible Euler equations. For instance, we refer to [10,6,11] for discontinuous Galerkin, [9,12,13,4] for continuous Galerkin and [14,5] for spectral element methods, where the artificial viscosity terms are constructed via physics based viscosity. In above-mentioned references, several high-order numerical algorithms for the construction of artificial viscosity coefficient are proposed and many realistic flow problems in different physics are solved.

However, since the Navier–Stokes fluxes do not involve any viscosity in the mass equation, the solution suffers from the so-called Gibbs phenomenon and introduces dissipation errors when using non-smooth initial data: or, more precisely, the so-called positivity property of density cannot be guarantied. By constructing very simple flow scenarios one can show that Navier–Stokes fluxes violate the positivity properties of the density and internal energy, and do not satisfy entropy inequalities, see [15, Sec 2.4.]. These theoretical findings question the usage of the Navier–Stokes fluxes to regularize the Euler equations. Recently, a general class of viscous regularization of the Euler equations, that satisfy all above-mentioned thermodynamic properties, was introduced by Guermond and Popov [15], which is coined as Guermond–Popov fluxes in this paper.

In [16], the Guermond–Popov fluxes are used to solve compressible flow in various speeds for linear finite element approximation. The main goal of this paper is to investigate the Guermond–Popov fluxes numerically in continuous finite element setting in arbitrary polynomial degrees and space dimensions. Since, this is an alternative to the standard physics based viscous terms, it is important to show at least numerically that the method converges to the entropy solution. For instance, computing correctly the position of shocks, contact discontinuities and rarefaction waves are one of the main requirements that need to be assessed.

Most applications involving the compressible Euler system require to impose the so-called slip boundary condition, i.e., normal velocity should vanish on the boundary. This boundary condition is challenging to implement for general complex geometries. In this paper two different approaches to impose this boundary condition are discussed, namely weakly and strongly, for compressible flows and compare their performance using a realistic flow problem. The strong implementation of slip boundary condition for incompressible flow was reported earlier in [17] and [18].

The problem formulation is stated in Section 2. Next, Section 3 recalls the standard Navier–Stokes fluxes and introduces the Guermond–Popov regularization, together with the finite element space discretization and the time