

## COMPUTATIONAL EULERIAN HYDRODYNAMICS AND GALILEAN INVARIANCE

BRANT E. ROBERTSON<sup>1,2,5</sup>, ANDREY V. KRAVTSOV<sup>1,2</sup>, NICKOLAY Y. GNEDIN<sup>1,3</sup>, AND TOM ABEL<sup>4</sup>

*Draft version August 5, 2009*

### ABSTRACT

Eulerian hydrodynamical simulations are a powerful and popular tool for modeling fluids in astrophysical systems. However, these methods have recently been criticized for producing numerical results that differ depending on a system's bulk velocity with respect to the computational grid. This behavior has been interpreted as evidence that Eulerian methods are not generically Galilean invariant. Given the ubiquity of supersonic bulk flows in astrophysical systems, such a failure would imply that Eulerian methods have limited applicability for modeling astrophysical problems, especially in the context of structure and galaxy formation. In this work, we critically examine the performance of Eulerian hydrodynamic simulations in the presence of a uniform bulk velocity. We demonstrate that numerical diffusion associated with advection of the fluid through a discrete computational grid with discrete time steps accounts for the effects previously identified as evidence for Galilean non-invariance in Eulerian methods. Consequently, for different bulk velocities different results are obtained *only when the spatial resolution of the simulation is kept fixed*. We demonstrate that with increased resolution Eulerian methods converge to the Galilean invariant result in problems where a well-defined convergent solution exists. In particular, we show that Kelvin-Helmholtz instabilities develop properly in realistic Eulerian simulations regardless of the bulk velocity, provided the problem is simulated with sufficient resolution (a factor of 2-4 increase compared to the case without bulk flows for realistic velocities). Our results show that high-resolution Eulerian methods can perform well in the presence of supersonic bulk flows, especially when aided by Adaptive Mesh Refinement, and should continue to be a highly competitive and attractive choice for modeling astrophysical fluids.

*Subject headings:*

### 1. INTRODUCTION

Eulerian methods have been the tool of choice in computational fluid dynamics for over five decades. Many successful Eulerian methods in popular use descended from the Godunov (1959) scheme that combines the analytical Riemann solution of the Euler equations<sup>6</sup> with the upwind scheme of Courant et al. (1952) to numerically evolve fluid systems on a discretized mesh. These Godunov-type schemes, as such methods are commonly called, have been further engineered to include higher-order spatial reconstructions of the fluid distribution based on piecewise linear (e.g., van Leer 1977), parabolic (e.g., PPM, Colella & Woodward 1984), or, more generally, higher-order weighted essentially non-oscillatory interpolation schemes (Liu et al. 1994). Eulerian methods have also become quite popular in addressing astrophysical problems (e.g., Fryxell et al. 1989; Cen et al. 1990; Bryan et al. 1994; Quilis et al. 1996; Yepes et al. 1997; Wada & Norman 1999; Ricker et al. 2000), especially in the framework of Adaptive Mesh Refinement (AMR, e.g., Bryan & Norman 1997; Khokhlov 1998; Truelove et al. 1998; Fryxell et al. 2000; Plewa & Müller 2001; Kravtsov et al. 2002; Teyssier 2002; Quilis 2004). Given their wide-spread use in computational

astrophysics, an understanding of the fundamental limitations of such codes is important for interpreting the astrophysics of hydrodynamical systems that cannot be accessed through laboratory experiments.

While Eulerian astrophysical simulation codes routinely demonstrate excellent performance on idealized test cases, some shortcomings of these methods are known (e.g., Quirk 1994, 2005). Recently, several studies have focused on the differences produced by Eulerian codes in reference frames moving with different velocities with respect to the computational grid. Wadsley et al. (2008) emphasized the role of diffusion in altering the development of Kelvin-Helmholtz instabilities in the FLASH code (Fryxell et al. 2000) simulations of buoyant, hot bubbles. Tasker et al. (2008) simulated the advection of otherwise static, self-gravitating gas clouds, and showed that the performance of FLASH and the PPM version of Enzo (O'Shea et al. 2004) in maintaining the centroid and density profile of the gas cloud depended on its velocity with respect to the static computational grid. Most recently, Springel (2009) motivated the development of the new Lagrangian-Eulerian moving-mesh code AREPO by demonstrating that with fixed grid Godunov solvers Kelvin-Helmholtz instabilities may not develop and evolve properly when the interface between the two fluids has a large bulk velocity with respect to the grid. These apparent failures of Eulerian codes have been discussed in terms of "Galilean non-invariance," which in this context means that for initial conditions that move with different uniform bulk velocities with respect to the computational grid but are otherwise identical, numerical solutions obtained with Eulerian codes may depend on the chosen bulk velocity.

Given the ubiquity of supersonic bulk motions in astrophysical scenarios, these results are potentially damning for the application of stationary mesh Eulerian codes to galaxy and

<sup>1</sup> Kavli Institute for Cosmological Physics, and Department of Astronomy and Astrophysics, University of Chicago, 933 East 56th Street, Chicago, IL 60637, USA

<sup>2</sup> Enrico Fermi Institute, 5640 South Ellis Avenue, Chicago, IL 60637, USA

<sup>3</sup> Particle Astrophysics Center, Fermilab, Batavia, IL 60510, USA

<sup>4</sup> Kavli Institute for Particle Astrophysics and Cosmology, Stanford University, 2575 Sand Hill Road, Menlo Park, CA 94025, USA

<sup>5</sup> Spitzer Fellow

<sup>6</sup> There are also Eulerian astrophysical hydrodynamics codes that do not use a Godunov scheme, such as the ZEUS code (Stone & Norman 1992a,b; Clarke 1996; Hayes et al. 2006) and the code by Ryu et al. (1993) based on the TVD flux-corrected method by Harten (1983)

structure formation. The purpose of this work is to critically examine the performance of Eulerian hydrodynamical codes for simulating systems with supersonic bulk motions, and to clarify both the nature and meaning of the velocity-dependent differences highlighted in previous studies. Specifically, we use the Eulerian mesh codes ART (Kravtsov et al. 2002) and Enzo (O’Shea et al. 2004) to simulate the development of Kelvin-Helmholtz instabilities in test calculations similar to those presented in Springel (2009). We employ statistical measures to quantify convergence and error of the calculations in addition to an extensive visual comparison of the solutions. We show that the effects discussed above are not a consequence of Galilean non-invariance of Riemann solvers, but rather a result of diffusive errors accumulated during advection of fluid through the computational grid. The effects of these errors are thus particularly acute in systems where perturbations and the interface between fluids are under-resolved. We demonstrate that with a proper initial setup the Eulerian methods produce a convergent solution at large bulk velocity as the resolution of the simulation is increased.

The paper is organized as follows. In §2, we discuss the origin of numerical diffusion in the Eulerian method and illustrate its effects using simulations of contact discontinuities. Readers familiar with the effects of numerical diffusion should proceed to §3, where we review previous simulations of Kelvin-Helmholtz instabilities and the related claims of Galilean non-invariance in Eulerian methods. In §4, we present a new, better-behaved test calculation of Kelvin-Helmholtz instability and study the development of the instability over a range of resolutions and supersonic bulk motions. We study the statistical and error properties of the Kelvin-Helmholtz simulations and use these statistics to critically examine the apparent Galilean non-invariance of Eulerian simulation codes. We discuss our results in §5 and present a summary in §6.

## 2. NUMERICAL DIFFUSION

Computational Eulerian hydrodynamical codes calculate the evolution of fluid systems using a discretized approximation to Euler’s equations. When modeling the conservative form of Euler’s equations, the change in quantities like density or energy integrated over cell units of size  $\Delta x$  in the discretized mesh over a time step  $\Delta t$  will correspond to the flux of those quantities across the cell boundaries over the same time interval. Fluid interactions between cells then fundamentally involve calculations of the fluxes, which can be approximated using solutions to the Riemann problem (i.e., Godunov 1959) or through other means (e.g., the flux corrected methods of Boris & Book 1973, and Harten 1983, see also Chapter 21 of Laney 1998). Since these numerical approximations to the physical fluxes exchanged between fluid volumes during the time interval  $\Delta t$  are discretized, there is a truncation error associated with the numerical approximation. Missing or extraneous higher order terms in the discretized numerical approximation can appear as an effective viscosity or thermal conductivity and lead to the smearing or dispersion of features in fluid flow. We will refer to smearing effects as numerical diffusion, while effects that change the wave speed of features in the fluid will be labeled numerical dispersion<sup>7</sup>. Clear discussions about the effects of numerical diffusion can be found in

Boris & Book (1973) and Laney (1998).

The strength of numerical diffusion depends on the method chosen to model fluid systems. Lagrangian methods integrate the convective derivative form of the mass conservation equation directly, and therefore suffer from small diffusive truncation errors. Eulerian methods calculate the advective term in the mass conservation equation explicitly, which can lead to an appreciable diffusive truncation error upon discretization. Some Eulerian methods, such as Flux-Corrected Transport algorithms (e.g., Boris & Book 1973), include an explicit numerical diffusion term proportional to a second spatial derivative that owes to their forced conservative and non-negative properties (the “flux correction” refers to the explicit artificial anti-diffusion used to correct this truncation error term).

For Godunov-type methods based on Riemann solvers, differences in the amount of numerical diffusion can arise from the approximations made in constructing the discrete representation of the local fluid flow on the computational mesh. In Godunov-type methods, the numerical flux between cells is determined by the known solution of the piecewise-constant Riemann problem. The resulting flux across the cell face is then determined by the properties of fluid states on either side of the face, the cell size, and the time step size. Resolution determines the region used to average the fluid properties for finding the initial states in the Riemann problem. The averaging procedure introduces numerical diffusion, and can be counter-acted by higher spatial resolution. Improving the quality of the approximation to the fluid states used in the Riemann problem can also decrease the amount of numerical diffusion, so the method used to model the shape of the fluid flow on the grid can change the diffusivity of the method. For instance, the local flow can be approximated by constant (Godunov 1959), linear (van Leer 1977), parabolic (Colella & Woodward 1984), or higher-order piecewise polynomial (Liu et al. 1994) interpolations on the discrete mesh. Higher-order interpolations improve the local approximations used in reconstructing the fluid flow and calculating the initial states to the Riemann problem, and therefore will suffer from less numerical diffusion. In general, the strength of numerical diffusion will also depend on the local flow velocity. This velocity dependence arises because, in the presence of a large advective flow, more time steps are used and more local averages are performed. Additionally, with a large bulk velocity a larger region of the fluid is averaged to calculate the input states for the Riemann problem.

We can illustrate how numerical diffusion affects the shape of the local fluid distribution by simulating the advection of contact discontinuities. In the absence of numerical diffusion, the square wave should be perfectly advected and the contact discontinuities would remain sharp. However, as these simple tests will illustrate, numerical diffusion will act to soften the contact discontinuities in a resolution- and velocity-dependent manner. The effects of numerical diffusion in these tests will prove to be informative for simulations of the Kelvin-Helmholtz instability in §3 and 4.

The simulations presented in this paper use the Eulerian code ART with piecewise-linear reconstruction and an exact Riemann solver (Colella & Glaz 1985), based on the adaptive refinement strategy developed by Khokhlov (1998), unless otherwise noted. For the following square wave advection simulations, time steps were determined using the Courant-Friedrichs-Lewy condition with a parameter  $cfl = 0.6$ . ART uses a dual-energy formulation similar to that of Bryan et al. (1995), such that the internal energy equation is followed sep-

<sup>7</sup> We note that the numerical diffusion owing to truncation error in the Eulerian method is very distinct from artificial viscosity employed in Smoothed Particle Hydrodynamics to improve shock capturing, and the two should not be confused.