A first order Godunov scheme
for special relativistic fluid dynamics

Danilo Durante, Giorgio Riccardi

Dept. of Aerospace and Mech. Eng., Second University of Naples, Italy

Introduction

The special relativistic equations of rest mass, momentum and energy conservation can be written in conservative form as

$$
\begin{align*}
\partial_t D + \partial_{x_k} (Du_k) &= 0 \\
\partial_t S_j + \partial_{x_k} (S_j u_k + p \delta_{jk}) &= 0 \\
\partial_t \tau + \partial_{x_k} (S_k - Du_k) &= 0
\end{align*}
$$

where: $D = \rho_0 w$, $S_j = D w u_j$ and $\tau = \rho_0 \mu w^2 - p - \rho_0 w$. The variables $\mu$ (total specific enthalpy) and $w$ (Lorentz factor) are $\mu = 1 + \gamma p / [\rho_0 (\gamma - 1)]$ ($\gamma$ is the specific heats ratio) and $w = 1 / \sqrt{1 - u_k u_k}$. As it is evident by the expression of the total specific enthalpy, the state equation of an ideal gas have been used.

In order to perform a numerical integration of (1), in the last decades Godunov type schemes have been widely adopted, because of their formal simplicity and great numerical stability. In the framework of a Godunov scheme, fluxes at the interface between numerical cells are evaluated by solving Riemann problems. The Riemann problem concerns the relaxation of a field which is discontinuous across a plane at the initial time $t = 0$. Three kind of waves arise, two of which can be rarefaction or shock waves and a third one which is a contact discontinuity. For the sake of clearness suppose that this waves travel along $x_1$ direction (i.e. the plane is the $(x_2, x_3)$ one), so that the solution of a Riemann problem can be analyzed in the meaningfull $(u_1, p)$-plane. Once the state ahead the wave (say state 1) is fixed, you can find all the possible states behind (say 2). The wave pattern which follows will be indicated as belonging to the first family; in this way shock waves or rarefactions of the first family will be referred respectively to SCN or RCN curves. Similarly we can consider the same state (say 4) as the one behind the wave, in order to find all the possible states ahead (say 3). In this last case it will be considered NCS or NCR curves of the second family.

The foundamental task of a Riemann solver is to find the intermediate pressure and velocity (state 2 and 3) in a Riemann problem. The knowledge of the wave pattern allows to strongly improve the performances of the solver itself. This issue has been deeply investigated in the paper of Rezzolla and Zanotti [2], where analytical bounds for all the possible wave patterns
have been deduced, by considering the relativistic relative velocity $v_{14}$ of the state 1 with respect to the one 4 (or viceversa). It is proved that $v_{14}$ is a monotonic increasing function of the pressure $p_2$ (or $p_3$) in the intermediate states 2 and 3. This analysis allows us to define a priori the kind of transformation; however it holds for vanishing tangent velocities ($u_2 = u_3 = 0$) only.

In the framework of the two shock approximation of Colella, the paper of Balsara [3] analyzes the use of two different numerical procedures: secant and Newton methods. By analyzing a lot of numerical tests, Balsara concluded that the Newton method is the best choice to obtain high accuracy, but he never talks about an initialization procedure. On the contrary, our numerical work shows that such procedure is needed, in order to avoid failures during time integration.

**Relativistic Riemann solver**

The Riemann problem consist in finding the pressure and the velocity into the intermediate state. It has been chosen to find them iteratively, in order to obtain an high precision Riemann solver, so a Newton algorithm has been built. The principal drawback of this code is the necessity of initialize it closer enough to the final solution. Some ad-hoc initialization techniques have been so developed. With a first technique the pressure is discretized with a step $\Delta p$. In correspondence with a value $p_k = k \Delta p$ of the pressure, the velocities for the first $u^I_k$ and second $u^{II}_k$ families of waves are evaluated. For each pressure interval $[p_{i-1}, p_i]$ the straight lines passing on the points $(u^I_{i-1}, p_{i-1})$, $(u^I_i, p_i)$ for the first family and on the relative points for the second one are considered and their intersection is evaluated. If the inequality $p_{i-1} \leq p_{\text{int}} < p_i$ is verified, the pressure $p_{\text{int}}$ is selected as initial data for a Newton method. As second technique, starting by the lower or the upper pressure $p_i$ between the states 1 and 4 (depending by the particular transformation), it can be traced one tangent to the first family curve (relative to the state 1) and another one to the second family curve (relative to the state 4). Once the intersection point of the two lines have been found, the middle pressure $(p_i + p_{\text{int}})/2$ is used as new starting point. In this manner the Newton algorithm can be initialized with a pressure as closer to the analytical one as needed. The fundamental difference between the two techniques lays on a great robustness of the first, versus a rather significant speed of the second.

**First order Godunov scheme**

The exact Riemann solver can be efficiently implemented to integrate the Euler equation for an ideal fluid. If you consider a one-dimensional problem along $x_1$ direction, the spatial domain can be subdivided in $n$ cells of $\Delta x$ length. By writing the Euler equation in integral form in the $j-$th cell and integrating this equation respect to the time, the following numerical scheme
follows
\[ U_j^{n+1} = U_j^n - \frac{\Delta t}{\Delta x} [F(U_{j+1/2}) - F(U_{j-1/2})] \]

named Godunov method, where \( U \equiv (D, S_1, \tau) \) are the volume averaged integration variables and \( F \equiv (Du, S_1u_1 + p, S_1 - Du_1) \) is the flux vector. Obviously this numerical scheme could be extended to two and three dimensional problems, by using the multidimensional extension of the riemann problem, as found by Martì and Müller [1].

One of the great problems raising in Godunov relativistic numerical scheme is the difficulty to recover the primitive variables \( u_1, p, \rho_0 \) by the integration ones \( D, S_1, \tau \). Two analytical recoverig procedures have been built. The first one works in velocity and starts with the position \( u_1 = \sin \theta \), which is useful to avoid the presence of the square root in the Lorentz factor. With this assumption, the primitive variables can be expressed as functions of \( \theta \): \( \rho_0 = D \cos \theta, p = S_1/\sin \theta - \tau' \) where \( \tau' = \tau + D \). In term of \( y = \tan \theta / 2 \neq 0 \), the fourth degree algebraic equation in \( y \) is obtained \( (\alpha = (\gamma - 1)/\gamma) \):

\[ S_1(1 - \alpha) y^4 - 2(\tau' + \alpha D) y^3 + 2S_1(\alpha + 1) y^2 - 2(\tau' - \alpha D) y - S_1(1 - \alpha) = 0. \]

Another method works in energy, starting from the definitions of \( S_1 \) and \( \tau' \), by which follows: \( u_1 = S_1/(\tau' + p) \). The fourth degree algebraic equation in \( \eta = \tau' + p \) is obtained \( (\alpha' = 1/\alpha) \):

\[ (\alpha' - 1)^2 \eta^4 - 2\alpha'(\alpha' - 1) \tau' \eta^3 + [\alpha'^2 \tau'^2 + 2(\alpha' - 1)S_1^2 - D^2] \eta^2 - 2\alpha' \tau' S_1^2 \eta + S_1^2(D^2 + S_1^2) = 0. \]

This equation is analytically solved, after a normalization of its coefficients through a division by \( \max \{D, |S_1|, \tau' \} \). It has been proved via the Sturm alghoritm that these equations have only one real root physically admissible.

**Relativistic mixing layer**

A mixing layer arises when the velocity field of two parallel streams, which stationary flow one over the other, is perturbed. The basic velocity field is obtained via an hyperbolic tangent as follows:

\[ u_1 = 1/2 \tanh(20x_2/L_{x_2}) , \quad u_2 = 0. \]

On the above basic flow a pertubation field is added. Following the work of Lumpp [4], a solenoidal deterministic perturbation \( u' \) is superimposed on the basic field:

\[ u'_1 = 20 \frac{\varepsilon x_2 \lambda}{\pi L_{x_1}} \cdot \sin(2\pi x_1/\lambda) \cdot \exp(-10x_2^2/L_{x_2}^2) , \quad u'_2 = \frac{\varepsilon}{2} \cdot \cos(2\pi x_1/\lambda) \cdot \exp(-10x_2^2/L_{x_2}^2) , \]

where \( \varepsilon \) is the amplitude and \( \lambda \) is the wavelength of the pertubation. The flow is periodic in \( x_1 \) direction, while slip conditions (vanishing normal fluxes) have been set on upper \( (x_2 = L_{x_2}/2) \) and lower \( (x_2 = -L_{x_2}/2) \) boundaries.
The density distribution depends on the basic velocity field $u_1$, while the pressure has a constant value depending on the Mach number. A simulation is shown below, where the formation of only one vortex is present. The same data of the Lumpp’s work have been used: $L_{x_1} = 20$, $L_{x_2} = 10$, $\varepsilon = 0.1$, $\lambda = L_{x_1}$. The grid is $600 \times 300$ cells ($\Delta x_1 = \Delta x_2$) and the time step is $\Delta t = 10^{-3}$. Similarly to the Lumpp’s result, a shock formation is evident by the density contours on the right of the picture, while a big vortex appears on the left. Because our code is based on a first order computational scheme, it seems to be more stable than the Lumpp’s one, which is a third order in space and time and presents for this reason some oscillations in the final solution. If the numerical viscosity characteristic of the Godunov first order scheme is excluded, no additional viscosity have been introduced.

![Figure 1: From the left to the right the isolines of the fields $\rho_0$, $p$ and vorticity/density ratio are drawn at time $t = 20$ (upper row) and $t = 40$ (lower).](image)

References


