SIMPLE-RF: SIMPLE FOR RELATIVISTIC FLUIDS

Aadyot Bhardwaj
CS21B001

Ganesh B
CH22B002

ABSTRACT

High-velocity flows can enter the relativistic regime, where classical fluid models break down and may produce unphysical results such as velocities exceeding the speed of light. In this work, we extend a classical nozzle flow problem into the special relativistic domain by solving the steady, one-dimensional conservation equations using the finite volume method. We modify an existing non-relativistic solver to incorporate relativistic formulations of continuity and energy conservation. Our results demonstrate that the relativistic model preserves the physical constraint $u < c$, unlike its classical counterpart, and reveals how relativistic effects alter the flow characteristics under extreme conditions.

We aim to break the difficulty barrier between non relativistic and special relativistic simulations by developing a simulation framework for high-speed steady-state flows in special relativity.

Introduction

Relativistic fluid dynamics is essential in modeling a wide range of high-energy astrophysical phenomena, especially those involving compact objects and high-velocity outflows. While general relativity is necessary in the presence of strong gravitational fields, there exists a broad class of problems such as relativistic jets [6], blast waves, and certain shock tube [5] setups where gravity is negligible, but fluid velocities approach the speed of light. In these scenarios, special relativity suffices and becomes crucial for accurate modeling .

In this project, we extend a classical nozzle flow problem originally presented in Example 6.2 of Versteeg and Malalasekera into the relativistic regime by considering extremely high pressures and velocities. The problem is formulated using the finite volume method (FVM), with appropriate modifications to incorporate relativistic continuity and conservation equations. By comparing the non-relativistic and relativistic solutions, we aim to highlight the effects of relativistic corrections on flow behavior.

The project involves solving the 1D steady flow equations in a variable-area duct, validating them against analytic solutions, and then exploring how relativistic effects influence quantities like velocity and pressure.This work serves as a foundation for tackling more complex relativistic flow problems and highlights the emergence of relativistic effects in systems that are traditionally treated using classical fluid dynamics.

This work can be extended to study high-velocity flows in astrophysical contexts, such as relativistic jets from active galactic nuclei and gamma-ray bursts, where nozzle-like acceleration regions are key to jet dynamics. Additionally, it can be applied to modeling high-energy laboratory astrophysics experiments, such as plasma-based accelerators. These areas, where relativistic effects play a significant role, would benefit from the insights gained through the relativistic flow modeling developed in this project.

One key observation is that classical solvers can yield unphysical results, such as velocities exceeding the speed of light, when applied to extreme conditions. In contrast, our relativistic formulation inherently respects the physical constraint $u < c$, ensuring that all computed velocities remain subluminal, as required by special relativity.

General relativity (GR) is inherently complex, and our understanding of it remains limited, making simulations in this framework challenging. While special relativity (SR) simulations have made significant progress in studying transients and Riemann problems [8], simulations for steady-state flows at high speeds, though relatively simple to perform, are hidden behind a barrier of difficult math of transients.

Our goal is to bridge this gap by developing a simulation framework for high-speed steady-state flows in SR, contributing to a better understanding of relativistic fluid dynamics in extreme conditions.

A Brief Introduction to Special Relativity

Special relativity (SR), introduced by Einstein in 1905, unifies space and time into a four-dimensional spacetime. A cornerstone of SR is the *Minkowski metric*, which defines the invariant spacetime interval:

$$*ds^2 = -c^2dt^2 + dx^2 + dy^2 + dz^2 \tag{1}$$

This is expressed using the metric tensor:

$$*\eta_{\mu\nu} = \text{diag}(-1,+1,+1,+1) \tag{2}$$

It ensures Lorentz invariance and distinguishes the time component from spatial ones.

The *four-velocity*, $u^\mu = \frac{dx^\mu}{d\tau}$, generalizes classical velocity to spacetime. It has four components time and three spatial—so that physical laws remain frame-invariant. It satisfies:

$$*u^\mu u_\mu = -c^2 \tag{3}$$

The *stress-energy tensor* $T^{\mu\nu}$ encodes energy density, momentum, and stress in a fluid:

$$T^{\mu\nu} = (\rho h)u^\mu u^\nu + p\eta^{\mu\nu} \tag{4}$$

where ρ is rest-mass density, p is pressure, and $h = 1 + \varepsilon + \frac{p}{\rho}$ is the specific enthalpy. The specific enthalpy accounts for internal and pressure energy per unit mass.

The *Lorentz factor* (γ) quantifies the time dilation, length contraction, and relativistic mass increase experienced by an object moving at a velocity v relative to an observer, and is given by:

$$\gamma = \frac{1}{\sqrt{1 - \frac{v^2}{c^2}}} \tag{5}$$

where v is the object's velocity and c is the speed of light. This factor appears in the transformation of physical quantities between reference frames moving at high velocities.

Problem of Interest

We consider Example 6.2 from **Versteeg and Malalasekera** [1], but extend it to the regime of extremely high pressures and velocities, where relativistic effects become significant. The computational domain is shown in Figure 1.

We vary the following parameters in our study:

1. The ratio of inlet to outlet cross-sectional areas
2. The pressure difference between the inlet and outlet
3. The number of computational cells

The resulting discretized equations are solved using the finite volume method (FVM).

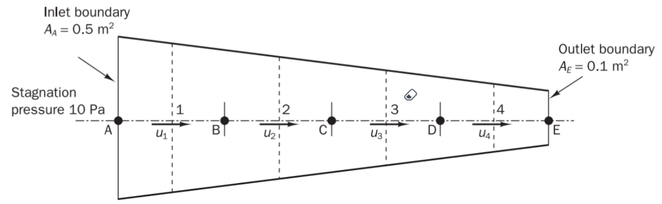

FIGURE 1: Computational domain

Governing Equations

Non- Relativistic Euler Equations

The Euler equations for a compressible, inviscid fluid with source terms in one dimension (conservative form) are:

$$\frac{\partial}{\partial t}\begin{pmatrix}\rho \\ \rho u \\ E\end{pmatrix} + \frac{\partial}{\partial x}\begin{pmatrix}\rho u \\ \rho u^2 + p \\ u(E+p)\end{pmatrix} = \begin{pmatrix}S_\rho \\ S_{\rho u} \\ S_E\end{pmatrix} \tag{6}$$

where:

ρ is the fluid density,
u is the fluid velocity,
p is the pressure,
E is the total energy density, given by $E = \frac{p}{\gamma-1} + \frac{1}{2}\rho u^2$,
γ is the adiabatic index,
$S_\rho, S_{\rho u}, S_E$ are source terms (e.g., due to gravity, heat addition, or other physics).

Relativistic Euler Equations

In special relativistic hydrodynamics, the Euler equations arise from the local conservation of the stress-energy tensor and the conservation of mass. These equations are in natural units (i.e. speed of light (c) = 1).

$$\partial_\mu T^{\mu\nu} = 0 \quad \text{(conservation of energy-momentum)} \tag{7}$$
$$\partial_\mu(\rho u^\mu) = 0 \quad \text{(conservation of mass)} \tag{8}$$

where:

ρ is the rest-mass density,
p is the pressure,
h is the specific enthalpy,
ε is the specific internal energy,
$\eta^{\mu\nu}$ is the Minkowski metric (in units where $c = 1$).

To solve the conservation equations, we *must* supply an *equation of state (EOS)* to relate thermodynamic quantities and close the system of equations.

$$p = (\gamma_{ad} - 1)\rho\varepsilon \tag{9}$$

Relativistic Euler Equations in Conservative Form

The conserved variables are:

$$D = \rho\gamma \tag{10}$$

$$S^i = (\rho h)\gamma^2 v^i \tag{11}$$

$$\tau = (\rho h)\gamma^2 - p - \rho\gamma \tag{12}$$

The relativistic Euler equations in conservative form can be written as:

$$\frac{\partial}{\partial t}\begin{pmatrix} D \\ S^i \\ \tau \end{pmatrix} + \partial_j \begin{pmatrix} Dv^j \\ S^i v^j + p\delta^{ij} \\ S^j \end{pmatrix} = \begin{pmatrix} S_\rho \\ S_{\rho u} \\ S_E \end{pmatrix} \tag{13}$$

where:

$D = \rho\gamma$ is the relativistic mass density,
$S^i = (\rho h)\gamma^2 v^i$ is the momentum density,
$\tau = (\rho h)\gamma^2 - p - \rho\gamma$ is the internal energy (excluding rest-mass),
v^j is the 3-velocity, δ^{ij} is the Kronecker delta.

These equations are closed with an equation of state, for example:

$$p = (\gamma_{\text{ad}} - 1)\rho\varepsilon \tag{14}$$

1D Steady Incompressible Relativistic Flow of Ideal Gas

We consider a one-dimensional, steady-state flow of an ideal gas in special relativity governed by the relativistic Euler equations. We make the following assumptions for this system:

Assumptions

1D system: All quantities depend only on x,
Steady state: $\partial_t = 0$,
Equation of state: Ideal gas with $\gamma_{ad} = \frac{4}{3}$,
Flat spacetime (special relativity).

Conservation Laws The conservation of mass, momentum, and energy in this system can be expressed as a single vector equation:

$$\frac{d}{dx}\begin{pmatrix} \rho\gamma v \\ (\rho h)\gamma^2 v^2 + p \\ (\rho h)\gamma^2 v \end{pmatrix} = 0 \tag{15}$$

where $\gamma = \frac{1}{\sqrt{1-\frac{v^2}{c^2}}}$ is the Lorentz factor, and h is the specific enthalpy.

Equation of State For an ideal gas with $\gamma_{ad} = \frac{4}{3}$, the specific enthalpy (in SI units) is:

$$h = 1 + 4\frac{p}{\rho c^2} \tag{16}$$

Continuity equation: Since the flow is steady, the continuity equation is given by:

$$\frac{d\bar{u}}{dx} = 0 \tag{17}$$

where

$$\bar{u} = \gamma u = \frac{u}{\sqrt{1 - \frac{u^2}{c^2}}} \tag{18}$$

.

Momentum equation: The relativistic momentum equation is:

$$\frac{d}{dx}\left[\left(\rho + \frac{4p}{c^2}\right)\bar{u}^2 + p\right] = 0 \tag{19}$$

Expanding this using the product rule:

$$\left(\frac{4\bar{u}^2}{c^2} + 1\right)\frac{dp}{dx} + \left(\rho + \frac{4p}{c^2}\right)\frac{d\bar{u}^2}{dx} = 0 \tag{20}$$

Equations 17 and 19 form a closed system that governs the flow of an ideal gas in this relativistic, incompressible regime.

We will solve for pressure p and generalised velocity $\bar{u}$. Equation 18 can be inverted to obtain the true velocity

$$u = \frac{\bar{u}}{\sqrt{1 + \frac{\bar{u}^2}{c^2}}} \tag{21}$$

.

SIMPLE Algorithm

The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm [7] is commonly used for solving incompressible flow problems by iterating over velocity and pressure correction. We generalized the SIMPLE algorithm to solve our governing equations.

Discretized Governing equations

We discretize the continuity and momentum equations over a staggered grid with velocity stored at cell faces and pressure stored at cell centres.

Continuity Equation: For a staggered grid, we discretize equation 17 as:

$$(A\bar{u})_{i+1} - (A\bar{u})_i = 0 \tag{22}$$

Momentum Equation: For a staggered grid, we discretize equation 19 as:

$$a_p\bar{u}_p = a_w\bar{u}_w + a_e\bar{u}_e + a_{pressure}(p_W - p_E) \tag{23}$$

Where a_p is the velocity at face p, a_w, a_e are velocity at weat and east neighbours, and p_W, p_E are pressures at cells that meet at face p from the west and east. Similarly A_x are corresponding cell areas.

The coefficients are given by using the upwind differencing scheme:

$$F_e = A_e\left(\rho + \frac{4P_e^*}{c^2}\right)\bar{u}_e^* \tag{24}$$

$$F_w = A_w\left(\rho + \frac{4P_w^*}{c^2}\right)\bar{u}_w^* \tag{25}$$

$$a_e = \max(0, -F_e) \tag{26}$$

$$a_w = \max(F_w, 0) \tag{27}$$

$$a_p = a_w + a_e + F_e - F_w \tag{28}$$

$$a_{pressure} = A_p\left(1 + \frac{4(\bar{u}_p^*)^2}{c^2}\right) \tag{29}$$

Note the difference in equations for $F_e, F_w, a_{pressure}$ compared to the standard SIMPLE algorithm. We use previous iteration values of velocity and pressure (denoted as starred variables) to handle the non-linearity of the equations.

Pressure correction equation: Pressure correction equation can be obtained by substituting the discrete momentum equation into the continuity equation.

$$a_P\, p'_P = a_W\, p'_W + a_E\, p'_E + b' \tag{30}$$

where

$$a_W = (\rho\, dA)_w, \tag{31}$$

$$a_E = (\rho\, dA)_e, \tag{32}$$

$$b' = F_w^* - F_e^*. \tag{33}$$

$$d_x = A/a_x \text{for cells x} = w, e \tag{34}$$

Programming

We found an open-source solver for the non-relativistic nozzle flow problem on GitHub by user tjburrows [4], which we used as a starting point. We made modifications to extend the solver into the relativistic regime by incorporating appropriate relativistic terms.

Link to our repository.

Validation

Since the problem setup is relatively simple, we implemented a basic Python script to solve a pair of equations derived from the Bernoulli equation and the continuity equation.

For the non-relativistic case, the governing equations are:

$$\rho A_1 u_1 = \rho A_2 u_2 \quad \text{(Continuity)} \tag{35}$$

$$\frac{1}{2}u_1^2 + \frac{P_1}{\rho} = \frac{1}{2}u_2^2 + \frac{P_2}{\rho} \quad \text{(Bernoulli)} \tag{36}$$

For the relativistic case, the equations are:

$$\rho A_1 \gamma_1 u_1 = \rho A_2 \gamma_2 u_2 \quad \text{(Continuity)} \tag{37}$$

$$\frac{1}{2}v_1^2\gamma_1^2 + h_1\gamma_1 = \frac{1}{2}v_2^2\gamma_2^2 + h_2\gamma_2 \quad \text{(Relativistic Bernoulli)} \tag{38}$$

where $\gamma_i = \frac{1}{\sqrt{1-u_i^2/c^2}}$ is the Lorentz factor and $h_i = 1 + \frac{\Gamma}{\Gamma-1}\frac{P_i}{\rho}$ is the specific enthalpy for an ideal gas with adiabatic index Γ.

These can be solved exactly for all positions over the domain using standard root-finding algorithms. We use fsolve available in the python's scipy library which uses a combination Newton's method and the Levenberg-Marquardt algorithm. The results have been compared with the results of the CFD simulation and are used for validating the simulation.

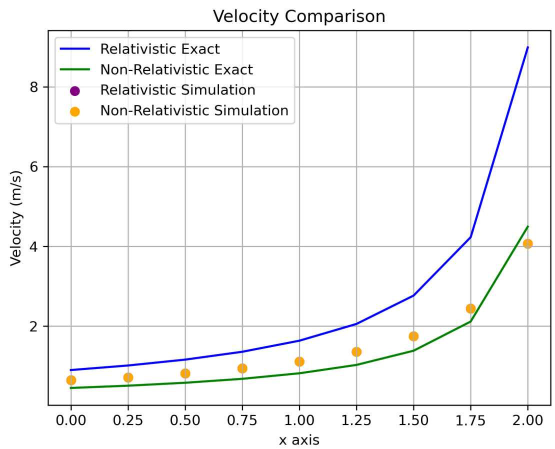

FIGURE 2: Velocity graph for N = 10, Area ratio = 10, Inlet pressure = 10 Pascals

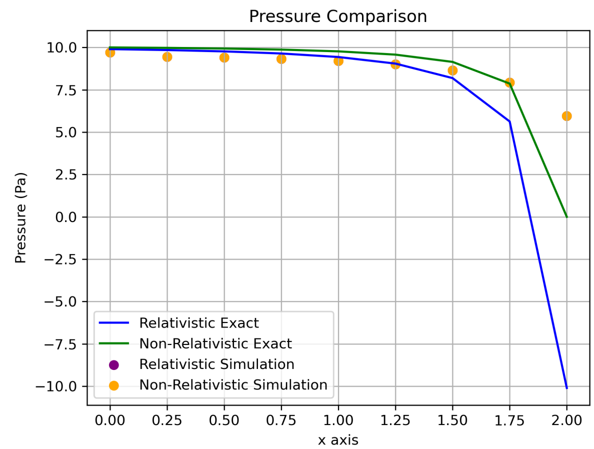

FIGURE 3: Pressure graph for N = 10, Area ratio = 10, Inlet pressure = 10 Pascals

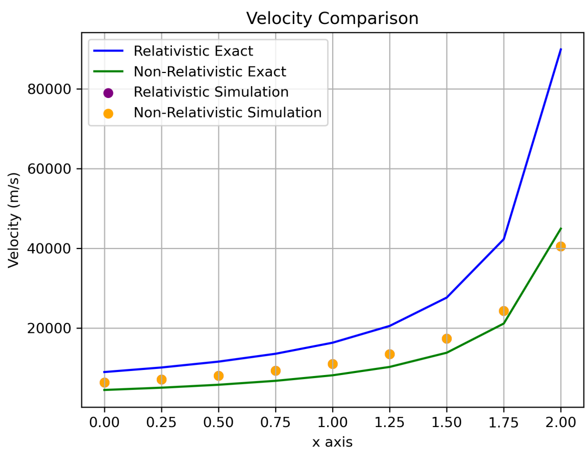

FIGURE 4: Velocity graph for N = 10, Area ratio = 10, Inlet pressure = 10^9 Pascals

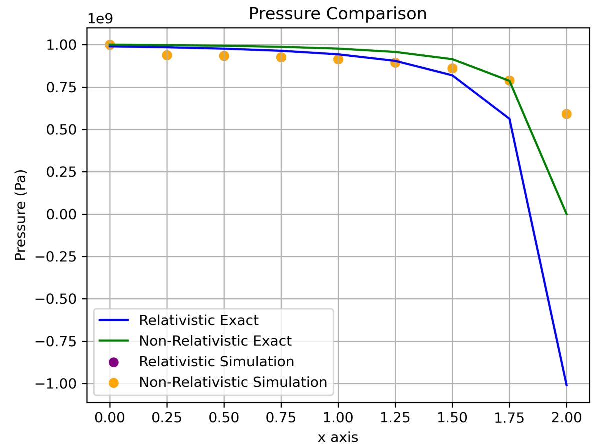

FIGURE 5: Pressure graph for N = 10, Area ratio = 10, Inlet pressure = 10^9 Pascals

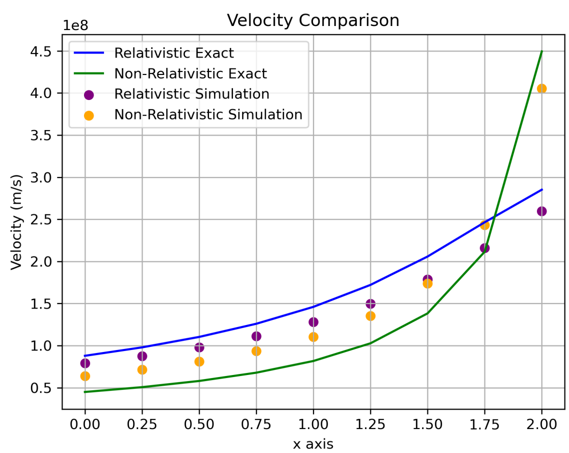

FIGURE 6: Velocity graph for N = 10, Area ratio = 10, Inlet pressure = 10^{17} Pascals

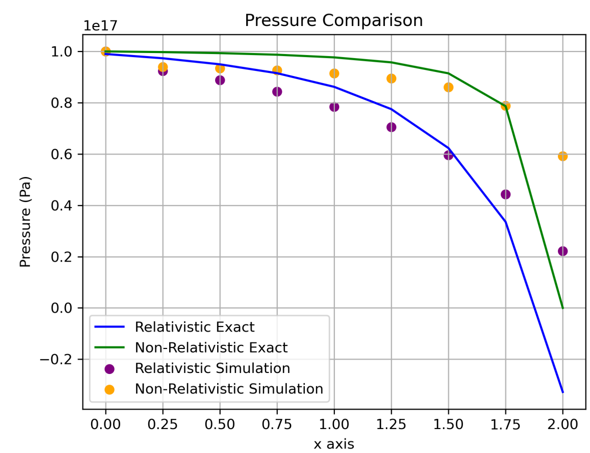

FIGURE 7: Pressure graph for N = 10, Area ratio = 10, Inlet pressure = 10^{17} Pascals

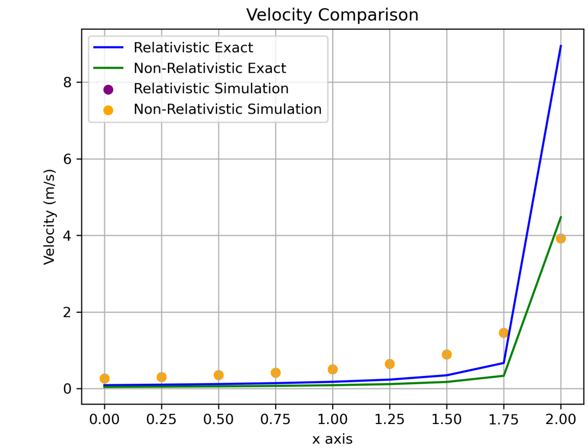

FIGURE 8: Velocity graph for N = 10, Area ratio = 100, Inlet pressure = 10 Pascals

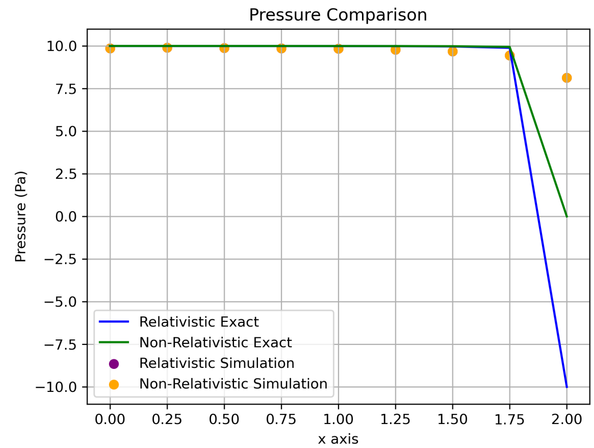

FIGURE 9: Pressure graph for N = 10, Area ratio = 100, Inlet pressure = 10 Pascals

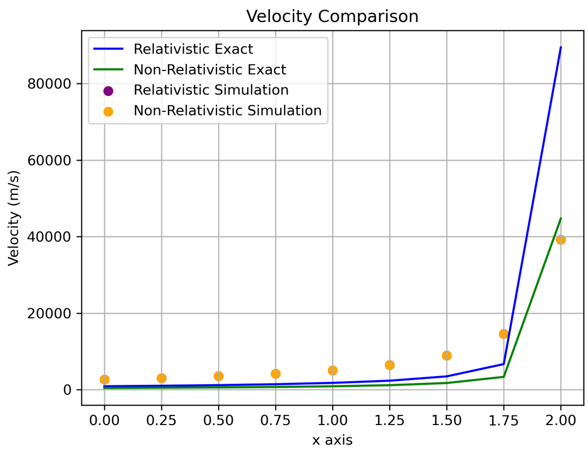

FIGURE 10: Velocity graph for N = 10, Area ratio = 100, Inlet pressure = 10^9 Pascals

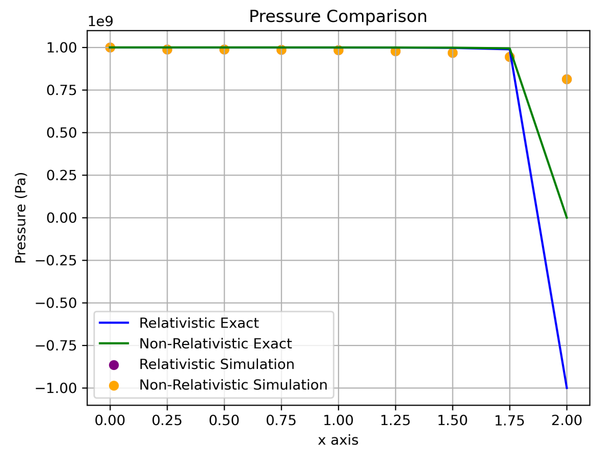

FIGURE 11: Pressure graph for N = 10, Area ratio = 100, Inlet pressure = 10^9 Pascals

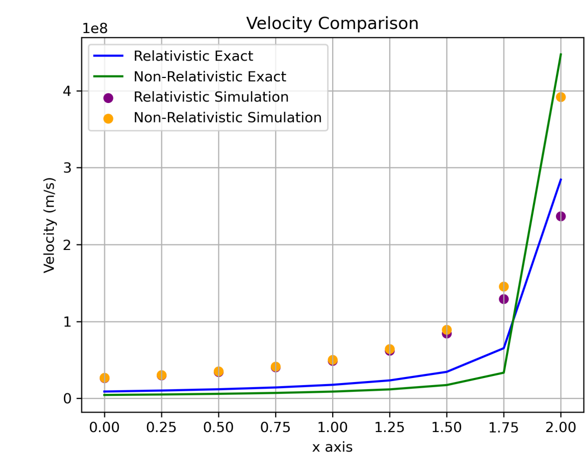

FIGURE 12: Velocity graph for N = 10, Area ratio = 100, Inlet pressure = 10^{17} Pascals

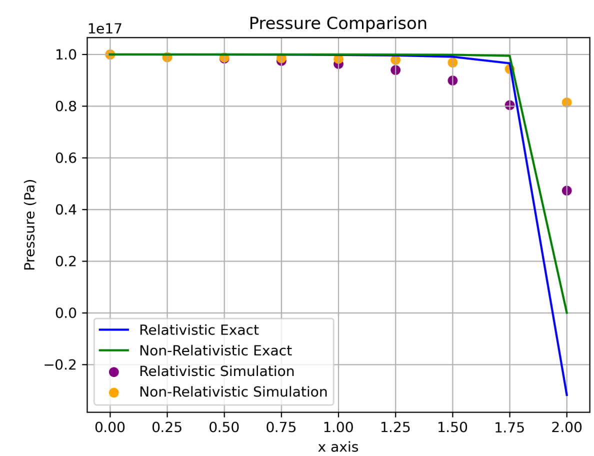

FIGURE 13: Pressure graph for N = 10, Area ratio = 100, Inlet pressure = 10^{17} Pascals

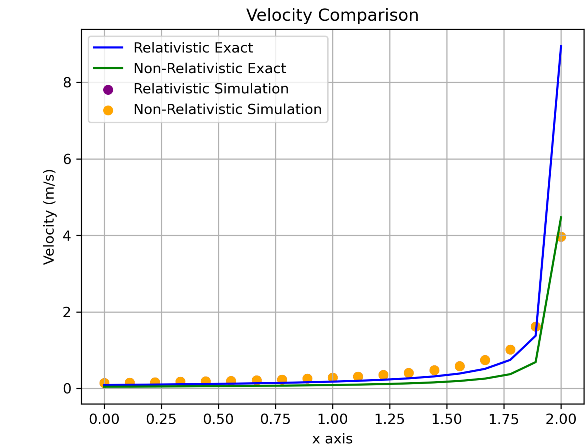

FIGURE 14: Velocity graph for N = 20, Area ratio = 100, Inlet pressure = 10 Pascals

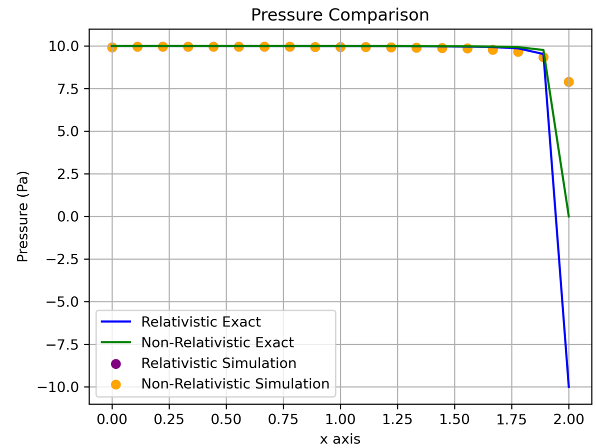

FIGURE 15: Pressure graph for N = 20, Area ratio = 100, Inlet pressure = 10 Pascals

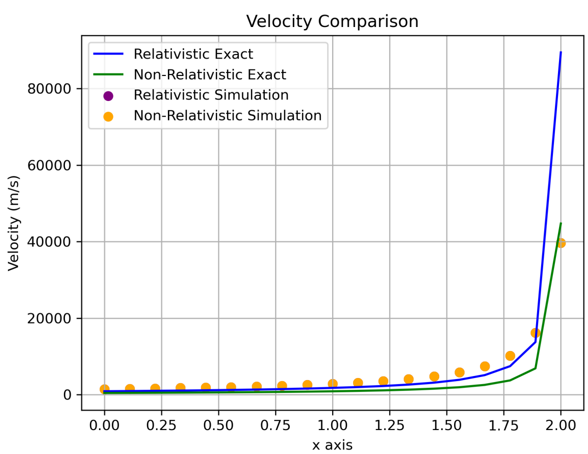

FIGURE 16: Velocity graph for N = 20, Area ratio = 100, Inlet pressure = 10^9 Pascals

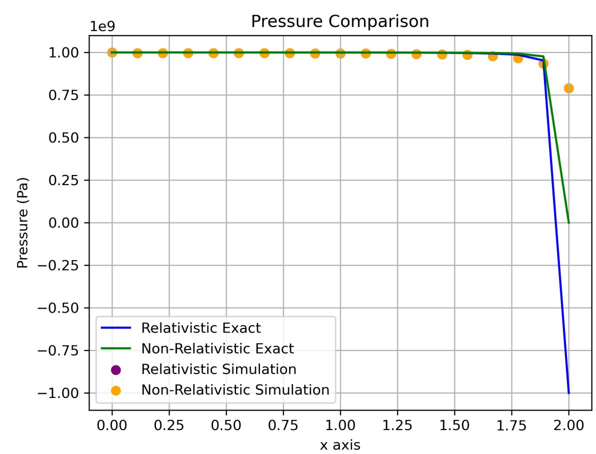

FIGURE 17: Pressure graph for N = 20, Area ratio = 100, Inlet pressure = 10^9 Pascals

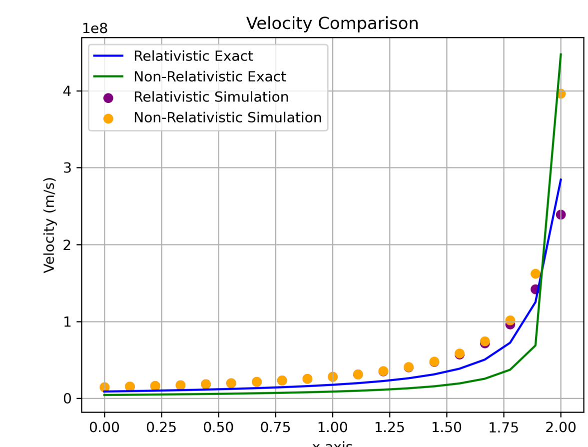

FIGURE 18: Velocity graph for N = 20, Area ratio = 100, Inlet pressure = 10^{17} Pascals

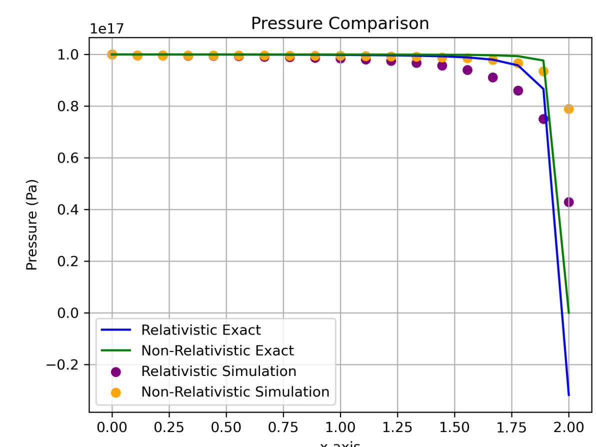

FIGURE 19: Pressure graph for N = 20, Area ratio = 100, Inlet pressure = 10^{17} Pascals

Results

Key Observations

1. Increasing the number of cells improves the accuracy of the simulation. the N=20 simulations match the analytic solutions more closely.
2. For cases $10^9, 10$ Pascals, the relativistic and non relativistic solvers give exactly the same solution. The yellow and purple dots overlap exactly. This is because the speeds are not high enough to be relativistically significant.
3. Increasing Pressure at inlet increases the outlet velocity by around the square-root of the increase. This order of magnitude estimate works as it can be easily seen from the non relativistic Bernoulli equation.
4. At low flow velocities, the relativistic and non-relativistic solutions closely match across all tested configurations. This is expected, as the Lorentz factor γ approaches 1 for small velocities, causing the relativistic equations to reduce to their classical counterparts.
5. Significant divergence between the relativistic and non-relativistic results becomes apparent only at high velocities, particularly when the inlet-outlet pressure difference is large.
6. For extreme pressure differences (e.g., $\sim 10^{27}$ Pa), the non-relativistic solver yields unphysical results, including flow velocities exceeding the speed of light. In contrast, the relativistic solver inherently respects the physical constraint $u < c$ due to the structure of the governing equations, and thus produces physically consistent results.
7. Both the relativistic and non-relativistic solvers show excellent agreement with their respective analytical (exact) solutions, validating the finite volume method implementation in both regimes.

Conclusion

We extended the idea of SIMPLE to solve Simple Relativistic Euler equations and showed that it handles relatvistic veloccties very well. Unlike its classical counterpart, it obeys the physical constraints of $u < c$. We also noticed that our modified solver behaves exactly like the classical one at low speeds. These results highlight the method's robustness and open up promising directions for efficient steady-state simulations in relativistic regimes.

REFERENCES

[1] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, 2nd ed., Pearson Education, 2007.

[2] J. A. Font, *An Introduction to Relativistic Hydrodynamics*, Departamento de Astronomía y Astrofísica, Universidad de Valencia.

[3] J. M. Martí and E. Müller, *Numerical Hydrodynamics in Special Relativity*, Living Rev. Relativity, **6**, 7 (2003). Available online: `http://relativity.livingreviews.org/lrr-2003-7`

[4] T. Burrows, *CFD Codes*, GitHub Repository, `https://github.com/tjburrows/CFD-codes`

[5] Kevin W. Thompson, *The special relativistic shock tube*, Journal of Fluid Mechanics, vol. 171, pp. 365–375, 1986.

[6] Orhan Dönmez and Refik Kayali, *Simulation of astrophysical jet using the special relativistic hydrodynamics code*, Astrophysics and Space Science, vol. 323, pp. 235–244, 2009.

[7] S. V. Patankar and D. B. Spalding, *A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows*, in *Numerical Prediction of Flow, Heat Transfer, Turbulence and Combustion*, pp. 54–73, Elsevier, 1983.

[8] A. Mignone and G. Bodo, *An HLLC Riemann solver for relativistic flows — I. Hydrodynamics*, Monthly Notices of the Royal Astronomical Society, 2005, 364(1), 126–136. https://doi.org/10.1111/j.1365-2966.2005.09546.x