Numerical modelling of hydrodynamical astrophysical outflows: application using the PLUTO code

I P van der Westhuizen[1], **B van Soelen**[1], **P J Meintjes**[1], **S J P K Riekert**[2] **and J H Beall**[3]

[1]Department of Physics, University of the Free State, Bloemfontein, 9301, SA
[2]High Performance Cluster, University of the Free State, Bloemfontein, 9301, SA
[3] St. John's Collage, Annapolis, MD, 21401, USA

E-mail: `VanDerWesthuizenIP@ufs.ac.za`

Abstract. In order to gain a better understanding of how structures form and evolve in astrophysical outflows it is necessary to build a numerical model to simulate the motion of material in these environments. Due to the large length scales of astrophysical outflows compared to the Debye radius of the ejected particles, a fluid dynamical approach provides a suitable model for the large scale motion of the outflow. In this study a 3 dimensional relativistic fluid dynamical model was set up on a structured mesh with dimensions of 64x64x64 units. A uniform ambient medium was defined on the mesh grid and a nozzle of radius 1 was created on the initial z boundary to inject a jet with a Lorentz factor of 10. The opensource magnetohydrodynamical code PLUTO was used to evolve the numerical model with time. The PLUTO code uses high resolution shock capturing schemes to evolve the time dependent partial differential conservation equations on the structured mesh. This model led to the formation of a collimated relativistic beam surrounded by a cocoon of backflow material. Asymmetric turbulence was found, which caused instabilities in the central beam and led to the formation of shocks in the jet. These results are consistent with those seen in previous studies, which were used in order to validate the simulation.

1. Introduction

Observational studies have shown that many astrophysical sources produce collimated outflows of material. Some of these objects include radio loud Active Galactic Nuclei (AGN), accreting binary systems and Young Stellar Objects (YSO) [1]. High resolution imaging of sources such as radio loud AGN have revealed the presence of complex small scale structures in the outflows. These structures include radio blobs which are time dependent and propagate through the outflows at superluminal velocities [2]. The formation and propagation of these time dependent structures may be a cause of variability in sources containing relativistic outflows. The processes which cause the formation of such structures within the relativistic outflows are not well understood and they are often too complex for analytical analysis. In such cases numerical simulations provide a powerful analysis tool to investigate the formation and evolution of these outflows. In this study we focus on AGN jets and their properties, since their structures have been observed over a wide range of the electromagnetic spectrum.

AGN are some of the most energetic objects in the universe and emit large amounts of radiation over a wide range of the electromagnetic spectrum. The non-thermal emission received from radio loud AGN is dominated by the jet component emerging from the central engine [3]. The biggest challenge with creating a numerical model for such sources is the large range of length scales which have to be taken into consideration. This includes the effects of ion interactions in the plasma on atomic length scales as well as the large scale hydrodynamic motion of the plasma, which in the case of AGN jets, can stretch over kiloparsec scales. We can, however, simplify the problem by considering a relativistic jet that has to conserve charge neutrality, which causes the Debye radius of the electrons and protons in the jet to be negligible when compared with the jet radius. This allows us to create an appropriate model using purely relativistic magnetohydrodynamics [4].

In this paper a preliminary relativistic outflow model was created and evolved using the PLUTO opensource code. The initial model was developed to test the computational intensiveness of the simulations on the University of the Free State High Performance Cluster and to validate the base code before incorporating more complex effects into the model. In this paper section 2 will focus on the numerical methods used to create the relativistic outflow model as well as the setup of the simulation, section 3 will give the preliminary results and compare our simulation to previous studies. Finally section 4 contains a short conclusion.

2. Numerical method

The simulation of a relativistic hydrodynamical outflow is created by numerically solving the fluid dynamical conservation equations on a structured mesh grid. We implemented this using the PLUTO *ver.* 4.0 opensource code [5]. The basis of computational fluid dynamics, the PLUTO code and our simulation setup is discussed in the sections below.

2.1. Computational fluid dynamics

The motion of a relativistic fluid can be described by a set of partial differential equations, which are called the hydrodynamical conservation equations and has the following general form

$$\frac{\partial \vec{U}}{\partial t} + \nabla \cdot \vec{\vec{T}}(U) = \vec{\vec{S}}(U), \tag{1}$$

where $\vec{U}$ is a column vector consisting of the conserved variables, $\vec{\vec{T}}(U)$ is a tensor containing the flux vectors as a function of the conserved variables in every direction and $\vec{\vec{S}}(U)$ is a tensor containing the source terms, which are additional terms introduced by effects such as viscosity and gravitational forces [6].

For an ideal relativistic fluid with no magnetic field the variables in equation 1 are described by:

$$\vec{U} = \begin{bmatrix} \rho\Gamma \\ \rho\Gamma^2 h\mathbf{v} \\ \rho\Gamma^2 h - p \end{bmatrix}, \quad \vec{\vec{T}}(U) = \begin{bmatrix} \rho\Gamma\mathbf{v} \\ \rho\Gamma^2 h\mathbf{v}\mathbf{v} + p\mathbf{I} \\ \rho\Gamma^2 h\mathbf{v} \end{bmatrix}, \quad \vec{\vec{S}}(U) = 0, \tag{2}$$

where ρ is the density, p is the pressure, h is the entropy, Γ is the Lorentz factor, $\mathbf{I}$ is a 3x3 unit matrix and $\mathbf{v}$ is the velocity vector.

In order to completely describe a fluid we need an equation of state (EoS), which relates different quantities in the fluid to each other. The EoS may vary from one substance to another, therefore the choice of equation plays an important role in the simulation. For example, the caloric EoS which relates the internal energy to the pressure and density of an ideal fluid is given by

$$e = \frac{p}{\rho(\gamma_{ad} - 1)}, \tag{3}$$

where e is the internal energy of the fluid and $\gamma_{ad} = C_p/C_v$ is the adiabatic index [7]. A complete introduction to computational fluid dynamics is given in [6].

Based on previous studies we chose to characterize the relativistic hydrodynamical simulation based on 5 variables namely, the Lorentz factor Γ, the Mach number M_b, the jet to ambient density ratio η and the adiabatic index γ_{ad}. Using these variables we can recover quantities such as the pressure, the energy and the velocity of the fluid [7].

For example by combining equation (3) with the definition of the sound speed in a fluid ($C_s = \frac{v}{M_b}$) we can solve for the pressure as,

$$p = \frac{(\gamma_{ad} - 1)\rho C_s^2}{\gamma_{ad}(\gamma_{ad} - 1 - C_s^2)}. \tag{4}$$

Therefore, using the proper EoS, all of the properties of the fluid can be calculated.

2.2. PLUTO relativistic magnetohydrodynamics code

To evolve the fluid dynamical simulation the opensource modular relativistic magnetohydrodynamical code PLUTO *ver.* 4.0 was used.[1] The code was specifically designed for supersonic time-dependent flows containing discontinuities, which makes it ideal for the simulation of relativistic astrophysical outflows. PLUTO uses High Resolution Shock Capturing (HRSC) algorithms to solve the fluid dynamic conservation equations and evolve them with time. It contains different modules for hydrodynamic (HD), relativistic hydrodynamic (RHD), magnetohydrodynamic (MHD) and relativistic magnetohydrodynamic (RMHD) models, which allows the code to incorporate different effects based on the conditions of the simulation. The modular nature of the code also allows the addition of effects such as gravity, viscosity and radiative cooling to be incorporated in the calculations [5].

The PLUTO code integrates and evolves equation (1) using three computational steps. The first step is to use interpolation to construct boundary values for each cell on the structured mesh. The interpolation is done based on the centre averaged values that are assigned for each cell. It then solves a Riemann problem to determine the flux vectors across each cell boundary. Finally the code evolves equation (1) with time. The PLUTO code follows these three steps regardless of which physics module is used. The effects of the different physics modules are incorporated by a conversion in the variables of equation (1) before the computational steps [5].

2.3. Simulation setup

For testing and validation purposes a 3 dimensional numerical model of a relativistic hydrodynamical jet was constructed. For this model we considered a Cartesian static mesh grid of $64 \times 64 \times 64$ units. In this simulation arbitrary units were assigned to all variables to avoid truncation errors when working with the extremely large or small values that accompany *cgs* units. In this simulation 1 unit length corresponds to the radius of the jet nozzle at the initial inlet. An initial rest background medium was assigned to the mesh grid at $t = 0$ with a uniform density and pressure. A nozzle containing the jet material was set up on the $z = 0$ boundary. Only RHD was used in this simulation and therefore no magnetic field was assigned. A pressure matched model was set up with the pressure of the material recovered using equation (4). The density of the jet medium was normalized in arbitrary units such that $\Gamma\rho_{jet} = 1$. To validate the simulation we used parameters similar to those of previous studies such as [7] and [9]. These parameters are listed in table 1.

The boundary condition for the $z = 0$ boundary was set to reflective to simulate the production of two symmetric jets on either side of the central engine. All other boundary

---

[1] The code was implemented by [5] and is available at http://plutocode.ph.unito.it/.

Table 1. Variables used in the set up of the initial conditions for the preliminary RHD jet simulation.

Parameter		Value (arbitrary units)
Lorentz factor	Γ	10
Density ratio	η	10^{-3}
Jet density	ρ_b	0.1
Ambient density	ρ_{am}	100
Mach number	M_b	3.0
Adiabatic index	γ_{ad}	5/3

conditions were set to outflow so that the matter could escape freely. The simulation was run at a resolution of 8 points per unit length. A constant inflow of jet material with $\Gamma = 10$ in the positive z direction was used. The piecewise parabolic interpolation method was used to determine the boundary values of each cell. The Hartman Lax van Leer Riemann solver with contact discontinuity (HLLC) was used to solve the Riemann problem for this model, while the time stepping was done using characteristic tracing [8].

3. Results and Discussion

The proper density, pressure and proper velocity distributions produced by the simulation were visualized as two dimensional slices through the y-axis. Figure 1 shows these slices at three different time steps. From this visualization we can see that the structure of the jet consists mainly of 6 regions. The outer region consists of the uniform unperturbed ambient medium. A terminal bow shock is formed around the body of the jet, produced by the inflow of material that compresses the surrounding ambient medium. As the ambient medium passes through the bow wave its pressure increases forming a shocked region. The high pressure and density of this region causes a discontinuity to form at the surface between the shocked medium and the jet material. This is called the working surface. The speed at which the working surface propagates through the medium determines the propagation speed of the jet.

The centre of the jet consists of the high Lorentz factor ($\Gamma = 10$) material that is flowing through the nozzle into the computational domain. This region will be referred to as the relativistic beam of the jet. As the jet material propagates through the beam of the jet it will reach the working surface at the head of the jet. At this boundary the interaction between the jet and the high pressure ambient material causes the jet material to decelerate, converting the kinetic energy into internal energy. The collision between the lower density jet material and the higher density ambient medium also causes a backflow of material surrounding the beam of the jet. This backflow forms a region of low density material between the jet and the ambient medium which prevents the interaction of jet material with the high pressure ambient medium. This region (called the cocoon) assists in keeping the jet collimated over the propagation distance [9].

The turbulent interaction of material at the head of the jet causes vortices to form in the cocoon. These vortices in the cocoon are further amplified by the formation of Kelvin-Helmholtz instabilities at the surface between the cocoon and the higher pressure ambient medium. Periodic shock waves are also formed in the central beam of the jet. These shocks are generated by perturbations in the boundary between the beam of the jet and the surrounding cocoon, and are the result of a pressure difference between material in the beam and the cocoon along with

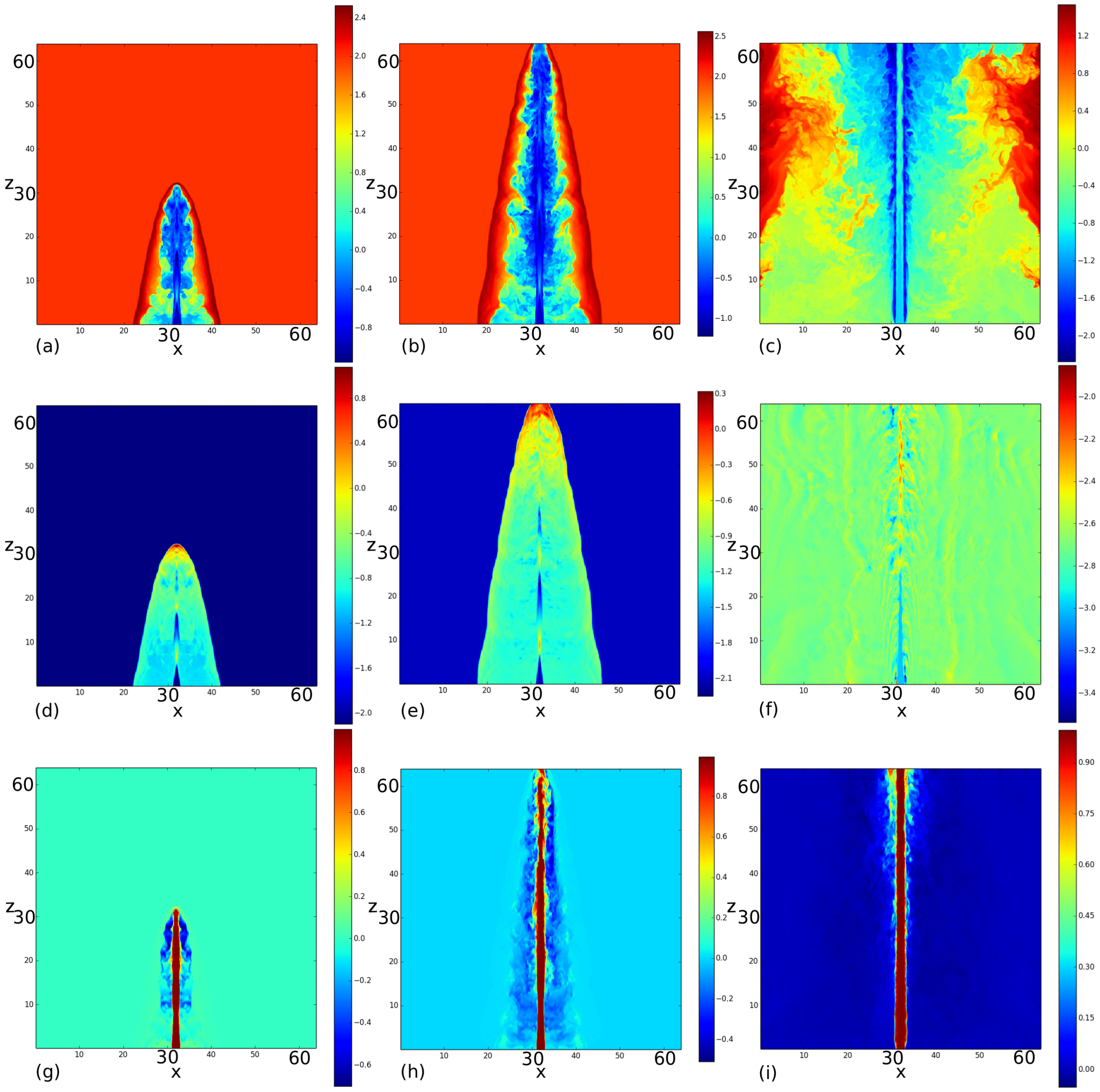

Figure 1. Two dimensional visualizations of the preliminary simulation through the y-axis of the jet, showing the proper density at (a) $t = 100$, (b) $t = 200$, (c) $t = 4300$; the pressure at (d) $t = 100$, (e) $t = 200$, (f) $t = 4300$, and the proper velocity component in the z direction at (g) $t = 100$, (h) $t = 200$, (I) $t = 4300$. Logarithmic scales are shown for the density and pressure plots in arbitrary units, while the velocity plots have a linear scaling in units of c.

the turbulence generated in the cocoon [7].

As the simulation evolves with time the cocoon of the jet expands outwards into the surrounding medium. The turbulence on the surface of the beam grows with time due to the interaction between the cocoon and jet material. This amplifies perturbations in the beam of the jet to form wavelike structures in the flow.

The simulation was run on the UFS HPC on 270 cores with a CPU time of 106098.27 h. This corresponded to 7384.7 units of simulation time. As seen in figure 1 the head of the jet leaves the computational domain at a time of 200 units, which allows for enough time for the formation of a stable jet. The real time necessary to complete the simulation amounted to 405.65 h. The simulation used a total of 112.8 Gb of memory. Higher resolution of the simulation was sacrificed for a decrease in computational resources. Additional computing power will be necessary when more complex effects such as magnetic fields are included in the simulation.

4. Conclusion

A 3D simulation of a relativistic outflow was created and evolved over time. The simulation shows a collimated central beam with little deceleration surrounded by an outer cocoon. Asymmetric structures formed in the jet, which is in accordance with observations of AGN sources and previous studies. The results shown are comparable to those in [7] and [9]. Small scale differences in the asymmetric turbulence are present in our results when compared to [7]. We attribute these differences in the jet morphology to a difference in the chosen parameters, a difference in grid structure and the fact that the model in that study was based on an axis symmetric two dimensional jet.

More complex effects such as viscosity, magnetic fields and a variable flow can now be incorporated into our model. The next step in our study will be to incorporate variable injection of jet material to investigate the formation of structures corresponding to a non-uniform flow inside the beam of the jet as well as how these structures propagate with time. This fluid dynamical simulation is not limited to the AGN case but can also be adopted for a variety of astrophysical objects such as microquasars, YSO, and X-ray binaries.

Acknowledgments

Acknowledgement is given to the HPC unit at the University of the Free State for their assistance in the installation and set up of software used to run simulations on the UFS HPC.

The financial assistance of the National Research Foundation (NRF) towards this research is hereby acknowledged. Opinions expressed and conclusions arrived at, are those of the author and are not necessarily to be attributed to the NRF.

This work is based on the research supported in part by the National Research Foundation of South Africa for the grant 87919. Any opinion, finding and conclusion or recommendation expressed in this material is that of the authors and the NRF does not accept any liability in this regard.

References

[1] De Gouveia Dal Pino E M 2005 Astrophysical jets and outflows *AdSpR* Issue 5 **35** 908-924

[2] Ghisellini G, Padovani P, Celotti A and Maraschi L 1993 Relativistic bulk motion in active galactic nuclei *ApJ* **407** 65-82

[3] Böttcher M 2011 Modeling the spectral energy distributions and variability of blazars *2011 Fermi & Jansky: Our Evolving Understanding of AGN* (*preprint arXiv:1205.0539*)

[4] Begelman M C, Blandford R D and Rees M J 1984 Theory of extragalactic radio sources *Rev. Mod. Phys.* **56** 255-351

[5] Mignone A, Bodo G, Massaglia S, Matsakos T, Tesileanu O, Zanni C and Ferrari A 2007 PLUTO: a numerical code for computational astrophysics *ApJ* **170** 228-42

[6] Toro E F 2009 *Riemann Solvers and Numerical Methods for Fluid Dynamics: A Practical Introduction Third Edition* (Berlin: Springer) Chapter 1 pp 1-40

[7] Marti J M, Mller E, Font J A, Ibez J M and Marquina 1997 Morphology and dynamics of relativistic jets *ApJ* **479** 151-163

[8] Mignone A and Bodo G 2006 An HLLC Riemann solver for relativistic flows - II. Magnetohydrodynamics *MNRAS* **368** 1040-54

[9] Rossi P, Mignone A, Bodo G, Massaglia S and Ferrari A 2008 Formation of dynamical structures in relativistic jets: the FRI case *A&A* **488** 795-806