New Astronomy 14 (2009) 71–87

Contents lists available at ScienceDirect

New Astronomy

journal homepage: www.elsevier.com/locate/newast

RAPID: A fast, high resolution, flux-conservative algorithm designed for planet–disk interactions

L.R. Mudryk *, N.W. Murray

Canadian Institute for Theoretical Astrophysics, Toronto, ON, Canada M5S 3H8

ARTICLE INFO

Article history:
Received 22 February 2008
Received in revised form 25 April 2008
Accepted 6 May 2008
Available online 15 May 2008
Communicated by W.B. Hubbard

PACS:
95.30.Lz
95.75.Pq
97.10.Gz
97.82.Jw

Keywords:
Accretion disks
Hydrodynamics
Methods: numerical
Planetary systems: formation

ABSTRACT

We describe a newly developed hydrodynamic code for studying accretion disk processes. The numerical method uses a finite volume, non-linear, Total Variation Diminishing (TVD) scheme to capture shocks and control spurious oscillations. It is second-order accurate in time and space and makes use of a FARGO-type algorithm to alleviate Courant–Friedrichs–Lewy time step restrictions imposed by the rapidly rotating inner disk region. OpenMP directives are implemented enabling faster computations on shared-memory, multi-processor machines. The resulting code is simple, fast and memory efficient. We discuss the relevant details of the numerical method and provide results of the code's performance on standard test problems. We also include a detailed examination of the code's performance on planetary disk–planet interactions. We show that the results produced on the standard problem setup are consistent with a wide variety of other codes.

1. Introduction

The study of almost all astronomical objects relies on an understanding of their hydrodynamics. Indeed, for many such objects the involved hydrodynamics are complex enough to require numerical modeling. The process of numerical modeling usually proceeds by writing partial differential equations describing the behavior of a continuous medium as an equivalent set of algebraic equations for a finite set of discretized elements. This discretization can generally be performed in two different ways. In an Eulerian approach, one discretizes the spatial domain into volumes termed grid cells. The fluid is considered to move through this fixed background grid. By contrast, in a Lagrangian approach the fluid is discretized into fluid elements (or 'particles') which can then move freely according to their initial velocities, and only their interactions need to be modeled. Lagrangian methods work well in situations with large background flows where Eulerian methods would spend the bulk of their time advecting the (uninteresting) balanced flow, accumulating numerical errors with the numerous iterations required. Lagrangian methods have a large dynamic range in length but not in mass, achieving good spatial resolution in high-density regions but performing poorly in low-density regions. In addition, the usual implementations of Lagrangian methods, based on smoothed particle hydrodynamics (SPH), do not easily allow the higher spatial accuracy that grid methods can employ nor do they capture shocks as accurately as grid methods. By contrast, Eulerian methods provide a large dynamic range in mass but not in length. In general they are also computationally faster by several orders of magnitude, easier to implement, and easier to parallelize.

The RAPID code (Rapid Algorithm for Planets In Disks), which we present here, uses an Eulerian approach, adapted for a cylindrical grid. While we focus on planet–disk interactions in this paper, the code is intended for the general study of accretion disks containing a dominant central mass. In such systems, the gas disk surrounding the central object is of a small enough mass that its self-gravity may be ignored. Such disks will have a roughly Keplerian velocity profile resulting from the mass of the central object. In order to obtain higher algorithm efficiency in the presence of this Keplerian flow, we make use of a FARGO-type algorithm (Masset, 2000). The algorithm's underlying strategy is to subtract off the bulk flow, which can be considered simply a translation of grid quantities, leaving the dynamically important residual velocity. RAPID is second-order

* Corresponding author.
E-mail addresses: mudryk@cita.utoronto.ca (L.R. Mudryk), murray@cita.utoronto.ca (N.W. Murray).

1384-1076/$ - see front matter © 2008 Elsevier B.V. All rights reserved.
doi:10.1016/j.newast.2008.05.002

accurate in space and time. Advection is accomplished through a non-linear Total Variation Diminishing (TVD) scheme, which helps to control spurious oscillations. Time-stepping is accomplished through a standard Runge–Kutta scheme. Operator splitting is used to account for multiple dimensions and source terms such as those due to gravitational potentials and viscosity.

In Section 2, we outline the fluid equations to be solved and discuss considerations of angular momentum important for accuracy on cylindrical grids. In Section 3, we discuss the details of the RAPID algorithm. We provide results of basic hydrodynamical tests in Section 4 and demonstrate the code's performance on typical planetary disk setups in Section 5. Conclusions are presented in Section 6.

2. Eulerian hydrodynamics

The Navier–Stokes equations may be written as

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \boldsymbol{u} = 0 \tag{1}$$

$$\frac{\partial \rho \boldsymbol{u}}{\partial t} + \nabla \cdot [\rho \boldsymbol{u}\boldsymbol{u} + p\boldsymbol{I}] = -\rho \nabla \phi + \nabla \cdot \boldsymbol{\sigma} \tag{2}$$

$$\frac{\partial \rho e}{\partial t} + \nabla \cdot [(\rho e + p)\boldsymbol{u}] = -\rho \boldsymbol{u} \cdot (\nabla \phi) + \nabla \cdot [\boldsymbol{u} \cdot \boldsymbol{\sigma}] - \nabla \cdot \boldsymbol{\psi}, \tag{3}$$

for the mass density ρ, momentum density $\rho \boldsymbol{u} = \rho(u_1, u_2, u_3)$, and total energy density $\rho e = \rho\varepsilon + \frac{1}{2}\rho \boldsymbol{u}^2$ of a fluid volume. The symbols ε and p represent the internal energy per mass and the pressure of the fluid, ϕ represents the potential due to a body force (such as that from an external gravitational field), $\boldsymbol{\sigma}$ represents the non-isotropic component of the stress–strain tensor for the fluid, and $\boldsymbol{\psi}$ represents any heat flux. We use the symbol $\boldsymbol{I}$ for the identity matrix and note that the combination $\rho \boldsymbol{u}\boldsymbol{u} \equiv \rho u_i u_j$ is a direct product yielding a matrix for the momentum fluxes.

These equations express the transfer of mass, linear momentum, and energy within the fluid volume written out in an arbitrary coordinate system. In terms of a general solution vector $\boldsymbol{q} = (\rho, \rho u_1, \rho u_2, \rho u_3, \rho e)$, flux tensor $\boldsymbol{F}(\boldsymbol{q})$, and source vector $\boldsymbol{S}$, these equations all have the same formally simple form

$$\frac{\partial \boldsymbol{q}}{\partial t} + \nabla \cdot \boldsymbol{F} = \boldsymbol{S}. \tag{4}$$

In the case where $\boldsymbol{S} = 0$ the equations reduce to the conservation form of the Euler equations, expressing non-dissipative advection of fluid quantities.

In Cartesian coordinates and for the solution vector $\boldsymbol{q} = (\rho, \rho u_x, \rho u_y, \rho u_z, \rho e)$, the Euler equations describe the evolution of five conserved scalar quantities. However, written in cylindrical coordinates, where $\boldsymbol{x} = (r, \theta, z)$, and for the natural choice of solution vector $\boldsymbol{q} = (\rho, \rho u_r, \rho u_\theta, \rho u_z, \rho e)$, only three components of this vector are conserved scalar quantities. The quantities ρu_r and ρu_θ are not conserved. We thus choose to solve for the solution vector $\boldsymbol{q} = (\rho, \rho u_r, \mathscr{H}, \rho u_z, \rho e)$, where the quantity $\mathscr{H} = \rho r(u_\theta + r\Omega)$ is the fluid's angular momentum in the inertial frame (we will refer to this quantity as the inertial angular momentum). It includes contributions from the fluid's angular velocity $\omega = u_\theta / r$, as well as the reference frame's angular velocity Ω, assumed to be oriented along the z-axis. Because inertial angular momentum is conserved in a rotating system, it is a more natural physical variable to use and doing so improves the accuracy of the results (see Section 5.4). In Appendix A we write out modified versions of Eqs. (1)–(3) for this choice of solution vector, expressed in cylindrical coordinates.

3. Numerical method

The solution method we describe is based on the relaxing Total Variation Diminishing (TVD) method by Jin and Xin (1995). This method has been successfully applied by Pen (1998) and Trac and Pen (2003, 2004) to solve the Euler equations on Cartesian grids in astrophysical simulations.

3.1. Relaxation system

The relaxing TVD method solves the Euler equations by assuming the equations may be split into components corresponding to leftward and rightward-travelling waves. In place of the Euler equation (Eq. (4) with $\boldsymbol{S} = 0$), the following coupled system is solved along a single grid direction for the solution vector $\boldsymbol{q}$:

$$\frac{\partial \boldsymbol{q}}{\partial t} + \frac{\partial}{\partial x}(c\boldsymbol{w}) = 0 \tag{5}$$

$$\frac{\partial \boldsymbol{w}}{\partial t} + \frac{\partial}{\partial x}(c\boldsymbol{q}) = 0. \tag{6}$$

The relations $\boldsymbol{q} = \boldsymbol{q}^{\mathrm{R}} + \boldsymbol{q}^{\mathrm{L}}$, and $\boldsymbol{w} = \boldsymbol{F}/c = \boldsymbol{q}^{\mathrm{R}} - \boldsymbol{q}^{\mathrm{L}}$, define the solution variables in terms of the leftward and rightward-travelling waves. Eq. (6) represents a separate equation for the evolution of the normalized flux vector $\boldsymbol{w}$. The variable c is a positive-definite function which has the interpretation of a speed associated with a particular grid cell. The solution is stable in the sense that its total variation (see Section 3.3) decreases as long as all values of c are greater than or equal to the largest eigenvalue of the flux Jacobian $\partial \boldsymbol{F}(\boldsymbol{q})/\partial \boldsymbol{q}$ (Jin and Xin, 1995). Because the waves are split into separate rightward and leftward components, the maximum eigenvalue of the Jacobian is limited for both components by the value $c_i = |u_i| + c_s$ where c_s is the sound speed for the cell. Substituting these definitions into Eqs. (5) and (6) decouples the system and yields

$$\frac{\partial \boldsymbol{q}}{\partial t} + \frac{\partial \boldsymbol{F}^{\mathrm{R}}}{\partial x} - \frac{\partial \boldsymbol{F}^{\mathrm{L}}}{\partial x} = 0, \tag{7}$$

where $\boldsymbol{F}^{\mathrm{L}} = c\boldsymbol{q}^{\mathrm{L}}$ and $\boldsymbol{F}^{\mathrm{R}} = c\boldsymbol{q}^{\mathrm{R}}$. The original coupled system, Eqs. (5) and (6), is then equivalent to the solutions of the two separate leftward- and rightward-moving waves given in Eq. (7). It is now possible to separately solve for each of the travelling waves and add the results to determine the full solution along a single direction.

3.2. Solution of wave-split, one-dimensional Euler equations

In order to solve for the advection of the separately travelling waves, we implement a second-order Runge–Kutta scheme which uses a TVD flux-interpolation scheme to control spurious oscillations. Consider the integral form of the classical Euler equations (Eq. (4) with $\boldsymbol{S} = 0$) in one dimension and for a single conserved fluid quantity so that $\boldsymbol{q}(\boldsymbol{x}, t) \equiv q(x, t)$. We can write this form as

$$\frac{\partial}{\partial t}\int_{x_A}^{x_B} q(x,t)\mathrm{d}x + \frac{\partial}{\partial x}\int_{x_A}^{x_B} F(x,t)\mathrm{d}x = 0. \tag{8}$$

We discretize the N-dimensional spatial domain into a uniformly spaced grid of points x_i, defined to be the centers of uniformly packed rectangular N-volumes (cells). Using this simple discretization, fluid quantities defined at the cell-centered grid points may be interpreted as the cell-averaged value of the solution vector for that grid point. For a single one-dimensional cell at location x_i and with boundaries at $x_A = x_{i-1/2}$ and $x_B = x_{i+1/2}$, the integrals $\int_{x_A}^{x_B} q\,\mathrm{d}x$ and $\int_{x_A}^{x_B} F(x,t)\mathrm{d}x$ in Eq. (8) represent the cell-averaged fluid quantities q_i and F_i. Discretizing Eq. (8) in terms of these cell-averaged quantities yields

$$\frac{q_i^{t+\Delta t} - q_i^t}{\Delta t} + \frac{F_{i+1/2}^t - F_{i-1/2}^t}{\Delta x} = 0, \tag{9}$$

where superscripts reference the specific time step and subscripts reference the spatial cell. Computing $q_i^{t+\Delta t}$ for any grid cell thus re-