

CRANFIELD UNIVERSITY

JAMIE F. TOWNSEND

ADVANCED NUMERICAL METHODS FOR DISSIPATIVE AND
NON-DISSIPATIVE RELATIVISTIC HYDRODYNAMICS

SCHOOL OF AEROSPACE
TRANSPORT AND MANUFACTURING
CENTRE FOR COMPUTATIONAL ENGINEERING SCIENCES



PHD THESIS

ACADEMIC YEAR: 2019–2020

SUPERVISOR: LÁSZLÓ KÖNÖZSY
ASSOCIATE SUPERVISOR: KARL W. JENKINS
MAY, 2020

CRANFIELD UNIVERSITY

SCHOOL OF AEROSPACE
TRANSPORT AND MANUFACTURING
CENTRE FOR COMPUTATIONAL ENGINEERING SCIENCES

PHD THESIS

ACADEMIC YEAR: 2019–2020

JAMIE F. TOWNSEND

ADVANCED NUMERICAL METHODS FOR DISSIPATIVE AND
NON-DISSIPATIVE RELATIVISTIC HYDRODYNAMICS

SUPERVISOR: LÁSZLÓ KÖNÖZSY
ASSOCIATE SUPERVISOR: KARL W. JENKINS
FEBRUARY, 2020

This thesis is submitted in partial fulfilment of the requirements for the degree of Doctor
in Philosophy in Aerospace.

©Cranfield University, 2020. All rights reserved. No part of this publication may be
reproduced without the written permission of the copyright holder.

Abstract

High-energy physical phenomena such as astrophysical events and heavy-ion collisions contain a hydrodynamic aspect in which a branch of fluid dynamics called relativistic hydrodynamics (RHD) is required for its mathematical description. The resulting equations must be, more often than not, solved numerically for scientists to ascertain useful information regarding the fluid system in question. This thesis describes and presents a two-dimensional computational fluid dynamics (CFD) solver for dissipative and non-dissipative relativistic hydrodynamics, i.e. in the presence and absence of physically resolved viscosity and heat conduction. The solver is based on a finite volume, Godunov-type, High-Resolution Shock-Capturing (HRSC) framework, containing a plethora of numerical implementations such as high-order Weighted-Essentially Non-Oscillatory (WENO) spatial reconstruction, approximate Riemann solvers and a third-order Total Variation Diminishing (TVD) Runge–Kutta method. The base numerical solver for the solution of non-dissipative RHD is extensively tested using a series of one-dimensional test cases, namely, a smooth flow problem and shock-tube configurations as well as the two-dimensional vortex sheet and Riemann problem test cases. For the case of non-dissipative relativistic hydrodynamics the relativistic CFD solver is found to perform well in terms of the orders of accuracy achieved and its ability to resolve shock wave patterns.

Numerical pathologies have been identified when the relativistic HLLC Riemann solver is used in multi-dimensions for problems exhibiting strong shock waves. This is attributed to the so-called Carbuncle problem which is shown to occur because of pressure differencing within the process of restoring the missing contact discontinuity of its predecessor, the HLL Riemann solver. To avoid this numerical pathology and improve the robustness of numerical solutions that make use of the HLLC Riemann solver, the development of a rotated-hybrid Riemann solver arising from the hybridisation of the HLL and HLLC (or Rusanov and HLLC) approximate Riemann solvers is presented. A standalone application of the HLLC Riemann solver can produce spurious numerical artefacts when it is employed in conjunction with Godunov-type high-order methods in the presence of discontinuities. It has been found that a rotated-hybrid Riemann solver with the proposed HLL/HLLC (Rusanov/HLLC) scheme could overcome the difficulty of the spurious numerical artefacts and presents a robust solution for the Carbuncle problem. The proposed rotated-hybrid Riemann solver provides sufficient numerical dissipation to capture the behaviour of strong shock waves for relativistic hydrodynamics. Therefore, focus is placed

on two benchmark test cases (odd-even decoupling and double-Mach reflection problems) and the investigation of two astrophysical phenomena, the relativistic Richtmyer–Meshkov instability and the propagation of a relativistic jet. In all presented test cases, the Carbuncle problem is shown to be eliminated by employing the proposed rotated-hybrid Riemann solver. This strategy is problem-independent, straightforward to implement and provides a consistent robust numerical solution when combined with Godunov-type high-order schemes for relativistic hydrodynamics.

An implementation of the causal theory of physical dissipation using the so-called Israel–Stewart theory is presented within the finite volume HRSC framework. This development relies on operator splitting in which the governing equations are decomposed into that of a perfect and non-perfect fluid. For the perfect fluid part, freedom is available to use any HRSC method such as approximate Riemann solvers and high-order methods. The dissipative part requires the solution of an advection–diffusion problem which contains a set of stiff equations. The stiff equations are solved using a piecewise exact solution (PES) that are stable for any time-step provided by the Courant–Friedrichs–Lewy (CFL) constraint. A series of one and two-dimensional test cases are performed to validate the implementation using one-dimensional smooth and shock-tube problems before a more detailed study exploring the role of high-order methods in the context of viscous RHD ensues.

The performance of high-order numerical methods in conjunction with physically resolved viscosity is investigated using the relativistic Kelvin–Helmholtz instability. Initially, numerical simulations are conducted for a perfect fluid such that the behaviour of the numerical schemes can be established without any contribution from physical viscosity, this enables a direct comparison between the numerical schemes that gauges the magnitude of numerical viscosity manifest within schemes. This is then repeated for the simulation runs in which physical viscosity is included, the contribution of which is evaluated when different numerical schemes are used and the effect that high-order schemes have on kinetic energy dissipation in the solution is considered. A potential advantage of higher-order schemes, namely that they can achieve commendable solutions using coarser grid resolutions given a high-enough order of schemes is employed, as opposed to a low-order scheme at a fine resolution, is studied when physical viscosity is present. Because the viscosity acts to smooth out small-scale grid perturbations it is possible to find reasonable agreement between results that employ a low-order scheme using a fine resolution and results that employ a high-order scheme at a coarser resolution, this finding is shown through both qualitative and quantitative insights. Finally, a method is presented that offers the possibility to measure the inherent numerical viscosity of a given scheme as a function of the grid resolution at early times. Overall it is suggested the high-order methods can present an advantageous solution methodology for problems in which physical viscosity is necessary for a complete description such as the study of the quark–gluon plasma. This is primarily due to the lower numerical viscosity that was found within the

high-order schemes.

Keywords

relativity – hydrodynamics – viscosity – shock waves – high-order – Riemann solver – instabilities

Acknowledgements

It's fair to say that I wouldn't have been able to achieve a finished PhD manuscript if it weren't for the many characters in my life that have helped me along the way.

A significant portion of this thesis was written under lockdown resulting from the COVID-19 pandemic. This has been a troublesome time for many and I first like to thank Alessandra and Paulo for accommodating me in their home during the early stages of lockdown, your hospitality and kindness will not be forgotten.

To my PhD supervisors László and Karl, thank you for always keeping faith in me and providing me with the necessary council, academic or otherwise, when I needed and for the opportunity to battle for a PhD degree alongside you. Both László and Karl always had time for me and I am truly grateful for their support.

During this PhD program I have been fortunate to visit Nagoya University in Japan on two occasions to pursue research with Professor Shu-ichiro Inutsuka. It was pleasure to join the TA-Lab within the Department of Physics and I am grateful to Prof. Inutsuka for allowing me this opportunity. I would also like to acknowledge Tsuyoshi Inoue for our engaging discussions as well as Ritsuko Watanabe for her support in making my time in Japan run smoothly. Furthermore, I would like to thank the Japan Society for the Promotion of Science (JSPS) and the British Council in Japan for providing the financial support to make these trips possible. A portion of this work was partially conducted as a JSPS International Research Fellow (Graduate School of Science, Nagoya University) [JSPS Summer Program 2019, SP19122].

The research group, the APU, thanks to all. It's always been a pleasure to spend time in the office with such a talented pool of unique individuals. Without the daily comradery it would have been difficult to remain cheerful through the tough times of the PhD and your support has always been well-received. The friendships that I have held during my PhD are all held dear and there are in fact too many to name and for fear of leaving anyone out I shall not name names. But if you're reading these acknowledgments expecting to see your name, then it consider it here, you all know who you are, particularly if you are an esteemed member of the Squidditch and Forest Club and special thanks also goes to Tom-Robin Teschner.

To my parents, thank you for all your support throughout my academic journey, without your confidence I wouldn't have made it this far.

Finally I would like to say thank you to Cranfield University. Thank you for allowing me to pursue this endeavour and I hope the eventual completion of this PhD thesis isn't our final goodbye.

May 2020

Jamie Francis Townsend
Liskeard, England

*To my Grandfather,
Gordon Francis Townsend
1927–2019*

Contents

Abstract	i
Acknowledgements	v
Dedication	viii
Nomenclature	xvii
List of Figures	xix
List of Tables	xxv
Research Outputs	xxvii
I Background	1
1 Introduction	3
1.1 CFD for High-Energy Physics	3
1.2 Current Challenges of Numerical Relativistic Hydrodynamics	5
1.3 Aims and Objectives of the Thesis	7
1.4 Thesis Overview	9
2 Governing Equations	11
2.1 Introductory Concepts	11
2.2 Relativistic Hydrodynamics: Perfect Fluid Approximation	13
2.3 Dissipative (Non-Perfect) Relativistic Hydrodynamics	15
3 The High-Resolution Shock-Capturing Framework	21
3.1 Hyperbolic Conservation Laws	21
3.2 A Finite Volume Godunov-Type Method	23
3.3 Numerical Flux Evaluation	24
3.3.1 The HLL Approximate Riemann Solver	26
3.3.2 The HLLC Approximate Riemann Solver	27

3.4	Spatial Reconstruction	29
3.4.1	WENO Reconstruction	30
3.4.2	MUSCL Reconstruction	35
3.5	Time Integration	35
3.6	Primitive Variable Recovery	36
II	Non-Dissipative Relativistic Hydrodynamics	39
4	Numerical Test Bench for Ideal Relativistic Hydrodynamics	41
4.1	One-Dimensional Test Cases	41
4.1.1	Smooth Flow Problem	41
4.1.2	One-Dimensional Shock Tube Problems	43
4.2	Two-Dimensional Test Cases	56
4.2.1	Vortex Sheet Evolution	56
4.2.2	Two-Dimensional Riemann Problem	57
4.3	Conclusions	59
5	A Rotated-Hybrid Riemann Solver for Relativistic Hydrodynamics	61
5.1	Riemann Solvers for Relativistic Hydrodynamics	61
5.2	The Rotated-Hybrid Riemann Solver	63
5.3	Numerical Test Cases	65
5.3.1	The Shock-Instability (Odd–Even Decoupling) Problem	65
5.3.2	The Relativistic Double-Mach Reflection Problem	68
5.4	Astrophysical Test Cases	72
5.4.1	Relativistic Richtmyer–Meshkov Instability	72
5.4.2	Propagation of a Relativistic Jet	75
5.5	Computational Timings	80
5.6	Conclusions	80
III	Dissipative Relativistic Hydrodynamics	83
6	Israel–Stewart Theory Implementation: Description and Validation	85
6.1	Introduction	85
6.2	Implementation Strategy for Causal Dissipative Relativistic Hydrodynamics	87
6.3	One-Dimensional Dissipative Test Cases	90
6.3.1	Viscous Shear Flow	90
6.3.2	Shock-Tube Problem	92
6.4	Viscous Vortex Sheet Evolution	97
6.5	Summary	98

7	High-Order Methods and Non-Perfect Relativistic Hydrodynamics	101
7.1	The Relativistic Kelvin–Helmholtz Instability	101
7.1.1	Problem Setup	103
7.1.2	Flow Diagnostics	104
7.2	The Inviscid Case	106
7.3	Influence of Shear Viscosity	109
7.4	On Estimating Numerical Viscosity using Physical Viscosity	115
7.4.1	Numerical Viscosity Estimations for WENO Reconstruction	116
7.4.2	Numerical Reynolds Number	117
7.5	An Accuracy and Cost Evaluation of High-Order Schemes with Physical Viscosity	124
7.6	Conclusions	128
IV	Closing Remarks	131
8	Thesis Summary	133
8.1	General Conclusions	133
8.2	Summary of Achievements	134
8.3	Future Work	134
	Bibliography	145

Nomenclature

Symbols

β	Thermodynamic coefficients
β_r	WENO smoothness indicator
$\mathbf{F}$	Numerical flux
$\mathbf{U}$	Conserved variables
$\mathbf{L}(\mathbf{U}_i)$	Spatial differential operator
Δt	Time-step
$\Delta x/y$	Grid Spacing
ϵ	Internal energy density
η	Shear viscosity coefficient
η_*	Numerical viscosity
Γ	Adiabatic index
$\gamma^{\mu\nu}$	Projection operator
γ_r	Linear WENO weights
κ	Heat conduction coefficient
$\lambda_{0/\pm}$	Eigenvalues
$\mathcal{C}$	Contact discontinuity
$\mathcal{D}$	Numerical dissipation
$\mathcal{R}^\mu$	Arbitrary four-vector
$\mathcal{S}^\mu$	Entropy four-vector

$\mathcal{W}$	Wave
∇_μ	Four-gradient
ω_r	Non-linear WENO weights
Π	Bulk viscosity
$\pi^{\mu\nu}$	Shear viscosity tensor
ρ	Rest-mass density
τ	Proper time
$\tau^{\mu\nu}$	Viscosity tensor
$\tau_\Pi/\pi/q$	Relaxation time
ξ	Bulk viscosity coefficient
c	Speed of light in a vacuum
c_{rj}	Constant WENO coefficients
c_s	Speed of sounds
D	Mass density
$d\tau$	Proper time interval
ds	Space interval
dt	Time interval
E	Energy density
E_k	Kinetic energy
$g^{\mu\nu}$	Metric tensor
h	Specific enthalpy
I^{ij}	Identity matrix
I_i	Computational cell
n	Proper particle number density
N^μ	Particle number density current
P	Pressure

p_n	Arbitrary polynomial
q^μ	Heat-flux four-vector
Re_*	Numerical cell Reynolds number
S^i	Momentum density
$S_{L/R}$	Left/Right wave speed
T	Temperature
t	Time
$T^{\mu\nu}$	Energy–momentum tensor
u^μ	Four-velocity
u^i	Three-velocity
V	Volume
W	Lorentz factor
x^μ	Four-dimensional spatial vector
NX	Grid points in the x -direction
NY	Grid points in the y -direction

Subscripts

*	Variable located within the star region of the Riemann fan
μ/ν	Index spanning 0 . . . 3
i/j	Index spanning 1 . . . 3
L/R	Left/Right
NS	Navier–Stokes
RMS	Root Mean Square

Abbreviations

AGN	Active Galactic Nuclei
CAFD	Computational Astrophysical Fluid Dynamics
CFD	Computational Fluid Dynamics

CFL Courant–Friedrichs–Lewy number
 CIT Classical Irreversible Thermodynamics
 DG Discontinuous Galerkin
 EIT Extended Irreversible Thermodynamics
 ENO Essentially Non-Oscillatory
 FD Finite Difference
 FV Finite Volume
 GRB Gamma-Ray Burst
 GRMHD General Relativistic Magnetohydrodynamics
 HLL Harten Lax and van Leer
 HLLC Harten Lax and van Leer Contact
 HLLE Harten Lax and van Leer Einfeldt
 HRSC High-Resolution Shock Capturing
 KT Kurganov and Tadmor
 LHC Large Hadron Collider
 MP Monotonicity Preserving
 MUSCL Monotone Upstream-centered Scheme for Conservation Laws
 PES Piecewise Exact Solution
 PV Primitive Variables
 QCD Quantum Chromodynamics
 QGP Quark–Gluon Plasma
 RHD Relativistic Hydrodynamics
 RHIC Relativistic Heavy-Ion Collider
 RK Runge–Kutta
 RMHD Relativistic Magnetohydrodynamics
 SHASTA Kurganov and Tadmor

SPH Smoothed Particle Hydrodynamics

TVD Total Variation Diminishing

WENO Weighted Essentially Non-Oscillatory

List of Figures

1.1	An example of the Carbuncle instability taken from Nishikawa & Kitamura (2008) that depicts the pressure contours of a high Mach-number flow interacting with a blunt body. The result shown on the left exhibits the Carbuncle instability as a jet-like structure emanating from the blunt body whereas the result on the right is Carbuncle free.	6
3.1	Riemann problem initial conditions specified by Equation (3.10) for the cell boundary $x_{i+\frac{1}{2}}$	25
3.2	Wave structure arising from the solution of the Riemann problem shown using the Riemann fan.	25
3.3	Schematic representation of the candidate stencils for the WENO3 reconstruction at $x_{i+\frac{1}{2}}$, i.e. when $k = 3$	31
4.1	Numerical solution profiles of the Riemann problem 1 from initial conditions given in Equation (4.6) at $t = 0.4$ using a grid resolution of $NX = 400$ and $CFL = 0.5$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used.	45
4.2	Numerical solution profiles of the Riemann problem 2 from initial conditions given in Equation (4.7) at $t = 0.4$ using a grid resolution of $NX = 400$ and $CFL = 0.5$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used.	46
4.3	Numerical solution profiles of the Riemann problem 3 from initial conditions given in Equation (4.8) at $t = 0.4$ using a grid resolution of $NX = 400$ and $CFL = 0.5$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used.	48
4.4	Numerical solution profiles of the Riemann problem 4 from initial conditions given in Equation (4.9) at $t = 0.4$ using a grid resolution of $NX = 400$ and $CFL = 0.5$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used.	49
4.5	Numerical solution profiles of the Riemann problem 5 from initial conditions given in Equation (4.10) at $t = 0.4$ using a grid resolution of $NX = 400$ and $CFL = 0.5$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used.	50

4.6	Numerical solution profiles of the Riemann problem 6 from initial conditions given in Equation (4.11) at $t = 0.35$ using a grid resolution of $NX = 400$ and $CFL = 0.5$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used.	51
4.7	Numerical solution profiles of the Riemann problem 7 from initial conditions given in Equation (4.12) at $t = 1.5$ using a grid resolution of $NX = 400$ and $CFL = 0.2$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used.	53
4.8	Rest-mass density contours using the HLL and HLLC approximate Riemann solvers shown in Sub-Figures 4.8a and 4.8b, respectively. Lorentz factor contours using the HLL and HLLC approximate Riemann solvers shown in Sub-Figures 4.8c and 4.8d, respectively. The solutions are reported at $t = 0.4$ using $CFL = 0.5$ at a grid resolution of 200×200 . The MP-WENO7 schemes was used throughout alongside a third-order TVD Runge–Kutta time integration method.	57
4.9	Rest-mass density contours using the HLL and HLLC approximate Riemann solvers shown in Sub-Figures 4.9a and 4.9b, respectively. Pressure field contours using the HLL and HLLC approximate Riemann solvers shown in Sub-Figures 4.9c and 4.9d, respectively. The solutions are reported at $t = 0.4$ using $CFL = 0.5$ at a grid resolution of 200×200 . The MP-WENO7 schemes was used throughout alongside a third-order TVD Runge–Kutta time integration method.	59
5.1	Definition of vector components for the rotated-hybrid Riemann solver. . . .	64
5.2	Rest-mass density contours of the odd-even decoupling using 10 contour levels a various simulation times using a first-order spatial reconstruction scheme. The discretisations consists of 800×40 cells. The Riemann solvers are: (5.2a)–(5.2c) standard HLL, (5.2d)–(5.2f) standard HLLC, (5.2g)–(5.2i) rotated-hybrid HLL/HLLC and (5.2j)–(5.2l) rotated-hybrid Rusanov/HLLC.	70
5.3	Rest-mass density contours of the relativistic double Mach reflection using 25 contour levels a $t = 4$. The WENO5 spatial reconstruction scheme with the WENO3 scheme used in the vicinity of shocks is deployed. The discretisations consists of 1600×400 cells. The Riemann solvers shown are the: HLL (5.3a), HLLC (5.3b), rotated-hybrid HLL/HLLC (5.3c) and the rotated-hybrid Rusanov/HLLC (5.3d).	71
5.4	Schematic diagram for the relativistic Richtmyer–Meshkov instability initial conditions showing the rest-mass density. W_b denotes the initial primitive variables behind the rightward propagating shock, W_L and W_R are the initial conditions to the left and right of the initial perturbation, respectively. The profile of the perturbed interface is given by equation (5.21) and initial conditions for the three regions are specified in Equation (5.22). The height H is used to compute the perturbation amplitude evolution in Figure 5.6 . . .	73

5.5	Rest-mass density contours of the relativistic Richtmyer–Meshkov instability at $t \approx 30$ using the WENO5 spatial reconstruction scheme. The computational domain shown has size $[10.5, 13] \times [0, 2.5]$. The discretisations consists of 1500×400 cells. The Riemann solver used are: (5.5a) standard Rusanov, (5.5b) standard HLL, (5.5c) standard HLLC, (5.5d) rotated-hybrid Rusanov/HLLC and (5.5e–5.5f) rotated-hybrid HLL/HLLC using the definition of $\mathbf{n}_1$ in Equation (5.4) and (5.6) stated as $\mathbf{n}_1^A$ and $\mathbf{n}_1^B$, respectively. . . .	75
5.6	Time evolution for the amplitude in initial perturbation for the HLL, HLLC and rotated-hybrid HLL/HLLC Riemann solvers and comparison to numerical data achieved in Zanotti & Dumbser (2015) , [Fig. 7 in Z&D (2015)]. Inset: time evolution for the amplitude in initial perturbation for the HLL Riemann solver at grid resolutions 750×200 , 1500×400 and 3000×800 . . .	76
5.7	One-dimensional power spectrum of the rest-mass density for the HLL, HLLC and rotated-hybrid HLL/HLLC Riemann solvers at $t = 30$ using a grid resolution of 1500×400 cells.	77
5.8	Logarithm of the rest-mass density contours for the relativistic jet at $t = 20$. The WENO3 spatial reconstruction scheme is used, reducing to a first-order reconstruction in the vicinity of shocks. The Riemann solver used are: HLL (5.8a), HLLC (5.8b) and the rotated-hybrid HLL/HLLC (5.8c) at a grid resolution of 600×600 cells. Note that only a portion of domain is shown in the y -direction: $[0, 5]$	78
5.9	Logarithm of the rest-mass density contours for the relativistic jet at $t = 20$. The WENO3 spatial reconstruction scheme is used, reducing to a first-order reconstruction in the vicinity of shocks. The Riemann solver used are the rotated-hybrid HLL/HLLC (5.9a–5.9b) using a finer resolution 1200×1200 cells and using the definitions for the vector $\mathbf{n}_1$ in Equations (5.4) and (5.6), respectively. Note that only a portion of domain is shown in the y -direction: $[0, 5]$	79
6.1	Profiles of the y -velocity component for varying grid resolutions using the MP-WENO5 scheme with a HLL Riemann solver and a third-order TVD Runge–Kutta method at $t = 4$	92
6.2	Profiles of the y -velocity component for the Riemann problem with initial conditions given in Equation (6.15) at $t = 0.4$ using $\text{CFL} = 0.5$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used. Figure 6.2a shows influence of using $\eta = 10^{-15}$, 0.01 and 0.04 with the MP-WENO5 scheme at $NX = 200$. Figure 6.2b shows the profile at $\eta = 0.01$ using the WENO3, MP-WENO5 and MP-WENO7 schemes at $NX = 100$. . .	94
6.3	Profiles of the x -velocity component (Figure 4.5c) for $\xi = 10^{-15}$, 0.01 and 0.04. Temperature profiles (figure 4.5d) using $\kappa = 10^{-15}$, 0.01 and 0.04. Results correspond to the initial conditions given in Equation (6.15) at $t = 0.4$ using $\text{CFL} = 0.5$ at $NX = 200$. A third-order TVD Runge–Kutta method, MP-WENO5 scheme and HLL approximate Riemann solver are used. . . .	95

6.4	Rest-mass density contours of the vortex evolution test case in the presence of physical viscosity. The MP-WENO5 scheme, HLL Riemann solver and third-order TVD Runge–Kutta time integration method were used throughout. The solutions are reported at $t = 2$ using $CFL = 0.5$ at a grid resolution of 200×200 . Sub-Figure 6.4a shows the evolution using $\eta = 0.01$, 6.4b using $\eta = 0.005$, 6.4c using $\eta = 0.001$ and 6.4d using $\eta = 0.0001$	98
7.1	Example of the time evolution of kinetic energy computed from Equation (7.5). The area between the blue and red line (Num_{Loss}) corresponds to energy lost due to numerical dissipation. The area between the blue yellow line ($Total_{Loss}$) corresponds to the total energy lost due to numerical and physical dissipation.	105
7.2	Evolution of the v_{RMS} for all numerical schemes when $\eta = 0$ using a grid resolution of 512×1024 . The inset shows the effect of grid refinement when the WENO3 spatial reconstruction method is used. The HLL Riemann solver is used throughout.	108
7.3	The rest-mass density power spectrum at $t = 4$ when $\eta = 0$ for all numerical schemes considered using a grid resolution of 512×1024	108
7.4	Rest-mass density contours at $t = 4$ for the MUSCL2 (7.4a), WENO3 (7.4b), MP-WENO5 (7.4c) and MP-WENO7 (7.4d) numerical schemes. For all cases the HLL approximate Riemann solver was used with a grid resolution of 512×1024	109
7.5	Evolution of the v_{RMS} for all numerical schemes when $\eta = 10^{-3}$, the inset shows the affect of grid refinement when the MP-WENO7 spatial reconstruction method is used also for $\eta = 10^{-3}$. The HLL Riemann solver is used throughout.	110
7.6	Rest-mass density contours at $t = 4$ for the MUSCL2 (7.6a), WENO3 (7.6b), MP-WENO5 (7.6c) and MP-WENO7 (7.6d) numerical schemes at $\eta = 10^{-3}$. For all cases the HLL approximate Riemann solver was used with a grid resolution of 512×1024	111
7.7	Time evolution of the kinetic energy defined in Equation (7.5) for when $\eta = 0$ and $\eta = 10^{-3}$ (left figures) using different grid resolutions: coarse–fine (top–bottom) and all spatial reconstruction schemes. The right-hand-side column shows the difference in kinetic energy whereby the solution when $\eta = 10^{-3}$ is subtracted from the solution when $\eta = 0$ for the time up until $t = 2$	112
7.8	Time evolution of the kinetic energy defined in Equation (7.5) for when $\eta = 0, 10^{-4}, 10^{-3}$ and $\eta = 10^{-2}$ (left figures) using the medium grid resolution all spatial reconstruction schemes. The right-hand-side column shows the difference in kinetic energy whereby the viscous solution is subtracted from the solution when $\eta = 0$ for the time up until $t = 2$	113
7.9	The rest-mass density power spectrum at $t = 4$ for varying η using the WENO3 (7.9a) and MP-WENO7 scheme (7.9b) at a grid resolution of 512×1024	114

7.10	Evolution of the v_{RMS} for the WENO3 scheme at varying value of η	114
7.11	Time-evolution of the total kinetic energy in the system at different grid resolutions and different values of η for the WENO3 (7.11a), MP-WENO5 (7.11b) and MP-WENO7 (7.11c) schemes.	117
7.12	The predicted numerical viscosity as a function of grid spacing computed from Equation (7.13) for the WENO3, MP-WENO5 and MP-WENO7 schemes.	119
7.13	Evolution of the v_{RMS} for different grid resolutions and values of η for the WENO3 (7.13a) and MP-WENO5 (7.13b) schemes.	120
7.14	Rest-mass density power spectra for different grid resolutions and values of η for the WENO3 (7.14a) and MP-WENO5 (7.14b) schemes.	121
7.15	Rest-mass density contours at $t = 2$ for the WENO3 scheme at different resolutions and values of η	121
7.16	Numerical cell Reynolds number contours at $t = 2$ for the WENO3 scheme at different spatial resolutions.	122
7.17	Rest-mass density contours at $t = 2$ for the MP-WENO5 schemes at different resolutions and values of η	122
7.18	Rest-mass density contours at $t = 2$ for the MP-WENO7 schemes at different resolutions and values of η	123
7.19	Rest-mass density contours at $t = 2$ for the MP-WENO5 scheme at high resolutions and finite values of η	124
7.20	Rest-mass density contours of the relativistic Kelvin–Helmholtz instability at $t = 4$. The solutions are obtained using the HLL approximate Riemann solver and a shear viscosity parameter of $\eta = 10^{-3}$. The solutions using a coarse grid resolution of 512×1024 and a higher-order scheme (left) are seen in Figures 7.20a and 7.20c whilst the results obtained with a finer grid resolution and lower-order scheme (right) are seen in Figures 7.20b and 7.20d.	125
7.21	Evolution of the v_{RMS} different numerical schemes at different grid resolutions using $\eta = 10^{-3}$	126
7.22	Evolution of the v_{RMS} and the volume-averaged kinetic energy loss contribution arising from the inclusion of viscosity for different numerical schemes at different grid resolutions using $\eta = 10^{-3}$	127
7.23	The rest-mass density power spectrum at $t = 4$ using high-order schemes at coarse grid resolutions and low-order schemes at fine grid resolutions. .	128

List of Tables

4.1	Rates of convergence for the different numerical schemes applied to the one-dimensional smooth flow problem. A third-order TVD Runge–Kutta time integration method and HLL approximate Riemann solver were used. The simulation is performed until a final time of $t = 2$ at which time the L_1 error norm was computed.	43
4.2	L_1 error norms and respective convergence rates for the one-dimensional Riemann problems presented throughout Chapter 4.1.2 using the HLL approximate Riemann solver. The rest-mass density was used for the error analysis.	54
4.3	L_1 error norms and respective convergence rates for the one-dimensional Riemann problems presented throughout Chapter 4.1.2 using the HLLC approximate Riemann solver. The rest-mass density was used for the error analysis. † Inflow velocity set to $u = 0.9$	55
5.1	Normalised CPU time for all presented simulations: A: Shock-Instability, B: Double-Mach Reflection, C: Richtmyer–Meshkov, D: Relativistic Jet.	80
6.1	L_1 error norms and respective convergence rates for the one-dimensional viscous shear flow problem given by Equation (6.11) using the HLL approximate Riemann solver and a third-order TVD Runge–Kutta method. The shear viscosity coefficient is studied at $\eta = 0.01$. The y -component of the velocity was used for the error analysis at $t = 4$	92
6.2	L_1 error norms and respective convergence rates for the one-dimensional Riemann problem given by Equation (6.15) using the HLL approximate Riemann solver. The parameters η , ξ and κ are firstly tested at 10^{-15} and later the shear viscosity coefficient is studied at $\eta = 0.01, 0.04$. The y -component of the velocity was used for the error analysis.	95
6.3	L_1 error norms and respective convergence rates for the one-dimensional Riemann problem given by Equation (6.15) using the HLL approximate Riemann solver. The bulk viscosity coefficient is studied at $\xi = 0.01, 0.04$. The x -component of the velocity was used for the error analysis.	96

6.4	L_1 error norms and respective convergence rates for the one-dimensional Riemann problem given by Equation (6.15) using the HLL approximate Riemann solver. The heat conduction coefficient is studied at $\kappa = 0.01, 0.04$. The temperature profile was used for the error analysis where $T \equiv p/\rho$. . .	96
7.1	Volume-averaged total energy losses for the cases using the WENO3 scheme.	118
7.2	Volume-averaged total energy losses for the cases using the MP-WENO5 scheme.	118
7.3	Volume-averaged total energy losses for the cases using the MP-WENO7 scheme.	118
7.4	Coefficients for $x \equiv \eta$ using Equation (7.12).	118
7.5	Coefficients for $x \equiv NX$ using Equation (7.12).	118
7.6	Normalised CPU time with respect to the MUSCL2 scheme for the high vs low-order schemes presented in this section.	128

Research Outputs

Journal Publications

Jamie F. Townsend, László Könözy, Karl W. Jenkins (2020), ‘On the Development of a Rotated-Hybrid HLL/HLLC Riemann Solver for Relativistic Hydrodynamics’, *Monthly Notices of the Royal Astronomical Society*, **492**(2), 2493–2505.

Conference Proceedings

Jamie F. Townsend, László Könözy, Karl W. Jenkins, *High-Order Numerical Methods for Relativistic Hydrodynamics — Part I. Theory and Methodology*, MultiScience-XXXIII. microCAD International Multidisciplinary Scientific Conference, University of Miskolc, Hungary, 23–24 April 2019.

Jamie F. Townsend, László Könözy, Karl W. Jenkins, *High-Order Numerical Methods for Relativistic Hydrodynamics — Part II. The Relativistic Richtmyer–Meshkov Instability*, MultiScience-XXXIII. microCAD International Multidisciplinary Scientific Conference, University of Miskolc, Hungary, 23–24 April 2019.

Other Presentations

Jamie F. Townsend, *Building Tools for Physicists: Testing the Untestable*, JSPS Farewell Party, Tokyo, Japan, 20 August 2019.

Jamie F. Townsend, Tsuyoshi Inoue, Shu-ichiro Inutsuka, László Könözy, Karl W. Jenkins, *The Relativistic Richtmyer–Meshkov Instability*, EASW9, Astrophysical and Laboratory Plasma Workshop, Nagoya University, Japan, 29 Aug–2 Sep 2019, (poster presentation).

Jamie F. Townsend, László Könözy, Karl W. Jenkins, *High-Order Numerical Simulations of the Relativistic Richtmyer–Meshkov Instability*, APS March Meeting, Boston, U.S.A., 2019, (abstract presentation).

Part I

Background

Chapter 1

Introduction

The scientific discipline known as Computational Fluid Dynamics (CFD) provides numerically engineered solutions to problems involving fluid flow. CFD enables the visualisation of fluid dynamics and permits quantitative analysis to evaluate the system in question. A traditional example of this would be the application of CFD to an aerofoil; the fluid flow can be visualised at a prescribed angle of attack and the lift/drag coefficients can be extracted. CFD therefore has a proven track-record in applications such as the aerospace and automotive industry, amongst many others, because it enables design changes to be virtually tested in any number of configurations before any specific hardware is manufactured and (expensive) experimental testing is performed. As a result of its success, the number of CFD software companies have proliferated, aiming to provide CFD analysis for a wide range of engineering applications containing fluid flow. In fact, the CFD industry is expected to be worth \$3.1 Billion by 2024, largely driven by the aforementioned aerospace and automotive industries¹.

It could be argued that two categories of CFD engineer exist: (1) those that apply CFD to engineering challenges to gain insight into problems and (2) those that develop the CFD software itself, investigating the capabilities of numerical methods, implementing a multi-physics capacity into a current software and of course, designing new numerical methods. In this work, the CFD research presented largely falls into the second category however knowledge of the underlying fluid dynamics is essential to evaluate the development of the numerical solver.

1.1 CFD for High-Energy Physics

Although the CFD research in the present work focuses on numerical and multi-physics methodologies, the target application deviates from a traditional CFD exercise. A plethora of astrophysical events that occur throughout the universe contain a hydrodynamic aspect, for example: gamma-ray bursts (GRB), core-collapse supernova, relativistic jets, neutron

¹[imarcgroup.com/computational-fluid-dynamics-market](https://www.imarcgroup.com/computational-fluid-dynamics-market)

stars, black holes, active galactic nuclei (AGN) and pulsar winds. For astrophysical physical events the typical time and length-scales of the problem are drastically different from terrestrial CFD problems. Despite this difference, what fundamentally separates these astrophysical applications with terrestrial ones is often the need for Einstein's theory of general and/or special relativity to complete their hydrodynamic description — this branch of fluid dynamics is known as relativistic hydrodynamics (RHD). Special relativity is required when the bulk fluid motion, or a section of it, travels at speeds comparable with the speed of light. General relativity is required when strong gravitational fields are present in regions surrounding compact objects such as black holes and neutron stars. In addition, strong magnetic fields are also encountered in such environments meaning general relativistic magnetohydrodynamics (GRMHD) must be considered. The present work is restricted to fluid flow described by special relativity and hydrodynamics only, a review of relativistic hydrodynamics for general relativity has been composed by [Font \(2000\)](#) to which the reader is referred.

Man-made relativistic flows are created at experimental facilities such as at the Relativistic Heavy Ion Collider (RHIC) at Brookhaven National Laboratory (USA), at the GSI Helmholtzzentrum für Schwerionenforschung (Germany) and in the Large Hadron Collider (LHC) at CERN in Switzerland. The experiments performed at these facilities ultimately seek a deeper understanding into the structures of matter and the early development of the universe. In particular, a theory known as quantum chromodynamics (QCD) is extensively studied via the collision of heavy ions — ions whose nuclei have an atomic weight heavier than carbon. This gives rise to the so-called quark–gluon Plasma (QGP) which is thought to have existed at around 10^{-5} s after the Big Bang. The heavy ions are accelerated to relativistic speeds such that their centre-of-mass energy is larger than their rest-mass energy and allowed to collide head-on with one another. At (proper) time $\tau_0 \sim 3.3 \times 10^{-24}$ s after the collision the constituents arising from the collision experience a sufficiently strong interaction such that local thermodynamic equilibrium is established thus permitting hydrodynamic flow. It was shown by [Baym et al. \(1983\)](#) that for collisions with energies in the range of 100–270 GeV that the dense matter produced as a result of the collisions would have a mean free path much smaller than typical dimension of the post-collision (central-rapidity) region at τ_0 . This supports the notion that a relativistic hydrodynamics description is valid for the study of the QGP in its hydrodynamic phase. The nearly perfect fluid was initially experimentally detected and studied via vast scientific collaborations, the STAR² and PHENIX³ collaborations. This was furthered by ALICE⁴, ATLAS⁵ and CMS⁶ experiments at the LHC⁷ in CERN. Investigations by authors such as [Heinz et al. \(2012\)](#) proceeded which sought to estimate the specific shear viscosity

²bnl.gov/rhic/STAR.aspt

³bnl.gov/rhic/PHENIX.asp

⁴home.cern/science/experiments/alice

⁵home.cern/science/experiments/atlas

⁶home.cern/science/experiments/cms

⁷home.cern/science/accelerators/large-hadron-collider

property of the QGP, making use of both experimental data and the results of numerical calculations.

Astrophysical and particle physics applications are the driving force behind the developments of CFD solvers for relativistic hydrodynamics. Both of these applications require a compressible flow description as strong shock waves are encountered and investigating the equation of state of a fluids such as the QGP is of interest to research scientists. Comprehensive reviews on the development of such numerical methodologies for special relativity were compiled by [Marti & Müller \(2003\)](#) and more recently by the same authors in [Martí & Müller \(2015\)](#). Early numerical developments began in the 1970s using explicit finite difference techniques and artificial viscosity methods to capture shock waves. Developments then ensued to avoid the artificial viscosity technique such as using the Lax–Wendroff scheme and Smoothed Particle Hydrodynamics (SPH). A major change in methodology occurred when High-Resolution Shock-Capturing (HRSC) methods were applied to the RHD equations in the early 90s. This led to rapid development in this field, which is also relied upon for aerospace applications containing shock waves. This thesis also aims to contribute to developments within the HRSC framework. More recently, discontinuous Galerkin methods have been applied to RHD problems and have showed great promise, see e.g. [Radice & Rezzolla \(2011\)](#) and [Núñez-de la Rosa & Munz \(2018\)](#).

1.2 Current Challenges of Numerical Relativistic Hydrodynamics

At present, modern-day finite volume (FV) relativistic hydrodynamic (RHD) and relativistic magnetohydrodynamics (RMHD) codes can either rely on the exact or approximate solution to the local Riemann problem. For larger-scale multi-dimensional problems which may also include different multi-physics considerations, approximate Riemann solvers are often preferred. One of the most popular choices of approximate Riemann solver is the HLL solver, originally formulated by [Harten \(1983\)](#) and first applied to relativistic flows by [Schneider et al. \(1993\)](#). A drawback on the HLL Riemann solver is that the intermediate contact discontinuity is not resolved, leading to excessive smearing in the solution. To tackle the issue of excessive smearing found using the HLL approximate Riemann solver, [Toro et al. \(1994\)](#) proposed the HLLC Riemann solver that seeks resolution of the missing contact discontinuity. This proved to be a very powerful strategy and was introduced in the context of RHD by [Mignone & Bodo \(2005\)](#) and later for RMHD by the same authors ([Mignone & Bodo 2006](#)). While the HLLC solver provides clear advantages over its predecessor in terms of accuracy due to a more completely resolved wave pattern, issues surround the appearance of the so-called *Carbuncle* problem under certain flow conditions. It was reported in [Wang et al. \(2008\)](#) that the relativistic Riemann HLLC solver was in fact “unsuitable” for multi-dimensional problems in RHD due to this numerical artefact. The Carbuncle problem resembles itself as a distortion in the flow field when a shock wave

is present. Figure 1.1 presents an example taken from [Nishikawa & Kitamura \(2008\)](#) that depicts the pressure contours of a high Mach-number flow interacting with a blunt body. The result shown on the left exhibits the Carbuncle instability as a jet-like structure emanating from the blunt body whereas the result on the right is Carbuncle free. It is, in effect, a numerical instability, completely non-physical and detrimental to high-order numerical simulations ([Quirk 1994](#)). The plaguing of numerical solutions due to the so-called Carbuncle problem is an area undergoing constant research for both relativistic and non-relativistic hydrodynamics. Consequently this hinders the robustness of the numerical strategy employed for problems involving strong shock waves.

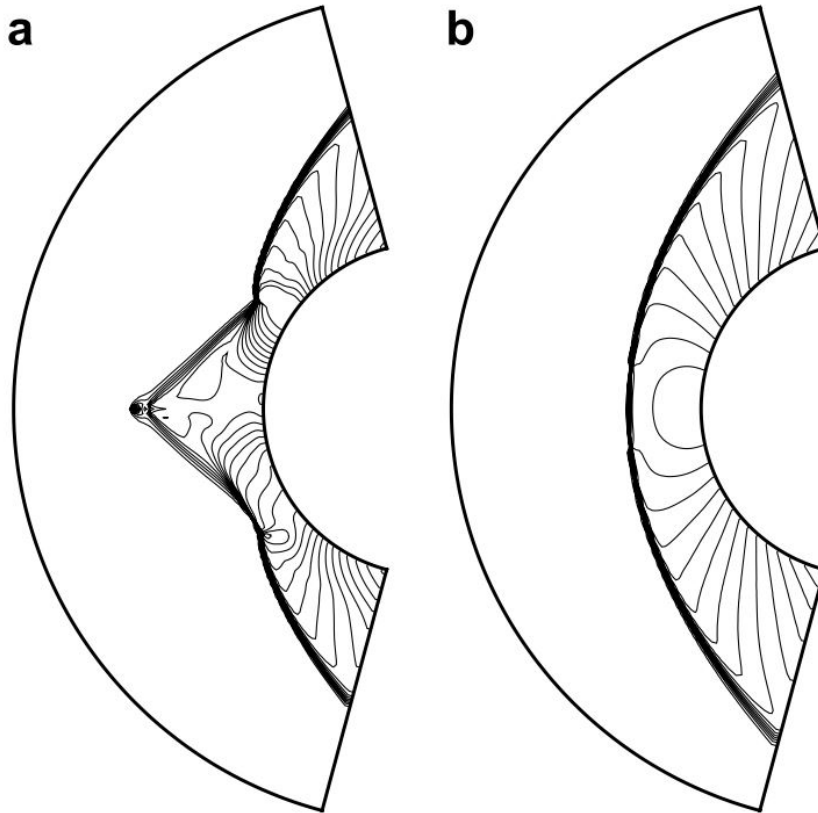


Figure 1.1: An example of the Carbuncle instability taken from [Nishikawa & Kitamura \(2008\)](#) that depicts the pressure contours of a high Mach-number flow interacting with a blunt body. The result shown on the left exhibits the Carbuncle instability as a jet-like structure emanating from the blunt body whereas the result on the right is Carbuncle free.

The inclusion of physical viscosity into RHD simulations is fraught with difficulty due to the increase in complexity of the governing equations and the disparity between the time-scales involved. Because the viscous time-scale is often much smaller than that of the macroscopic fluid motion the numerical solution to a series of stiff equations is required.

Furthermore, the theoretical framework of a dissipative theory that is compatible with special and/or general relativity provides a source of mathematical intricacy. It was noted in (Rezzolla & Zanotti 2013, p. 313) however that "...we foresee that the next years will witness rapid progress both in the numerical solution of relativistic-hydrodynamic equations of non-perfect fluids...". The numerical solution to this challenging multi-physics aspect of relativistic hydrodynamics has been considered by past authors outside of the HRSC framework e.g. Baier & Romatschke (2007), Schenke et al. (2010, 2011), Roy & Chaudhuri (2012). Only recently did authors start to use numerical strategies resembling those used for a perfect fluid within the HRSC framework, this was thanks to Takamoto & Inutsuka (2011) who proposed the governing equations could be split into a perfect and non-perfect component. Traditional finite volume Godunov-type methods could be employed for the perfect part and an alternative treatment would be used for the non-perfect part. This strategy has proved very successful thus far and has been used by Karpenko et al. (2014), Akamatsu et al. (2014) and Okamoto et al. (2016) for their studies relating to the quark–gluon plasma. So far however, the role of high-order methods in conjunction with physical viscosity in the relativistic context is not well established. This would be important considering the known trait of higher-order methods to possess a lower numerical viscosity which could be an advantage in correctly measuring the shear viscosity of the quark–gluon plasma.

1.3 Aims and Objectives of the Thesis

Four key objectives are presented for the research described in this thesis. Two of which concern the development of a relativistic CFD solver, one that seeks to improve the robustness of current numerical practices and the final objective investigates the performance of numerical schemes when multi-physics implementations are included within the developed framework. They are summarised as follows.

Develop and implement the current state-of-the-art numerical methods used for High-Resolution Shock-Capturing (HRSC) Godunov-type schemes into a numerical software capable of numerically solving the governing equations of relativistic hydrodynamics (RHD) for a perfect fluid in one- and two-dimensions.

- Implement the HRSC framework consisting of several orders of WENO reconstruction, the HLL and HLLC approximate Riemann solvers, a third-order TVD Runge–Kutta method and an appropriate primitive variable recovery scheme using the finite volume methodology.
- Perform validation simulations on a series of one-dimensional and two-dimensional shock-tube problems as well as a smooth test case for comparison to analytical solutions where available. This serves to establish a degree of confidence in the current numerical implementation.

- For the aforementioned one-dimensional test problems also perform grid convergence studies and assess the order of convergence using an appropriate error measurement for both the shock-tube and smooth problem.

Understand, investigate and suggest a possible remedy to the so-called Carbuncle problem exhibited by the relativistic HLLC approximate Riemann solver when shock waves are present to improve the robustness of numerical solutions.

- Establish a possible cause for the Carbuncle problem in the relativistic HLLC approximate Riemann solver based on its numerical formulation.
- Modify the original odd-even decoupling problem for the data structure of the developed code to test the previously established reasoning for the cause of the Carbuncle problem.
- Implement a rotated-hybrid HLL/HLLC Riemann solver as a possible remedy for this issue as performed in the non-relativistic case.
- Test and compare the rotated-hybrid Riemann solver through a series of numerical (odd-even decoupling and the double Mach reflection) and astrophysical test cases (the relativistic Richtmyer–Meshkov instability and relativistic jets).

Include a causal description of physical dissipation (viscosity and heat conduction) within the current HRSC framework based the extended irreversible thermodynamics (EIT) Israel–Stewart theory.

- Implement the Israel–Stewart theory of physical dissipation using an operator-splitting and subsequent Strang-splitting technique to include the Israel–Stewart telegrapher equations and resolve the resulting stiff equations.
- Test the implementation through a series of one-dimensional shock-tube problems using varying dissipation coefficients and compare with results already available in the current literature. Perform a two-dimensional vortex sheet simulation to assess the implementation in two-dimensions.

Assess and evaluate the performance of high-order spatial reconstruction techniques when used in conjunction with physically resolved viscosity using the relativistic Kelvin–Helmholtz instability.

- Identify relevant quantitative measures to evaluate the performance of the different numerical schemes in the context of the Kelvin–Helmholtz instability.
- Perform initial simulations without physically resolved viscosity to gauge the behaviour of the numerical schemes.
- Assess what effect the inclusion physical viscosity has on the instability development for different values of the shear viscosity transport coefficient and at different grid resolutions.

- Identify what, if any, benefits can be achieved by the use of high-order methods in the presence of physical viscosity in terms of the necessary grid size needed to resolve accurate flow physics. In other words, can the use of high-order methods at a lower grid resolution produce comparable results to those at a high resolution using a high-order method?
- Determine a strategy for estimating the numerical viscosity of high-order schemes using physically resolved viscosity.
- Establish the legitimacy of secondary flow features reported for the presented initial conditions of the relativistic Kelvin–Helmholtz instability.

1.4 Thesis Overview

This thesis describes the development of a High-Resolution Shock-Capturing (HRSC) finite volume (FV) numerical solver for the equations of dissipative and non-dissipative relativistic hydrodynamics written using the FORTRAN programming language. The numerical solver is built using high-order numerical methods both in space and time as well as a host of different approximate Riemann solver formulations; physical dissipation is included in this framework following the phenomenological Israel–Stewart theory. This thesis challenges the robustness of the state-of-the-art numerical methodologies, offering both an explanation and possible solution for their improvement. Furthermore, this thesis explores the role that high-order methods play when combined with physical dissipation in the context of relativistic hydrodynamics and provides a initial benchmark for the future improvement of such multi-physics methodologies as well as estimates for the inherent numerical viscosity of high-order schemes.

In Chapters 2–3 the governing equations for both dissipative and non-dissipative relativistic hydrodynamics are given, alongside a brief primer on Israel–Stewart theory. The general framework for High-Resolution Shock-Capturing (HRSC) methods used in this work is also described, this features numerical aspects such as finite volume Godunov-type methods, WENO schemes, Riemann solvers, the Runge–Kutta method and the primitive variable recovery procedure necessary in numerical relativity.

Chapter 4 presents the validation and verification of the developed numerical solver for non-dissipative relativistic hydrodynamics based on the HRSC methods presented in Chapter 3. This includes one-dimensional smooth problems, one-dimensional Riemann problems (i.e. the shock-tube) as well as a selection of two-dimensional test cases. Building on this platform, Chapter 5 scrutinises the robustness of the HLLC approximate Riemann solver when used in multi-dimensions in conjunction with high-order methods. It is found that the HLLC Riemann solver is susceptible to the so-called Carbuncle problem. An explanation for why this may be as well as a possible remedy using the concept of a

rotated-hybrid Riemann solver is presented.

In Chapter 6 validation steps are taken in order to determine whether physically correct behaviour emerging from the implementation of Israel–Stewart theory is achieved — unlike the non-dissipative case, exact solutions are seldom attainable for viscous RHD. Both a selection of one and two-dimensional test cases are provided and a description of the implementation into the HRSC framework using a splitting method is given. By use of Israel–Stewart theory, the role of high-order methods together with physical viscosity is explored in Chapter 7 using the relativistic Kelvin–Helmholtz instability as a two-dimensional test case. Relationships are established and insight is provided into the benefits of high-order methods, especially when physical viscosity is included in the numerical solution. Furthermore, an attempt is made to quantify the numerical viscosity for a given scheme as a function of the grid resolution through empirical means.

Finally, in Chapter 8 the thesis is closed with general comments about the work completed in this thesis and an outlook on future work in this field.

Chapter 2

Governing Equations

This chapter presents the governing equations of relativistic hydrodynamics for a perfect and non-perfect fluid in conservation law form. The governing equations for a relativistic perfect fluid, i.e. where heat conduction and viscosity are neglected, can be found in Chapter 2.2 and are formulated according to the so-called (3+1) Valencia formulation along with the corresponding eigenvalues of the system. Chapter 2.3 then describes the governing equations for non-perfect relativistic hydrodynamics when dissipative effects are considered such as viscosity and heat conduction. In the realm of classical physics, the inclusion of dissipative effects in the Euler equations leads to the well-known Navier–Stokes equations. This extension alters the mathematical character of the governing equations in a manner incompatible with relativity. Therefore, to extend the relativistic Euler equations to include dissipative effects a different approach is needed which, in the present work, is provided by Israel–Stewart theory which represents the most active theory in use to-date.

2.1 Introductory Concepts

Relativity introduces the notion that space and time are not two independent dimensions but instead interwoven into a single entity known as spacetime. The classical Euler and Navier–Stokes equations that are employed for traditional CFD applications do not make any use of this relationship between space and time. Because most terrestrial applications of CFD only require a Newtonian description of reality, spacetime is not accommodated in the mathematical formulation of the governing equations. In contrast, for certain astrophysical and high-energy physics applications, the notion of spacetime must be considered which requires modification of the governing equations from first principles. This then gives rise to the relativistic hydrodynamics (RHD) governing equations which are compatible with the concept of spacetime and therefore, in the case presented herein, special relativity. Throughout this work the validity of the continuum hypothesis is maintained which is applicable for a plethora of computational astrophysical fluid dynamics (CAFD) applications.

In this work, natural units are used such that the speed of light is set equal to unity ($c = 1$), Roman indices run from $1 \dots 3$ which correspond to a three-dimensional Cartesian space, i.e. $x^i = (x^1, x^2, x^3) \equiv (x, y, z)$. Greek indices run from $0 \dots 4$ and correspond to a four-dimensional space that amalgamates the three spatial dimensions with time, i.e. $x^\mu = (t, x^i)$, these are also referred to as four-vectors. Furthermore, all thermodynamic quantities are defined in the fluid rest-frame and are therefore considered as invariant scalars.

Special relativity follows two key axioms which must be incorporated into the governing hydrodynamic equations, these can be summarised as (Rezzolla & Zanotti 2013, p. 24):

Axiom 1. *“There is no privileged inertial reference frame and the laws of physics are invariant when changing among (inertial) reference frames.”*

Axiom 2. *“The speed of light c is the maximum speed at which physical bodies and physical perturbations can move or propagate.”*

These axioms imply the speed of light must be the same in all reference frames which means a different definition of distance is required when working with special relativity. Consider two events a and b that occur in a four-dimensional space. An interval between those two events can be defined by

$$\begin{aligned} s^2 &= -(t_a - t_b)^2 + (x_a - x_b)^2 + (y_a - y_b)^2 + (z_a - z_b)^2, \\ ds^2 &= -(dt)^2 + (dx)^2 + (dy)^2 + (dz)^2. \end{aligned} \quad (2.1)$$

This relationship is invariant because another observer using a different reference frame (t', x', y', z') would find an equivalent interval

$$ds'^2 = ds^2 = -(dt')^2 + (dx')^2 + (dy')^2 + (dz')^2.$$

This distance presents a new interesting concept, that is, it can be negative despite its seemingly analogous nature to a 3-dimensional distance in space. Depending on the sign of the distance calculated, one can categorise the type of distance or interval in question according to

$$\begin{aligned} ds^2 &> 0 \text{ the interval is spacelike} \\ ds^2 &< 0 \text{ the interval is timelike} \\ ds^2 &= 0 \text{ the interval is lightlike, also called null} \end{aligned}$$

An interval is spacelike if there exists a reference frame where two events can occur at the same time in different locations. An interval is null (or lightlike) if a reference frame does not exist and finally, an interval is timelike if a reference frame exists such that two events occur at the same location at different times. A more concise way to summarise Equation (2.1) is by using the summation convention

$$ds^2 = g_{\mu\nu} dx^\mu dx^\nu, \quad (2.2)$$

where $g_{\mu\nu}$ is the metric tensor for a Minkowski (flat) spacetime, often seen written as $\text{diag}(-,+,+,+)$, which is explicitly given by

$$g_{\mu\nu} = \begin{pmatrix} g_{00} & g_{01} & g_{02} & g_{03} \\ g_{10} & g_{11} & g_{12} & g_{13} \\ g_{20} & g_{21} & g_{22} & g_{23} \\ g_{30} & g_{31} & g_{32} & g_{33} \end{pmatrix} = \begin{pmatrix} -1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{pmatrix}. \quad (2.3)$$

If two events are to be considered as timelike separated then a minimum time interval $d\tau$ exists between them that is defined by an observer who assigns the same coordinate system to each event. If $dx = dy = dz = 0$ this would then imply that $ds^2 = -dt^2 = -d\tau^2$ which entails

$$d\tau^2 = -g_{\mu\nu}dx^\mu dx^\nu = dt^2 - ds^2 = dt^2 \left[1 - \left(\frac{dx^i}{dt} \right)^2 \right] = dt^2 \left[1 - u_i u^i \right], \quad (2.4)$$

that then leads to the relationship

$$d\tau = \frac{dt}{\sqrt{1 - u_i u^i}} = d\tau W \text{ and therefore } dt > d\tau, \quad (2.5)$$

where u^i is the fluid three-velocity and W is the Lorentz factor. The fluid four-velocity is defined via the relation

$$u^\mu := \frac{dx^\mu}{d\tau}, \quad (2.6)$$

Hence,

$$u^\mu = \frac{dt}{d\tau} \frac{dx^\mu}{dt} = \frac{1}{\sqrt{1 - u_i u^i}} (1, u^i) = W(1, u^i). \quad (2.7)$$

A mass current N^μ and energy–momentum tensor $T^{\mu\nu}$ are required to establish the governing equations of a relativistic fluid system. In analogy to the stress tensor for Newtonian hydrodynamics, contributions of $T^{\mu\nu}$ provide information pertaining to the energy and momentum density, i.e. the conserved variables, as well as to the flux of energy and momentum thus describing the forces exerted on a fluid element. The mass current N^μ is described by considering that baryon density n is defined in the rest-frame. Hence the baryon density in the co-moving is nu^0 which for a generic reference frame yields $N^\mu = nu^\mu$.

2.2 Relativistic Hydrodynamics: Perfect Fluid Approximation

The perfect fluid approximation of RHD neglects contributions from dissipative phenomena such as viscosity and thermal conductivity. Therefore for a perfect fluid, the particle

number density current N^μ and energy–momentum tensor $T^{\mu\nu}$ are given by

$$N^\mu = nu^\mu, \quad (2.8a)$$

$$T^{\mu\nu} = \rho hu^\mu u^\nu + pg^{\mu\nu}, \quad (2.8b)$$

where n is the proper particle number density, u^μ is the fluid four-velocity that satisfies the condition $u^\mu u_\mu = -1$, p is the fluid pressure and ρ is the fluid rest-mass density. In this work an ideal fluid is assumed, meaning that an equation of state corresponding to an ideal gas of the form $p = p(\rho, \epsilon)$ is used throughout, where ϵ denotes the specific internal energy of the fluid. The equation of state and specific enthalpy expression h are given by

$$p = \rho\epsilon(\Gamma - 1) \quad (2.9)$$

and

$$h = 1 + \frac{\Gamma}{\Gamma - 1} \frac{p}{\rho}, \quad (2.10)$$

respectively. The constant adiabatic index Γ used herein represents the ratio of specific heats and is typically either provided as $\Gamma = 5/3$ or $\Gamma = 4/3$ which span its region of admissibility (Taub 1948). The latter value corresponds to an ultra-relativistic gas at very high temperatures, i.e. the internal energy is much higher than rest-mass energy. The relativistic speed of sound c_s is consequently expressed as

$$c_s^2 = \frac{\Gamma p}{\rho h}. \quad (2.11)$$

The governing equations for a relativistic perfect fluid follow from the conservation laws corresponding to the conservation of mass,

$$\nabla_\mu N^\mu = \nabla_\mu (\rho u^\mu) = 0, \quad (2.12)$$

and the conservation of energy and momentum,

$$\nabla_\mu T^{\mu\nu} = \nabla_\mu [\rho hu^\mu u^\nu + pg^{\mu\nu}] = 0. \quad (2.13)$$

In the present work the governing equations are presented and subsequently solved numerically in conservation-law form. The origins of this strategy date back to the work of Martí et al. (1991) who exploited their hyperbolic mathematical character and applied previously well-established numerical techniques already developed for Newtonian hydrodynamics, which is in fact still the tactic nowadays, see Chapter 3. By defining a set of conserved variables the equations could easily be cast into conservation-law form, such that we have a system containing expressions for the conservation of mass D , momentum S^i and energy E according to (Banyuls et al. 1997)

$$U^i \equiv \mathbf{U} = \begin{bmatrix} D \\ S^i \\ E \end{bmatrix} = \begin{bmatrix} \rho W \\ \rho h W^2 u^i \\ \rho h W^2 - p \end{bmatrix}, \quad (2.14)$$

where the Lorentz factor is recalled as

$$W = \frac{1}{\sqrt{1 - u^i u_i}}. \quad (2.15)$$

By use of Equations (2.12), (2.13) and (2.14), the governing equations can be written in the (3+1) “Valencia” conservative form using a Cartesian coordinate system as

$$\frac{\partial}{\partial t} \begin{pmatrix} D \\ S^i \\ E \end{pmatrix} + \frac{\partial}{\partial x^j} \begin{pmatrix} Du^j \\ S^i u^j + p I^{ij} \\ S^i \end{pmatrix} = 0,$$

Many authors adopt the practice of subtracting the rest-mass term from the energy term and similarly the mass flux with the energy flux to yield the following set of equations,

$$\frac{\partial}{\partial t} \begin{pmatrix} D \\ S^i \\ E - D \end{pmatrix} + \frac{\partial}{\partial x^j} \begin{pmatrix} Du^j \\ S^i u^j + p I^{ij} \\ S^i - Du^j \end{pmatrix} = 0. \quad (2.16)$$

This practice is purely numerical and is adopted in the present work. Through extensive numerical testing it was found that this modification of the equations provided the most accurate solutions as opposed the cases without such alteration. The author notes that this is not a general statement as many authors have achieved accurate solutions using HRSC methods for RHD without this modification to the energy terms, see e.g. [Nunez-de la Rosa & Munz \(2016\)](#).

A final remark is made regarding the eigenstructure of Equations (2.16), see also Chapter 3. The eigenvalues for the system of conservation laws given in Equation (2.16) for the x -direction are,

$$\lambda_0 = u^x, \quad (2.17a)$$

$$\lambda_{\pm} = \frac{1}{1 - u^i u_i c_s^2} \left\{ u^x (1 - c_s^2) \pm c_s \sqrt{(1 - u^i u_i) [1 - u^x u^x - (u^i u_i - u^x u^x) c_s^2]} \right\}. \quad (2.17b)$$

The eigenvalues are all real and distinct, thus implying that the system of governing equations for a perfect relativistic fluid is hyperbolic. Symmetry considerations yield the corresponding eigenvalues in the y -direction. The eigenvalues in Equation (2.17) correspond to the wave speeds within the system and the maximum velocities permissible.

2.3 Dissipative (Non-Perfect) Relativistic Hydrodynamics

The perfect fluid description in Chapter 2.2 arises from consideration of the Lorentz transformation, conservation laws and the assumption of local thermodynamic equilibrium.

This latter assumption however is restrictive in the sense that it does not permit effects on the fluid flow due to physical dissipation; because all fluids are susceptible in experiencing dissipation then the notion of local thermodynamic equilibrium is never strictly true. Recently [Trachenko & Brazhkin \(2020\)](#) presented findings that indicate a universal lower bound on the amount of viscosity present in a fluid and [Romatschke & Romatschke \(2007\)](#) demonstrated how viscous contributions explain the hydrodynamic behaviour of the quark–gluon plasma resulting from relativistic heavy ion collisions. Breaking the notion of a local thermodynamic equilibrium requires the consideration of irreversible thermodynamics such that the entropy production rate increases over time. Irreversible thermodynamic processes occur during the motion of a fluid which give rise to dissipative effects, heat is exchanged through the fluid elements meaning that an individual fluid element may not be in equilibrium with the entire fluid, moreover the fluid elements in relative motion can generate friction too.

Including physical dissipation within the relativistic framework first requires a renewed definition for the fluid four-velocity, u^μ . No such definition is needed under the perfect fluid approximation in which the fluid rest-frame is chosen to comply with the fact that a co-moving frame can be defined such that there is no flow of energy, no particle flow and no increase in entropy. Unlike the perfect fluid approximation however, both energy and particle diffusion are present hence the definition of a local rest frame becomes ambiguous. Additionally, the rest-mass density current N^μ , the energy–momentum tensor $T^{\mu\nu}$ and the entropy current S^μ are considered to be primary field variables with no *a priori* relation to the four-velocity or any rest frame. Therefore, two physical perspectives are commonly used to define the fluid four-velocity. The so-called Landau frame in which the four-velocity is specified as having the same direction as the energy flux vector allows for particle diffusion but not energy diffusion in the local rest frame ([Landau & Lifshitz 1987](#)). On the other hand the so-called Eckart frame can be used as the local rest-frame in which the four-velocity u^μ is said to be parallel to the particle flux vector, meaning there is no particle diffusion and therefore the rest-mass density current remains unchanged and energy diffusion is present ([Eckart 1940](#)). In practice however it can be shown that the resolved physics is the same despite the choice of local rest-frame as demonstrated in [Akamatsu et al. \(2014\)](#) who showed that the particle diffusion in the Landau frame is equivalent to the heat diffusion in the Eckart frame.

Using the Eckart decomposition, the particle number density current N^μ and energy–momentum tensor $T^{\mu\nu}$ are written as

$$N^\mu = nu^\mu, \quad (2.18a)$$

$$T^{\mu\nu} = \rho hu^\mu u^\nu + pg^{\mu\nu} + q^\mu u^\nu + q^\nu u^\mu + \tau^{\mu\nu}, \quad (2.18b)$$

where q^μ is the heat-flux vector and $\tau^{\mu\nu}$ is the viscosity tensor granting contributions from dissipative phenomena. The viscous tensor $\tau^{\mu\nu}$ is decomposed into contributions arising

from shear and bulk viscosity related to the traceless and trace part of the viscosity tensor,

$$\tau^{\mu\nu} \equiv \Pi\gamma^{\mu\nu} + \pi^{\mu\nu}, \quad (2.19)$$

where $\gamma^{\mu\nu} \equiv g^{\mu\nu} + u^\mu u^\nu$ is the projection operator that projects any tensor on the hypersurface normal to u^μ , Π is the bulk viscosity and $\pi^{\mu\nu}$ is the shear viscosity tensor.

The conservation laws in Equation (2.13) are now reformulated using the particle number density current N^μ and energy-momentum tensor $T^{\mu\nu}$ in Equation (2.18) such that the conservation laws for non-perfect relativistic hydrodynamics arise from

$$\nabla_\mu N^\mu = \nabla_\mu (\rho u^\mu) = 0, \quad (2.20)$$

$$\nabla_\mu T^{\mu\nu} = \nabla_\mu [\rho h u^\mu u^\nu + p g^{\mu\nu} + q^\mu u^\nu + q^\nu u^\mu + \tau^{\mu\nu}] = 0. \quad (2.21)$$

Explicitly writing Equations (2.20) and (2.21) in a form that also encompasses the governing equations for a perfect fluid in Equation (2.16), the full governing equations as a system of evolved quantities in time that correspond to both inviscid and dissipative fluxes is given by,

$$\frac{\partial}{\partial t} \begin{pmatrix} S^i + q^0 u^i + q^i u^0 + \tau^{0i} \\ E - D + 2q^0 u^0 + \tau^{00} \end{pmatrix} + \frac{\partial}{\partial x^j} \begin{pmatrix} S^i u^j + p I^{ij} + q^i u^j + q^j u^i + \tau^{ij} \\ S^j - D u^j + q^0 u^j + q^j u^0 + \tau^{0j} \end{pmatrix} = 0. \quad (2.22)$$

Equation (2.22) therefore provides the fundamental governing equation for non-perfect relativistic hydrodynamics in the presence of special relativity. While this system of equations is sound, definitions for the bulk viscosity Π , heat-flux vector q^μ and shear viscosity tensor $\pi^{\mu\nu}$ are still needed.

The second law of thermodynamics states that the change in entropy of a closed thermodynamic system will always be positive or zero. Hence the entropy production rate is,

$$\nabla_\mu S^\mu \geq 0, \quad (2.23)$$

where S^μ the entropy four-vector which, along with N^μ and $T^{\mu\nu}$, comprise the state variables of the system for a non-perfect fluid. In the case of a perfect fluid the entropy current is defined as $S^\mu = s\rho u^\mu$. According to [Eckart \(1940\)](#), for the entropy production rate to remain non-zero an additional contribution is required such that

$$S^\mu = s\rho u^\mu + \frac{R^\mu}{T} \quad (2.24)$$

where T is the temperature and R^μ is some four-vector with non-zero divergence and is assumed to be a function of the dissipative fluxes Π , $\pi^{\mu\nu}$, q^μ which in turn vanish under the perfect fluid approximation.

The theory of classical irreversible thermodynamics (CIT) assumes a linear dependence of the four-vector R^μ on the thermodynamics fluxes. In the Eckart frame it is possible to construct a generic four-vector R^μ from the thermodynamics fluxes and the four-velocity. This results in an entropy current for CIT given by

$$S^\mu = s\rho u^\mu + \frac{q^\mu}{T}. \quad (2.25)$$

The entropy generation rate from Equation (2.23) must remain positive which, after some algebra, lead to the following relations for the thermodynamic fluxes under the assumption of an ideal equation of state given by Equation (2.9), ([Takamoto & Inutsuka 2011](#)),

$$q_{\text{NS}}^\mu = \kappa \gamma^{\mu\nu} \left(\frac{\partial T}{\partial x_\nu} - \frac{T}{\rho h} \frac{\partial p}{\partial x_\nu} \right), \quad (2.26a)$$

$$\begin{aligned} \tau_{\text{NS}}^{\mu\nu} &= \pi_{\text{NS}}^{\mu\nu} + \Pi_{\text{NS}} \gamma^{\mu\nu}, \\ \tau_{\text{NS}}^{\mu\nu} &= -\gamma^{\mu\rho} \gamma^{\nu\sigma} \left[\eta \left(\frac{\partial u_\rho}{\partial x_\sigma} + \frac{\partial u_\sigma}{\partial x_\rho} - \frac{2}{3} g_{\rho\sigma} \frac{\partial u^\lambda}{\partial x_\lambda} \right) \right] - \xi \frac{\partial u^\lambda}{\partial x_\lambda} \gamma^{\mu\nu}, \end{aligned} \quad (2.26b)$$

that ensure the positivity of the entropy generation rate if the so-called transport coefficients κ , η and ξ are positive. These transport coefficients account for the thermal conductivity κ , the shear viscosity η and the bulk viscosity ξ . The expressions in Equation (2.26) are known as the constitutive equations of CIT. These equations will hereafter be referred to as the Navier–Stokes terms appearing in the non-perfect formulation, as indicated by the subscript NS.

Although these equations are written in covariant form, the theory of CIT exhibits undesirable tendencies which are attributed to the mathematical nature of this formulation. The constitutive relations in Equation (2.26) exhibits a parabolic mathematical character which implies any disturbances in those fields can propagate at infinite speeds. [Romatschke \(2010\)](#) demonstrated this feature through perturbation analysis, concluding that the diffusion speed was linearly proportional to wave number which permits superluminal speeds and therefore violates a basic premise of relativity theory. High wave numbers were associated with instabilities in the numerical solution as found by [Hiscock & Lindblom \(1983\)](#). Therefore, the relativistic Navier–Stokes equations alone do not deliver a causal theory of physical dissipation in a relativistic fluid.

To overcome these troublesome numerical and physical deficiencies the theory known as extended irreversible thermodynamics (EIT) can be employed. The entropy four-current from Equation (2.25) was identified to be insufficient and so for stability and causality the entropy four-current S^μ required second-order terms of the thermodynamic fluxes. This gave rise to the general entropy four-current ([Hiscock & Lindblom 1983](#)),

$$S^\mu = s\rho u^\mu + \frac{q^\mu}{T} - (\beta_0 \Pi^2 + \beta_1 q_\nu q^\nu + \beta_2 \pi_{\alpha\beta} \pi^{\alpha\beta}) \frac{u^\mu}{2T} + \alpha_0 \frac{\Pi q^\mu}{T} + \alpha_1 \frac{\pi^{\mu\nu} q_\nu}{T}, \quad (2.27)$$

where the terms β_0 , β_1 and β_2 are thermodynamic coefficients that model the scalar, vector and tensor dissipative contributions to the entropy four-current, respectively. Changes in entropy relating to viscosity and heat flux coupling are measured by the terms α_0 and α_1 . After taking the divergence of the entropy four-current the second law of thermodynamics is again satisfied by imposing linear relations between the thermodynamics fluxes and extended forces. As a consequence of the Israel–Stewart phenomenological theory the dissipation variables for heat flow and viscosity are considered as independent variables that depend on convective derivatives of the dissipative variables themselves whilst ensuring that the entropy in the system does not decrease. This leads to a set of Israel–Stewart evolution equations which are presented in their truncated form according to [Muronga \(2007\)](#) as,

$$\hat{\mathcal{D}}\Pi = \frac{1}{\tau_\Pi}(\Pi_{\text{NS}} - \Pi) - I_\Pi, \quad (2.28a)$$

$$\hat{\mathcal{D}}\pi^{\mu\nu} = \frac{1}{\tau_\pi}(\pi_{\text{NS}}^{\mu\nu} - \pi^{\mu\nu}) - I_\pi^{\mu\nu}, \quad (2.28b)$$

$$\hat{\mathcal{D}}q^\mu = \frac{1}{\tau_q}(q_{\text{NS}}^\mu - q^\mu) - I_q^\mu, \quad (2.28c)$$

where the convective derivative is defined as $\hat{\mathcal{D}} = u^\alpha \nabla_\alpha$. The source terms I_Π , $I_\pi^{\mu\nu}$ and I_q^μ represent the higher-order contributions to the entropy four-current S^μ that are introduced through Israel–Stewart theory and τ_Π , τ_π and τ_q are relaxation times; they correspond to the mean collision time, the inverse of which relates to the mean-free flight time of a particle. For astrophysical applications the source terms on the right-hand-side of Equation (2.28) can be neglected such that simple telegrapher equations emerge ([Takamoto & Inutsuka 2011](#)),

$$W\left(\frac{\partial}{\partial t} + u^j \frac{\partial}{\partial x^j}\right)\Pi = \frac{1}{\tau_\Pi}(\Pi_{\text{NS}} - \Pi), \quad (2.29a)$$

$$W\left(\frac{\partial}{\partial t} + u^j \frac{\partial}{\partial x^j}\right)\pi^{\mu\nu} = \frac{1}{\tau_\pi}(\pi_{\text{NS}}^{\mu\nu} - \pi^{\mu\nu}), \quad (2.29b)$$

$$W\left(\frac{\partial}{\partial t} + u^j \frac{\partial}{\partial x^j}\right)q^\mu = \frac{1}{\tau_q}(q_{\text{NS}}^\mu - q^\mu), \quad (2.29c)$$

where the convective derivative has been written explicitly. Because the relaxation times depend on the collisions between constituent particles only cases close to equilibrium are considered as the continuum hypothesis is assumed. The relaxation times are computed using the dissipation transport coefficients Π , π and κ and the coefficients β_0 , β_1 and β_2 which should be computed from kinetic theory and are a unique property of a particular fluid. If the characteristic time-scale τ is much shorter than the fluid time-scale τ_{fluid} then this results in a set of stiff equations. In this work the relaxation times are computed according to ([Takamoto & Inutsuka 2011](#)),

$$\tau_\Pi = \frac{2\xi\beta^2}{5p}, \quad \tau_\pi = \frac{\eta}{p}, \quad \tau_q = \frac{2\kappa T\beta}{5p} \quad \text{if} \quad \Gamma = \frac{5}{3} \quad (2.30)$$

or

$$\tau_{\Pi} = \frac{72\xi}{\beta^2 p}, \quad \tau_{\pi} = \frac{3\eta}{2p}, \quad \tau_q = \frac{5\kappa T}{4p} \quad \text{if } \Gamma = \frac{4}{3}, \quad (2.31)$$

where $\beta = \rho/T$.

This completes the description of the governing equations for a relativistic non-perfect fluid, the requisite equations for which are summarised below.

In summary, the governing equations for a relativistic non-perfect fluid used in this work are comprised of the conservation laws,

$$\frac{\partial}{\partial t} \begin{pmatrix} D \\ S^i + q^0 u^i + q^i u^0 + \tau^{0i} \\ E - D + 2q^0 u^0 + \tau^{00} \end{pmatrix} + \frac{\partial}{\partial x^j} \begin{pmatrix} Du^i \\ S^i u^j + pI^{ij} + q^i u^j + q^j u^i + \tau^{ij} \\ S^j - Du^j + q^0 u^j + q^j u^0 + \tau^{0j} \end{pmatrix} = 0.$$

which must be solved in conjunction with the telegrapher equations for the dissipative variables

$$\begin{aligned} W \left(\frac{\partial}{\partial t} + v^j \frac{\partial}{\partial x^j} \right) \Pi &= \frac{1}{\tau_{\Pi}} (\Pi_{\text{NS}} - \Pi), \\ W \left(\frac{\partial}{\partial t} + v^j \frac{\partial}{\partial x^j} \right) \pi^{\mu\nu} &= \frac{1}{\tau_{\pi}} (\pi_{\text{NS}}^{\mu\nu} - \pi^{\mu\nu}), \\ W \left(\frac{\partial}{\partial t} + v^j \frac{\partial}{\partial x^j} \right) q^{\mu} &= \frac{1}{\tau_q} (q_{\text{NS}}^{\mu} - q^{\mu}), \end{aligned}$$

which makes use of the so-called Navier–Stokes terms

$$\begin{aligned} q_{\text{NS}}^{\mu} &= \kappa \gamma^{\mu\nu} \left(\frac{\partial T}{\partial x_{\nu}} - \frac{T}{\rho h} \frac{\partial p}{\partial x_{\nu}} \right), \\ \tau_{\text{NS}}^{\mu\nu} &= \pi_{\text{NS}}^{\mu\nu} + \Pi_{\text{NS}} \gamma^{\mu\nu}, \\ \tau_{\text{NS}}^{\mu\nu} &= -\gamma^{\mu\rho} \gamma^{\nu\sigma} \left[\eta \left(\frac{\partial u_{\rho}}{\partial x_{\sigma}} + \frac{\partial u_{\sigma}}{\partial x_{\rho}} - \frac{2}{3} g_{\rho\sigma} \frac{\partial u^{\lambda}}{\partial x_{\lambda}} \right) \right] - \xi \frac{\partial u^{\lambda}}{\partial x_{\lambda}} \gamma^{\mu\nu}, \end{aligned}$$

and the relaxation times computed according to

$$\tau_{\Pi} = \frac{2\xi\beta^2}{5p}, \quad \tau_{\pi} = \frac{\eta}{p}, \quad \tau_q = \frac{2\kappa T\beta}{5p} \quad \text{if } \Gamma = \frac{5}{3},$$

or

$$\tau_{\Pi} = \frac{72\xi}{\beta^2 p}, \quad \tau_{\pi} = \frac{3\eta}{2p}, \quad \tau_q = \frac{5\kappa T}{4p} \quad \text{if } \Gamma = \frac{4}{3}.$$

Chapter 3

The High-Resolution Shock-Capturing Framework

In this chapter, the basic building-blocks for a High-Resolution Shock-Capturing (HRSC) methodology imposed on a Godunov-type finite volume (FV) framework is discussed in the context of hyperbolic conservation laws. Two approximate Riemann solvers, the HLL and HLLC are presented which are used to compute the numerical fluxes at the cell interfaces. The initial conditions for the flux computations are provided by the spatial reconstruction scheme in which the present work makes extensive use of Weighted Essentially Non-Oscillatory (WENO) reconstruction which is also presented. Finally, details are provided concerning the time advances of the numerical solution due to an explicit Runge–Kutta method and the primitive variable recovery algorithm.

3.1 Hyperbolic Conservation Laws

The governing equations of inviscid (perfect) relativistic hydrodynamics (RHD) can be cast into a system of hyperbolic partial differential equations in conservative form such that

$$\frac{\partial}{\partial t} \mathbf{U} + \frac{\partial}{\partial x^a} \mathbf{F}^a(\mathbf{U}) = 0, \quad (3.1)$$

whereby the superscript a denotes the dimensions of the problem within a three-dimensional Cartesian coordinate system, i.e. $a = \{1, 2, 3\} \equiv \{x, y, z\}$. The variable $\mathbf{U}$ is attributed to a set of conserved variables which are evolved over time t according to the fluxes denoted by $\mathbf{F}^a$ that are a function of $\mathbf{U}$. Writing the RHD equations in conservation-law form is crucial for the correct physical solution of problems involving shocks and other discontinuities which often feature in relativistic flows. This conjecture is supported by two well-established theorems that can be summarised as follows ([Rezzolla & Zanotti 2013](#), pp. 363–364):

Theorem 1. “Conservative numerical schemes, if convergent, do converge to the weak solution of the problem.”

Theorem 2. *“Non-conservative schemes, i.e. schemes in which the equations are not written in the conservative form, see Equation (3.1), do not converge to the correct solution if a shock wave is present in the flow.”*

Solutions involving shocks can be considered as weak solutions to the conservative formulation. Therefore, for convergence to a physically correct numerical solution, the RHD equations in the present work are expressed in the conservative form.

The conservation law in Equation (3.1) is said to be a hyperbolic conservation law if the Jacobian of the numerical fluxes have real and distinct eigenvalues as well as a complete set of corresponding right eigenvectors (Toro 2009). The Jacobian matrices for each spatial direction are computed via

$$\mathbf{A}^a(\mathbf{U}) = \frac{\partial \mathbf{F}^j}{\partial \mathbf{U}}, \quad (3.2)$$

from which it is then possible to extract the eigenvalues λ associated with the system from the characteristic polynomial

$$|\mathbf{A}^a - \lambda \mathbf{I}| = 0, \quad (3.3)$$

where $\mathbf{I}$ is simply the identity matrix. A set of eigenvalues can then be found for each spatial direction at a point (x, t) which in turn provides the finite velocities at which information propagates within the system. The eigenvalues are therefore associated with the propagation speeds of flow disturbances which for hydrodynamics are comprised of sound and entropy waves (Martí & Müller 2015).

This eigen (spectral) decomposition has been performed for non-dissipative RHD compatible with special relativity by several authors. It was shown by Martí et al. (1991) that the system of RHD equations could be written in a conservative form by adopting an appropriate set of conserved variables, subsequent analysis of the conservative system in one-dimension revealed that it is of hyperbolic mathematical character. Banyuls et al. (1997) also showed that for general relativistic hydrodynamics the system demonstrated these same hyperbolic properties. This was later followed by Font et al. (1994) who gave the eigenvalues alongside corresponding right eigenvectors for the three-dimensional system and applied high-resolution shock-capturing (HRSC) methodologies to a multidimensional system. The full eigen decomposition was first presented for the three-dimensional system by Donat et al. (1998) in which a summary of the eigenvalues with corresponding left and right eigenvectors is shown by Martí & Müller (2015).

Because the system of conservation laws used to describe non-dissipative RHD possess the mathematical character of a hyperbolic system, well-established numerical techniques already in use for the solution of the classical Euler equations can be applied in the relativistic context, see e.g. Toro (2009). The application of HRSC methods first applied by Font et al. (1994) brought about a new technical revolution in the field. Exploiting the

hyperbolic character enabled an improved numerical treatment of discontinuities in the flow-field, associated to shocks, that are a characteristic attribute in non-linear hyperbolic systems. In the following sections the numerical methods required for the solution of hyperbolic conservation laws is discussed.

3.2 A Finite Volume Godunov-Type Method

In the present work, the numerical solution to the relativistic hydrodynamics equations is obtained using a finite volume (FV) Godunov-type method. This strategy ensures that the solution is found using a conservative numerical scheme and presents a forceful platform for building a HRSC numerical solver.

Following [Rezzolla & Zanotti \(2013\)](#), consider a one-dimensional numerical cell of size $\Delta x := x_{i+1/2} - x_{i-1/2}$. Using the finite volume method implies that the flow quantities $\mathbf{U}_i^n$ are defined as the cell (volume) averages at time-level t^n such that

$$\mathbf{U}_i^n := \frac{1}{\Delta x} \int_{x_{i-1/2}}^{x_{i+1/2}} \mathbf{U}(x, t^n) dx. \quad (3.4)$$

Integrating the conservation law from Equation (3.1), considering only the x -direction, over a space-time control volume specified by $[x_{i-1/2}, x_{i+1/2}] \times [t^n, t^{n+1}]$ where $\Delta t := t^{n+1} - t^n$ one finds,

$$\int_{t^n}^{t^{n+1}} \int_{x_{i-1/2}}^{x_{i+1/2}} \frac{\partial}{\partial t} \mathbf{U}(x, t) dx dt = - \int_{t^n}^{t^{n+1}} \int_{x_{i-1/2}}^{x_{i+1/2}} \frac{\partial}{\partial x} \mathbf{F}(\mathbf{U}(x, t)) dx dt, \quad (3.5)$$

from which the value of $\mathbf{U}_i^{n+1}$ can be isolated by taking the appropriate time and spatial differences over the relevant derivatives such that

$$\int_{x_{i-1/2}}^{x_{i+1/2}} \mathbf{U}_i^{n+1}(x, t) dx = \int_{x_{i-1/2}}^{x_{i+1/2}} \mathbf{U}_i^n(x, t) dx + \int_{t^n}^{t^{n+1}} \mathbf{F}(\mathbf{U}(x_{i-1/2}, t)) dt - \int_{t^n}^{t^{n+1}} \mathbf{F}(\mathbf{U}(x_{i+1/2}, t)) dt. \quad (3.6)$$

By defining the numerical fluxes as

$$\mathbf{F}_{i\pm\frac{1}{2}} := \frac{1}{\Delta t} \int_{t^n}^{t^{n+1}} \mathbf{F}(\mathbf{U}(x_{i\pm\frac{1}{2}}, t)) dt, \quad (3.7)$$

the finite volume Godunov method can be written as

$$\mathbf{U}_i^{n+1} = \mathbf{U}_i^n + \frac{\Delta t}{\Delta x} (\mathbf{F}_{i-\frac{1}{2}} - \mathbf{F}_{i+\frac{1}{2}}). \quad (3.8)$$

The numerical fluxes at each cell boundary $\mathbf{F}_{i\pm\frac{1}{2}}^n$ are dependent on the states either side of the boundary; this property acts to ensure that a numerical scheme applied to Equation

(3.8) is a conservative numerical scheme. Therefore, the numerical fluxes are dependent only on the values in the adjacent cells, i.e.

$$\mathbf{F}_{i-\frac{1}{2}} = \mathcal{F}(\mathbf{U}_{i-1}^n, \mathbf{U}_i^n), \quad (3.9a)$$

$$\mathbf{F}_{i+\frac{1}{2}} = \mathcal{F}(\mathbf{U}_i^n, \mathbf{U}_{i+1}^n). \quad (3.9b)$$

Godunov (1959) made the realisation that the interface between each numerical cell gave rise to a local Riemann problem separating the left and right states at $\mathbf{U}_{i\pm\frac{1}{2}}^n$. A Godunov-type method is known as any method which solves a local Riemann problem at the cell interface to obtain the fluxes $\mathbf{F}_{i\pm\frac{1}{2}}^n$. By using the solution to a local Riemann problem that relies on the wave structure at each point in space and time a desirable upwind property can also be maintained, one that correctly captures the direction of information contained within the flow-field.

3.3 Numerical Flux Evaluation

At the core of a Godunov-type method lies the solution to a local Riemann problem at each cell interface. This initial-value problem is capable of determining the flow pattern imminent from constant and discontinuous initial data for non-linear hyperbolic systems. At each numerical interface a set of initial conditions can be specified at each instance in time, e.g. surrounding $x_{i+\frac{1}{2}}$ one finds

$$\bar{\mathbf{U}}(x, t) = \begin{cases} \mathbf{U}_L & \text{if } x_{i-\frac{1}{2}} < x < x_{i+\frac{1}{2}}, \\ \mathbf{U}_R & \text{if } x_{i+\frac{1}{2}} < x < x_{i+\frac{3}{2}}, \end{cases} \quad (3.10)$$

where the subscripts L and R denote the initial states to the left and right of the interface, respectively. Figure 3.1 depicts the initial conditions for the local Riemann problem from Equation (3.10). The elementary non-linear waves present in hydrodynamics are comprised of shock and rarefaction waves which will always be separated by a contact discontinuity which can be illustrated using a Riemann fan. Figure 3.2 depicts a Riemann fan with two waves $\mathcal{W}$ travelling in opposite directions separated by a contact discontinuity $\mathcal{C}$. Four constant states surround the waves, two of which are the initial states whilst the remaining two are known as the intermediate states indicated by the subscript $*$. The region encompassing the two intermediate states is referred to as the star region in which a constant pressure and velocity is found throughout. The solution of the Riemann problem is self-similar and dependent entirely on the initial conditions which, when coupled with the (relativistic) Rankine–Hugoniot conditions, the intermediate states can be related to the initial states accordingly. At present, modern-day finite volume RHD codes can either rely on the exact or approximate solution to the local Riemann problem. The first appearance of an exact Riemann solver for RHD dates back to the analytical solution presented by Martí & Müller (1994). The solution was then successfully validated using test cases involving strong shocks, ultra-relativistic flows and interacting contact discontinuities (Martí

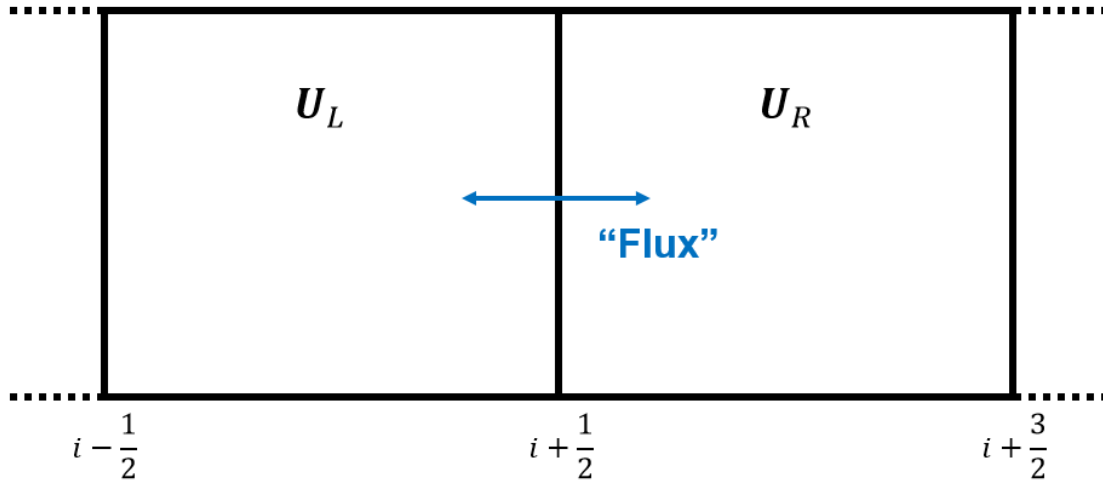


Figure 3.1: Riemann problem initial conditions specified by Equation (3.10) for the cell boundary $x_{i+\frac{1}{2}}$.

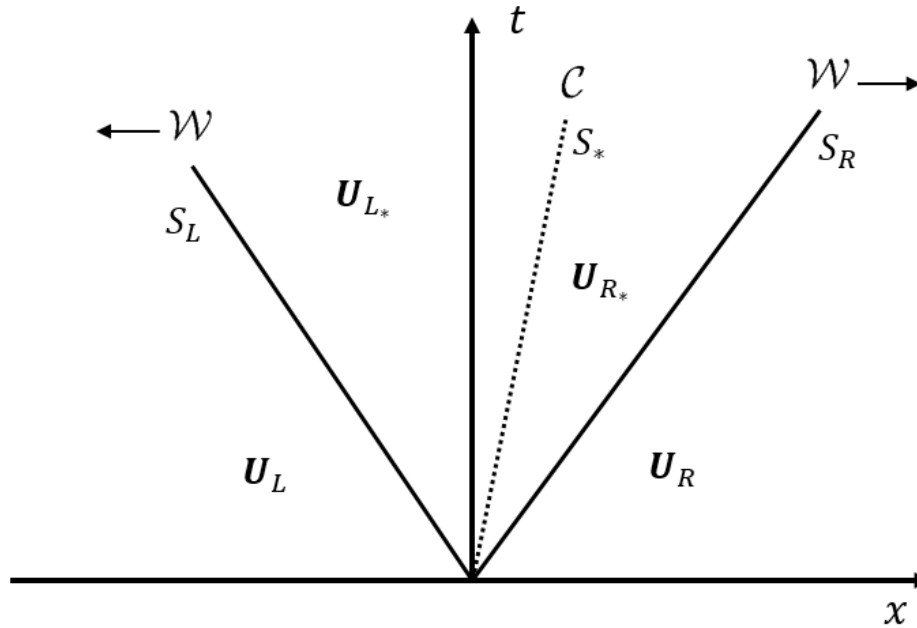


Figure 3.2: Wave structure arising from the solution of the Riemann problem shown using the Riemann fan.

& Müller 1996). It was then generalised for compatibility with multi-dimensional problems by Pons et al. (2000) such that tangential flow velocities were accommodated. A method to determine the correct wave pattern from the initial conditions before finding the pressure across the contact discontinuity led to the exact Riemann solver of Rezzolla & Zanotti (2001). The resulting methodology not only proved more efficient but also simpler for nu-

merical implementation. The Roe Riemann solver, originally developed by Roe (1981), finds the exact solution to the local Riemann problem using a linearised original system of equations. This approach was extended to general relativistic flows by Eulderink & Mellema (1995).

For larger-scale multi-dimensional problems which may also include different multi-physics considerations, approximate Riemann solvers are often preferred. One of the most popular choices of approximate Riemann solver is the HLL solver, originally formulated by Harten (1983) and first applied to relativistic flows by Schneider et al. (1993). Choosing only the smallest and largest wave speeds to determine the bounds of the intermediate state leads to a very simple implementation with good performance (Einfeldt 1988). Be that as it may, due to the fact that a contact discontinuity is not resolved, excessive smearing may be present in such cases (Rezzolla & Zanotti 2013). An even simpler alternative to the HLL Riemann solver is the approximate Riemann solver of Rusanov (1961) that requires only the maximum wave speed in the system. It still, however, possess the disadvantages of the HLL solver due to the inherent larger numerical dissipation. To tackle the issue of excessive smearing found using the HLL approximate Riemann solver, Toro et al. (1994) proposed the HLLC Riemann solver that seeks resolution of the missing contact discontinuity. This proved to be a very powerful strategy and was introduced in the context of RHD by Mignone & Bodo (2005) and later for RMHD by the same authors in Mignone & Bodo (2006).

The numerical RHD solver developed in this work makes use of the Rusanov, HLL and HLLC approximate Riemann solvers as well as variants thereof. In what follows, the HLL and HLLC Riemann solvers are summarised with a brief note on the Rusanov solver included within the HLL description.

3.3.1 The HLL Approximate Riemann Solver

A popular choice for the solution of the local Riemann problem is the HLL Riemann solver of Harten (1983), later modified by Einfeldt (1988) to yield the HLLC approximate Riemann solver. Its simple implementation and proven track-record of reliable flux computations have earned it the position of the standard workhorse within the majority of RHD research codes.

The HLLC (hereafter HLL) Riemann solver assumes that after the initial discontinuity decay, only two waves emanate that propagate in opposite directions with velocities S_L and S_R leaving a constant single-state in-between. In other words, from Figure 3.2, $U_{L*} = U_{R*} = U_*$. With knowledge of the smallest and largest characteristic wave speeds,

i.e. $S_L \leq 0$ and $S_R \geq 0$, the numerical flux at the cell boundary can be computed via

$$\mathbf{F}_{i \pm \frac{1}{2}} = \begin{cases} \mathbf{F}_L & \text{if } 0 \leq S_L, \\ \mathbf{F}_{\text{HLL}} & \text{if } S_L \leq 0 \leq S_R, \\ \mathbf{F}_R & \text{if } 0 \geq S_R, \end{cases} \quad (3.11)$$

where $\mathbf{F}_L = \mathbf{F}(\mathbf{U}_L)$, $\mathbf{F}_R = \mathbf{F}(\mathbf{U}_R)$ and $\mathbf{F}_{\text{HLL}}$ is the numerical HLL flux in the constant intermediate state given by

$$\mathbf{F}_{\text{HLL}} = \frac{S_R \mathbf{F}_L - S_L \mathbf{F}_R + S_L S_R (\mathbf{U}_R - \mathbf{U}_L)}{S_R - S_L}. \quad (3.12)$$

A further comment should be made about the choice of wave speeds. S_L and S_R should be chosen accordingly, in this work that selection is made via

$$S_L := \min(0, \lambda_-(\mathbf{U}_L), \lambda_-(\mathbf{U}_R)), \quad (3.13a)$$

$$S_R := \max(0, \lambda_+(\mathbf{U}_L), \lambda_+(\mathbf{U}_R)), \quad (3.13b)$$

where λ_- and λ_+ are provided by the expressions in Equation (2.17).

The HLL approximate Riemann solver is a robust strategy for numerical flux computations, is exact for single shocks and demonstrates good performance at critical sonic rarefactions (Rezzolla & Zanotti 2013, Martí & Müller 2015). On the other hand, in areas surrounding contact discontinuities, it is very dissipative. This is because the middle wave in the Riemann fan is ignored which subsequently prohibits the correct wave pattern when a contact discontinuity is present in the solution. The HLLC approximate Riemann solver presented in Chapter 3.3.2 offers a remedy to this issue.

Another simple numerical flux implementation is the Riemann solver of Rusanov (1961). Unlike the HLL Riemann solver only one wave speed is required for the numerical flux computation, this is found from, $S = \max(|S_L|, |S_R|)$, where S_L and S_R are given by Equation (3.13). The numerical flux is then computed via

$$\mathbf{F}_{\text{Rusanov}} = \frac{1}{2}(\mathbf{F}_L + \mathbf{F}_R) - \frac{1}{2}S(\mathbf{U}_R - \mathbf{U}_L). \quad (3.14)$$

Similarly to the HLL solver, the Rusanov solver also has the property of high numerical dissipation. That being said, when the HLL and Rusanov solvers are used in conjunction with high-order spatial reconstruction, troublesome numerical artefacts such as the Carbuncle phenomenon can be avoided, see e.g. Quirk (1994), Maccormack (2011).

3.3.2 The HLLC Approximate Riemann Solver

The HLLC approximate Riemann solver, where ‘C’ stands for *contact*, was introduced by Toro et al. (1994) to resolve the missing intermediate contact discontinuity absent in the

HLL construction. Within the relativistic framework the HLLC solver was re-formulated firstly for RHD by [Mignone & Bodo \(2005\)](#) and later for RMHD by the same authors ([Mignone & Bodo 2006](#)). The implementation strategy adopted closely follows [Mignone & Bodo \(2005\)](#) and is summarised here.

Firstly the HLL state and flux vectors must be computed,

$$\mathbf{U}_{\text{HLL}} = \begin{bmatrix} D_{\text{HLL}} \\ m_{\text{HLL}}^x \\ m_{\text{HLL}}^y \\ \tau_{\text{HLL}} \end{bmatrix} = \frac{S_R \mathbf{U}_R - S_L \mathbf{U}_L + \mathbf{F}_L - \mathbf{F}_R}{S_R - S_L}, \quad (3.15)$$

along with the corresponding HLL fluxes,

$$\mathbf{F}_{\text{HLL}} = \begin{bmatrix} F_{\text{HLL}}^D \\ F_{\text{HLL}}^{m^x} \\ F_{\text{HLL}}^{m^y} \\ F_{\text{HLL}}^\tau \end{bmatrix} = \frac{S_R \mathbf{F}_L - S_L \mathbf{F}_R + S_L S_R (\mathbf{U}_R - \mathbf{U}_L)}{S_R - S_L}, \quad (3.16)$$

and where the momentum vector S^i has been denoted as m^i for this discussion to avoid any confusion with the wave speeds. The intermediate wave speed in the x -direction is found via the quadratic equation,

$$\left(F_{\text{HLL}}^\tau + F_{\text{HLL}}^D \right) (S_*)^2 - \left[(\tau + D) + F_{\text{HLL}}^{m^x} \right] S_* + m^x = 0. \quad (3.17)$$

The positive root of the quadratic formula then provides a suitable intermediate wave speed, S_* . By consideration of the relativistic Rankine–Hugonit conditions the remainder of the conserved quantities within the intermediate star region can be obtained in each spatial direction,

$$\mathbf{F}_{L*} - \mathbf{F}_L = S_L (\mathbf{U}_{L*} - \mathbf{U}_L), \quad (3.18a)$$

$$\mathbf{F}_{R*} - \mathbf{F}_R = S_R (\mathbf{U}_{R*} - \mathbf{U}_R), \quad (3.18b)$$

and finding the pressure within the star region p_* from averaging the pressure computed in the left and right star regions,

$$p_K = \frac{[S_K(\tau + D) - m^x] S_* - (S_*(S_K - v^x) - p_K)}{1 - S_K S_*}, \quad (3.19)$$

where $K = L, R$. The corresponding HLLC numerical fluxes are then given by,

$$\mathbf{F}_{i \pm \frac{1}{2}} = \begin{cases} \mathbf{F}_L & \text{if } S_L \geq 0, \\ \mathbf{F}_L^* & \text{if } S_L \leq 0 \leq S_*, \\ \mathbf{F}_R^* & \text{if } S_* \leq 0 \leq S_R, \\ \mathbf{F}_R & \text{if } S_R \leq 0, \end{cases} \quad (3.20)$$

The HLLC Riemann solver is capable of accurate resolution of shocks in one-dimensional problems and, in general, provides superior restoration of the contact and tangential discontinuities when compared with the HLL Riemann solver. However, as discussed by [Wang et al. \(2008\)](#), numerical pathologies in higher-dimensions may emerge when shocks are present in the solution.

3.4 Spatial Reconstruction

Initial conditions required for the solution of the local Riemann problem in Equation (3.10) are obtained by means of a spatial reconstruction method. The techniques used for this operation are equally applicable for hyperbolic problems in classical hydrodynamics and relativistic hydrodynamics, hence modification of their original formulation is not required. If a piecewise constant representation is used, in other words, the initial conditions to the Riemann problem in Equation (3.10) are obtained by the two neighboring cell averages from Equation (3.23), then any information contained within the cell is lost. To circumvent this information loss, a polynomial representation of the flow variables within each cell can be used that in turn provides better approximations for the left and right states required for the local Riemann problem.

The original method of [Godunov \(1959\)](#) made use of the piecewise constant approximation of the solution data, a low-order method, which was capable of resolving discontinuities in a monotonic fashion and did not generate spurious oscillations. However this method presents the disadvantage that it is highly dissipative which could smooth out flow features and smear discontinuities within the flow. Godunov then posed the theorem, hereafter referred to as Godunov's theorem stating that:

Theorem 3 (Godunov's Theorem). *A linear and monotonicity-preserving scheme is at most first-order accurate.*

A historical effort then ensued which saw the steady development of methods capable of breaking the first-order barrier set by Godunov. An idea to use piecewise linear slopes was championed by [van Leer \(1979\)](#) which gave rise to the Monotone Upstream-centered Scheme for Conservation Laws (MUSCL) that provided second-order spatial accuracy. These methods are known as slope limiter methods which act to maintain the monotonicity and TVD properties of the solution. The next major advancement came from the development of the piecewise parabolic method (PPM) originally devised by [Colella & Woodward \(1984\)](#) and first applied to RHD by [Martí & Müller \(1996\)](#) and [Mignone et al. \(2005\)](#). Using the PPM method, a parabolic interpolation procedure is used. By default the method is not TVD hence monotonicity constraints must be included in the solution. This method is capable of achieving third-order accuracy in smooth flow regions however as authors such as [Baiotti et al. \(2009\)](#) have shown, the order of convergence for the PPM rarely exceeds second-order.

In the present work two differing methodologies are used for the spatial reconstruction procedure. The first, and most prolific family of schemes used throughout this work, is the so-called Weighted Essentially Non-Oscillatory (WENO) schemes, originally devised by [Liu et al. \(1994\)](#) from the extension of the ENO scheme proposed by [Harten et al. \(1987\)](#). The second is the aforementioned second-order MUSCL scheme, which is generally used as a benchmark to compare higher-order methodologies against.

3.4.1 WENO Reconstruction

WENO schemes are well-suited for problems containing discontinuities and complex intricate smooth flow features. These schemes use a combination of adaptive stencils in which each stencil contains a polynomial constructed from the cell-average flow quantities. The sub-stencils surround the interface for which the spatial reconstruction is being applied, i.e. where the Riemann problem is being solved. Then, the stencils are assigned a specific non-linear weight depending on the local smoothness surrounding each stencil. When a stencil contains non-smooth data, i.e. a discontinuity, its weight are set to, essentially, zero. On the other hand, when a solution contains smooth data, optimal weights are prescribed. This results in an weighted combination of the global stencil based on the polynomials constructed from each sub-stencil. The order of accuracy for WENO schemes is determined by the number of cells in each sub-stencil, k , such that the entire reconstruction polynomial is of order $(2k - 1)$.

Polynomial Construction

Consider a uniform grid such that $x_i = i\Delta x$ and positions at the cell interface defined by $x_{i+\frac{1}{2}} = \frac{1}{2}(x_i + x_{i+1})$. By choosing an order of accuracy k , it is possible to define at any given cell a stencil $S_n(i)$ in which

$$r + s + 1 = k, \quad (3.21)$$

where r and s denote the number of cells to the left and right, respectively, such that the stencil takes the form

$$S_n(i) \equiv \{I_{i-r}, \dots, I_{i+s}\}. \quad (3.22)$$

Figure 3.3 provides a schematic example of the candidate stencils when $k = 3$ for the WENO3 scheme, reconstructed at $i + \frac{1}{2}$. The assumption that the cell averages are used throughout the WENO procedure is again reiterated,

$$\bar{u}_i = \frac{1}{\Delta x} \int_{x_{i-\frac{1}{2}}}^{x_{i+\frac{1}{2}}} u(x) dx. \quad (3.23)$$

Following [Shu \(1997\)](#) and [Shu \(2009\)](#), it is possible to specify a unique polynomial $p_n(x)$ of degree at most $k - 1 = r + s$, corresponding to the stencil $S_n(i)$ and containing the cell

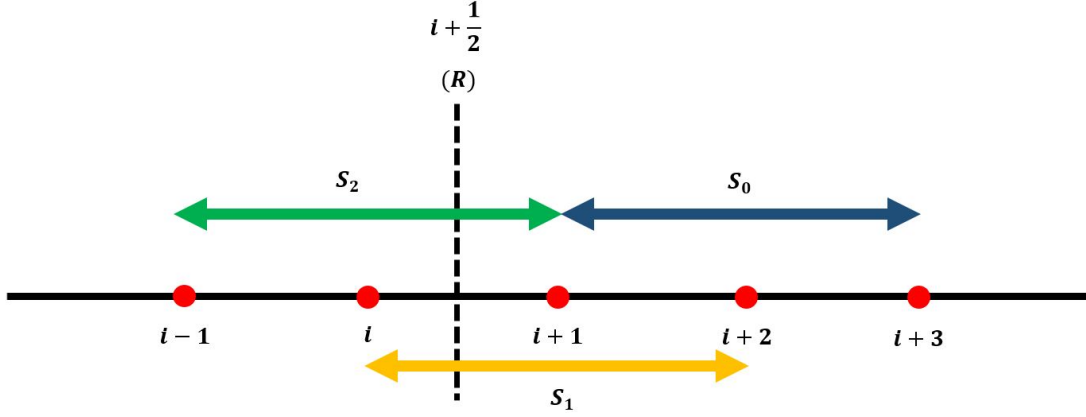


Figure 3.3: Schematic representation of the candidate stencils for the WENO3 reconstruction at $x_{i+\frac{1}{2}}$, i.e. when $k = 3$.

averages, such that

$$\frac{1}{\Delta x_j} \int_{x_{j-\frac{1}{2}}}^{x_{j+\frac{1}{2}}} p_n(\zeta) d\zeta = \bar{u}_j, \quad j = i-r, \dots, i+s, \quad (3.24)$$

in the interval $I_i = (x_{i-\frac{1}{2}}, x_{i+\frac{1}{2}})$. The polynomial in Equation (3.24) provides a k^{th} -order approximation of $u(x)$ inside the cell I_i , if the function $u(x)$ is smooth inside the stenciled region — see [Shu \(1997\)](#) for a proof.

An approximation of the function $u(x)$ is required at the cell boundaries, i.e. $u(x_{i+\frac{1}{2}})$. By consideration of the primitive function

$$U(x) = \int_{x-\frac{1}{2}}^x u(\zeta) d\zeta, \quad (3.25)$$

in which the lower limit is irrelevant, it follows that

$$U(x_{i+\frac{1}{2}}) = \int_{x_{i-\frac{1}{2}}}^{x_{i+\frac{1}{2}}} u(\zeta) d\zeta = \sum_{l=0}^i \int_{x_{i-\frac{1}{2}}}^{x_{i+\frac{1}{2}}} u(\zeta) d\zeta = \sum_{l=0}^i \Delta x \bar{u}_l. \quad (3.26)$$

All this implies that with knowledge of the cell averages within the stencil, an approximation of $U(x_{i+\frac{1}{2}})$ can be found at the cell interface. One can then find an interpolation polynomial of the primitive function $U(x)$ where its derivative approximates $u(x) = U'(x)$. By substitution of $u^{(n)}(x_{i+\frac{1}{2}}) \equiv p_n(x_{i+\frac{1}{2}})$ expressions for the values at $u_{i+\frac{1}{2}}^L$ and $u_{i-\frac{1}{2}}^R$ using the constant coefficients c_{rj} and $\tilde{c}_{rj}$, respectively, considering the left shift of the stencil r and the order of accuracy k of the stencil $S_n(i)$, such that

$$u_{i+\frac{1}{2}}^- = \sum_{j=0}^{k-1} c_{rj} \bar{u}_{i-r+j}, \quad u_{i-\frac{1}{2}}^+ = \sum_{j=0}^{k-1} \tilde{c}_{rj} \bar{u}_{i-r+j}. \quad (3.27)$$

The superscripts $\pm$ arise due to the different stencil possibilities at the same location. If the left shift r is then instead associated with the point of reconstruction $x_{i+\frac{1}{2}}$, then the superscripts become redundant and there is no need to consider $\tilde{c}$ since

$$\tilde{c}_{rj} = c_{r-1,j}, \quad (3.28)$$

which when the grid is uniform, i.e. $x_i = \Delta x$, the constant values of c_{rj} for the reconstruction at $x_{i+\frac{1}{2}}$ can be pre-computed from

$$c_{rj} := \sum_{m=j+1}^k \frac{\sum_{l=0, l \neq m}^k \prod_{q=0, q \neq m, l}^k (r - q + 1)}{\prod_{l=0, l \neq m}^k (m - l)}. \quad (3.29)$$

Linear Weights

Polynomial approximations calculated over the large stencil can be written as a linear convex combination of the smaller candidate stencils $s^n(i)$ by

$$u_{i+\frac{1}{2}} = \sum_{r=0}^{k-1} \gamma_r u_{i+\frac{1}{2}}^{(r)}, \quad (3.30)$$

where the constant coefficients γ_r satisfies $\sum_{r=0}^{k-1} \gamma_r = 1$.

Non-Linear Weights

Unlike its ENO predecessor, WENO schemes make use of a convex combination of the different candidate stencils which can be expressed through

$$u_{i+\frac{1}{2}} = \sum_{r=0}^{k-1} \omega_r u_{i+\frac{1}{2}}^{(r)}, \quad (3.31)$$

where $\omega \geq 0$ and $\sum_{r=0}^{k-1} \omega_r = 1$ is required for stability and consistency.

If the function $u(x)$ is smooth across all stencils, it is desirable to have

$$\omega_r = \gamma_r + \mathcal{O}(\Delta x^{k-1}), \quad r = 0, \dots, k-1, \quad (3.32)$$

which entails an accuracy of $(2k-1)$ for smooth flows,

$$u_{i+\frac{1}{2}} = \sum_{r=0}^{k-1} \omega_r u_{i+\frac{1}{2}}^{(r)} = u(x_{i+\frac{1}{2}}) + \mathcal{O}(\Delta x^{2k-1}), \quad (3.33)$$

as shown by [Shu \(1997\)](#).

The non-linear weights in Equation (3.32) satisfy the follow two criteria:

- $\omega_r \approx \gamma_r$ if $u(x)$ is smooth across the large stencil.
- $\omega_r \approx 0$ if $u(x)$ contains a discontinuity in any of the candidate stencils $S_n(i)$.

The non-linear weights themselves should be smooth functions and computed in the most efficient manner possible, this led to the following,

$$\omega_r = \frac{\alpha_r}{\sum_{s=0}^{k-1} \alpha_s}, \quad r = 0, \dots, k-1 \quad (3.34)$$

with

$$\alpha_r = \frac{\gamma}{(\epsilon + \beta_r)^2} \quad (3.35)$$

where ϵ is simply a very small constant number to avoid any division by zero, in the present work $\epsilon = 10^{-6}$. The new term, β_r , is known as a *smoothness indicator* which is computed for each of the candidate stencils. The smoothness indicators measure the relative smoothness of each the function $u(x)$ in each stencil, the larger the value of β_r , the less smooth the solution is within that stencil. The smoothness indicator can be computed from

$$\beta_r = \sum_{l=1}^{k-1} \int_{x_{i-\frac{1}{2}}}^{x_{i+\frac{1}{2}}} \Delta x^{2l-1} \left(\frac{\partial^l p_r(x)}{\partial^l x} \right)^2 dx. \quad (3.36)$$

Equation (3.36) is therefore comprised of the sum of the squares of scaled L_2 norms for the polynomial $p_r(x)$ in the interval $(x_{i-\frac{1}{2}}, x_{i+\frac{1}{2}})$. A scaling factor of Δx^{2l-1} is used to remove any dependency on the grid size in the calculation of the smoothness indicator.

Monotonicity Preserving Considerations

The promising developments by [Liu et al. \(1994\)](#) and [Jiang & Shu \(1996\)](#) led to the introduction of (essentially) non-oscillatory high-order spatial reconstruction methods within a Godunov-type scheme. Despite their early success, it was noticed that for schemes with a theoretical order above five, the monotonicity properties of the schemes were lost. To circumvent these issues, [Balsara & Shu \(2000\)](#) were able to devise a new class of high-order WENO schemes with an additionally monotonicity preserving (MP) functionality based on the work of [Suresh & Huynh \(1997\)](#). This new breed of high-order WENO schemes were suitably named MP-WENO and demonstrated the desirable monotonicity preserving properties at orders higher than five.

Although high-order WENO schemes have been applied to a variety of RHD problems by authors such as [Tchekhovskoy et al. \(2007\)](#), [Nunez-de la Rosa & Munz \(2016\)](#) and [Radice & Rezzolla \(2012\)](#), an explicit monotonicity procedure has not been included for the very high-orders. Instead, for example, [Nunez-de la Rosa & Munz \(2016\)](#) opted to identify troubled regions surrounding discontinuities using a shock detector and then locally reduce

the order of the WENO scheme. This approach is practical and requires minimal implementation effort whilst also providing robustness to the high-order schemes. Although the MP procedure is applied to WENO orders above five by [Balsara & Shu \(2000\)](#), the MP procedure is applied to orders of five (inclusive) and above such that an MP-WENO5 and MP-WENO7 scheme is used in the present work, the third-order WENO3 scheme remains unchanged.

The monotonicity procedure presented by [Balsara & Shu \(2000\)](#) applied to the cell interface at $x_{i-\frac{1}{2}}$ is described, since the procedure at $x_{i+\frac{1}{2}}$ can be found in the original text. The monotonicity procedure requires both the minmod and median functions

$$\text{minmod}(a_1, a_2, \dots, a_n) = \frac{1}{n} \left[\sum_k^n \text{sign}(a_k) \right] \min(|a_1|, \dots, |a_n|), \quad (3.37a)$$

$$\text{median}(a, b, c) = a + \text{minmod}(b - a, c - a). \quad (3.37b)$$

Curvature measures are then chosen using the cell averages defined in Equation (3.23),

$$d_i = u_{i+1} - 2u_i + u_{i-1}, \quad (3.38)$$

which can be used also to define the curvature at the zone boundary. A definition of the curvature at the cell interface is required such that smooth local extrema are not clipped which would result in a loss in the order of accuracy. Therefore, a limited curvature is defined from which the M4X strategy is adopted which offers the largest domain of support from those presented in the original literature,

$$d_{i-\frac{1}{2}}^{\text{M4X}} = \text{minmod}(4d_{i-1} - d_i, 4d_i - d_{i-1}, d_{i-1}, d_i, d_{i-2}, d_{i+1}). \quad (3.39)$$

Maximum and minimum bounds $u_{i-\frac{1}{2}}^{R,\min}$ and $u_{i-\frac{1}{2}}^{R,\max}$, respectively, need to be found in which the solution $u_{i-\frac{1}{2}}^R$ must lie. The right-sided upper limit (UL) is found from

$$u_{i-\frac{1}{2}}^{\text{UL}} = u_i - \alpha(u_{i+1} - u_i), \quad (3.40)$$

where in the present work $\alpha = 2$. The median (MD) is given by

$$u_{i-\frac{1}{2}}^{\text{MD}} = \frac{1}{2}(u_{i-1} - u_i) - \frac{1}{2}d_{i-\frac{1}{2}}^{\text{MD}}. \quad (3.41)$$

Then the right-sided large curvature (LC) is given by

$$u_{i-\frac{1}{2}}^{\text{LC}} = u_i + \frac{1}{2}(u_i - u_{i+1}) + \frac{\beta}{3}d_{i-\frac{1}{2}}^{\text{LC}}, \quad (3.42)$$

where in the present work $\beta = 4$. Finally, expressions for $u_{i-\frac{1}{2}}^{R,\min}$ and $u_{i-\frac{1}{2}}^{R,\max}$ are provided by

$$u_{i-\frac{1}{2}}^{R,\min} = \max \left[\min \left(u_i, u_{i-1}, u_{i-\frac{1}{2}}^{\text{MD}} \right), \min \left(u_i, u_{i-\frac{1}{2}}^{\text{LC}}, u_{i-\frac{1}{2}}^{\text{UL}} \right) \right], \quad (3.43)$$

$$u_{i-\frac{1}{2}}^{R,\max} = \min \left[\max \left(u_i, u_{i-1}, u_{i-\frac{1}{2}}^{\text{MD}} \right), \max \left(u_i, u_{i-\frac{1}{2}}^{\text{LC}}, u_{i-\frac{1}{2}}^{\text{UL}} \right) \right], \quad (3.44)$$

from which the monotonicity preserving value of $u_{i-\frac{1}{2}}^R$ is finally obtained,

$$u_{i-\frac{1}{2}}^R = \text{median}(u_{i-\frac{1}{2}}^R, u_{i-\frac{1}{2}}^{R,\min}, u_{i-\frac{1}{2}}^{R,\max}). \quad (3.45)$$

3.4.2 MUSCL Reconstruction

The left and right states at the cell interfaces for $i - 1/2$ using the second-order MUSCL scheme are found from

$$\bar{u}_{i-\frac{1}{2}}^L = \bar{u}_i + \frac{1}{2}\phi(r_{i-1})(\bar{u}_i - \bar{u}_{i-1}), \quad (3.46a)$$

$$\bar{u}_{i-\frac{1}{2}}^R = \bar{u}_i - \frac{1}{2}\phi(r_i)(\bar{u}_{i+1} - \bar{u}_i), \quad (3.46b)$$

where

$$r_i = \frac{\bar{u}_i - \bar{u}_{i-1}}{\bar{u}_{i+1} - \bar{u}_i}. \quad (3.47)$$

A limiting function ϕ is applied to slope of the piecewise approximations to maintain the TVD property of the solution. In the present work the Minmod limiter function is used ([Roe 1986](#)),

$$\phi(r_i) = \max[0, \min(1, r)]. \quad (3.48)$$

The reconstruction at $i + 1/2$ is found from simply resolving Equation (3.46) at $i = i + 1$. This reconstruction is straightforward to implement and provides second-order accuracy.

3.5 Time Integration

To advance the hyperbolic system of conservation laws given by Equation (3.1) forward in time, the method of lines is employed. The high-order methods discussed thus far only apply to the spatial discretisation yet no consideration for the time discretisation strategy which is also essential to provide numerical solutions with a high-order of accuracy has been given. The Godunov method presented in Chapter 3.2 contains the idea that the solution is discretised in space but left continuous in time. This resulted in an ODE provided in Equation (3.8) that advances the solution in time, which, written explicitly for a one-dimensional case would be

$$\frac{d\mathbf{U}_i(t)}{dt} = \frac{1}{\Delta x} (\mathbf{F}_{i-\frac{1}{2}} - \mathbf{F}_{i+\frac{1}{2}}), \quad (3.49a)$$

$$=: \mathbf{L}(\mathbf{U}_i), \quad (3.49b)$$

where $\mathbf{L}(\mathbf{U}_i)$ denotes the spatial differential operator. This work relies on the explicit multi-stage Runge–Kutta method devised by [Shu & Osher \(1988\)](#). In particular, a third-order Runge–Kutta (RK3) method is used, which, from time-level t^n to time-level t^{n+1} can be summarised as

$$\mathbf{U}^{(1)} = \mathbf{U}^n + \Delta t \mathbf{L}(\mathbf{U}_i), \quad (3.50a)$$

$$\mathbf{U}^{(2)} = \frac{1}{2} \left[3\mathbf{U}^n + \mathbf{U}^{(1)} + \Delta t \mathbf{L}(\mathbf{U}^{(1)}) \right], \quad (3.50b)$$

$$\mathbf{U}^{n+1} = \frac{1}{3} \mathbf{U}^n + \frac{2}{3} \mathbf{U}^{(2)} + \frac{2}{3} \Delta t \mathbf{L}(\mathbf{U}^{(1)}), \quad (3.50c)$$

where the subscript notation has been dropped purely for convenience. This method is Total Variation Diminishing (TVD) meaning that the total variation of the solution defined by

$$\text{TV}(\mathbf{U}^n) = \sum_{i=-\infty}^{+\infty} |\mathbf{U}_{i+1}^n - \mathbf{U}_i^n| \quad (3.51)$$

will not increase over time like

$$\text{TV}(\mathbf{U}^n) \leq \text{TV}(\mathbf{U}^{n-1}) \leq \text{TV}(\mathbf{U}^0). \quad (3.52)$$

The third-order Runge–Kutta method is a TVD if the time-step is chosen according to

$$\Delta t = \frac{\text{CFL}}{\text{Dim}} \min \left(\frac{\Delta x^a}{|\lambda_a|} \right) \quad (3.53)$$

where Δx^a is the grid spacing and λ_a corresponds to the maximum wave speed from Equations (2.17) in the directions: $a = \{1, 2, 3\} \equiv \{x, y, z\}$. The variable *Dim* denotes the number of dimensions being modelled, in the current work this is either one or two. Finally, the CFL (Courant–Friedrichs–Lewy) condition will maintain stability provided the constraint

$$\text{CFL} \leq 1 \quad (3.54)$$

is met. Only information from the previous time-step is required for the TVD RK3 method so its implementation does not require large amounts of memory.

3.6 Primitive Variable Recovery

The conserved variables are the variables evolved in time by Equation (3.1) and therefore a procedure is required to obtain the primitive variables. Unlike in classical hydrodynamics, a closed-form mapping between the conserved and primitive variables does not exist, hence an additional numerical effort is needed. The procedure of obtaining the primitive variables from the conserved ones is known as primitive variable recovery, whereby

an additional procedure is implemented within the numerical code to do so. The primitive variable recovery algorithm as implemented within the developed high-order HRSC RHD code for an ideal equation of state of the form $h = h(p, \rho)$ follows that described in [Mignone & Bodo \(2005\)](#) which can be summarised as follows.

The relation between the primitive variables of interest $\mathbf{V} = (\rho, u, v, p)$ and the conserved variables is recalled as

$$D = \rho W, \quad (3.55a)$$

$$S^i = \rho h W^2 u^i, \quad (3.55b)$$

$$E = \rho h W^2 - p - D, \quad (3.55c)$$

where the Lorentz factor is defined as

$$W = \frac{1}{\sqrt{1 - u^i u_i}}, \quad (3.56)$$

and the energy density in Equation (3.55c) is also re-formulated such that $E = E + D$ before the primitive variable recovery procedure begins. An ideal gas is considered in the present work whereby the specific enthalpy takes the form

$$h = 1 + \frac{p\Gamma}{\rho(\Gamma - 1)}. \quad (3.57)$$

The inverse relation of Equation (3.55) resides in the solution of the non-linear equation for the pressure such that

$$f(p) = p + E - \mathcal{X} \left[D + \frac{p\Gamma}{(\Gamma - 1)} \mathcal{X} \right] = 0, \quad (3.58)$$

where

$$\mathcal{X} = \frac{1}{\sqrt{1 - |S^i|^2 / (E + p)^2}}. \quad (3.59)$$

The function presented in Equation (3.58) can be solved using a one-dimensional implicit Newton–Raphson root-finding procedure

$$p^{n+1} = p^n - \frac{f(p^n)}{f'(p^n)}, \quad (3.60)$$

where the derivative of the function is

$$f'(p) = 1 - \frac{\Gamma}{\Gamma - 1} \mathcal{X}^2 + \left[D + \frac{2\Gamma}{\Gamma - 1} p \mathcal{X} \right] \left[\frac{|S|^2}{(E + p)^3 \mathcal{X}^3} \right]. \quad (3.61)$$

Upon finding the pressure value within an acceptable region of accuracy, in this case when the difference in pressure from the previous Newton–Raphson iteration achieves

the criterion $|p^{n+1} - p^n| < p^n \times 10^{-6}$, the remaining primitive variables are computed. The velocities in the x - and y -directions are found from the relationship

$$u^i = \frac{S^i}{(E + p)}, \quad (3.62)$$

from which the Lorentz factor in Equation (3.56) is calculated accordingly. It then follows that the rest-mass density can be computed from

$$\rho = \frac{D}{W}, \quad (3.63)$$

and finally the updated specific enthalpy is re-computed from Equation (3.57) using the newly obtained primitive variables.

The above procedure applies only when the perfect RHD equations are considered. For the case of dissipative RHD in which the evolved conserved variables also include dissipative terms a small modification in this procedure is made. Following the same logic used by [Takamoto & Inutsuka \(2011\)](#), the assumption that the computed four-velocity u^μ only experiences a very small change during the dissipative stage of the solution is made. Therefore, the dissipative conserved variables U_{dissip} are firstly computed by using the primitive variables after the inviscid step. Next the ideal conserved variables are approximated by $U_{\text{ideal}} = U - U_{\text{dissip}}$. The above procedure can then be applied, updating the calculated value of U_{dissip} using the newly obtain primitive variables at each iteration.

Part II

Non-Dissipative Relativistic Hydrodynamics

Chapter 4

Numerical Test Bench for Ideal Relativistic Hydrodynamics

In this chapter a series of one and two-dimensional numerical test cases are performed on the developed Godunov-type HRSC numerical solver. These test cases aim to demonstrate the solver's vocation for reproducing high-fidelity solutions to problems involving both smooth flow profiles and those involving discontinuities. The following tests evaluate the implementation and performance of the numerical schemes introduced in Chapter 3 and are solely applied to the relativistic hydrodynamics equations for a perfect fluid. The test cases considered are ones used by various authors for the validation of their codes, an informative and up-to-date review on a selection of cases is presented by [Martí & Müller \(2015\)](#). Overall, the developed numerical solver is found to be competitive with those finite difference and finite volume solvers currently available in terms of its robustness and shock-capturing capability.

4.1 One-Dimensional Test Cases

4.1.1 Smooth Flow Problem

A one-dimensional smooth flow problem is used to determine the formal order of accuracy achievable by the different spatial reconstruction strategies implemented within the numerical framework. A simple smooth wave with a known exact solution is resolved for increasingly finer grid resolutions such that an order of convergence can be found from use of a suitable error norm. This common test has been performed particularly by those authors investigating the behavior of high-order schemes in the context of relativistic flows see e.g. [Tchekhovskoy et al. \(2007\)](#), [Wang et al. \(2008\)](#), [Radice & Rezzolla \(2012\)](#), [Bugner et al. \(2016\)](#), [Nunez-de la Rosa & Munz \(2016\)](#). A smooth sine wave propagating with constant velocity through a one-dimensional periodic domain of size $x \in [-1, 1]$ with uniform discretisation is considered. The initial conditions follow those used by [Bugner](#)

et al. (2016) which are given by

$$\rho(x, t) = 1.0 + 0.2 \sin(2\pi(x - ut)), \quad (4.1a)$$

$$u(x, t) = 0.2, \quad (4.1b)$$

$$p(x, t) = 1.0. \quad (4.1c)$$

An ideal equation of state with constant adiabatic index $\Gamma = 5/3$ is used as well as the third-order TVD Runge–Kutta time integration method and the HLL approximate Riemann solver. The numerical simulation is run until time $t = 2$ from which a comparison is made between the rest-mass density field of the numerical solution and the exact solution using Equation (4.1a). The L_1 error norm is used for this analysis which is found from

$$L_1(\rho) = \frac{\sum |\rho_{\text{numerical}} - \rho_{\text{exact}}|}{\sum |\rho_{\text{exact}}|}. \quad (4.2)$$

To demonstrate the formal order of accuracy a time-step size computed according to Equation (3.53) is used. Titarev & Toro (2004) remarked that a time-step computed according to

$$\Delta t \approx \Delta x^{(2r-1)/3}, \quad (4.3)$$

where r represented the smallest stencil size should be used for this test to recover the formal order of accuracy. A large, unpractical, computational cost is associated with the use of Equation (4.3) hence its use is avoided for the tests presented herein.

Table 4.1 presents the L_1 error norms and subsequent orders of convergence for the one-dimensional smooth problem. The second-order MUSCL scheme alongside the third, fifth and seventh-order WENO implementations are assessed. Furthermore, the fifth and seventh-order WENO schemes are assessed when an additional monotonicity preserving (MP) algorithm is included in the reconstruction procedure. While all reconstruction schemes achieve the desired order of accuracy, the monotonicity preserving WENO schemes fall short of their standard WENO counterparts. This is likely due to the inherent limiting procedure manifest within the MP-WENO scheme. Interestingly, the error norm reported when using the MP-WENO schemes are two (three) order(s) of magnitude lower when a comparison is made between the relevant orders.

NX	Standard Reconstruction								Monotonicity Preserving			
	MUSCL2		WENO3		WENO5		WENO7		MP-WENO5		MP-WENO7	
	L_1 Error	Order	L_1 Error	Order	L_1 Error	Order	L_1 Error	Order	L_1 Error	Order	L_1 Error	Order
10	1.18×10^{-1}		1.20×10^{-1}		7.32×10^{-2}		4.54×10^{-2}		1.06×10^{-1}		9.68×10^{-2}	
20	7.76×10^{-2}	0.60	7.38×10^{-2}	0.70	7.74×10^{-3}	3.24	9.22×10^{-4}	5.62	1.32×10^{-2}	4.12	6.09×10^{-3}	3.99
40	2.64×10^{-2}	1.56	2.08×10^{-2}	1.83	3.96×10^{-4}	4.29	2.18×10^{-5}	5.40	1.67×10^{-2}	1.22	5.68×10^{-3}	0.10
80	1.07×10^{-2}	1.30	9.25×10^{-3}	1.17	1.25×10^{-5}	4.98	3.93×10^{-6}	2.47	5.53×10^{-3}	2.81	2.39×10^{-3}	1.25
160	3.54×10^{-3}	1.59	2.46×10^{-3}	1.91	7.72×10^{-6}	0.70	7.83×10^{-7}	2.33	1.67×10^{-3}	3.23	5.90×10^{-4}	2.02
320	9.96×10^{-4}	1.83	6.07×10^{-4}	2.02	1.53×10^{-6}	2.33	1.71×10^{-7}	2.20	4.53×10^{-4}	2.88	2.26×10^{-4}	1.38

Table 4.1: Rates of convergence for the different numerical schemes applied to the one-dimensional smooth flow problem. A third-order TVD Runge–Kutta time integration method and HLL approximate Riemann solver were used. The simulation is performed until a final time of $t = 2$ at which time the L_1 error norm was computed.

4.1.2 One-Dimensional Shock Tube Problems

The vocation of a numerical solver to resolve non-linear wave patterns can be assessed via evaluation of a one-dimensional Riemann problem. The Riemann problem consists of two constant and discontinuous initial states given by

$$U(x, 0) = \begin{cases} U_L & \text{if } x < 0, \\ U_R & \text{if } x > 0, \end{cases} \quad (4.4)$$

where U_L and U_R denote the two constant left and right states, respectively. Bernhard Riemann deduced that four possible wave patterns are produced from initial conditions specified by Equation (4.4) that are comprised of either shock waves, rarefactions waves and a contact discontinuity. The two former waves will propagate in opposite directions and a contact discontinuity will exist between them. The wave pattern evolution of the left (L) and right (R) initial states can be concisely represented at $t > 0$ by

$$LR, \quad \text{at } t = 0, \quad (4.5a)$$

$$L\mathcal{W}_{\leftarrow}L_*\mathcal{C}R_*\mathcal{W}_{\rightarrow}R, \quad \text{at } t > 0, \quad (4.5b)$$

where $\mathcal{W}$ denotes a shock or rarefaction wave that propagates either toward the left ($\leftarrow$) or right ($\rightarrow$) with respect to the initial discontinuity. Two new hydrodynamic states L_* and R_* are formed behind the two waves propagating in opposite directions which are separated by a contact discontinuity $\mathcal{C}$. The states L_* and R_* have the same values of velocity and pressure but different values of density — this wave structure is shown schematically in Figure 3.2.

In what follows, a series of one-dimensional Riemann problems are considered that cover all the possible wave patterns emerging from the solution of Equation (4.4). A one-dimensional closed domain of size $[0, 1]$ is used in which a diaphragm placed at $x = 0.5$ for $t = 0$ that separates the left $\mathbf{U}_L$ and right $\mathbf{U}_R$ initial states. Unless otherwise stated, the numerical solutions are directly compared to the solutions obtained with the exact Riemann solver provided by [Martí & Müller \(2015\)](#). Figures 4.1–4.7 depict numerical solutions obtained using the HLL approximate Riemann solver, a third-order TVD Runge–Kutta time integration scheme and three orders of WENO reconstruction.

A table summarising the L_1 error norms computed with respect to the aforementioned exact solution for the HLL Riemann solver at successive grid resolutions is found in Table 4.2. Table 4.3 provides the same analysis but using the HLLC approximate Riemann solver. A convergence order of no more than unity is expected for a grid convergence study that contains discontinuities as all numerical schemes will reduce to first-order accurate in those regions. The tables presented indicate that, generally, first-order convergence was achieved however for certain test cases this was lower for both the HLL and HLLC Riemann solvers. The errors computed for the HLLC Riemann solver are typically marginally smaller which can be attributed to an improved accuracy of the contact discontinuity resolution. Overall, the HLL and HLLC Riemann solvers performed similarly across all test cases except for Riemann problem 7, the planar shock reflection, in which the HLLC Riemann solver could not correctly and stably resolve the same Lorentz factor as the HLL Riemann solver.

Riemann Problem 1: Diverging Shock Waves

The first Riemann problem consists of initial conditions

$$(\rho, u, v, p) = \begin{cases} (1.0, 0.9, 0.0, 1.0) & \text{if } x < 0, \\ (1.0, 0.0, 0.0, 10.0) & \text{if } x > 0, \end{cases} \quad (4.6)$$

with $\Gamma = 4/3$. The ensuing wave pattern is comprised of two shock waves, one propagating to the left and one to the right, which are separated by a contact discontinuity. The simulation is run until $t = 0.4$ using $\text{CFL} = 0.5$ and all three WENO reconstruction schemes. Dirichlet boundary conditions are applied to the left and right boundaries of the computational domain. Figure 4.1 reports the rest-mass density (Figure 4.1a), pressure (Figure 4.1b), x -velocity (Figure 4.1c) and Lorentz factor (Figure 4.1d) profiles using $N_X = 400$ at $t = 0.4$. The numerical solution finds good agreement with the exact solution, the highest-order MP-WENO7 scheme providing the most accurate resolution of the contact discontinuity. Spurious oscillations are present throughout the solutions in the star region of the wave pattern. These oscillations occur regardless of the spatial reconstruction order however the amplitude of the oscillations is smaller when the higher-order WENO schemes are used, most likely due to the monotonicity procedure manifest in the reconstruction scheme which acts to reduce the so-called Gibbs phenomenon.

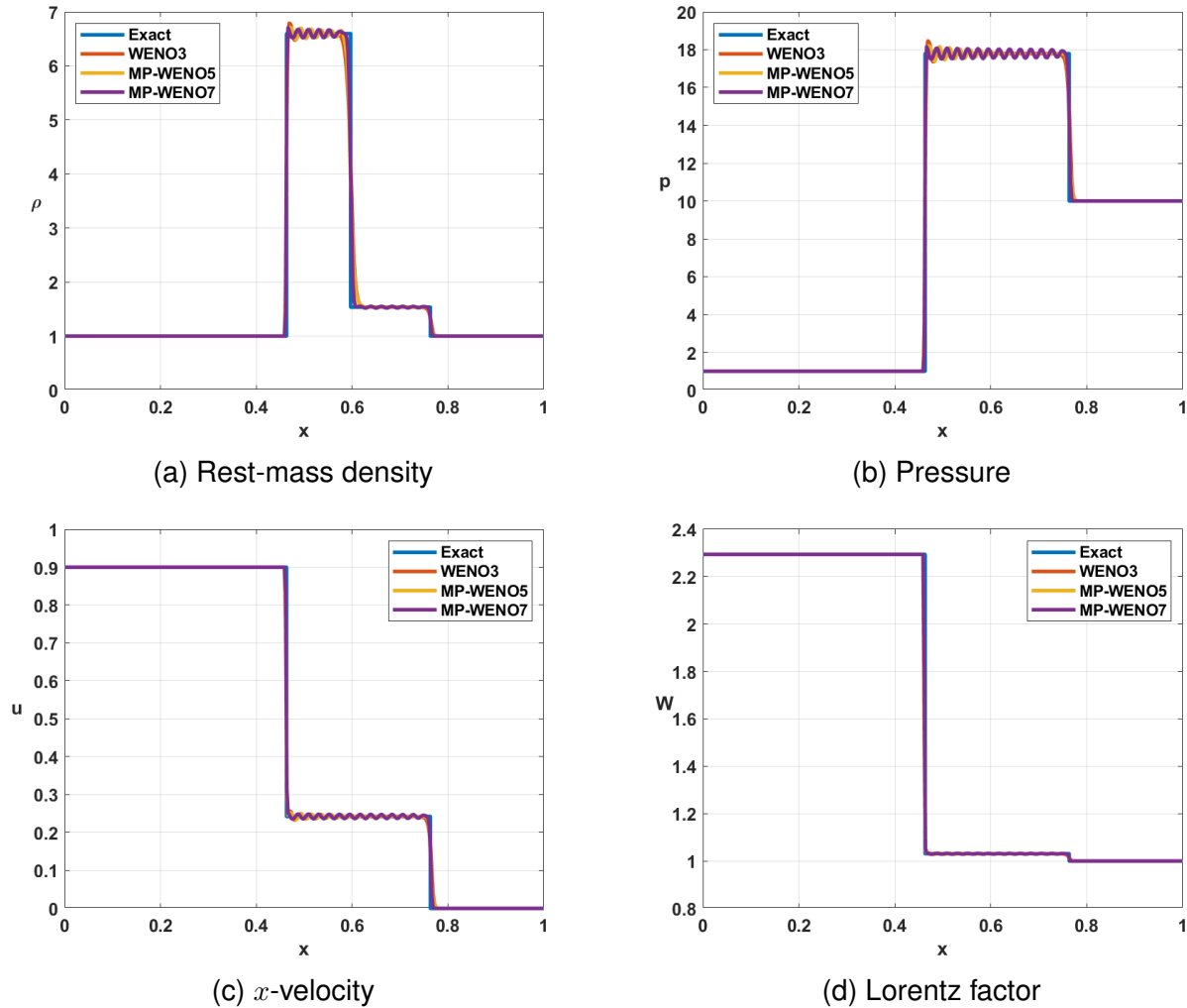


Figure 4.1: Numerical solution profiles of the Riemann problem 1 from initial conditions given in Equation (4.6) at $t = 0.4$ using a grid resolution of $NX = 400$ and $CFL = 0.5$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used.

Riemann Problem 2: Diverging Rarefaction Waves

The second Riemann problem consists of initial conditions

$$(\rho, u, v, p) = \begin{cases} (1.0, -0.6, 0.0, 10.0) & \text{if } x < 0, \\ (10.0, 0.5, 0.0, 20.0) & \text{if } x > 0, \end{cases} \quad (4.7)$$

with $\Gamma = 5/3$. The ensuing wave pattern is comprised of two rarefaction waves, one propagating to the left and one to the right, which are separated by a contact discontinuity. The simulation is run until $t = 0.4$ using $CFL = 0.5$ and all three WENO reconstruction schemes. Dirichlet boundary conditions are applied to the left and right boundaries of the

computational domain. Figure 4.2 reports the rest-mass density (Figure 4.2a), pressure (Figure 4.2b), x -velocity (Figure 4.2c) and Lorentz factor (Figure 4.2d) profiles using $NX = 400$ at $t = 0.4$. The numerical solutions provide a good agreement with the exact solution, the MP-WENO7 scheme captures the flow profiles more accurately than its lower-order counterparts and no spurious oscillations are present in the solution.

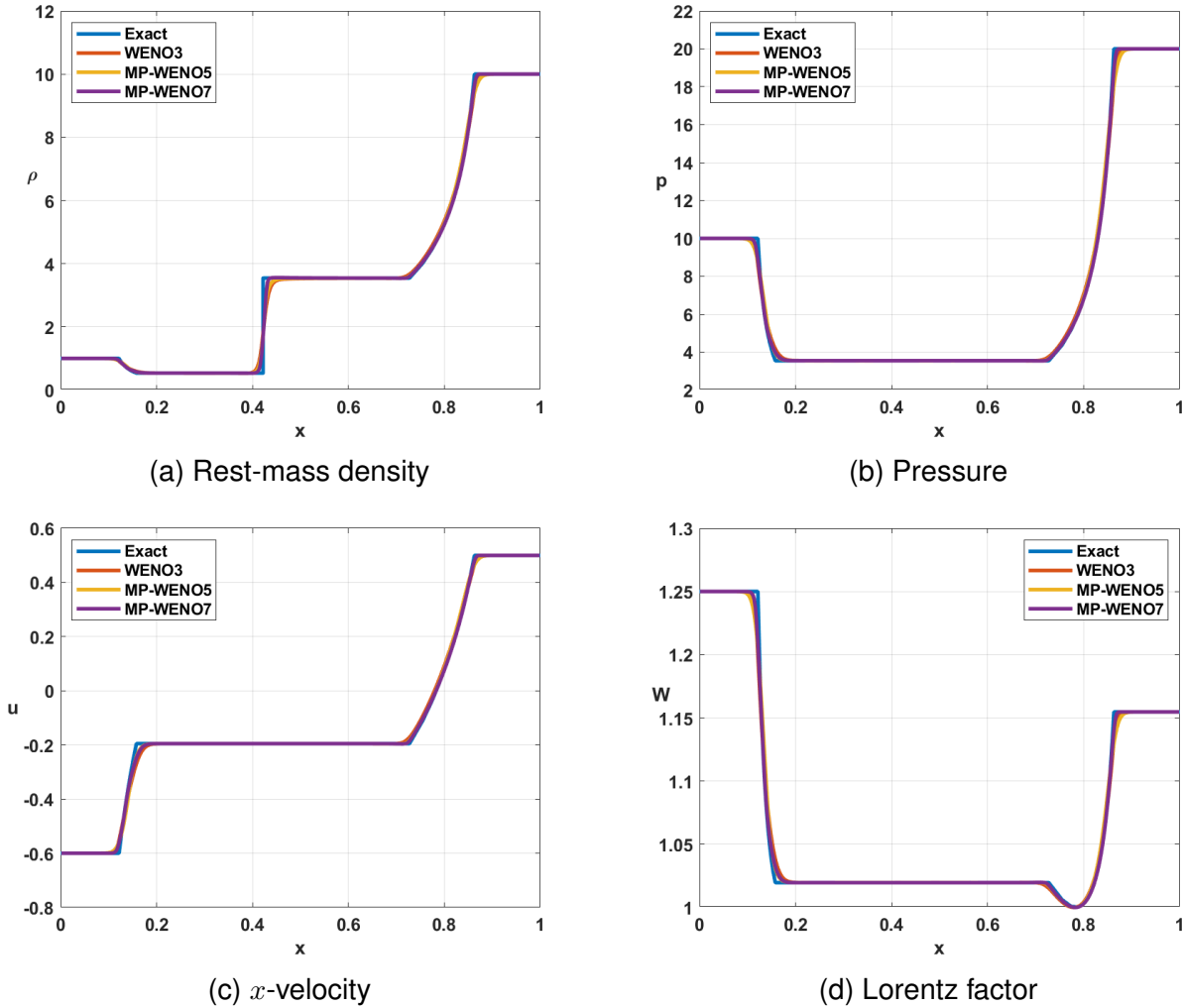


Figure 4.2: Numerical solution profiles of the Riemann problem 2 from initial conditions given in Equation (4.7) at $t = 0.4$ using a grid resolution of $NX = 400$ and $CFL = 0.5$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used.

Riemann Problem 3: Mildly Relativistic Blast Wave

The third Riemann problem, initially presented by [Schneider et al. \(1993\)](#), consists of initial conditions

$$(\rho, u, v, p) = \begin{cases} (10, 0, 0, 40/3) & \text{if } x < 0, \\ (1, 0, 0, 2/3^{-6}) & \text{if } x > 0, \end{cases} \quad (4.8)$$

with $\Gamma = 4/3$. The ensuing wave pattern is comprised of a left-going rarefaction wave and a right-going shock wave separated by a contact discontinuity. This Riemann problem gives rise to a dense shell propagating rightward at $u \approx 0.72c$. The simulation is run until $t = 0.4$ using $\text{CFL} = 0.5$ and all three WENO reconstruction schemes. Dirichlet boundary conditions are applied to the left and right boundaries of the computational domain. Figure 4.3 reports the rest-mass density (Figure 4.3a), pressure (Figure 4.3b), x -velocity (Figure 4.3c) and Lorentz factor (Figure 4.3d) profiles using $\text{NX} = 400$ at $t = 0.4$. The numerical findings all demonstrate good agreement with the exact solution, capturing the high-density shell well. Some spurious oscillations are present for the MP-WENO7 scheme across the high-density shell, similarly oscillations appear at the beginning of the rarefaction wave for the velocity (and therefore Lorentz factor) field.

Riemann Problem 4: Highly Relativistic Blast Wave

The fourth Riemann problem consists of initial conditions

$$(\rho, u, v, p) = \begin{cases} (1, 0, 0, 10^3) & \text{if } x < 0, \\ (1, 0, 0, 10^{-2}) & \text{if } x > 0, \end{cases} \quad (4.9)$$

with $\Gamma = 5/3$. The ensuing wave pattern is comprised of a left-going rarefaction wave and a right-going shock wave separated by a contact discontinuity. The large pressure gradient gives rise to a thin dense shell propagating rightward at $u \approx 0.96c$ in a similar fashion to Riemann problem 3. However, the specific enthalpy is much higher in this problem which implies a more thermodynamically relativistic configuration. The simulation is run until $t = 0.4$ using $\text{CFL} = 0.5$ and all three WENO reconstruction schemes. Dirichlet boundary conditions are applied to the left and right boundaries of the computational domain. Figure 4.4 reports the rest-mass density (Figure 4.4a), pressure (Figure 4.4b), x -velocity (Figure 4.4c) and Lorentz factor (Figure 4.4d) profiles using $\text{NX} = 400$ at $t = 0.4$. The thin dense shell that results from this Riemann problem is a challenge for many numerical codes, including the code presented here. The rest-mass density profile in Figure 4.4a shows a clear under-prediction for the magnitude of the high-density shell. Since the shell corresponds to about 1% of the grid length, accurate resolution of the shell clearly favours an approach with high grid resolution. It is noted that the MP-WENO7 scheme provides a better predication for the high-density shell than the WENO3 scheme which reproduces a much wider shell at a lower magnitude of rest-mass density. Like in Riemann problem 3, minor spurious oscillations are found at the beginning of the rarefaction wave for the x -velocity field.

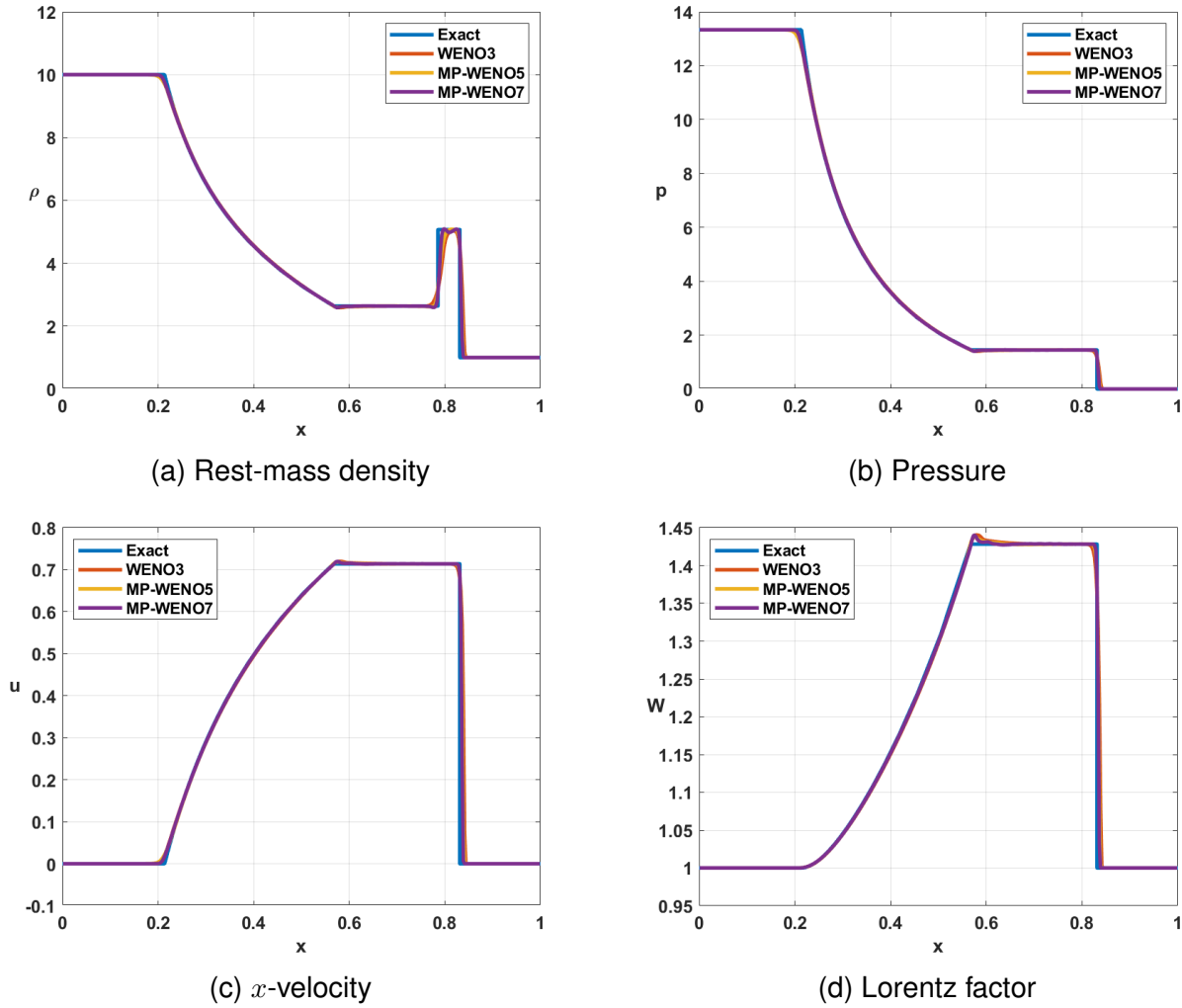


Figure 4.3: Numerical solution profiles of the Riemann problem 3 from initial conditions given in Equation (4.8) at $t = 0.4$ using a grid resolution of $NX = 400$ and $CFL = 0.5$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used.

Riemann Problem 5: Transverse Blast Wave

The fifth Riemann problem consists of initial conditions

$$(\rho, u, v, p) = \begin{cases} (1, 0, 0, 10^3) & \text{if } x < 0, \\ (1, 0, 0.99, 10^{-2}) & \text{if } x > 0, \end{cases} \quad (4.10)$$

with $\Gamma = 5/3$. The ensuing wave pattern is comprised of a left-going rarefaction wave and a right-going shock wave separated by a contact discontinuity. Unlike classical hydrodynamics, the relativistic equations experience a coupling in the momentum terms due to the Lorentz factor. [Pons et al. \(2000\)](#) presented solutions to the Riemann problem in

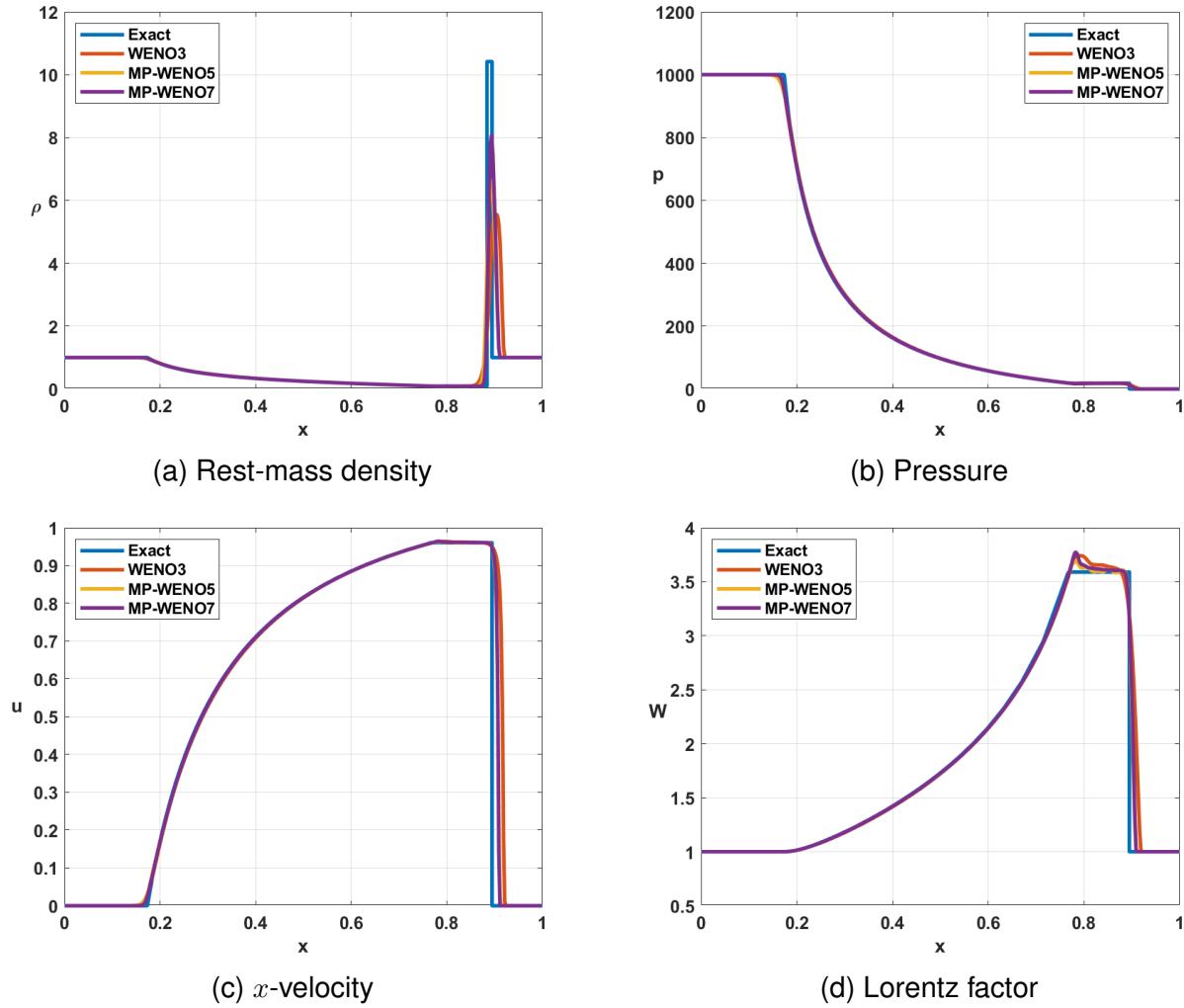


Figure 4.4: Numerical solution profiles of the Riemann problem 4 from initial conditions given in Equation (4.9) at $t = 0.4$ using a grid resolution of $NX = 400$ and $CFL = 0.5$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used.

which a transverse velocity component was included, realising that a drastically different flow profile emerged, especially for ultra-relativistic transverse flows. Therefore, any transverse component to the local of a local Riemann problem can lead to different flow dynamics when multi-dimensional flows are considered and directional splitting is used. Dirichlet boundary conditions are applied to the left and right boundaries of the computational domain. Figure 4.5 reports the rest-mass density (Figure 4.5a), pressure (Figure 4.5b), x -velocity (Figure 4.5c) and y -velocity (Figure 4.5d) profiles using $NX = 400$ at $t = 0.4$. Good agreement is found with all schemes compared to the exact solution with the higher-order schemes faring better in the regions surrounding the contact discontinuity.

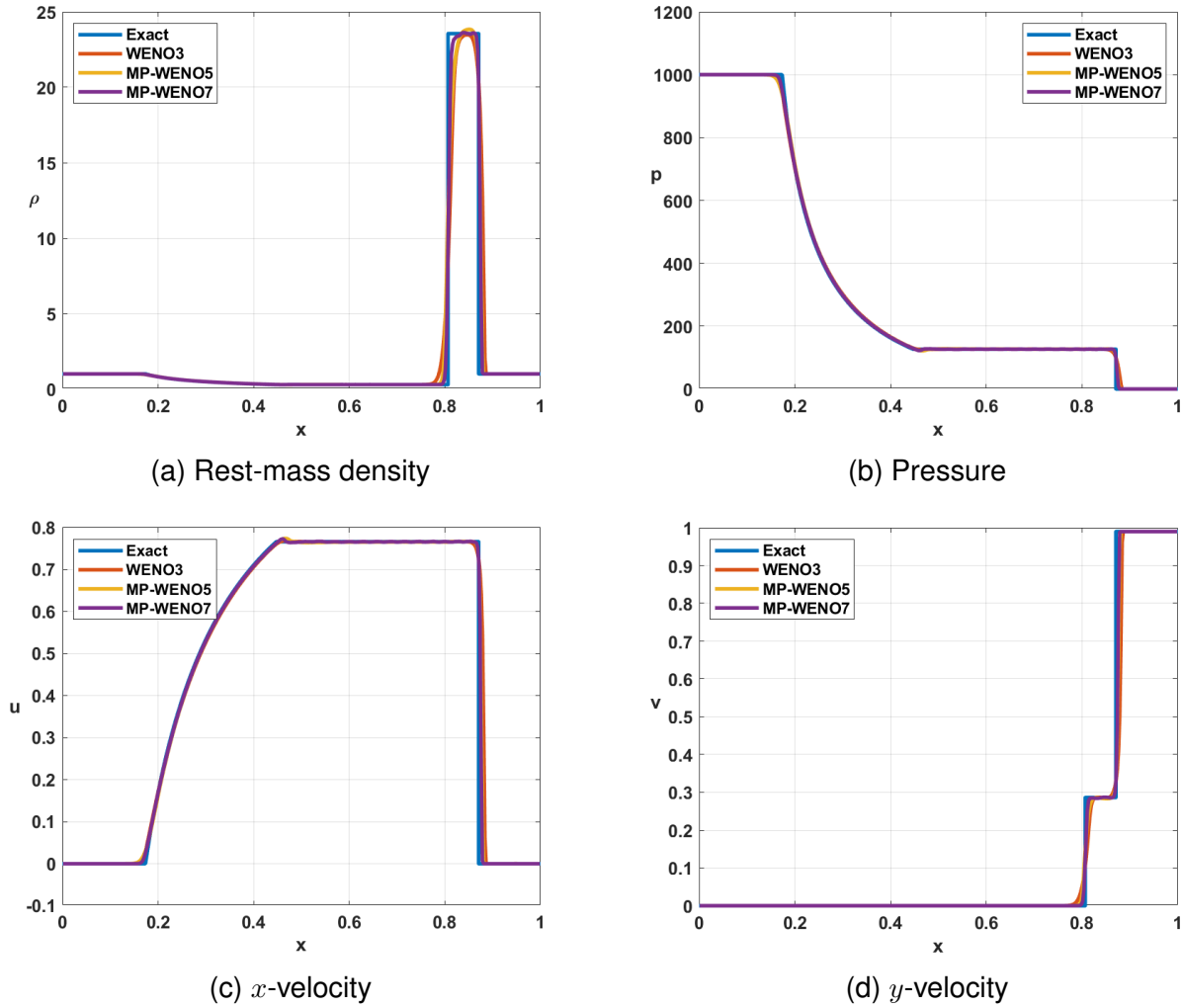


Figure 4.5: Numerical solution profiles of the Riemann problem 5 from initial conditions given in Equation (4.10) at $t = 0.4$ using a grid resolution of $NX = 400$ and $CFL = 0.5$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used.

Riemann Problem 6: Sinusoidal Rest-Mass Density Perturbation

The sixth Riemann problem, similar to Riemann problem 3, consists of initial conditions provided by [Del Zanna & Bucciantini \(2002\)](#) as

$$(\rho, u, v, p) = \begin{cases} (5, 0, 0, 50) & \text{if } x < 0, \\ (2 + 0.3 \sin 50x, 0, 0, 5) & \text{if } x > 0, \end{cases} \quad (4.11)$$

with $\Gamma = 5/3$. [Dolezal & Wong \(1995\)](#) proposed the application of a sinusoidal perturbation to the rest-mass density field to understand how well a numerical scheme can transport a smooth flow profile across a discontinuity. An exact solution is not available for

this problem and so, following the aforementioned authors, a reference solution is used instead which adopts the WENO3 scheme using $NX = 10,000$. Dirichlet boundary conditions are applied to the left and right boundaries of the computational domain. Figure 4.6 reports the rest-mass density (Figure 4.6a), pressure (Figure 4.6b), x -velocity (Figure 4.6c) and Lorentz factor (Figure 4.6d) profiles using $NX = 400$ at $t = 0.35$. Overall good agreement is found between the numerical and reference solutions with the MP-WENO7 demonstrated as the most capable for accurately transporting the smooth flow profile across the discontinuity.

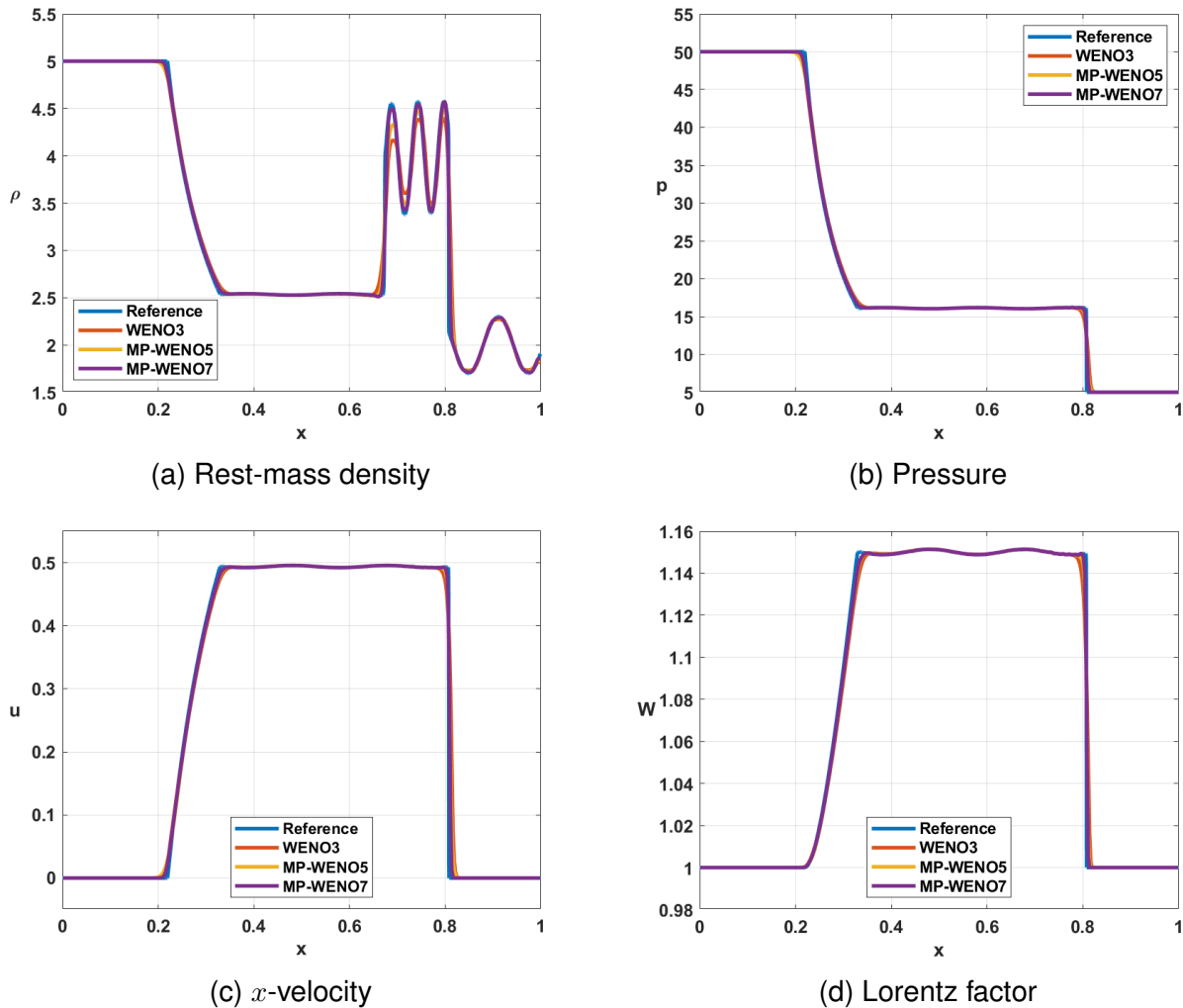


Figure 4.6: Numerical solution profiles of the Riemann problem 6 from initial conditions given in Equation (4.11) at $t = 0.35$ using a grid resolution of $NX = 400$ and $CFL = 0.5$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used.

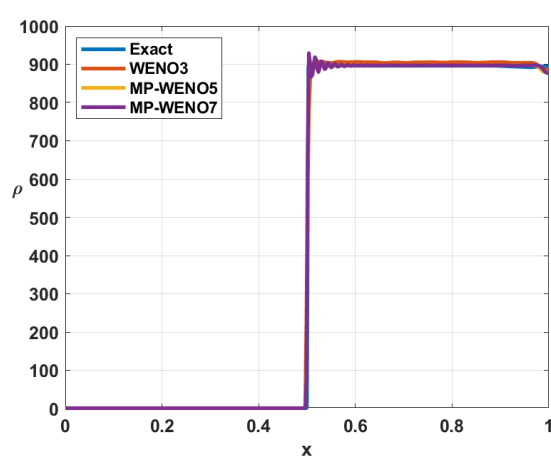
Riemann Problem 7: Planar Shock Reflection

The final and most demanding Riemann problem has initial conditions

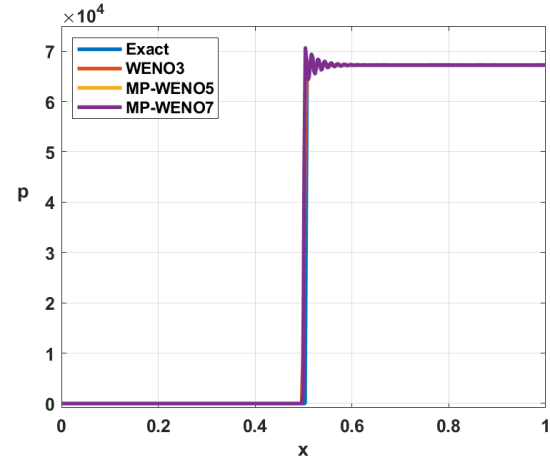
$$(\rho, u, v, p) = (1, 0.99999, 0, 10^{-2}) \quad (4.12)$$

with $\Gamma = 4/3$ and is known as the planar shock reflection (or shock heating) problem. This test sees an initially cold fluid colliding with a wall at $x = 1$ which is prescribed a reflecting boundary condition, a Dirichlet boundary condition is applied at $x = 0$. The reflected shock wave propagates leftwards and leaves the gas behind the shock at rest. Large jumps in the rest-mass density and pressure field ensue. In this test an initial x -velocity is used that corresponds to a Lorentz factor of $W \approx 223$, this makes the developed numerical framework competitive with previously presented RHD codes in terms of the maximum permissible Lorentz factor achievable. Figure 4.7 reports the rest-mass density (Figure 4.7a), pressure (Figure 4.7b) and x -velocity (Figure 4.6c) profiles using $NX = 400$ and $CFL = 0.2$ at $t = 1.5$. Overall, the numerical schemes capture the flow profile well but spurious oscillation occur only when the MP-WENO7 scheme is used. Figure 4.7d evaluates two different shock capturing strategies. The MP-WENO5 scheme and the standard WENO5 scheme without any monotonicity procedure but with a reduction to a first-order piecewise constant spatial reconstruction in the region surrounding the shock, as determined by the shock indicator of [Jameson et al. \(1981\)](#). It is clearly seen that the MP-WENO5 scheme provides more accurate results with the absence of spurious oscillations.

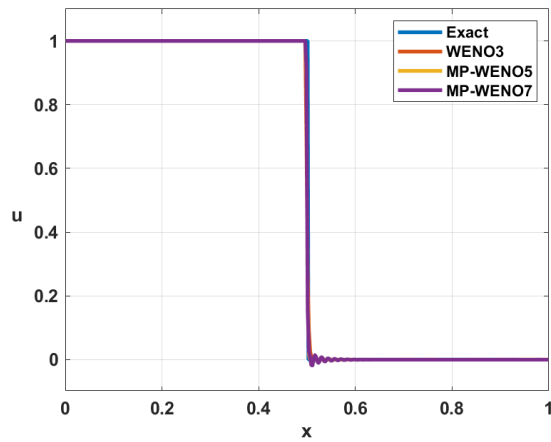
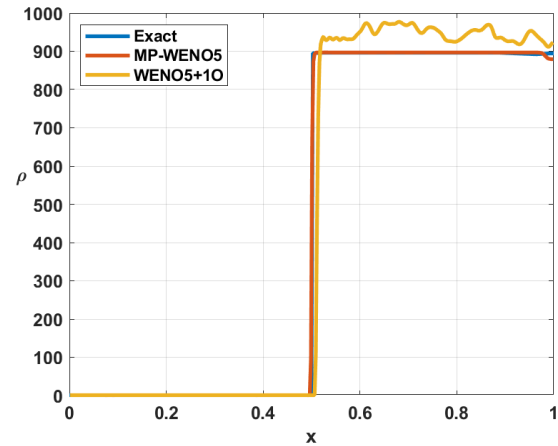
At this Lorentz factor the problem proved more challenging for the HLLC Riemann solver and so this test case was instead performed using an inflow velocity of $u = 0.9$ which was found to provided a stable solution across all spatial reconstruction schemes at $CFL = 0.5$.



(a) Rest-mass density



(b) Pressure

(c) x -velocity

(d) Rest-mass density

Figure 4.7: Numerical solution profiles of the Riemann problem 7 from initial conditions given in Equation (4.12) at $t = 1.5$ using a grid resolution of $NX = 400$ and $CFL = 0.2$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used.

NX	WENO3		MP-WENO5		MP-WENO7	
	L_1 error	L_1 order	L_1 error	L_1 order	L_1 error	L_1 order
Riemann Problem 1						
100	5.77×10^{-2}		5.05×10^{-2}		4.51×10^{-2}	
200	3.67×10^{-2}	0.65	3.39×10^{-2}	0.57	2.98×10^{-2}	0.60
400	2.26×10^{-2}	0.70	2.33×10^{-2}	0.55	2.13×10^{-2}	0.49
800	1.79×10^{-2}	0.33	1.92×10^{-2}	0.28	2.07×10^{-2}	0.04
Riemann Problem 2						
100	4.49×10^{-2}		3.92×10^{-2}		2.15×10^{-2}	
200	2.39×10^{-2}	0.91	1.93×10^{-2}	1.02	1.14×10^{-2}	0.91
400	1.28×10^{-2}	0.90	1.00×10^{-2}	0.94	5.51×10^{-3}	1.05
800	6.78×10^{-3}	0.92	5.24×10^{-3}	0.94	2.41×10^{-3}	1.19
Riemann Problem 3						
100	3.45×10^{-2}		2.70×10^{-2}		1.93×10^{-2}	
200	1.94×10^{-2}	0.83	1.18×10^{-2}	1.20	9.25×10^{-3}	1.06
400	8.39×10^{-3}	1.21	4.82×10^{-3}	1.29	3.89×10^{-3}	1.25
800	4.53×10^{-3}	0.89	3.88×10^{-3}	0.31	3.45×10^{-3}	0.17
Riemann Problem 4						
100	4.05×10^{-1}		3.67×10^{-1}		3.57×10^{-1}	
200	3.59×10^{-1}	0.17	2.59×10^{-1}	0.51	2.77×10^{-1}	0.37
400	2.84×10^{-1}	0.34	1.69×10^{-1}	0.61	1.58×10^{-2}	0.81
800	1.98×10^{-1}	0.52	1.13×10^{-1}	0.58	7.94×10^{-2}	0.99
Riemann Problem 5						
100	5.53×10^{-1}		3.48×10^{-1}		2.67×10^{-1}	
200	3.21×10^{-1}	0.78	2.06×10^{-1}	0.76	1.39×10^{-1}	0.94
400	1.82×10^{-1}	0.82	1.18×10^{-1}	0.81	7.32×10^{-1}	0.92
800	1.03×10^{-1}	0.83	6.61×10^{-2}	0.83	3.87×10^{-1}	0.92
Riemann Problem 6						
100	4.62×10^{-2}		3.73×10^{-2}		2.77×10^{-2}	
200	2.83×10^{-2}	0.71	1.79×10^{-2}	1.06	1.20×10^{-2}	1.21
400	1.40×10^{-2}	1.01	8.45×10^{-3}	1.09	4.97×10^{-3}	1.27
800	6.33×10^{-3}	1.15	4.02×10^{-3}	1.07	2.28×10^{-3}	1.12
Riemann Problem 7						
100	2.68×10^{-2}		1.59×10^{-2}		1.94×10^{-2}	
200	1.72×10^{-2}	0.64	7.15×10^{-3}	1.16	9.00×10^{-3}	1.11
400	1.29×10^{-2}	0.41	3.51×10^{-3}	1.03	4.62×10^{-3}	0.96
800	1.08×10^{-2}	0.26	1.91×10^{-3}	0.88	2.83×10^{-3}	0.71

Table 4.2: L_1 error norms and respective convergence rates for the one-dimensional Riemann problems presented throughout Chapter 4.1.2 using the HLL approximate Riemann solver. The rest-mass density was used for the error analysis.

NX	WENO3		MP-WENO5		MP-WENO7	
	L_1 error	L_1 order	L_1 error	L_1 order	L_1 error	L_1 order
Riemann Problem 1						
100	5.30×10^{-2}		5.19×10^{-2}		4.69×10^{-2}	
200	3.15×10^{-2}	0.75	3.16×10^{-2}	0.71	3.12×10^{-2}	0.59
400	2.04×10^{-2}	0.63	2.19×10^{-2}	0.53	2.24×10^{-2}	0.48
800	1.74×10^{-2}	0.23	1.87×10^{-2}	0.22	2.12×10^{-2}	0.08
Riemann Problem 2						
100	3.94×10^{-2}		3.58×10^{-2}		1.87×10^{-2}	
200	2.09×10^{-2}	0.92	1.75×10^{-2}	1.03	1.02×10^{-2}	0.87
400	1.12×10^{-2}	0.90	9.04×10^{-3}	0.96	4.91×10^{-3}	1.06
800	5.88×10^{-3}	0.92	4.62×10^{-3}	0.97	2.14×10^{-3}	1.20
Riemann Problem 3						
100	3.43×10^{-2}		2.69×10^{-2}		1.92×10^{-2}	
200	1.93×10^{-2}	0.83	1.27×10^{-2}	1.09	9.16×10^{-3}	1.07
400	8.35×10^{-3}	1.21	4.82×10^{-3}	1.40	3.87×10^{-3}	1.24
800	4.50×10^{-3}	0.89	3.90×10^{-3}	0.30	3.43×10^{-3}	0.18
Riemann Problem 4						
100	4.02×10^{-1}		3.63×10^{-1}		3.54×10^{-1}	
200	3.55×10^{-1}	0.18	2.56×10^{-1}	0.51	2.73×10^{-1}	0.37
400	2.80×10^{-1}	0.34	1.71×10^{-1}	0.59	1.58×10^{-2}	0.79
800	1.97×10^{-1}	0.50	1.16×10^{-1}	0.55	8.03×10^{-2}	0.97
Riemann Problem 5						
100	5.49×10^{-1}		3.54×10^{-1}		2.66×10^{-1}	
200	3.21×10^{-1}	0.77	2.07×10^{-1}	0.77	1.40×10^{-1}	0.93
400	1.83×10^{-1}	0.82	1.18×10^{-1}	0.81	7.47×10^{-2}	0.91
800	9.69×10^{-2}	0.91	6.68×10^{-2}	0.82	3.96×10^{-2}	0.91
Riemann Problem 6						
100	3.87×10^{-2}		3.19×10^{-2}		2.35×10^{-2}	
200	2.40×10^{-2}	0.69	1.43×10^{-2}	1.12	1.03×10^{-2}	1.19
400	1.19×10^{-2}	1.01	6.81×10^{-3}	1.07	4.38×10^{-3}	1.24
800	5.74×10^{-3}	1.05	3.34×10^{-3}	1.03	2.02×10^{-3}	1.12
Riemann Problem 7 [†]						
100	2.01×10^{-2}		1.86×10^{-2}		2.56×10^{-2}	
200	9.94×10^{-2}	1.01	9.21×10^{-3}	1.02	1.29×10^{-2}	0.99
400	4.84×10^{-3}	1.04	4.54×10^{-3}	1.02	6.83×10^{-3}	0.92
800	2.23×10^{-3}	1.12	1.97×10^{-3}	1.20	4.27×10^{-3}	0.68

Table 4.3: L_1 error norms and respective convergence rates for the one-dimensional Riemann problems presented throughout Chapter 4.1.2 using the HLLC approximate Riemann solver. The rest-mass density was used for the error analysis. [†] Inflow velocity set to $u = 0.9$.

4.2 Two-Dimensional Test Cases

A selection of two-dimensional test cases are presented that seek to validate the numerical implementation of the numerical schemes discussed thus far in multi-dimensions. Numerical test cases such as the odd–even decoupling and double-Mach reflection problem as well as astrophysical test cases are presented in Chapter 5. In this section a two-dimensional vortex evolution and two-dimensional Riemann problem are presented for both the HLL and HLLC Riemann solvers using the high-order MP-WENO7 scheme in conjunction with the third-order TVD Runge–Kutta time integration method.

A computational domain of size $[0, 1] \times [0, 1]$ is divided into four constant discontinuous states at $t = 0$ according to initial conditions specified by

$$U(x, 0) = \begin{cases} U_{NE} & \text{if } x, y > 0.5, \\ U_{NW} & \text{if } x < 0.5 < y, \\ U_{SW} & \text{if } x, y < 0.5, \\ U_{SE} & \text{if } y < 0.5 < x. \end{cases} \quad (4.13)$$

4.2.1 Vortex Sheet Evolution

The evolution of four vortex sheets is considered following the initial conditions provided by [Nunez-de la Rosa & Munz \(2016\)](#),

$$(\rho, u, v, p) = \begin{cases} (0.5, 0.5, -0.5, 5) & \text{if } x, y > 0.5, \\ (1, 0.5, 0.5, 5) & \text{if } x < 0.5 < y, \\ (3, -0.5, 0.5, 5) & \text{if } x, y < 0.5, \\ (1.5, -0.5, -0.5, 5) & \text{if } y < 0.5 < x, \end{cases} \quad (4.14)$$

with $\Gamma = 5/3$. Outflow Neumann-type boundary conditions are used at all boundaries in the computational domain. A grid resolution of 200×200 computational cells is used and the simulation is run until $t = 0.4$ using $CFL = 0.5$. Figure 4.8 presents the solutions obtained using the HLL and HLLC approximate Riemann solvers using the MP-WENO7 scheme. Figures 4.8a and 4.8b display the rest-mass density contours using the HLL and HLLC Riemann solvers, respectively. In addition, Figures 4.8c and 4.8d display the Lorentz factor contours using the HLL and HLLC Riemann solvers, respectively. Any difference between the solutions obtained with either Riemann solver is not immediately discernible by eye however good agreement is found between the achieved results and those available in the current literature. In both cases, the low density region formed at the vortex core does not report any non-physical negative densities, this was also found to be the case for all numerical schemes.

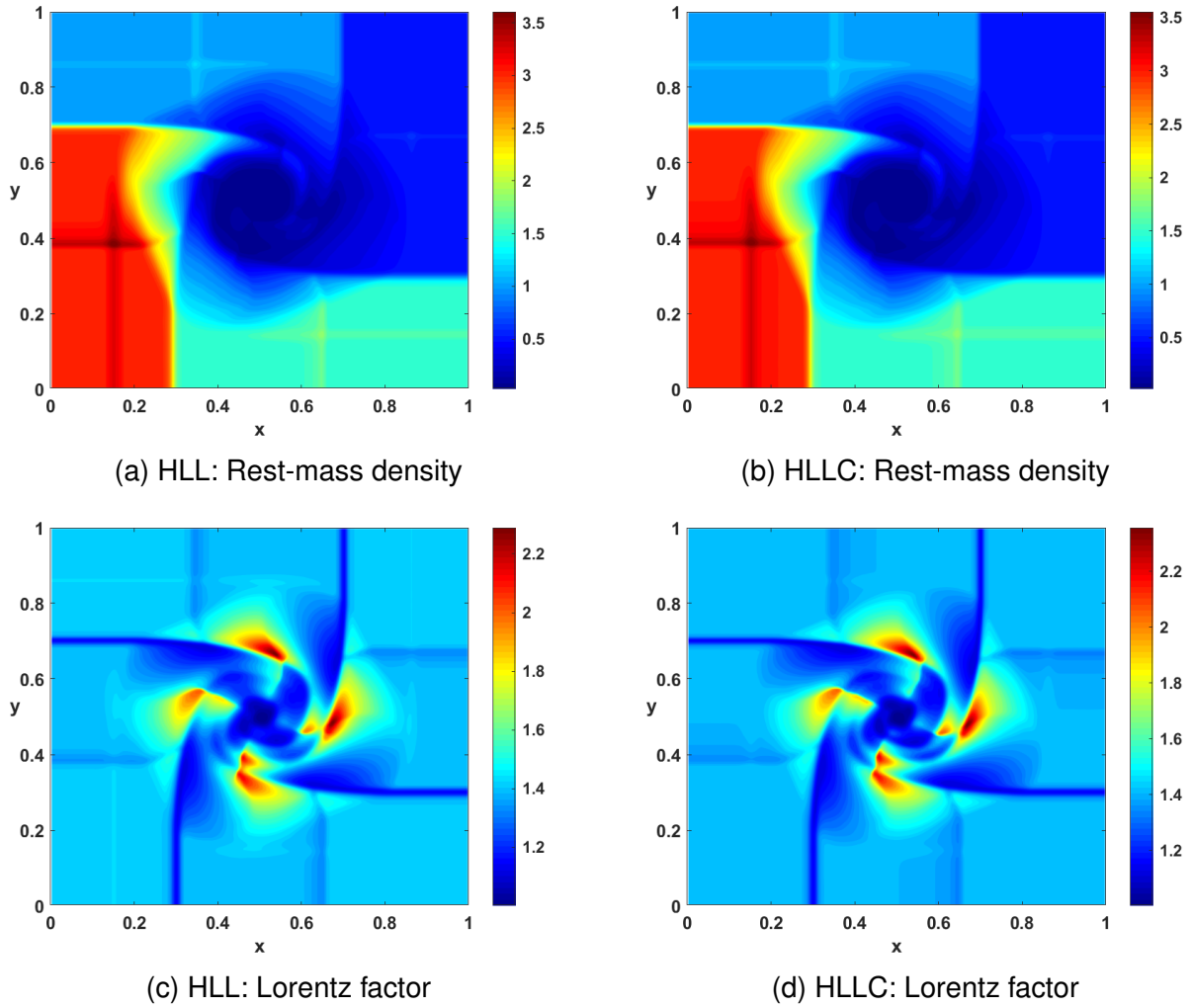


Figure 4.8: Rest-mass density contours using the HLL and HLLC approximate Riemann solvers shown in Sub-Figures 4.8a and 4.8b, respectively. Lorentz factor contours using the HLL and HLLC approximate Riemann solvers shown in Sub-Figures 4.8c and 4.8d, respectively. The solutions are reported at $t = 0.4$ using $\text{CFL} = 0.5$ at a grid resolution of 200×200 . The MP-WENO7 schemes was used throughout alongside a third-order TVD Runge–Kutta time integration method.

4.2.2 Two-Dimensional Riemann Problem

A two-dimensional Riemann problem with initial conditions following [Del Zanna & Bucciantini \(2002\)](#) is given by

$$(\rho, u, v, p) = \begin{cases} (0.5, 0.5, -0.5, 5) & \text{if } x, y > 0.5, \\ (1, 0.5, 0.5, 5) & \text{if } x < 0.5 < y, \\ (3, -0.5, 0.5, 5) & \text{if } x, y < 0.5, \\ (1.5, -0.5, -0.5, 5) & \text{if } y < 0.5 < x, \end{cases} \quad (4.15)$$

with $\Gamma = 5/3$. Outflow Neumann-type boundary conditions are used at all boundaries in the computational domain. A grid resolution of 200×200 computational cells is used and the simulation is run until $t = 0.4$ using $\text{CFL} = 0.5$. Figure 4.9 presents the solutions obtained using the HLL and HLLC approximate Riemann solvers and the MP-WENO7 scheme. Figures 4.9a and 4.9b display the rest-mass density contours using the HLL and HLLC Riemann solvers, respectively. In addition, Figures 4.9c and 4.9d display the pressure contours using the HLL and HLLC Riemann solvers, respectively. After the decay of the initial discontinuity, two contact discontinuities emerge surrounding the south west quadrant. It is clear from the rest-mass density contours in the HLLC case in Figure 4.9b that is much more effective at capturing a sharp profile for the contact discontinuity than the HLL Riemann solver in Figure 4.9a which shows a more diffusive contact discontinuity. Furthermore, the jet-like structure appearing at the centre of the domain that propagates in a south westerly direction is sharply resolved in the HLLC case whereas the HLL Riemann solver produces yet again a more diffusive flow structure. The two shock fronts in the north east quadrant posses no differing features discernible by eye between the two considered Riemann solvers. The pressure contours reported in Figures 4.9c and 4.9d for the HLL and HLLC Riemann solvers, respectively, do not again showcase any major qualitative differences however a small difference in pressure can be seen at the location of the jet-like structure, the HLLC Riemann solver reporting the lower pressure value. For both cases however good agreement is found with results previously obtained in the literature by [Del Zanna & Bucciantini \(2002\)](#) as well as [Nunez-de la Rosa & Munz \(2016\)](#).

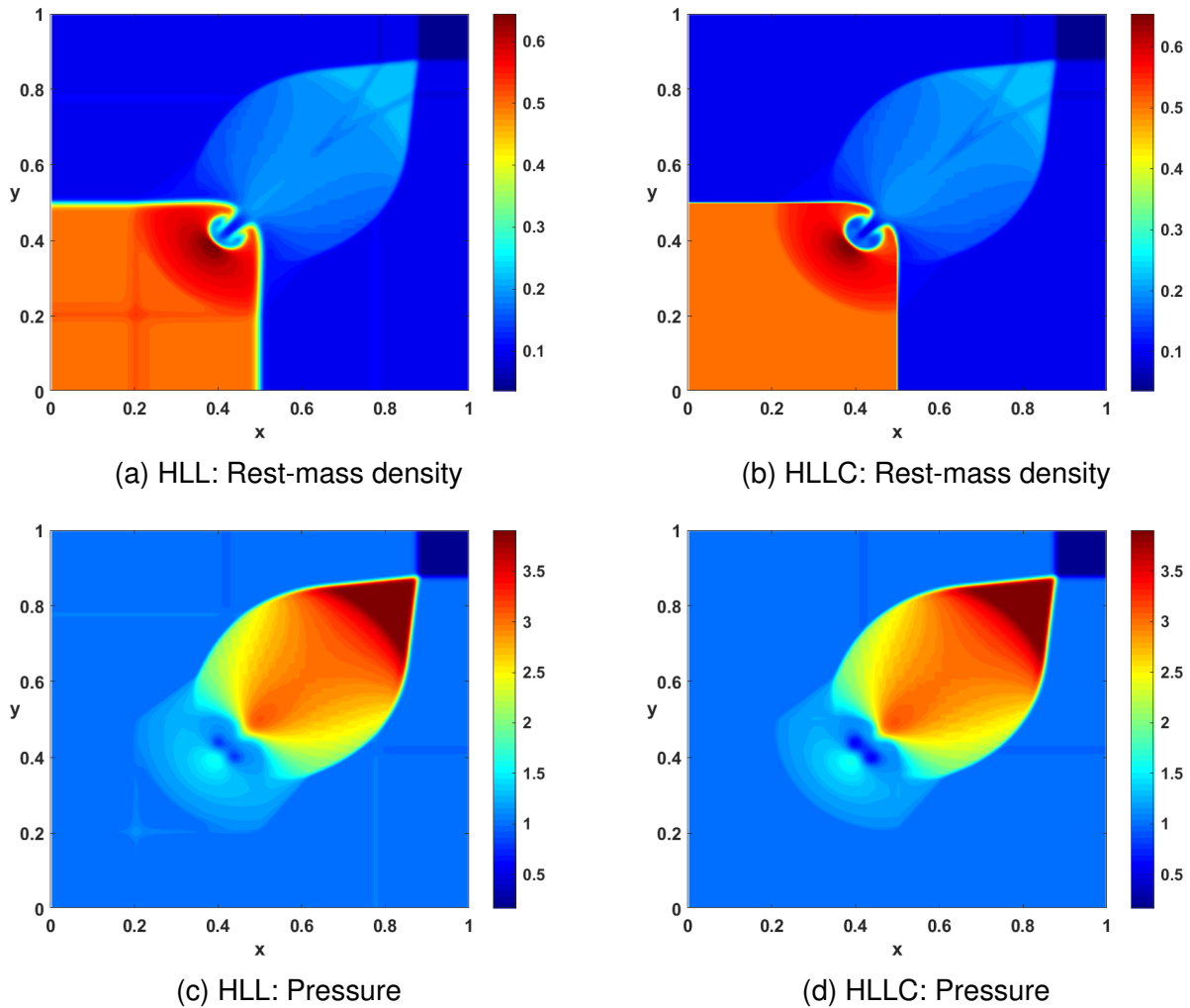


Figure 4.9: Rest-mass density contours using the HLL and HLLC approximate Riemann solvers shown in Sub-Figures 4.9a and 4.9b, respectively. Pressure field contours using the HLL and HLLC approximate Riemann solvers shown in Sub-Figures 4.9c and 4.9d, respectively. The solutions are reported at $t = 0.4$ using $CFL = 0.5$ at a grid resolution of 200×200 . The MP-WENO7 schemes was used throughout alongside a third-order TVD Runge–Kutta time integration method.

4.3 Conclusions

The developed Godunov-type HRSC numerical solver has been validated through a series of one and two-dimensional numerical test cases to determine its ability to resolve different wave patterns arising from discontinuities as well as its fidelity regarding smooth flow profiles. It was found that the previously discussed numerical methods from Chapter 3 have been correctly implemented and good performance was found across all test

cases. It was found that the HLLC Riemann solver performed similarly to the HLL Riemann solver throughout the one-dimensional test cases however for the planar shock reflection case it was unable to reach such a high Lorentz factor. Otherwise for the two-dimensional test case the HLLC Riemann solver demonstrated excellent ability to sharply resolve the contact discontinuity.

Chapter 5

A Rotated-Hybrid Riemann Solver for Relativistic Hydrodynamics

This chapter presents the development of a rotated-hybrid Riemann solver for solving relativistic hydrodynamics problems with the hybridisation of the HLL and HLLC (or Rusanov and HLLC) approximate Riemann solvers. A standalone application of the HLLC Riemann solver can produce spurious numerical artefacts when it is employed in conjunction with Godunov-type high-order methods in the presence of discontinuities. It has been found that a rotated-hybrid Riemann solver with the proposed HLL/HLLC (Rusanov/HLLC) scheme could overcome the difficulty of the spurious numerical artefacts and presents a robust solution for the Carbuncle problem. The proposed rotated-hybrid Riemann solver provides sufficient numerical dissipation to capture the behaviour of strong shock waves for relativistic hydrodynamics. Therefore, this chapter focuses on two benchmark test cases (odd–even decoupling and double-Mach reflection problems) and investigates two astrophysical phenomena, the relativistic Richtmyer–Meshkov instability and the propagation of a relativistic jet. In all presented test cases, the Carbuncle problem is shown to be eliminated by employing the proposed rotated-hybrid Riemann solver. This strategy is problem-independent, straightforward to implement and provides a consistent robust numerical solution when combined with Godunov-type high-order schemes for relativistic hydrodynamics.

5.1 Riemann Solvers for Relativistic Hydrodynamics

At present, modern-day finite volume (FV) RHD and RMHD codes can either rely on the exact or approximate solution to the local Riemann problem. The first appearance of an exact Riemann solver for RHD dates back to the analytical solution presented by [Martí & Müller \(1994\)](#). The solution was then successfully validated using test cases involving strong shocks, ultra-relativistic flows and interacting contact discontinuities, [Martí & Müller \(1996\)](#). It was then generalised for compatibility to multi-dimensional problems by [Pons et al. \(2000\)](#) such that tangential flow velocities were accommodated. A method to

determine the correct wave pattern from the initial conditions before finding the pressure across the contact discontinuity led to the exact Riemann solver of [Rezzolla & Zanotti \(2001\)](#). The resulting methodology not only proved more efficient but also simpler for numerical implementation. The Roe Riemann solver, originally developed by [Roe \(1981\)](#), finds the exact solution to the local Riemann problem using a linearised original system of equations. This approach was extended to general relativistic flows by [Eulderink & Mellema \(1995\)](#).

For larger-scale multi-dimensional problems which may also include different multi-physics considerations, approximate Riemann solvers are often preferred. One of the most popular choices of approximate Riemann solver is the HLL solver, originally formulated by [Harten \(1983\)](#) and first applied to relativistic flows by [Schneider et al. \(1993\)](#). Choosing only the smallest and largest wave speeds to determine the bounds of the intermediate state leads to a very simple implementation with good performance ([Einfeldt 1988](#)). Be that as it may, due to the fact that a contact discontinuity is not resolved, excessive smearing may be present in such cases ([Rezzolla & Zanotti 2013](#)). An even simpler alternative to the HLL Riemann solver is the approximate Riemann solver of [Rusanov \(1961\)](#) that requires only the maximum wave speed in the system. It still, however, possess the disadvantages of the HLL solver due to the inherent larger numerical dissipation. To tackle the issue of excessive smearing found using the HLL approximate Riemann solver, [Toro et al. \(1994\)](#) proposed the HLLC Riemann solver that seeks resolution of the missing contact discontinuity. This proved to be a very powerful strategy and was introduced in the context of RHD by [Mignone & Bodo \(2005\)](#) and later for RMHD by the same authors in [Mignone & Bodo \(2006\)](#).

While the HLLC solver provides clear advantages over its predecessor in terms of accuracy due to a more completely resolved wave pattern, issues surround the appearance of the so-called *Carbuncle* problem under certain flow conditions, see e.g. [Tsoutsanis \(2019\)](#). It was reported in [Wang et al. \(2008\)](#) that the relativistic Riemann HLLC solver was in fact “unsuitable” for multi-dimensional problems in RHD due to this numerical artefact. In the realm of classical physics, measures have been taken to improve the robustness of the HLLC Riemann solver using weighted average flux strategies by [Kim et al. \(2009\)](#) and by using selective wave modification by [Simon & Mandal \(2019\)](#) however the latter requires tuning and may not be applicable as an out-of-the-box solution strategy. The Carbuncle problem resembles itself as a distortion in the flow field when a shock wave is present. It is, in effect, a numerical instability, completely non-physical and detrimental to high-order numerical simulations (see e.g. [Quirk \(1994\)](#)). It has been recognised that the appearance of this phenomenon occurs when methods with insufficient numerical dissipation are applied.

As a strategy aimed at keeping the advantages provided by state-of-the-art numerical solutions to the Riemann problem, [Nishikawa & Kitamura \(2008\)](#) presented the rotated-

hybrid Riemann solver concept. This development was based on the rotated Riemann solver approach investigated by [Davis \(1984\)](#), [Levy et al. \(1993\)](#) and [Ren \(2003\)](#) who made use of only a single Riemann solver set to resolve the numerical flux chosen in an adaptively suitable direction. Based on the location of a shock wave or shear layer with respect to a cell boundary, the rotated-hybrid Riemann solver allows for contributions of two base Riemann solvers. By pre-determining whether a cell-normal is orthogonal to a shock wave or parallel to a shear layer, a Carbuncle-free base Riemann solver can be applied in that direction. They demonstrated that Carbuncle-free solutions were obtained by this strategy whilst maintaining numerical robustness.

Throughout this chapter, a direct numerical evaluation of the most popular approximate Riemann solvers is presented. Each approximate Riemann solver is presented in turn followed by a summary of the rotated-hybrid Riemann solver methodology. A rotated-hybrid Riemann solver aimed at circumventing the undesired numerical traits of the relativistic HLLC Riemann solver is then evaluated. Considerations are given to the presence of the Carbuncle problem for flows encountering strong shocks and smooth flows in which numerical dissipation properties can be directly compared.

5.2 The Rotated-Hybrid Riemann Solver

The solution methodology for the rotated-hybrid Riemann solver closely follows that of [Nishikawa & Kitamura \(2008\)](#). The motivation for including the rotated-hybrid Riemann solver philosophy are also identical, that is, to increase robustness of the numerical solution of the RHD equations for a perfect fluid, see Equation (2.16). For completeness their strategy is summarised which can be easily implemented using the aforementioned base Riemann solvers for RHD presented in Chapter 3.3. The rotated-hybrid Riemann solver requires a definition of a normal vector $\mathbf{n}$ with respect to a given cell-face. This vector is built such that it can be decomposed into two orthogonal directions, $\mathbf{n}_1$ and $\mathbf{n}_2$. By definition $\mathbf{n}_1 \cdot \mathbf{n}_2 = 0$ for any choice of $\mathbf{n}$. In addition, $|\mathbf{n}_1| = |\mathbf{n}_2| = 1$ is imposed, see Figure 5.1. The projection of the cell-face normal onto the two orthogonal vectors provides,

$$\mathbf{n} = \alpha_1 \mathbf{n}_1 + \alpha_2 \mathbf{n}_2, \quad (5.1)$$

where the scalar values α_1 and α_2 are computed from

$$\alpha_1 = \mathbf{n} \cdot \mathbf{n}_1, \quad (5.2a)$$

$$\alpha_2 = \mathbf{n} \cdot \mathbf{n}_2. \quad (5.2b)$$

The numerical flux for the rotated-hybrid concept is then formulated based on the decomposition given by,

$$\mathbf{F}(\mathbf{n}) = \alpha_1 \mathbf{F}(\mathbf{n}_1) + \alpha_2 \mathbf{F}(\mathbf{n}_2). \quad (5.3)$$

Two independent Riemann solvers can therefore be utilised for the flux computation in directions $\mathbf{n}_1$ and $\mathbf{n}_2$. The amount of *weight* placed on each flux contribution is specified

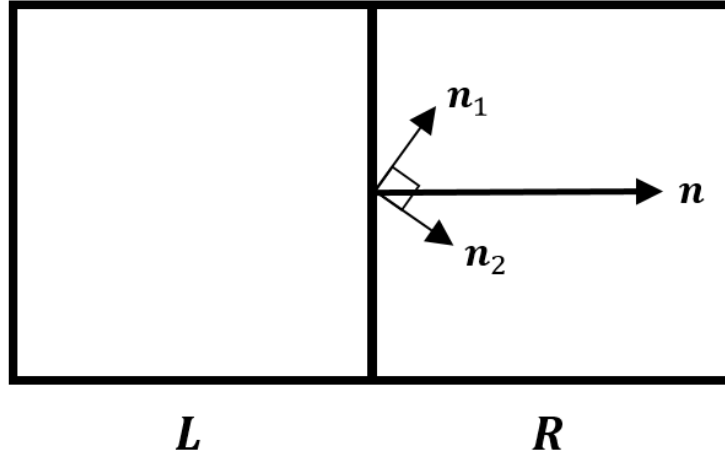


Figure 5.1: Definition of vector components for the rotated-hybrid Riemann solver.

by the coefficients α_1 and α_2 which is determined by the direction of vector $\mathbf{n}_1$ at the cell boundary. This concept allows a Riemann solver to be applied in a direction not restricted by the grid geometry, contrary to standard Riemann solver implementations on a structured Cartesian grid. Following [Levy et al. \(1993\)](#), the vector $\mathbf{n}_1$ is computed using velocity differencing such that it directs either in the direction normal to a shock or parallel to a shear layer,

$$\mathbf{n}_1 = \begin{cases} \frac{\Delta \vec{q}}{\|\Delta \vec{q}\|}, & \text{if } \|\Delta \vec{q}\| \geq \epsilon, \\ \mathbf{n}, & \text{otherwise.} \end{cases} \quad (5.4)$$

The velocity difference vector $\Delta \vec{q}$ and its modulus are defined by

$$\Delta \vec{q} = (\Delta u, \Delta v) = (u_R - u_L, v_R - v_L), \quad (5.5a)$$

$$\|\Delta \vec{q}\| = \sqrt{(\Delta u)^2 + (\Delta v)^2} \quad (5.5b)$$

and ϵ is a small number set to $\epsilon = 10^{-12}(1 + U_{\text{ref}})$ in the present work, where U_{ref} is a reference velocity magnitude. Should a shock and a shear layer be present at the cell-interface then the shock will propagate in the direction of $\mathbf{n}_1$ whilst the shear layer will travel in the direction of $\mathbf{n}_2$ ([Ren 2003](#)). Furthermore, if a strong rarefaction wave is to exist then this would also invoke the application of the dissipative Riemann solver. This choice of $\mathbf{n}_1$ is designed to ensure that sufficient numerical dissipation is applied in the vicinity of shocks to avoid the Carbuncle phenomenon and ensure that excessive numerical dissipation is not present in shear layers to ensure important flow features are not artificially smoothed out. The vector $\mathbf{n}_2$ is then consequently specified to be perpendicular to $\mathbf{n}_1$ and the scalar values α_1 and α_2 are then found from Equations in (5.2) whereby $\alpha_1 \geq 0$ and $\alpha_2 \geq 0$.

An alternative definition of $\mathbf{n}_1$ consists of defining the flux in the tangential direction to a shear layer such that the HLLC Riemann solver is also deployed if the streamwise velocity is smooth. In other words, the dissipative Riemann solver is turned off in regions

surrounding shear layers to recover only the numerical flux arising from the HLLC Riemann solver. This modification of $\mathbf{n}_1$ is made such that

$$\mathbf{n}_1 = \begin{cases} \frac{\Delta \vec{q}}{\|\Delta \vec{q}\|}, & \text{if } \|\Delta \vec{q}\| \geq \epsilon, \\ \mathbf{n}_\perp, & \text{otherwise} \end{cases} \quad (5.6)$$

where $\mathbf{n}_\perp$ is the tangential direction to the geometric face normal. It is found that, in general, this alternative formulation has only a small influence on the solution for the test cases considered. This is to be somewhat expected as it is the cross-diffusion rather than the streamwise diffusion that is known to have a greater impact on the numerical solution at shear layers (Nishikawa & Kitamura 2008). Unless otherwise state, the definition of $\mathbf{n}_1$ will always be chosen as that in Equation (5.4).

With the rotated-hybrid Riemann solver concept described it is now possible to amalgamate a choice of base Riemann solvers to complete the numerical implementation. In this work, the HLLC is chosen as primary Riemann solver and the Rusanov or HLL approximate Riemann solvers are adopted in the troublesome direction perpendicular to a shock. In general, the rotated-hybrid Riemann solver can be constructed via,

$$\mathbf{F} = \alpha_1 \mathbf{F}_{\text{HLL}}(\mathbf{n}_1) + \alpha_2 \mathbf{F}_{\text{HLLC}}(\mathbf{n}_2), \quad (5.7a)$$

$$\mathbf{F} = \alpha_1 \mathbf{F}_{\text{Rusanov}}(\mathbf{n}_1) + \alpha_2 \mathbf{F}_{\text{HLLC}}(\mathbf{n}_2), \quad (5.7b)$$

for the rotated-hybrid HLL/HLLC and Rusanov/HLLC formulations, respectively. The dissipative (HLL or Rusanov) Riemann solver acts predominantly in the direction normal to a shock or parallel to shear layers, should they exist, based on the definition of $\mathbf{n}_1$ from Equation (5.4) whilst the HLLC Riemann solver is used in all other regions of the flow. If the direction of $\mathbf{n}_1$ is chosen from Equation (5.6) then the HLLC Riemann solver will be utilised parallel to a shear layer if the streamwise velocity is smooth.

This implementation is simple, straightforward to implement and is independent of problem-specific parameters meaning that no *ad-hoc* configuration is required for different relativistic problems.

5.3 Numerical Test Cases

A selection of two numerical test cases is presented. The odd–even decoupling problem due to Quirk (1994) and the relativistic double Mach reflection formulated for the relativistic case by Zhang & MacFadyen (2006). Two astrophysical test cases follow, the relativistic Richtmyer–Meshkov instability and the propagation of a relativistic jet.

5.3.1 The Shock-Instability (Odd–Even Decoupling) Problem

Insight into whether or not a Riemann solver is susceptible to the Carbuncle instability is provided by Liou (2000) who investigated the numerical diffusivity of various numerical

flux schemes. Liou states that the mass flux can be written in the general form

$$\dot{\mathbf{m}} = \langle \dot{\mathbf{m}} \rangle - \frac{1}{2} \mathcal{D}(\mathbf{U}_L, \mathbf{U}_R) \quad (5.8)$$

in which $\langle \dot{\mathbf{m}} \rangle$ is a centrally weighted average that can be simply expressed by

$$\langle \dot{\mathbf{m}} \rangle = \frac{1}{2} \left[(\rho u W)_L + (\rho u W)_R \right] \quad (5.9)$$

in the context of the ideal RHD equations. $\mathcal{D}$ is defined as the numerical dissipation term which can be expressed as differences of primitive variables. Considering only ρ, u^i and p one would find,

$$\mathcal{D} = \mathcal{D}^{(\rho)}(\bar{\mathbf{U}}) \Delta \rho + \sum_h \mathcal{D}^{(u_h)}(\bar{\mathbf{U}}) \Delta u_h + \mathcal{D}^{(p)}(\bar{\mathbf{U}}) \Delta p \quad (5.10)$$

which enables the extraction of dissipative contributions from those primitive variables. Liou determined that a pressure contribution in the dissipative part of the mass flux (continuity) equation relating to the numerical scheme is the culprit for this numerical instability. In other words, schemes that have the property $\mathcal{D}^{(p)} = 0$ in the mass flux are shock-stable and are not prone to this numerical instability. To illustrate ideas, the dissipative pressure term $\mathcal{D}^{(p)}$ is analysed for the relativistic HLL and HLLC Riemann solvers in turn. The HLL flux can be written as a single formula as

$$\mathbf{F}^{\text{HLL}} = \frac{1}{2} \left[\mathbf{F}_L + \mathbf{F}_R \right] + \underbrace{\frac{S_R S_L}{S_R - S_L} \Delta \mathbf{U} - \frac{1}{2} \frac{S_R + S_L}{S_R - S_L} \Delta \mathbf{F}}_{\text{Dissipation Terms}}, \quad (5.11)$$

where the structure of the flux contains a contribution from the mass flux and dissipative terms. Using the expressions in Equation (2.14) and the fluxes from Equation (2.16) of the RHD governing equations in the x -direction the above formula becomes,

$$\begin{aligned} \mathbf{F}_{\text{cont.}}^{\text{HLL}} = \frac{1}{2} \left[(\rho u W)_L + (\rho u W)_R \right] + \frac{S_R S_L}{S_R - S_L} \left[(\rho W)_R - (\rho W)_L \right] \\ - \frac{1}{2} \frac{S_R + S_L}{S_R - S_L} \left[(\rho u W)_R - (\rho u W)_L \right]. \end{aligned} \quad (5.12)$$

By inspection it is possible to see from Equation (5.12) that there are no pressure difference terms in the continuity equation for the HLL scheme and therefore $\mathcal{D}^{(p)} = 0$ and the scheme is shock-stable. The HLLC Riemann solver can be re-expressed as a single formula, see [Kim et al. \(2010\)](#),

$$\mathbf{F}^{\text{HLLC}} = C_1 \mathbf{F}_L + C_2 \mathbf{F}_R + A(\mathbf{U}_L^* - \mathbf{U}_L) + B(\mathbf{U}_R^* - \mathbf{U}_R) \quad (5.13)$$

where C_1, C_2, A and B are given by [Kim et al. \(2010\)](#) as,

$$C_1 = \frac{\max(0, S^*)}{S^*}; \quad C_2 = \frac{\min(0, S^*)}{S^*},$$

$$A = S_L \left(C_1 - \frac{\max(0, S_L)}{S_L} \right); \quad B = S_R \left(C_2 - \frac{\min(0, S_R)}{S_R} \right).$$

The mass flux for the perfect RHD equations can then be analysed through use of Equation (5.13),

$$\mathbf{F}_{\text{cont.}}^{\text{HLLC}} = C_1(\rho u W)_L + C_2(\rho u W)_R + \underbrace{A(\rho u W)_L \frac{S_* - u_L}{S_L - S_*} + B(\rho u W)_R \frac{S_* - u_R}{S_R - S_*}}_{\text{Dissipation Term}}, \quad (5.15)$$

which allows for the numerical dissipation terms to be isolated,

$$\mathcal{D} = A(\rho u W)_L \frac{S_* - u_L}{S_L - S_*} + B(\rho u W)_R \frac{S_* - u_R}{S_R - S_*}. \quad (5.16)$$

Considering only the contributions of pressure in Equation (5.16) it can be seen that differencing in p is not explicitly present. In the case of classical hydrodynamics the intermediate wave speed S_* is found from an expression that contains differencing in p . However, when S_* is computed in the RHD case it is found from the solution of a quadratic Equation (3.17). Within Equation (3.17) are the HLL fluxes found from Equation (3.16) which include differences in left and right conserved variables, which means pressure differencing is manifest in the calculation of the intermediate wave speed and therefore $\mathcal{D}^{(p)} \neq 0$. This then implies by the logic presented by [Liou \(2000\)](#) that the relativistic HLLC Riemann solver of [Mignone & Bodo \(2005\)](#) is shock unstable.

To evaluate the shock instability resulting from the relativistic HLLC Riemann solver the odd–even decoupling problem is considered. First introduced by [Quirk \(1994\)](#), the odd–even decoupling problem consists of perturbing an otherwise uniform grid such that a disjointed cross-section coincides with a moving shock wave. Instead of a clean shock profile, oscillations surrounding the shock interface emerge in the density field, which eventually deteriorate the solution. This well-known shock instability has been reported within the classical literature for the Roe and HLLC Riemann solvers, see [Quirk \(1994\)](#), [Ren \(2003\)](#), [Kim et al. \(2009\)](#), [Simon & Mandal \(2019\)](#).

Since the data structure of the presented solver does not permit grid perturbations an alternative demonstration is presented to evaluate the relativistic Riemann solvers. Given that the numerical dissipation term $\mathcal{D}^{(p)}$ is used to explain the appearance of the Carbuncle phenomenon, an analogue of the original odd–even decoupling problem is presented. Quirk showed theoretically how perturbations in the pressure field, however small, can grow linearly to influence the density and velocities fields to corrupt the solution. Hence,

a test case consisting of a pressure field perturbation that disturbs the moving shock is presented. A two-dimensional channel of size $[0, 16] \times [0, 0.5]$ contains a rightward propagating shock initially positioned at $x = 0.1$. The left boundary condition is designated as an inflow with Dirichlet conditions prescribed as those behind the shock. The right boundary is prescribed as a Neumann-type outflow whilst the top and bottom boundaries are periodic. The initial primitive variable conditions ahead and behind the shock are denoted with subscript a and b , respectively, and are given by:

$$\mathbf{W}_a = (\rho, u, v, p)^T = (1.0, 0.0, 0.0, 0.1)^T, \quad (5.17a)$$

$$\mathbf{W}_b = (\rho, u, v, p)^T = (6.59, 0.748, 0.0, 2.4)^T. \quad (5.17b)$$

The initial conditions of the shock are such that the shock propagates with a velocity $v_s = 0.831$ with corresponding Lorentz factor $W_s = 1.8$ and relativistic Mach number $M_s = 4.61$. A small perturbation is applied to the pressure field $p_{i,j}$ at a position ahead of initial shock at $i = 100$ as follows

$$p_{100,j} = \begin{cases} p_j = 0.1 + \hat{p}, & \text{if } j = \text{even}, \\ p_j = 0.1, & \text{otherwise,} \end{cases} \quad (5.18)$$

where $\hat{p}$ is chosen to be 0.01. Simulations are then performed until time $t = 18$ for the HLL, HLLC, rotated-hybrid HLL/HLLC and rotated-hybrid Rusanov/HLLC Riemann solvers. Figure 5.2 reports the condition of the incident shock as it propagates through the domain at different times with the different single and rotated-hybrid Riemann solver configurations. For all cases a first-order Runge–Kutta method (also known as the Euler method) is used for time integration and a first-order spatial reconstruction scheme is used — this ensures that any numerical dissipation arises predominantly from the chosen Riemann solver and not the spatial reconstruction or time integration method. The HLL Riemann solver in Figures 5.2a–5.2c yields a clean and consistent shock profile at all times in the simulation. The HLLC Riemann solver on the other hand in Figures 5.2d–5.2f shows a disturbed solution. The incident shock experiences fluctuations in the rest-mass density field behind the shock caused by the pressure perturbation which worsens over time. When, on the other hand, the rotated-hybrid method is used, it can be seen in Figures 5.2g–5.2i that this numerical instability is avoided. Because the HLL or Rusanov Riemann solver provides the dominant flux contribution in the vicinity of the shock the inherent numerical dissipation is enough to overcome the growth of the pressure perturbations in a consistent manner.

5.3.2 The Relativistic Double-Mach Reflection Problem

To evaluate the credibility of a numerical solver designed to accurately capture shock waves, the double-Mach reflection test case must be a component in the numerical test bench. Originally devised by [Woodward & Colella \(1984\)](#) and later applied to RHD by [Zhang & MacFadyen \(2006\)](#), the test sees an incident shock wave with a 60° angle to the

x -axis propagate rightward through the domain. The shock then interacts with a reflective boundary condition imposed along the lower x -axis at $x > 1/6$. At the left boundary, an inflow boundary condition is implemented that mimics the conditions behind the shock and an outflow boundary condition is prescribed at the right domain boundary. The top boundary condition is set to either a pre- or post-shock value depending on the shock position. The computational domain is of size $[0, 4] \times [0, 1]$ and is discretised by 1600×400 computational cells; for all test cases the MP-WENO5 scheme is employed. All simulations are run for a total time of $t = 4$ and the initial conditions are specified in terms of the primitive variables, [Nunez-de la Rosa & Munz \(2016\)](#):

$$\mathbf{W}_L = (\rho, u, v, p)^T = (8.564, 0.4247 \sin 60^\circ, -0.4247 \cos 60^\circ, 0.3808)^T, \quad (5.19a)$$

$$\mathbf{W}_R = (\rho, u, v, p)^T = (1.4, 0.0, 0.0, 0.0025)^T \quad (5.19b)$$

and an adiabatic index of $\Gamma = 7/5$. The beginning of the reflecting wall, denoted x_0 , and the exact solution of the shock wave position, $h(x, t)$ travelling with velocity v_s is given by

$$h(x, t) = \sqrt{3} \left(x - x_0 \right) - 2v_s t, \quad x_0 = \frac{1}{6}, \quad v_s = 0.4984 \quad (5.20)$$

which must be considered for the correct implementation of the top boundary condition.

The results presented within Figure 5.3 showcase the rest-mass density contours for the relativistic double-Mach reflection with the standard base Riemann solvers and their rotated-hybrid modifications. Figure 5.3a provides an overview of the complete problem whereas Figures 5.3b–5.3d show only the portion of the domain in the interaction zone. The appearance of a kinked Mach stem, seen in Figure 5.3b when the HLLC Riemann is employed, is an indication of the numerical scheme's sensitivity to the Carbuncle instability. It has been reported in the classical literature that use of the Roe and HLLC Riemann solvers can result in the appearance of a kinked Mach step (see e.g. [Quirk \(1994\)](#), [Tsoutsanis \(2019\)](#)) and the same finding can now be seen when the relativistic HLLC Riemann solver of [Mignone & Bodo \(2005\)](#) is in use. Note that when the relativistic HLL Riemann solver is used then a kinked Mach stem is not present. This extends to the rotated-hybrid Riemann solvers seen in Figures 5.3c and 5.3d that do not display a kinked Mach stem due to the HLL or Rusanov Riemann solver providing the dominant flux contribution in the direction of the shock and adequate numerical dissipation in the region of the angled shock.

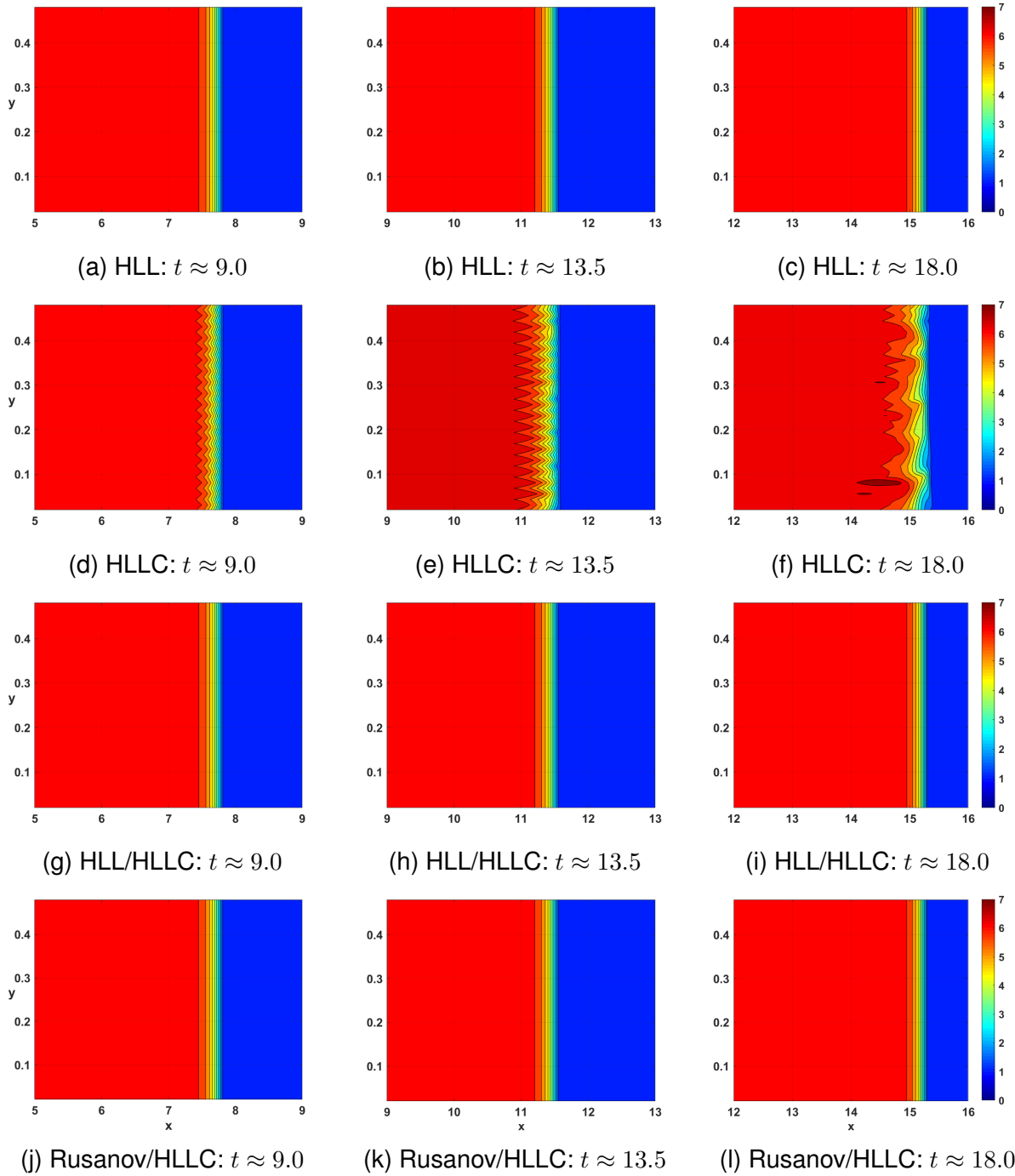
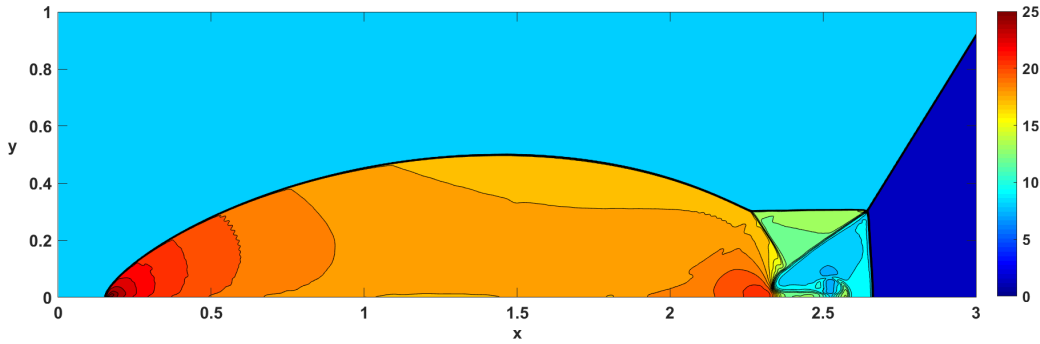
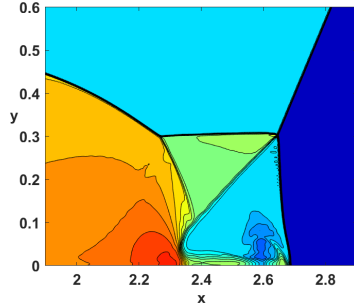


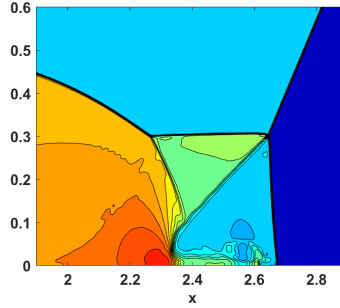
Figure 5.2: Rest-mass density contours of the odd-even decoupling using 10 contour levels a various simulation times using a first-order spatial reconstruction scheme. The discretisations consists of 800×40 cells. The Riemann solvers are: (5.2a)–(5.2c) standard HLL, (5.2d)–(5.2f) standard HLLC, (5.2g)–(5.2i) rotated-hybrid HLL/HLLC and (5.2j)–(5.2l) rotated-hybrid Rusanov/HLLC.



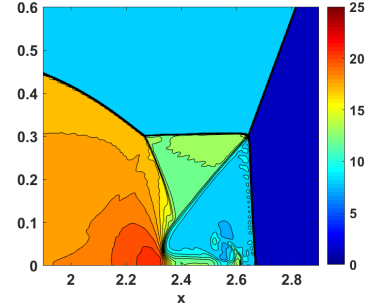
(a) Full solution using the standard HLL Riemann solver, the computational domain shown has size $[0, 3] \times [0, 1]$.



(b) HLLC



(c) HLL/HLLC



(d) Rusanov/HLLC

Figure 5.3: Rest-mass density contours of the relativistic double Mach reflection using 25 contour levels at $t = 4$. The WENO5 spatial reconstruction scheme with the WENO3 scheme used in the vicinity of shocks is deployed. The discretisation consists of 1600×400 cells. The Riemann solvers shown are the: HLL (5.3a), HLLC (5.3b), rotated-hybrid HLL/HLLC (5.3c) and the rotated-hybrid Rusanov/HLLC (5.3d).

5.4 Astrophysical Test Cases

5.4.1 Relativistic Richtmyer–Meshkov Instability

The Richtmyer–Meshkov instability (RMI) occurs when an impulsive acceleration, such as a shock wave, is applied across a perturbed interface with contrasting density. This instability was theoretically predicted by Richtmyer (1960) and later experimentally confirmed by Meshkov (1972). Unlike the Rayleigh–Taylor instability (RTI) which develops under the condition of constant acceleration directed towards the lighter fluid, the RMI can develop when such an impulsive acceleration is directed towards either the heavy or lighter fluid. As the incident shock wave initially coincides with the interface the perturbation is compressed. This is then followed by a period of linear growth as the shock wave drags the interface rightward leaving a trailing spike. A velocity shear is generated along the spike and eventually a transition to turbulence and subsequent mixing occur. While the vast majority of studies concerning the RMI are centred around its application to inertial confinement fusion (ICF), applications within high-energy astrophysical environments are also of important scientific interest. Inoue (2012) has considered implications of the RMI in the context of RMHD for pulsar wind nebulae, gamma-ray bursts and active galactic nuclei, Matsumoto & Masada (2013) studied the role of the RTI and RMI on the structure of relativistic jets and Kane et al. (1999) evaluates what role the RMI plays in supernova remnant formation.

To date, only a handful of studies have been conducted that seek understanding of the RMI in conjunction with relativistic effects. Initially Mohseni et al. (2014) studied the relativistic RMI via a particle-based solver. Most notably however was the extension of the previous work and subsequent confirmation of suppression due to Lorentz contraction as the Lorentz factor of the incident shock exceeded a critical value by Zanotti & Dumbser (2015). Following Zanotti & Dumbser (2015), simulations of the relativistic RMI are performed and the prescribed Riemann solver is varied between the HLL, HLLC and the rotated-hybrid HLL/HLLC Riemann solvers.

Figure 5.4 shows a schematic diagram representing the initial conditions of the relativistic Richtmyer–Meshkov instability. The total size of the domain is $[0, 15] \times [0, 2.5]$ where a rightward propagating shock is placed initially at $x = 1$. Ahead of the shock a perturbed interface separates a discontinuous change in rest-mass density given by

$$x = x_0 + a \sin \left(\frac{\pi}{2} + \frac{2\pi y}{\lambda} \right) \quad (5.21)$$

where $a = 0.1$ is the perturbation amplitude, $x_0 = 3$ and $\lambda = 2.5$ corresponds to the perturbation wavelength. Initial conditions left of the interface are denoted with the subscript L whereas conditions to the right are denoted with the subscript R . Similarly, initial conditions behind the shock are denoted with subscript b . The initial conditions are specified in

terms of the primitive variables and are given by

$$\mathbf{W}_b = (\rho, u, v, p)^T = (5.13, 0.633, 0.0, 1.35)^T, \quad (5.22a)$$

$$\mathbf{W}_L = (\rho, u, v, p)^T = (1.0, 0.0, 0.0, 0.1)^T, \quad (5.22b)$$

$$\mathbf{W}_R = (\rho, u, v, p)^T = (35.0, 0.0, 0.0, 0.1)^T. \quad (5.22c)$$

This problem setup leads to a positive Atwood number configuration of $A = 0.98$ in which the shock propagates rightward with relativistic Mach number 3.44 and incident Lorentz factor 1.5. Periodic boundary conditions are used for the top and bottom boundaries, the left boundary is prescribed as an inflow, set to the conditions behind the shock, whilst the right boundary is prescribed as an outflow. The computational domain is discretised with 1500×400 cells unless stated otherwise and an adiabatic index of $\Gamma = 4/3$ is used throughout.

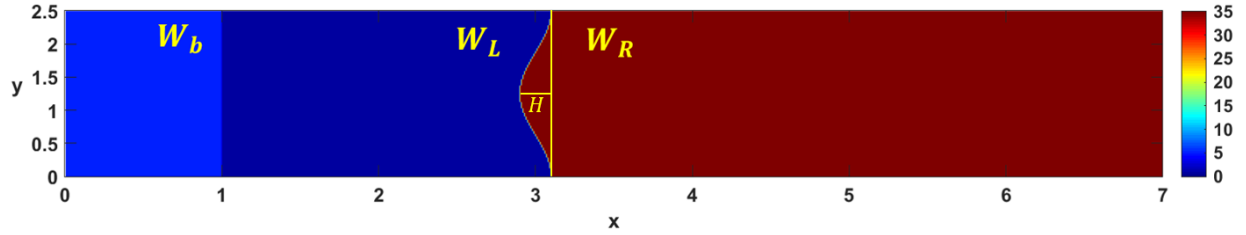


Figure 5.4: Schematic diagram for the relativistic Richtmyer–Meshkov instability initial conditions showing the rest-mass density. $\mathbf{W}_b$ denotes the initial primitive variables behind the rightward propagating shock, $\mathbf{W}_L$ and $\mathbf{W}_R$ are the initial conditions to the left and right of the initial perturbation, respectively. The profile of the perturbed interface is given by equation (5.21) and initial conditions for the three regions are specified in Equation (5.22). The height H is used to compute the perturbation amplitude evolution in Figure 5.6

Figure 5.5 provides a snapshot of the growing RMI at $t = 30$, highlighting the difference between the Rusanov, HLL, HLLC and rotated-hybrid Rusanov/HLLC and HLL/HLLC Riemann solvers. The HLLC Riemann solver in Figure 5.5c shows a clear protrusion jetting out of the growing RMI spike which is attributed to the Carbuncle phenomenon. This protrusion is absent in the HLL Riemann solver simulation seen in Figure 5.5b and for the rotated-hybrid HLL/HLLC simulation in Figure 5.5e. This demonstrates the suppression of the Carbuncle problem when the more dissipative HLL Riemann solver or the rotated-hybrid approach is used, similar conclusions can be seen from the Rusanov and rotated-hybrid Rusanov/HLLC Riemann solvers in Figures 5.5a and 5.5d, respectively. It is possible that due to the protrusion emanating from the spike head an inaccurate growth-rate of the instability may be reported. The height of the instability, subsequently used to compute the growth-rate, is defined as the extent of the mixing region with respect to the moving interface, this is denoted as H in Figure 5.4 (Zanotti & Dumbser 2015). By this

definition, the growth of all cases is shown in Figure 5.6. Note that for the HLLC Riemann solver a larger value of perturbation amplitude is found as the growth of the instability becomes non-linear, i.e. after $t \approx 15$. This is attributed to the spike protrusion as well as the oscillatory interface which are likely to corrupt the computation of the perturbation height. The inset of Figure 5.6 shows a grid study in which the HLL Riemann was used at coarse, medium and fine resolutions, 750×200 , 1500×400 and 3000×800 , respectively. Note that for the coarser resolution the perturbation height falls marginally shorter when compared to the two finer resolutions which are in good agreement with one another. From this it is concluded that the resolution used to compare the HLL, HLLC and rotated-hybrid Riemann solvers is sufficient and that the HLLC Riemann solver over-predicts the perturbation growth due to this numerical artefact. Furthermore, note that good agreement is found between the HLL and rotated-hybrid HLL/HLLC Riemann solvers, both of which are Carbuncle-free. To further support this statement, the evolution of the perturbation amplitude from the numerical simulations of [Zanotti & Dumbser \(2015\)](#) is also displayed in Figure 5.6. This data is in good agreement with the Carbuncle-free results in the linear growth stages and subsequent transition to the non-linear growth stage.

Another feature differing between each of the Riemann solver formulations apparent in Figure 5.5 is the typical “mushroom” structure of the RMI. This structure appears more diffusive in the case of the HLL, whereas in the HLLC case the structure is somewhat unorganised as smaller-scale structures appear in the solution. A similar discussion may be found in [Radice & Rezzolla \(2012\)](#) in the context of the relativistic Kelvin–Helmholtz instability where it has been shown that additional flow features may appear when schemes with less numerical dissipation are used. Similar conclusions have also been drawn by [Mignone et al. \(2009\)](#) and [Beckwith & Stone \(2011\)](#) in the magnetised version. The rotated-hybrid HLL/HLLC formulation however presents a result giving greater detail in the RMI mushroom but without the disorganised smaller-scale structures providing a contribution to the rest-mass density field. To quantify the power residing at the smaller scales and the inherent numerical viscosity of the schemes, the power spectrum density is reported for each case following [Beckwith & Stone \(2011\)](#) and [Radice & Rezzolla \(2012\)](#), which is summarised in Chapter 7. Figure 5.7 displays the spectra of the rest-mass density for the Rusanov, HLL, HLLC, rotated-hybrid Rusanov/HLLC and rotated-hybrid HLL/HLLC Riemann solver formulations by applying the Fourier transform in the y -direction, then computing the integrated power spectrum in the x -direction. The standalone Rusanov and HLL Riemann solver demonstrates the largest amount of numerical dissipation whereas the HLLC reports the least. The rotated-hybrid Rusanov/HLLC and HLL/HLLC then fall in between the HLLC and Rusanov/HLL however they are less diffusive at higher wavenumbers than the Rusanov and HLL implementations. This result supports the idea that while the rotated-hybrid approach is in general more dissipative but is still able to resolve small-scale structures where required. The alternative definition of the vector $\mathbf{n}_1$ using Equation (5.6) is also shown in Figure 5.5f along with its corresponding density power spectrum in Figure 5.7. While the interface separating the light and heavy fluid at the base of the

instability is less sharp, the schemes perform in a very similar manner both in terms of its qualitative outcome and density power spectrum.

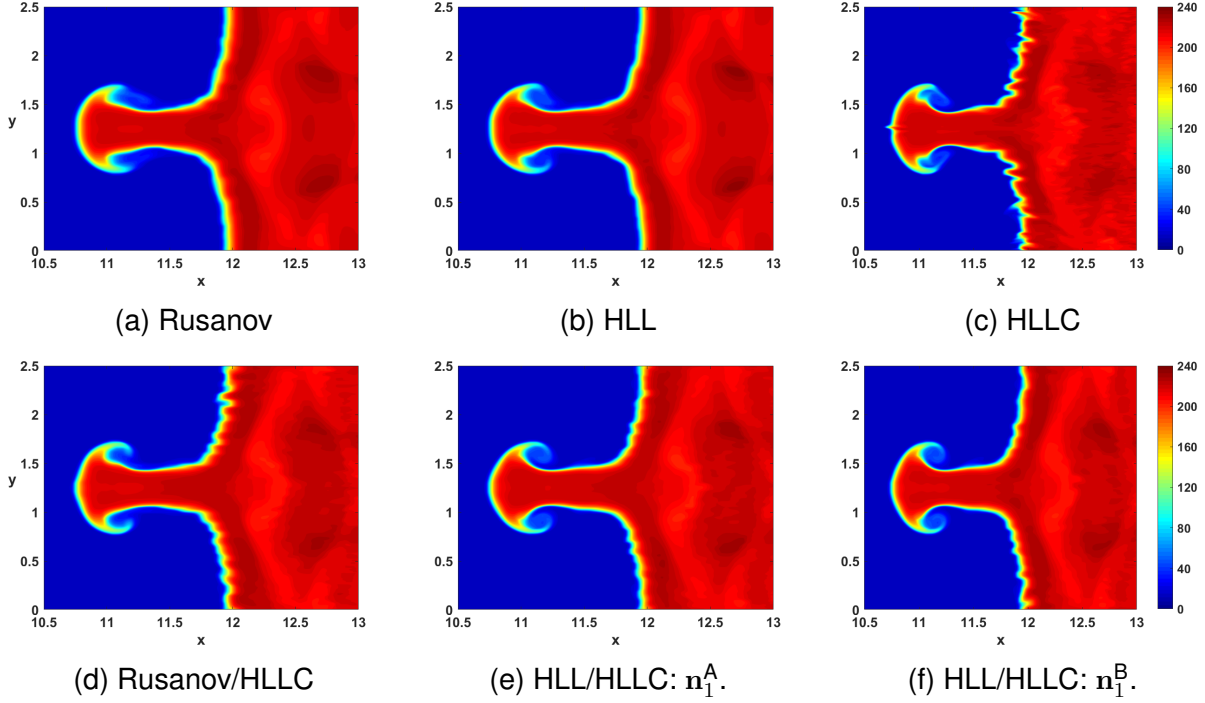


Figure 5.5: Rest-mass density contours of the relativistic Richtmyer–Meshkov instability at $t \approx 30$ using the WENO5 spatial reconstruction scheme. The computational domain shown has size $[10.5, 13] \times [0, 2.5]$. The discretisations consists of 1500×400 cells. The Riemann solver used are: (5.5a) standard Rusanov, (5.5b) standard HLL, (5.5c) standard HLLC, (5.5d) rotated-hybrid Rusanov/HLLC and (5.5e–5.5f) rotated-hybrid HLL/HLLC using the definition of $\mathbf{n}_1$ in Equation (5.4) and (5.6) stated as $\mathbf{n}_1^A$ and $\mathbf{n}_1^B$, respectively.

5.4.2 Propagation of a Relativistic Jet

As a final example an application of the rotated-hybrid Riemann solver to simulate the propagation of relativistic jets is proposed. The initial conditions closely follow those described by [Nunez-de la Rosa & Munz \(2016\)](#) and are summarised as follows. A computational domain of size $[0, 15] \times [-15, 15]$ is used where at $x = 0$ the relativistic jet is injected into the domain through a nozzle of radius r_b with a beam velocity v_b . The nozzle radius r_b at the left boundary is centred around $y = 0$ with the nozzle inlet described by $|y/r_b| < 1$ where $r_b = 0.2$. Outflow boundary conditions are applied everywhere in the domain except at the plane $x = 0$ in which a reflecting boundary condition is used surrounding the injection zone. The initially homogeneous ambient medium is given conditions

$$\mathbf{W} = (\rho, u, v, p)^T = (1.0, 0.0, 0.0, 0.001)^T \quad (5.23)$$

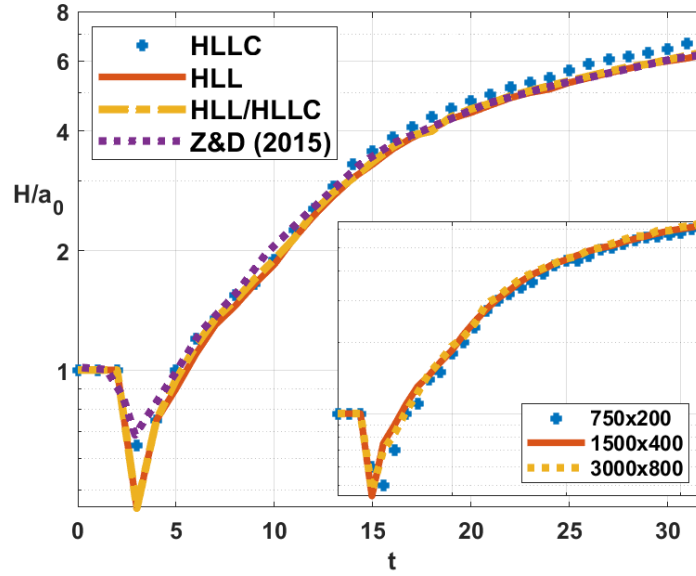


Figure 5.6: Time evolution for the amplitude in initial perturbation for the HLL, HLLC and rotated-hybrid HLL/HLLC Riemann solvers and comparison to numerical data achieved in [Zanotti & Dumbser \(2015\)](#), [Fig. 7 in Z&D (2015)]. **Inset:** time evolution for the amplitude in initial perturbation for the HLL Riemann solver at grid resolutions 750×200 , 1500×400 and 3000×800 .

and the inflow boundary condition representing the jet nozzle is given conditions

$$\mathbf{W}_b = (\rho, u, v, p)^T = (0.01, 0.999, 0.0, 0.001)^T \quad (5.24)$$

with a constant adiabatic index of $\Gamma = 5/3$. Therefore, the ratio of rest-mass density of the beam fluid and ambient medium is $\eta = 0.01$ and the ratio of pressure of the beam fluid and ambient medium is $\kappa = 1$. The simulation is run for a time of $t = 20$ using a WENO3 spatial reconstruction scheme that reduces to a first-order spatial reconstruction in the vicinity of shocks, detected via equation the shock detector of [Jameson et al. \(1981\)](#). This choice was made to guarantee a stable solution in the case of the HLLC Riemann solver to provide a comparison with the other Riemann solver strategies.

Figure 5.9 contains the logarithm of the rest-mass density contours for all schemes at a resolution of 600×600 cells in Figures 5.8a–5.8c for the HLL, HLLC and rotated-hybrid HLL/HLLC, respectively. A higher resolution of 1200×1200 for the rotated-hybrid HLL/HLLC is shown in Figure 5.9a. The HLL Riemann solver in Figure 5.8a reproduces a diffused jet structure likely caused by the diffusive nature of the Riemann solver itself and the first-order scheme applied to shocked regions. The HLLC Riemann solver in Figure 5.8b on the other hand reproduces a much narrower jet that propagates further in the same time. A protrusion is also visible at the jet head as seen in the case of the Richtmyer–Meshkov instability case in Chapter 5.4.1 which is again attributed to the

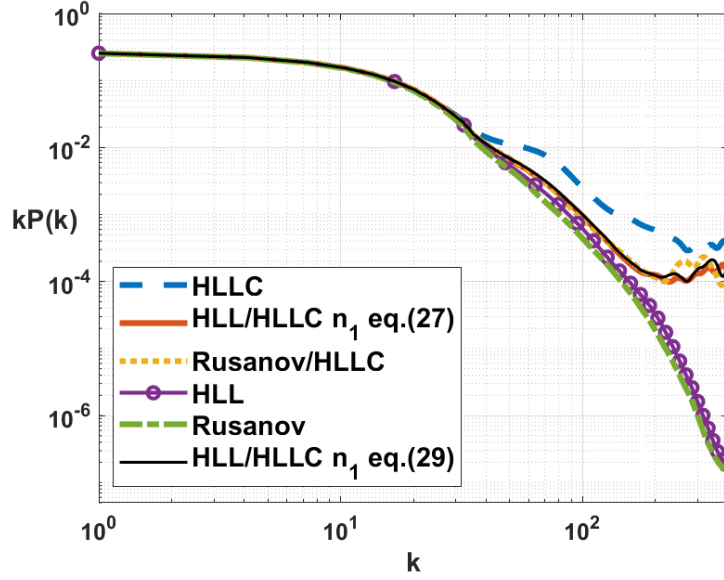


Figure 5.7: One-dimensional power spectrum of the rest-mass density for the HLL, HLLC and rotated-hybrid HLL/HLLC Riemann solvers at $t = 30$ using a grid resolution of 1500×400 cells.

Carbuncle phenomenon and deemed non-physical. When the rotated-hybrid HLL/HLLC approach is used, see Figure 5.8c, the protrusion is absent. Furthermore, a more detailed inner structure of the jet is visible when compared to the HLL Riemann solver in Figure 5.8a, likely due to the reduction in numerical dissipation provided by the HLLC Riemann solver and witnessed in the density power spectrum analysis reported for the Richtmyer–Meshkov instability in Figure 5.7. A test is also conducted to determine if such a protrusion may occur at higher resolutions. Figure 5.9a shows the rotated-hybrid HLL/HLLC simulation at twice the resolution of 1200×1200 cells in which the Carbuncle problem in the form of a protrusion is absent. The overall structure and morphology of the jet has changed as expected which is also observed by [Nunez-de la Rosa & Munz \(2016\)](#) in their resolution study. Figures 5.9a and 5.9b show the effect of changing the definition of the vector $\mathbf{n}_1$ using Equations (5.4) and (5.6), respectively, for which negligible qualitative difference is found.

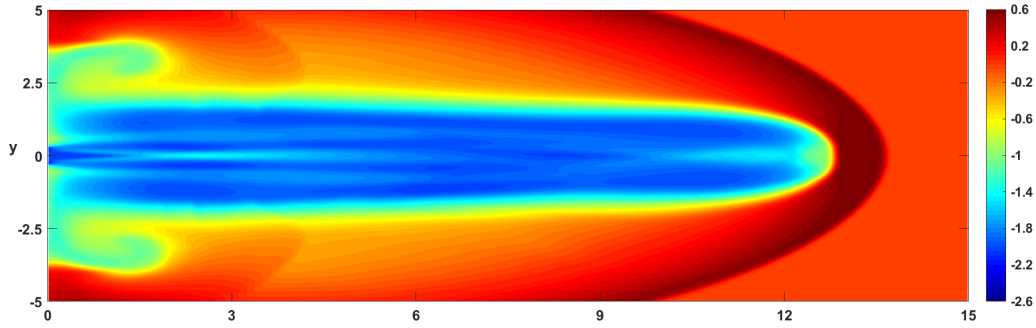
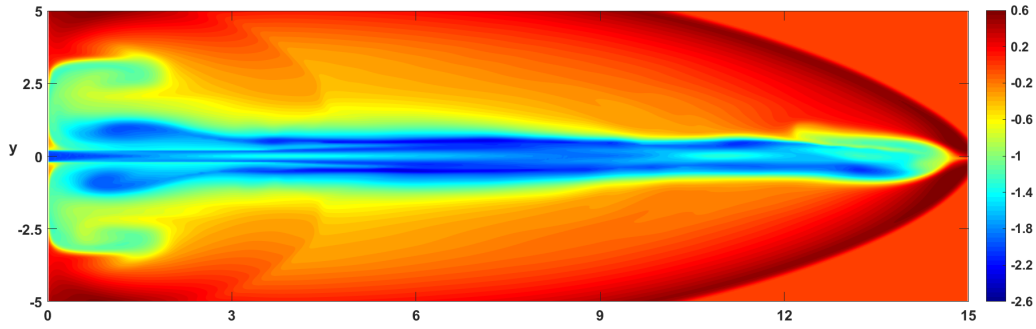
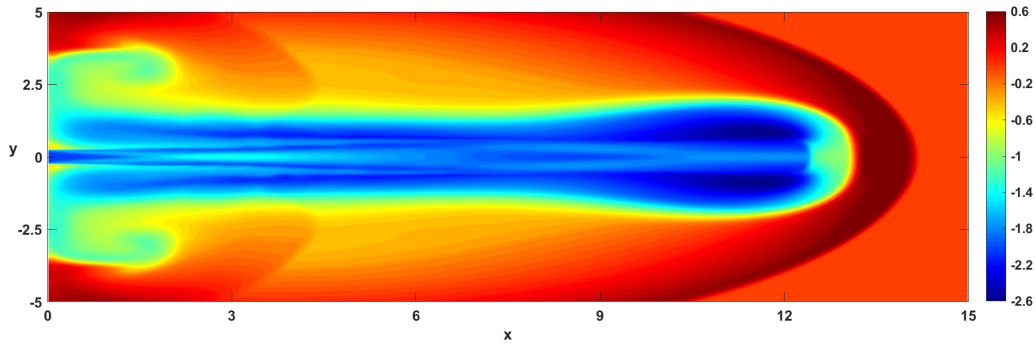
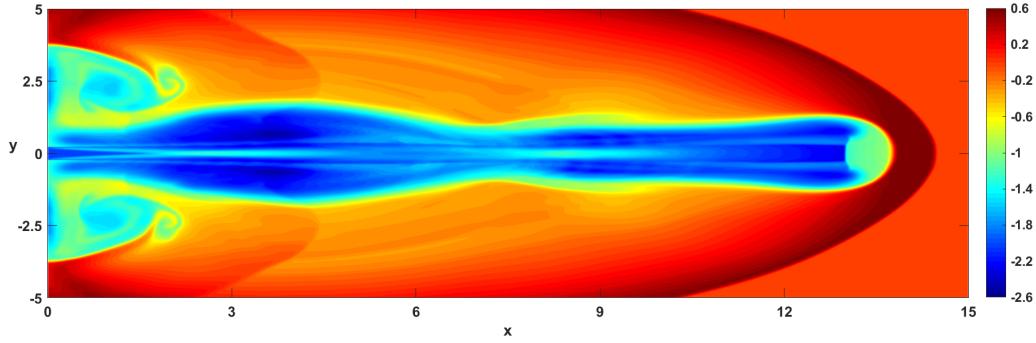
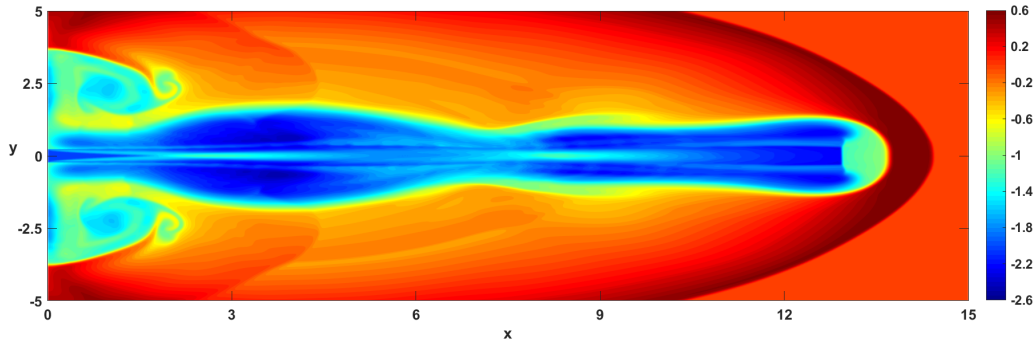
(a) Standard HLL Riemann solver using 600×600 cells.(b) Standard HLLC Riemann solver using 600×600 cells.(c) Rotated-hybrid HLL/HLLC Riemann solver using 600×600 cells.

Figure 5.8: Logarithm of the rest-mass density contours for the relativistic jet at $t = 20$. The WENO3 spatial reconstruction scheme is used, reducing to a first-order reconstruction in the vicinity of shocks. The Riemann solver used are: HLL (5.8a), HLLC (5.8b) and the rotated-hybrid HLL/HLLC (5.8c) at a grid resolution of 600×600 cells. Note that only a portion of domain is shown in the y -direction: $[0, 5]$.



(a) Rotated-hybrid HLL/HLLC Riemann solver using 1200×1200 cells using the definition of $\mathbf{n}_1$ in equation (5.4).



(b) Rotated-hybrid HLL/HLLC Riemann solver using 1200×1200 cells using the definition of $\mathbf{n}_1$ in equation (5.6).

Figure 5.9: Logarithm of the rest-mass density contours for the relativistic jet at $t = 20$. The WENO3 spatial reconstruction scheme is used, reducing to a first-order reconstruction in the vicinity of shocks. The Riemann solver used are the rotated-hybrid HLL/HLLC (5.9a–5.9b) using a finer resolution 1200×1200 cells and using the definitions for the vector $\mathbf{n}_1$ in Equations (5.4) and (5.6), respectively. Note that only a portion of domain is shown in the y -direction: $[0, 5]$.

5.5 Computational Timings

Here the normalised computational efforts required for each case previously discussed and the chosen Riemann solver formulation is summarised in Table 5.1. The odd–even decoupling problem reports a much longer CPU time when the HLLC and rotated-hybrid HLL/HLLC Riemann solver are used, approximately twice as long for the latter. The remainder of the test cases report about a 25% increase in computational effort for the rotated-hybrid HLL/HLLC strategy when compared to the standalone HLLC Riemann solver.

Case	No. of Cells	HLL	HLLC	HLL/HLLC
A	800×40	1	1.79	2.10
B	1600×400	1	1.24	1.36
C	1500×400	1	1.20	1.48
D	600×600	1	1.16	1.37

Table 5.1: Normalised CPU time for all presented simulations: A: Shock-Instability, B: Double-Mach Reflection, C: Richtmyer–Meshkov, D: Relativistic Jet.

5.6 Conclusions

The relativistic HLLC Riemann solver has been shown to exhibit numerical pathologies associated with the Carbuncle phenomenon through theoretical analysis and numerical test cases. To overcome this problem, a rotated-hybrid Riemann solver using the hybridisation of the HLL and HLLC approximate Riemann solvers (or the Rusanov and HLLC) has been proposed in the present work for solving relativistic hydrodynamics problems. In the case of the numerical shock instability problem (the odd–even decoupling problem), due to the fact that the HLLC scheme has not been used in the direction of the shock, the rotated-hybrid scheme provides enough numerical dissipation such that perturbations in the pressure field do not grow over time to distort the rest-mass density field. Therefore, the proposed rotated-hybrid scheme does not produce spurious numerical artefacts. In the case of the double-Mach reflection, the kinked Mach stem phenomenon does not appear in the numerical solution when the rotated-hybrid Riemann solver is employed but it does exist when the standalone HLLC Riemann solver is used. The proposed rotated-hybrid Riemann solver provides a robust numerical framework to consistently predict the perturbation growth of the relativistic Richtmyer–Meshkov instability. This is predominantly due to the elimination of the Carbuncle problem which causes a non-physical protrusion in the evolution of the spike tip. A resolution study using the stable HLL Riemann solver was conducted to determine a grid independent evaluation of the perturbation growth. It was then found that due to the Carbuncle problem an over-prediction in the growth was found when using the HLLC Riemann solver as the perturbation height is determined by

taking the distance from the tip of the spike to the base of the bubble. Furthermore, the rotated-hybrid approach demonstrated the ability to better resolve smaller-scale structures than the standalone Rusanov or HLL Riemann solvers which was quantified via the use of the density power spectrum. In the case of the relativistic jet propagation, it has been observed that the use of the rotated-hybrid HLL/HLLC Riemann solver eliminates the non-physical Carbuncle protrusion at the head of the jet. In addition, a more detailed inner structure of the jet is visible when compared to the HLL Riemann solver, likely due to the reduction in numerical dissipation witnessed in the in the density power spectrum analysis reported for the Richtmyer–Meshkov instability. Finally, it is shown that the choice of normal vector $\mathbf{n}_1$ used in defining the rotated-hybrid flux has negligible impact on the final solution when either choice is made. In addition to the oscillation-free behaviour of the proposed rotated-hybrid scheme, the computational time is increased compared to the standalone HLLC Riemann solver by roughly 25% for the astrophysical test cases when compared to the HLLC Riemann solver. Overall, this strategy is problem-independent, straightforward to implement and provides a robust numerical solution when combined with Godunov-type high-order schemes for relativistic hydrodynamics. It is possible that this strategy could be applied to relativistic magnetohydrodynamics (RMHD) in an analogous fashion without modification of the rotated-hybrid methodology; this possibility is left for future work.

Part III

Dissipative Relativistic Hydrodynamics

Chapter 6

Israel–Stewart Theory Implementation: Description and Validation

This chapter describes and assesses the implementation of the causal theory of physical dissipation presented by [Israel & Stewart \(1979\)](#) within a HRSC framework. The developed numerical solver makes use of a Cartesian coordinate system and the causal theory of physical dissipation presented by [Israel & Stewart \(1979\)](#). A series of one and two-dimensional test cases are performed to validate the implementation before a more detailed study exploring the role of high-order methods in the context of viscous RHD is given in Chapter 7. Validation and verification of the solution to the non-dissipative (perfect) RHD equations was given in Chapter 4.

6.1 Introduction

Early motivation for the inclusion of physical viscosity largely resides in its necessity for the study of the quark–gluon plasma (QGP) created through the collision of heavy ions. Early development of numerical solvers for study of the QGP opted for numerical schemes not within the HRSC framework. Authors such as [Baier & Romatschke \(2007\)](#) relied on a central difference method and [Schenke et al. \(2011\)](#) incorporated Israel–Stewart theory in a second-order accurate Riemann solver-free framework (see [Schenke et al. \(2010\)](#)) using the Kurganov and Tadmor (KT) central scheme, they note that negative values of the rest-mass density sometimes plague their numerical simulations. [Roy & Chaudhuri \(2012\)](#) employed the SHASTA scheme developed by [Boris & Book \(1973\)](#) which relies on the concept of flux-corrected transport and again makes no use of a Riemann solver. While these numerical solvers were able to fathom information of interest regarding the QGP the conservative and shock-capturing properties could have been improved if HRSC numerical schemes were employed. Some investigations concerned with heavy-ion collisions did in fact make use of HRSC methods but did not resolve physical viscosity in their simulations.

Due to the success of Godunov-type methods applied to the RHD equations, a numerical methodology was sought in which the advantages of this change in methodology could be applied for physical viscosity. This is not an entirely straightforward task because the HRSC methods rely on the hyperbolicity of the conservation equations such that the conserved variables can be evolved in time according to the spatial fluxes. Including a causal viscous theory such as Israel–Stewart theory disrupts this methodology because the dissipative variables must also be evolved in time according to a relaxation timescale which is typically much shorter than the hydrodynamic timescale. To circumvent this issue [Takamoto & Inutsuka \(2011\)](#) proposed to split the governing equations into a dissipative and non-dissipative part, this practice meant that the HRSC developments discussed hitherto could be exploited. Authors such as [Karpenko et al. \(2014\)](#), [Akamatsu et al. \(2014\)](#) and [Okamoto et al. \(2016\)](#) also adopted this strategy in their numerical solvers tailored to study the QGP via the use of a Milne coordinate system. [Del Zanna et al. \(2013\)](#) also make use of the HRSC finite difference framework by applying a higher-order scheme to the fluxes themselves without splitting the governing equations into a perfect/viscous part.

High-order numerical methods have found increasing use within the field of RHD since they have shown promising numerical solutions to systems of hyperbolic conservation laws, as is the case for the perfect fluid equations. For the sake of clarity, a high-order scheme in the present work is defined as those with an order of accuracy of three or above. The increasing demand placed on the computational physics community for high-fidelity numerical simulations requires consideration of the computational power and available memory necessary as a parameter of efficacy which may encumber numerical simulations. It is well-known that second-order accurate schemes, which are simple and robust, at sufficiently high resolutions are capable of reproducing solutions with an error as small as desired and that all methods are only as good as first-order accurate near discontinuities ([Rezzolla & Zanotti 2013](#)). High-order schemes are often surrounded with the idea that they are complex and therefore troublesome to implement, less robust and computationally more expensive when used at the same resolution as their lower-order counterparts. It has been shown in [Shu \(2003\)](#) that the use of high-order schemes can actually reduce computational cost as their deployment on a lower resolution grid can yield results competitive with a low-order scheme on a finer grid. For flows containing a range of difference length-scales, the use of high-order methods may then prove extremely powerful. Flows of this kind are commonplace in a turbulent environment, a research topic which is still in its infancy in the context of RHD and requires a consistent description of physical viscosity for its fundamental study, see [Mignone et al. \(2009\)](#), [Zhang et al. \(2009\)](#), [Beckwith & Stone \(2011\)](#), [Radice & Rezzolla \(2012\)](#), [Zrake & MacFadyen \(2013\)](#), [Radice & Rezzolla \(2013\)](#).

By applying the perfect/viscous splitting methodology devised by [Takamoto & Inutsuka \(2011\)](#) in conjunction with high-order numerical methods within the Godunov-type framework, an high-order dissipative relativistic hydrodynamics code is developed.

6.2 Implementation Strategy for Causal Dissipative Relativistic Hydrodynamics

In Chapter 2 the governing equations for both dissipative and non-dissipative relativistic hydrodynamics were presented, the latter set of equations reliant on the causal theory of dissipation presented by [Israel & Stewart \(1979\)](#). Recall that the complete set of governing equations representing the conservation of mass, momentum and energy in a Cartesian coordinate system using the dissipative framework are given by

$$\frac{\partial}{\partial t} \begin{pmatrix} D \\ S^i + q^0 u^i + q^i u^0 + \tau^{0i} \\ E - D + 2q^0 u^0 + \tau^{00} \end{pmatrix} + \frac{\partial}{\partial x^j} \begin{pmatrix} Du^i \\ S^i u^j + pI^{ij} + q^i u^j + q^j u^i + \tau^{ij} \\ S^j - Du^j + q^0 u^j + q^j u^0 + \tau^{0j} \end{pmatrix} = 0, \quad (6.1)$$

where the indices $i, j = \{1, 2, 3\} \equiv \{x, y, z\}$. The dissipative terms q^μ and $\tau^{\mu\nu} \equiv \Pi g^{\mu\nu} + \pi^{\mu\nu}$ contain both the time and spatial components, each of which are denoted with the superscript 0 or a Roman index to indicate their respective component. To obtain a numerical solution that still permits the use of the HRSC methods presented in Chapter 3, Equation (6.1) is split into its comprising non-dissipative (perfect) part and dissipative (viscous) part. The non-dissipative branch of the governing equations remains unchanged from Chapter 2, given by

$$\frac{\partial}{\partial t} \begin{pmatrix} D \\ S^i \\ E - D \end{pmatrix} + \frac{\partial}{\partial x^j} \begin{pmatrix} Dv^j \\ S^i v^j + pI^{ij} \\ S^i - Dv^j \end{pmatrix} = 0. \quad (6.2)$$

The dissipative arm of the governing equations then reads as

$$\frac{\partial}{\partial t} \begin{pmatrix} D \\ S^i + q^0 u^i + q^i u^0 + \tau^{0i} \\ E - D + 2q^0 u^0 + \tau^{00} \end{pmatrix} + \frac{\partial}{\partial x^j} \begin{pmatrix} 0 \\ q^i u^j + q^j u^i + \tau^{ij} \\ q^0 u^j + q^j u^0 + \tau^{0j} \end{pmatrix} = 0. \quad (6.3)$$

Extensive testing was performed for the numerical solution of Equation (6.2) using a variety of benchmark test cases in Chapter 4, in what follows, the numerical methods used herein are the same. The dissipative part in Equation (6.3) contains the dissipative fluxes and components of the dissipative terms evolved in time as well as the original conserved variables. The solution of the overall system is based on the strategy employed by [Akamatsu et al. \(2014\)](#) which was originally pioneered by [Takamoto & Inutsuka \(2011\)](#). A global set of conserved variables is defined as

$$\mathbf{U} = \mathbf{U}_{\text{perf}} + \mathbf{U}_{\text{vis}} \quad (6.4)$$

where

$$\mathbf{U}_{\text{perf}} = \begin{bmatrix} D \\ S^i \\ E - D \end{bmatrix}, \quad \mathbf{U}_{\text{vis}} = \begin{bmatrix} 0 \\ q^0 u^i + q^i u^0 + \tau^{0i} \\ 2q^0 u^0 + \tau^{00} \end{bmatrix}, \quad (6.5)$$

denote the conserved variables corresponding to the non-dissipative (perfect) part and the contributing variables from the dissipative (viscous) part. Similarly, the primitive variables are also denoted as $\mathbf{V}_{\text{perf}} = (\rho, u^i, p)^T$ and $\mathbf{V}_{\text{vis}} = (q^\mu, \pi^{\mu\nu}, \Pi)^T$ for the non-dissipative and dissipative primitive variables, respectively.

[Israel & Stewart \(1979\)](#) devised that for a causal theory of dissipation, telegrapher equations for the dissipative variables were required. In the present work these are given by

$$W \left(\frac{\partial}{\partial t} + v^j \frac{\partial}{\partial x^j} \right) \Pi = \frac{1}{\tau_\Pi} (\Pi_{\text{NS}} - \Pi), \quad (6.6a)$$

$$W \left(\frac{\partial}{\partial t} + v^j \frac{\partial}{\partial x^j} \right) \pi^{\mu\nu} = \frac{1}{\tau_\pi} (\pi_{\text{NS}}^{\mu\nu} - \pi^{\mu\nu}), \quad (6.6b)$$

$$W \left(\frac{\partial}{\partial t} + v^j \frac{\partial}{\partial x^j} \right) q^\mu = \frac{1}{\tau_q} (q_{\text{NS}}^\mu - q^\mu). \quad (6.6c)$$

which constitutes a stiff equation meaning that the characteristic time-scale τ is much shorter than the fluid time-scale τ_{fluid} . If this is the case then stringent time-step restrictions must be imposed to guarantee the numerical solution remains stable. To ensure stability the time-step size must be shorter than the characteristic time-scale, i.e. $\Delta t \lesssim \tau \ll \tau_{\text{fluid}}$. This is unfeasible for relativistic hydrodynamics in multiple dimensions as this would lead to unpractical computational costs. One possible solution to this numerical challenge is to use an implicit-explicit Runge–Kutta time integration method, see [Pareschi & Russo \(2005\)](#). Stable numerical schemes for hyperbolic conservation laws have been devised by [Svärd & Mishra \(2011\)](#) for the classical Euler equations with a stiff source term. In this work a different approach is considered. [Takamoto & Inutsuka \(2011\)](#) proposed splitting Equation (6.6) using the Strang-splitting method into an advection part and a relaxation (stiff) part. The advection equation reads

$$\left(\frac{\partial}{\partial t} + v^j \frac{\partial}{\partial x^j} \right) \Pi = 0, \quad (6.7a)$$

$$\left(\frac{\partial}{\partial t} + v^j \frac{\partial}{\partial x^j} \right) \pi^{\mu\nu} = 0, \quad (6.7b)$$

$$\left(\frac{\partial}{\partial t} + v^j \frac{\partial}{\partial x^j} \right) q^\mu = 0, \quad (6.7c)$$

which is solved using a simple first-order upwind method. In the present work a high-resolution upwind method is used as recommended by [Okamoto & Nonaka \(2017\)](#). The

remaining part of Equation (6.6) is

$$\frac{\partial}{\partial t} \pi^{\mu\nu} = \frac{\pi_{\text{NS}}^{\mu\nu} - \pi^{\mu\nu}}{\tau_\pi W}, \quad (6.8a)$$

$$\frac{\partial}{\partial t} q^\mu = \frac{q_{\text{NS}}^\mu - q^\mu}{\tau_q W}, \quad (6.8b)$$

$$\frac{\partial}{\partial t} \Pi = \frac{\Pi_{\text{NS}} - \Pi}{\tau_\Pi W}, \quad (6.8c)$$

which constitutes the stiff part equation. Following [Takamoto & Inutsuka \(2011\)](#), this can be solved in a stable manner using the Piecewise Exact Solution (PES) via

$$\pi^{\mu\nu} = (\pi_0^{\mu\nu} - \pi_{\text{NS}}^{\mu\nu}) \exp \left[-\frac{\Delta t}{\tau_\pi W} \right] + \pi_{\text{NS}}^{\mu\nu}, \quad (6.9a)$$

$$q^\mu = (q_0^\mu - q_{\text{NS}}^\mu) \exp \left[-\frac{\Delta t}{\tau_q W} \right] + q_{\text{NS}}^\mu, \quad (6.9b)$$

$$\Pi = (\Pi_0 - \Pi_{\text{NS}}) \exp \left[-\frac{\Delta t}{\tau_\Pi W} \right] + \Pi_{\text{NS}}, \quad (6.9c)$$

where the subscript 0 denotes the value at the previous time-step. The terms denoted with the subscript NS refers to the Navier–Stokes dissipation variables which are found from Equation (2.26). The time derivative in the calculation of the Navier–Stokes terms is achieved using a first-order explicit scheme and the spatial derivatives are found through use of central differencing.

The solution procedure is summarised in three stages as follows:

1. The non-dissipative (perfect) part of the conservation laws are evolved over the full time-step Δt , such that: $\mathbf{U}_{\text{perf}}(t) \rightarrow \mathbf{U}_{\text{perf}}^*(t + \Delta t)$, $\mathbf{V}_{\text{perf}}(t) \rightarrow \mathbf{V}_{\text{perf}}^*(t + \Delta t)$. The asterisk * denotes the fact that these variables at $t + \Delta t$ are not the final variables corresponding to the perfect part. Dissipative forces have yet to play any role in, for example, the evolution of the energy density, hence the asterisk variables are temporary values.

A vast weaponry of different options are available for this step, as presented in the previous chapters. This means that high-order numerical schemes and sophisticated Riemann solvers can be easily included into the numerical solution in place of the lower-order methods common in previous works relating to dissipative RHD.

2. The Israel–Stewart telegrapher equations are solved at a half time-step. First the advection Equations (6.7) are solved at a quarter time-step using the high-resolution

upwind method, i.e. $\mathbf{V}_{\text{vis}}(t) \rightarrow \mathbf{V}_{\text{vis}}(t + \Delta t/4)$. The Navier–Stokes values are computed using $\mathbf{V}_{\text{perf}}(t)$ and $\mathbf{V}_{\text{perf}}^*(t + \Delta t)$ which are then fed into the solution of the PES in Equation (6.9) alongside the values $\mathbf{V}_{\text{vis}}(t + \Delta t/4)$ from the advection part. The solution of the PES equation is then fed back into the advection equation and solved at a quarter time-step yielding $\mathbf{V}_{\text{vis}}(t + \Delta t/4) \rightarrow \mathbf{V}_{\text{vis}}(t + \Delta t/2)$.

3. The conservation Equation (6.3) is then evolved such that $\mathbf{U}_{\text{perf}}^*(t + \Delta t) + \mathbf{U}_{\text{vis}}(t) \rightarrow \mathbf{U}(t + \Delta t)$. The dissipative fluxes are calculated from the values of $\mathbf{V}_{\text{vis}}(t + \Delta t/2)$ achieved in the previous step. Primitive variable recovery follows the procedure outlined in Chapter 3.6.

The time-step for the case in which Israel–Stewart theory is invoked follows the provisional time-step proposed by [Takamoto & Inutsuka \(2011\)](#). They note that computing an exact time-step relation is difficult since the dissipative RHD equations are highly non-linear. The time-step used in this work which has demonstrated stable results thus far is given by

$$\Delta t^{(1)} = \min\left(\frac{\Delta x_a}{|\lambda_a|}\right), \quad (6.10a)$$

$$\Delta t^{(2)} = \min\left(\Delta t^{(1)}, \frac{\Delta x_a^2}{\max(\eta, \xi, \kappa)}\right), \quad (6.10b)$$

$$\Delta t = \frac{\text{CFL}}{\text{Dim}} \min\left(\Delta t^{(1)}, \Delta t^{(2)}\right). \quad (6.10c)$$

$$(6.10d)$$

6.3 One-Dimensional Dissipative Test Cases

Two one-dimensional test cases are presented in order to verify the correct implementation of Israel–Stewart theory in conjunction the Godunov-type high-order relativistic hydrodynamics numerical solver. The first is the viscous shear flow in Chapter 6.3.1 for which an exact solution is obtained. Secondly a shock-tube problem is tested in Chapter 6.3.2 with a tangential velocity component whereby all three dissipation coefficients are varied and their influence on the resulting solution profiles is studied. These tests are presented by [Takamoto & Inutsuka \(2011\)](#), a variant of the latter test is also performed by [Karpenko et al. \(2014\)](#).

6.3.1 Viscous Shear Flow

A one-dimensional viscous shear flow with corresponding exact solution reported by [Takamoto & Inutsuka \(2011\)](#) is considered. Because exact solutions in the presence

of physical dissipation are troublesome to achieve, a simplified case is examined for a very low velocity flow. [Takamoto & Inutsuka \(2011\)](#) pose that the inclusion of shear viscosity will enable the solution to relax to that of the non-relativistic case if the velocity is low enough. For this test the initial conditions are given by

$$(\rho, u, v, p) = \begin{cases} (1, 0, -0.1, 1) & \text{if } x < 0, \\ (1, 0, 0.1, 1) & \text{if } x > 0, \end{cases} \quad (6.11)$$

with adiabatic index $\Gamma = 4/3$ and a computational domain of size $x \in [-1, 1]$ is used. The y -momentum equation in the dissipative RHD governing equations then reduces to

$$\rho h \frac{\partial}{\partial t} [v W^2] + \frac{\partial}{\partial x} \tau^{xy} = 0, \quad (6.12)$$

where it is implied that the pressure and rest-mass density remain constant at all times. Because the velocity is low, it is assumed that the Lorentz factor is unity. This then gives rise to a viscous stress tensor of

$$\tau^{xy} = -\eta \frac{\partial v}{\partial x}. \quad (6.13)$$

Therefore, following [Takamoto & Inutsuka \(2011\)](#), an exact solution can be obtained as

$$u^y = u_0^y \operatorname{erf} \left[\frac{|x - x_0|}{\sqrt{4\theta t}} \right], \quad \theta = \frac{\eta}{\rho h}, \quad (6.14)$$

where $u_0^y = 0.1$ and $x_0 = 0$ are the reference variables used in this case. The numerical test evaluates the solution until time $t = 4$ at $\text{CFL} = 0.5$ and a constant shear viscosity parameter of $\eta = 0.01$. The solution is performed using a succession of grid resolutions and the L_1 error norms are computed with respect to Equation (6.14) which, alongside their corresponding convergence orders, can be found in Table 6.1. A positive convergence order is achieved yet the higher-order schemes demonstrate a lower convergence order than the WENO3 scheme. In addition, the WENO3 actually reports a lower error norm at the finest grid resolutions. Figure 6.1 provides the y -velocity solution at $t = 4$ using the MP-WENO5 scheme and HLL Riemann solver at a selection of grid resolutions as well as the exact solution from Equation (6.14). Overall the numerical solution demonstrates good agreement with the exact solution.

NX	WENO3		MP-WENO5		MP-WENO7	
	L_1 error	L_1 order	L_1 error	L_1 order	L_1 error	L_1 order
25	8.38×10^{-2}		6.35×10^{-2}		2.48×10^{-2}	
50	2.63×10^{-2}	1.67	1.61×10^{-2}	1.98	2.48×10^{-3}	3.21
100	6.92×10^{-3}	1.92	4.62×10^{-3}	1.80	1.46×10^{-3}	1.06
200	1.47×10^{-3}	2.23	1.53×10^{-3}	1.59	5.66×10^{-3}	1.29
400	3.17×10^{-4}	2.21	5.18×10^{-4}	1.56	1.24×10^{-4}	1.41
800	1.99×10^{-4}	0.67	2.64×10^{-4}	0.97	1.24×10^{-4}	0.77

Table 6.1: L_1 error norms and respective convergence rates for the one-dimensional viscous shear flow problem given by Equation (6.11) using the HLL approximate Riemann solver and a third-order TVD Runge–Kutta method. The shear viscosity coefficient is studied at $\eta = 0.01$. The y -component of the velocity was used for the error analysis at $t = 4$.

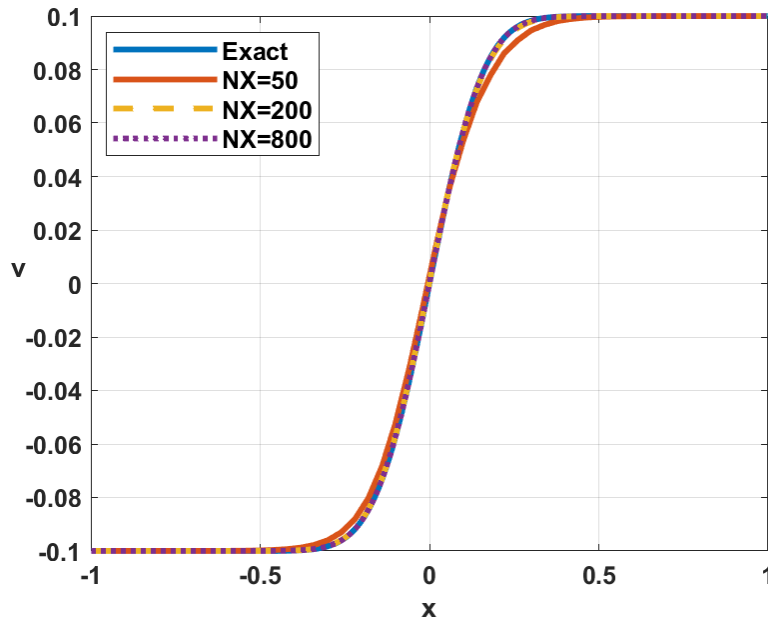


Figure 6.1: Profiles of the y -velocity component for varying grid resolutions using the MP-WENO5 scheme with a HLL Riemann solver and a third-order TVD Runge–Kutta method at $t = 4$.

6.3.2 Shock-Tube Problem

A one-dimensional shock-tube problem with a transverse velocity component is used to evaluate the performance of the implemented Israel–Stewart framework in the presence

of a discontinuity. The initial conditions for this test are the same used by [Takamoto & Inutsuka \(2011\)](#) which are given by

$$(\rho, u, v, p) = \begin{cases} (10, 0, 0.2, 10) & \text{if } x < 0, \\ (1, 0, -0.2, 1) & \text{if } x > 0, \end{cases} \quad (6.15)$$

with $\Gamma = 5/3$. The ensuing wave pattern is comprised of a left-going rarefaction wave and a right-going shock wave separated by a contact discontinuity. The influence of the physical dissipation parameters η , ξ and κ are of particular interest. In Chapter 4 it was clear that the lower-order schemes resolved a more diffusive wave profile, especially in the regions surrounding the contact discontinuity. When the grid resolution was increased the reported error norms with respect to the analytical solution also decreased. Both the numerical scheme and the grid resolution influence the amount of numerical dissipation present, hence smearing of the discontinuity can be attributed to numerical dissipation. In reality however, the profile depends on the physical dissipation parameters η , ξ and κ which is assessed in conjunction with the high-order numerical schemes.

Numerical solutions to shock-tube are presented in Figures 6.2 and 6.3. Figure 6.2a presents the y -component of the velocity using $\eta = 10^{-15}, 0.01$ and 0.04 with the MP-WENO5 scheme at $NX = 200$. It is clear that by increasing the shear viscosity coefficient smearing of the transverse velocity profile is experienced, across both the contact discontinuity and the shock front. Figure 6.2b also presents the y -velocity profile but with $\eta = 0.01$, $NX = 100$ and varying WENO reconstruction schemes. The result shows that even when a finite η is used, deviations in the numerical profiles may still arise due to numerical dissipation, in this case from the numerical scheme itself. Note that the MP-WENO7 scheme provides the “sharpest” reproduction of the y -velocity profile and is known to be the scheme with the least numerical dissipation. Figure 6.3a shows the x -velocity profile with a varying bulk viscosity coefficient set to $\xi = 10^{-15}, 0.01$ and 0.04 . Note that the increase in bulk viscosity influences the shock front and ends of the rarefaction wave, thus demonstrating how the fluid expansion is altered. Figure 6.3b demonstrates how the temperature profile defined by $T \equiv p/\rho$ changes when the heat conduction coefficient varies using $\kappa = 10^{-15}, 0.01$ and 0.04 . It is clear that the temperature profile at the shock front remains unchanged whereas across the contact discontinuity the temperature profiles smooths out as the dissipation coefficient is increased. Overall it is found that results agree well with those obtained by [Takamoto & Inutsuka \(2011\)](#) who initially presented this test case.

Tables 6.2–6.4 report the L_1 error norms alongside their respective convergence orders for different configurations of the physical dissipation parameters. Because an analytical solution does not exist for the dissipative case, the L_1 error norm is computed with respect to the numerical solution at double the grid resolution. Hence in practice, the solution was computed at a grid resolution of $NX = 50, 100, 200, 400$ and 800 , the latter not reported in the tables as this was the highest resolution used. The process is often called “self-

convergence” and makes the assumption that a finer grid resolution provides a better numerical solution. This assumption holds true in the cases presented since a positive convergence order is always achieved.

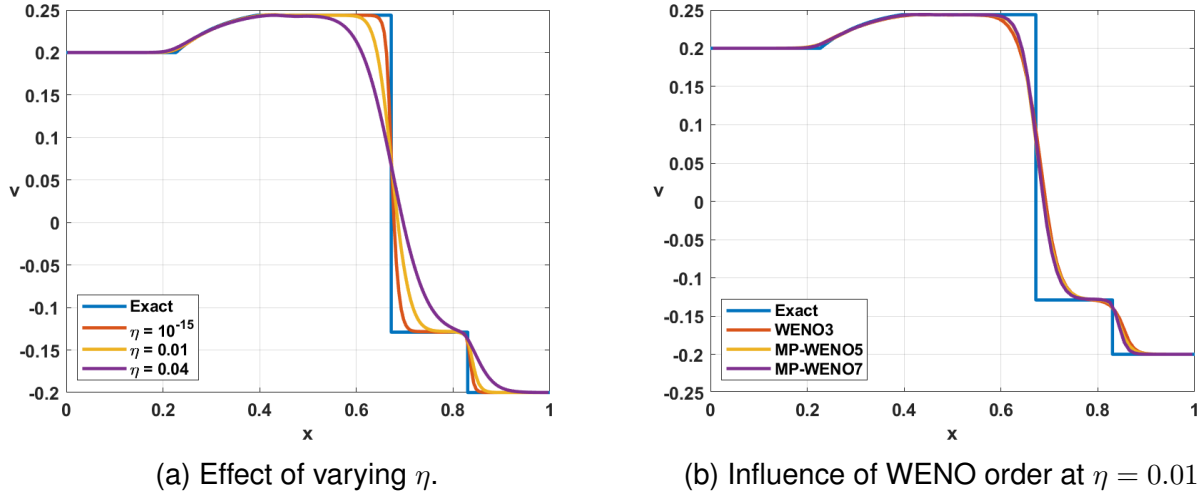


Figure 6.2: Profiles of the y -velocity component for the Riemann problem with initial conditions given in Equation (6.15) at $t = 0.4$ using $CFL = 0.5$. A third-order TVD Runge–Kutta method and HLL approximate Riemann solver are used. Figure 6.2a shows influence of using $\eta = 10^{-15}$, 0.01 and 0.04 with the MP-WENO5 scheme at $NX = 200$. Figure 6.2b shows the profile at $\eta = 0.01$ using the WENO3, MP-WENO5 and MP-WENO7 schemes at $NX = 100$.

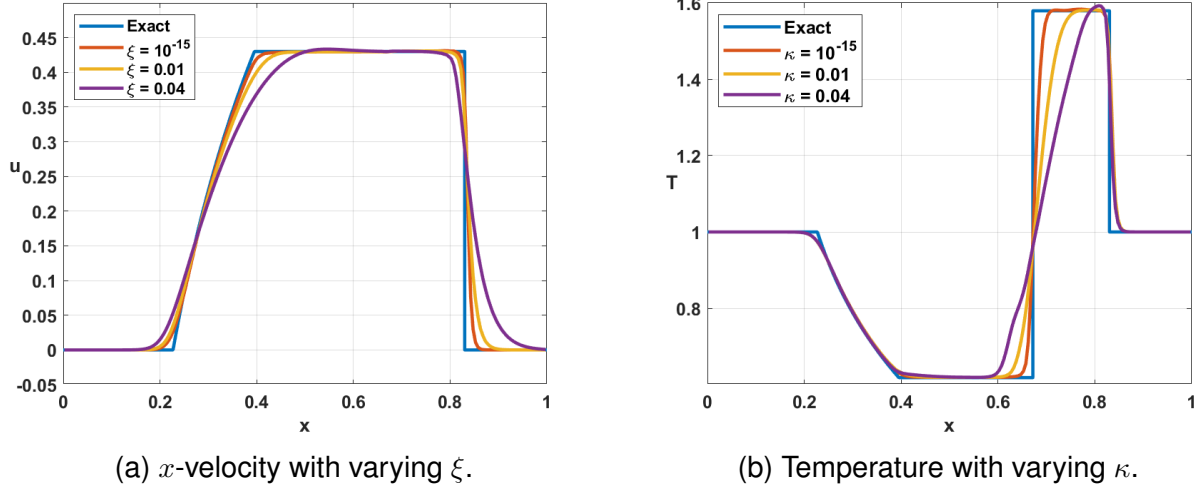


Figure 6.3: Profiles of the x -velocity component (Figure 4.5c) for $\xi = 10^{-15}$, 0.01 and 0.04. Temperature profiles (figure 4.5d) using $\kappa = 10^{-15}$, 0.01 and 0.04. Results correspond to the initial conditions given in Equation (6.15) at $t = 0.4$ using $\text{CFL} = 0.5$ at $\text{NX} = 200$. A third-order TVD Runge–Kutta method, MP-WENO5 scheme and HLL approximate Riemann solver are used.

NX	WENO3		MP-WENO5		MP-WENO7	
	L_1 error	L_1 order	L_1 error	L_1 order	L_1 error	L_1 order
$\eta, \xi, \kappa = 10^{-15}$						
50	3.19×10^{-2}		2.78×10^{-2}		2.05×10^{-2}	
100	1.74×10^{-2}	0.88	1.42×10^{-2}	0.97	1.29×10^{-2}	0.66
200	1.10×10^{-2}	0.66	8.36×10^{-3}	0.76	6.27×10^{-3}	1.04
400	6.55×10^{-3}	0.74	4.80×10^{-3}	0.80	3.44×10^{-3}	0.87
$\eta = 0.01$						
50	2.44×10^{-2}		1.84×10^{-2}		1.16×10^{-2}	
100	1.32×10^{-2}	0.88	8.13×10^{-3}	1.18	6.39×10^{-3}	0.86
200	5.05×10^{-3}	1.39	2.85×10^{-3}	1.51	2.12×10^{-3}	1.59
400	1.43×10^{-3}	1.82	9.57×10^{-4}	1.58	4.86×10^{-4}	2.13
$\eta = 0.04$						
50	1.67×10^{-2}		1.06×10^{-2}		8.35×10^{-3}	
100	6.37×10^{-3}	1.39	3.58×10^{-3}	1.57	2.61×10^{-3}	1.68
200	1.75×10^{-3}	1.86	9.42×10^{-4}	1.92	5.93×10^{-4}	2.14
400	5.08×10^{-4}	1.79	3.28×10^{-4}	1.52	2.58×10^{-4}	1.20

Table 6.2: L_1 error norms and respective convergence rates for the one-dimensional Riemann problem given by Equation (6.15) using the HLL approximate Riemann solver. The parameters η , ξ and κ are firstly tested at 10^{-15} and later the shear viscosity coefficient is studied at $\eta = 0.01, 0.04$. The y -component of the velocity was used for the error analysis.

NX	WENO3		MP-WENO5		MP-WENO7	
	L_1 error	L_1 order	L_1 error	L_1 order	L_1 error	L_1 order
$\xi = 0.01$						
50	3.71×10^{-2}		3.17×10^{-2}		2.48×10^{-2}	
100	2.43×10^{-2}	0.61	1.83×10^{-2}	0.79	1.46×10^{-2}	0.77
200	1.14×10^{-2}	1.09	8.42×10^{-3}	1.12	5.66×10^{-3}	1.37
400	4.15×10^{-3}	1.46	3.12×10^{-3}	1.43	1.24×10^{-3}	2.19
$\xi = 0.04$						
50	3.14×10^{-2}		2.57×10^{-2}		1.90×10^{-2}	
100	1.41×10^{-2}	1.16	1.03×10^{-2}	1.32	6.50×10^{-3}	1.55
200	4.89×10^{-3}	1.53	3.45×10^{-3}	1.57	1.82×10^{-3}	1.84
400	1.85×10^{-3}	1.40	1.18×10^{-3}	1.55	6.17×10^{-4}	1.56

Table 6.3: L_1 error norms and respective convergence rates for the one-dimensional Riemann problem given by Equation (6.15) using the HLL approximate Riemann solver. The bulk viscosity coefficient is studied at $\xi = 0.01, 0.04$. The x -component of the velocity was used for the error analysis.

NX	WENO3		MP-WENO5		MP-WENO7	
	L_1 error	L_1 order	L_1 error	L_1 order	L_1 error	L_1 order
$\kappa = 0.01$						
50	1.94×10^{-2}		1.50×10^{-2}		1.05×10^{-2}	
100	8.65×10^{-3}	1.16	8.45×10^{-3}	0.83	6.99×10^{-3}	0.58
200	4.99×10^{-3}	0.79	4.00×10^{-3}	1.08	3.01×10^{-3}	1.22
400	3.18×10^{-3}	0.65	2.50×10^{-3}	0.68	1.88×10^{-3}	0.68
$\kappa = 0.04$						
50	1.49×10^{-2}		1.39×10^{-2}		1.28×10^{-2}	
100	1.02×10^{-2}	0.55	8.42×10^{-3}	0.72	6.99×10^{-3}	0.87
200	6.27×10^{-3}	0.69	5.25×10^{-3}	0.68	4.48×10^{-3}	0.64
400	3.52×10^{-3}	0.83	2.78×10^{-3}	0.92	2.59×10^{-3}	0.79

Table 6.4: L_1 error norms and respective convergence rates for the one-dimensional Riemann problem given by Equation (6.15) using the HLL approximate Riemann solver. The heat conduction coefficient is studied at $\kappa = 0.01, 0.04$. The temperature profile was used for the error analysis where $T \equiv p/\rho$.

6.4 Viscous Vortex Sheet Evolution

The two-dimensional vortex sheet presented in Chapter 4.2 is performed with the inclusion of physical shear viscosity. The test serves to confirm the implementation of the Israel–Stewart framework in two-dimensions before the developed numerical code is used for further numerical investigations. Unlike the results presented in Chapter 4.2, the x - and y -velocity components are set to a smaller initial value, otherwise the initial conditions remain identical to that already used. Therefore, the initial conditions for the two-dimensional viscous vortex sheet are given by

$$(\rho, u, v, p) = \begin{cases} (0.5, 0.2, -0.2, 5) & \text{if } x, y > 0.5, \\ (1, 0.2, 0.2, 5) & \text{if } x < 0.5 < y, \\ (3, -0.2, 0.2, 5) & \text{if } x, y < 0.5, \\ (1.5, -0.2, -0.2, 5) & \text{if } y < 0.5 < x, \end{cases} \quad (6.16)$$

with $\Gamma = 5/3$. Outflow Neumann-type boundary conditions are used at all boundaries in the computational domain. A grid resolution of 200×200 computational cells is used and the simulation is run until $t = 2$ using $\text{CFL} = 0.5$.

Figure 6.4 presents the solutions obtained using the MP-WENO5 scheme with a HLL Riemann solver and third-order TVD Runge–Kutta method at varying values of η that contribute to the level of physical shear viscosity present. Sub-Figure 6.4a shows the evolution using $\eta = 0.01$, 6.4b using $\eta = 0.005$, 6.4c using $\eta = 0.001$ and 6.4d using $\eta = 0.0001$. It is noticeable that a larger amount of physical shear viscosity hinders the vortex roll-up; this effect diminishes as the value of η decreases. This could be attributed to the loss of energy through physical dissipation arising from shear which subsequently impedes the vortex sheet evolution. In other words, a higher η implies more fluid resistance between the differing layers of rest-mass density which leads to a less-developed central vortex. Note also that no negative values of the rest-mass density were produced, as was also the case for the results presented in Chapter 4.2. No further evaluation of the numerical schemes used is considered, this test only serves to gain a level of trust in the implemented Israel–Stewart framework in two-dimensions. Analysis regarding the use of physical viscosity and high-order numerical methods is considered in Chapter 7.

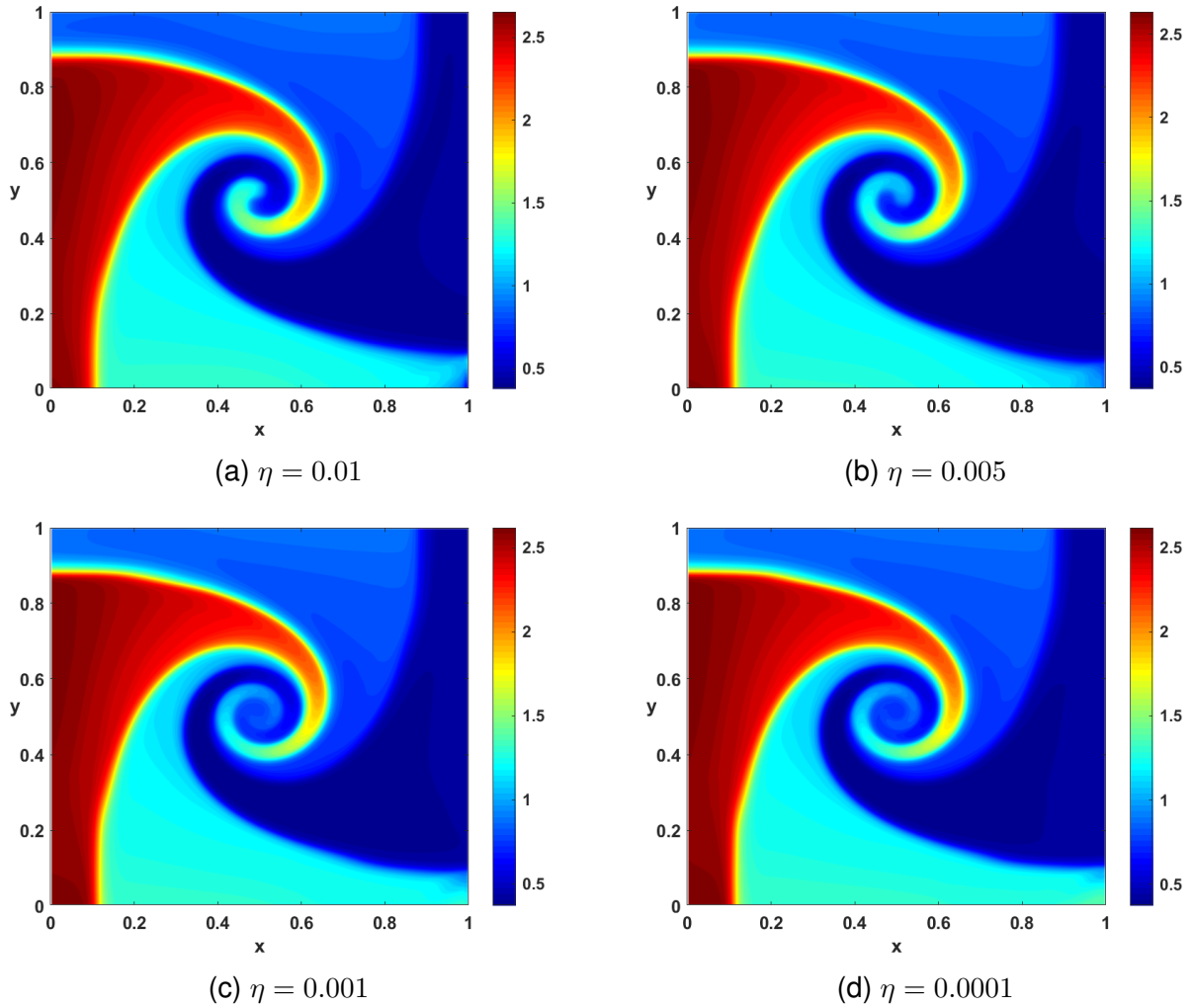


Figure 6.4: Rest-mass density contours of the vortex evolution test case in the presence of physical viscosity. The MP-WENO5 scheme, HLL Riemann solver and third-order TVD Runge–Kutta time integration method were used throughout. The solutions are reported at $t = 2$ using $\text{CFL} = 0.5$ at a grid resolution of 200×200 . Sub-Figure 6.4a shows the evolution using $\eta = 0.01$, 6.4b using $\eta = 0.005$, 6.4c using $\eta = 0.001$ and 6.4d using $\eta = 0.0001$.

6.5 Summary

In this chapter, an implementation for the causal theory of dissipation in the Eckart frame according to [Israel & Stewart \(1979\)](#) has been described. This causal theory of physical dissipation has been implemented within a finite volume HRSC framework for, to the best of the author’s knowledge, the first time. The lack of exact solutions limits the degree to which numerical solutions can be verified meaning the testing process is reliant

on results currently available in the literature for direct comparisons. Be that as it may, a one-dimensional shear flow test case for a sufficiently low velocity is used whereby it is assumed the flow becomes Newtonian as the Lorentz factor is deemed to be close to unity. This meant that an exact solution could be postulated to which the developed numerical solver presented good agreement with. A series of one-dimensional benchmark tests were performed using a Riemann problem with a transverse velocity component. In these tests the dissipation parameters η , ξ and κ corresponding to the shear viscosity, bulk viscosity and heat conductivity, respectively, were varied and their influence on the solution was assessed. This provided confidence in the current implementation as physically reasonable behaviour was encountered as the dissipation parameters were varied. Grid studies for both one-dimensional cases analysed through the concept of self-convergence, which always remained positive, gave further credence to the solver's ability. To examine the two-dimensional implementation a vortex sheet was considered. Again this test reproduced physically correct behaviour of the vortex evolution as the shear viscosity parameter was increased. All these tests contribute to an acceptable level of trust in the developed high-order HRSC dissipative RHD solver.

Chapter 7

High-Order Methods and Non-Perfect Relativistic Hydrodynamics

This Chapter presents an analysis of the dissipative properties of high-order numerical methods for relativistic hydrodynamics (RHD). A causal theory of physical dissipation is included within a finite volume High-Resolution Shock-Capturing (HRSC) framework based on the Israel–Stewart theory to assess the performance of high-order WENO (Weighted-Essentially Non-Oscillatory) schemes for simulating the relativistic Kelvin–Helmholtz instability. The decay of kinetic energy is used as a flow diagnostic to provide an estimation of the numerical dissipation of high-order schemes based on results obtained both with and without physically resolved dissipation for three orders of WENO reconstruction. In other words, an empirical relationship between the numerical dissipation and the grid resolution is proposed to quantify the dissipative properties of high-order schemes. The appearance of secondary flow features is considered within the evolution of the Kelvin–Helmholtz instability and it is determined that they are numerical artefacts based on the initial conditions. There is a potential advantage of using high-order schemes in terms of their accuracy and computational cost on coarser grid resolutions when directly compared to low-order schemes on a fine grid in the presence of physical viscosity. A sufficient amount of viscosity acts to smooth out small-scale grid perturbations for high-order schemes, thus achieving a converged solution. Therefore, it is possible to find a reasonable agreement between numerical results that employ low-order schemes using a fine grid resolution and results that employ high-order schemes at a coarser grid resolution. Overall, the present analysis gives an insight into the numerical dissipation of high-order shock-wave capturing schemes which can be relevant to computational studies of the quark–gluon plasma (QGP) in its hydrodynamics phase.

7.1 The Relativistic Kelvin–Helmholtz Instability

The Kelvin–Helmholtz instability occurs due to velocity shear within a continuous fluid or across an interface separating two different fluids. This instability is well-known to occur

within astrophysical environments as well as terrestrial engineering applications, see e.g. [Rana et al. \(2011\)](#). The shear leads to vorticity development which then grows with time and subsequently invokes turbulence. Not only is this test case used as a means to evaluate the performance of numerical methods, it is also studied in itself within the context of relativistic fluid flow. [Martí et al. \(1997\)](#) demonstrated through numerical simulations that relativistic jets were stable due to the suppression of the Kelvin–Helmholtz instability at high-enough Lorentz factors, confirming the analysis of [Ferrari et al. \(1978\)](#). [Okamoto & Nonaka \(2017\)](#) discussed the possible development of the Kelvin–Helmholtz instability during relativistic heavy-ion collisions and indicated that a numerical code with a lower numerical viscosity is required for its study and [Mourão Roque & Lugones \(2018\)](#), in a similar spirit, under conditions applicable to the early evolution of the Universe, assessed the role of the instability at the QCD epoch. Using RMHD, [Bucciantini & Del Zanna \(2006\)](#) investigated the development of the instability under conditions representative of a Pulsar Wind Nebulae environment. Analytical investigations have also been conducted by [Bodo et al. \(2004\)](#) who determined analytical expressions for the dispersion relation when an appropriate reference frame is used. Parametric studies have been performed by [Tian & Chen \(2016\)](#) using non-relativistic MHD to compare the dependencies of various physical parameters, such as viscosity, to investigate how the evolution changes in the context of astrophysical applications.

Numerical error is often associated with either numerical dispersion or numerical viscosity. The numerical dispersion contributes to phase errors into the solution while the latter acts to smooth out flow features via smearing; in this study only the latter is considered. An analogy between physical and numerical viscosity is often made as they have a similar effect on the solution in that they bear some resemblance to one another, i.e. smoothing. A certain degree of numerical viscosity is required in order to maintain a stable solution, consequences of not doing so can result in issues such as the Carbuncle problem (see Chapter 5 and [Townsend et al. \(2020\)](#)) however if the manifest numerical viscosity is too large this can also have a negative impact on the solution fidelity.

The numerical viscosity present within a numerical scheme was considered by [Beckwith & Stone \(2011\)](#) and [Radice & Rezzolla \(2012\)](#) using their simulations of the relativistic Kelvin–Helmholtz instability by employing the rest-mass density power spectrum to quantify the power residing at small-scales. Similarly for the non-relativistic case [San & Kara \(2015\)](#) also reported the energy power spectrum for the same reason as well as tracking the evolution of the kinetic energy over time, enabling quantitative insight into the dissipative properties of the numerical schemes used. [Tsoutsanis et al. \(2015\)](#) were able to isolate the energy losses arising from numerical and physical dissipation for the numerical solution of the non-relativistic Navier–Stokes equations applied to the two-dimensional vortex-pairing problem to study the contrasting behavior between numerical schemes as well as structured and unstructured grid strategies. Numerical viscosity measurements were thus not attainable *a priori* which led [Rembiasz et al. \(2017\)](#) to seek a general formula capable of estimating the numerical viscosity (and resistivity in their case) based on

knowledge of the numerical schemes and grid resolution, this fared well during the 1D and 2D simulations in their work. More recently, [Obergaullinger & Aloy \(2020\)](#) sought to determine the rates of numerical dissipation for the non-relativistic Kelvin–Helmholtz instability by analysis of the instability growth for simulations with only numerical viscosity present and those with a finite viscosity. They were able to determine a relationship in terms of the growth that led to an expression capable of estimating the numerical viscosity contained within the solutions.

This Chapter investigates the dissipative properties of three flavors of high-order WENO spatial reconstruction scheme that have witnessed increasing use within the RHD and RMHD community. The relativistic Kelvin–Helmholtz instability is used as a test case hence an investigation into the diffusive nature of the schemes in the presence of shock waves is not made. Instead, the aim is to provide an estimation of the inherent numerical viscosity and evaluate the performance of high-order numerical methods in the presence of physically resolved viscosity.

7.1.1 Problem Setup

Initial conditions for the two-dimensional relativistic Kelvin–Helmholtz instability closely follow those used by previous authors [Beckwith & Stone \(2011\)](#) and [Radice & Rezzolla \(2012\)](#) with the exception of the prescribed shear velocity, V_{shear} . A periodic computational domain of size $[-0.5, 0.5] \times [-1, 1]$ is initialised with a uniform pressure field set to $p = 1$ with a low-density strip across the centre of the domain. The shear velocity profile is then given by

$$u^x(y) = \begin{cases} V_{\text{shear}} \tanh[(y - 0.5)/a], & \text{if } y > 0, \\ -V_{\text{shear}} \tanh[(y + 0.5)/a], & \text{if } y \leq 0, \end{cases} \quad (7.1)$$

in the x -direction with $a = 0.01$ as the characteristic size of the shear layer and $V_{\text{shear}} = 0.3$. A small perturbation in the transverse velocity component is prescribed using

$$u^y(x, y) = \begin{cases} A_0 V_{\text{shear}} \sin(2\pi x) \exp \left[- (y - 0.5)^2 / \sigma \right], & \text{if } y > 0, \\ -A_0 V_{\text{shear}} \sin(2\pi x) \exp \left[- (y + 0.5)^2 / \sigma \right], & \text{if } y \leq 0, \end{cases} \quad (7.2)$$

where the perturbation amplitude is set to $A_0 = 0.1$ and $\sigma = 0.1$ is the characteristic length-scale. The initial conditions for the rest-mass density field are uniform in the x -direction and are applied according to

$$\rho(y) = \begin{cases} \rho_0 + \rho_1 \tanh[(y - 0.5)/a], & \text{if } y > 0, \\ \rho_0 - \rho_1 \tanh[(y + 0.5)/a], & \text{if } y \leq 0, \end{cases} \quad (7.3)$$

where $\rho_0 = 0.505$ and $\rho_1 = 0.495$. Finally, an ideal equation of state is used in which the adiabatic index is set to $\Gamma = 4/3$.

The relativistic Kelvin–Helmholtz instability is used to study the contrasting behaviour of the different high-order numerical schemes in conjunction with physically resolved shear viscosity. Unless otherwise stated, throughout all numerical tests hereafter the CFL number is set to 0.5 and the HLL approximate Riemann solver is used alongside the third-order TVD Runge–Kutta time integration method.

7.1.2 Flow Diagnostics

A consistent quantitative comparison between results is made using a selection of analysis techniques previously employed by various authors for similar investigations. A supplementary technique is also used to estimate the contributions arising from numerical dissipation when different numerical schemes are used on different grid and varying values of η .

In order to track the growth of the instability, the time evolution of the root-mean-square (RMS) value of the transverse velocity component is computed at regular time intervals throughout the simulation run. Following [Radice & Rezzolla \(2012\)](#), this is found from

$$v_{\text{RMS}} = \langle |v|^2 \rangle \equiv \frac{1}{V} \int_V |v|^2 dV, \quad (7.4)$$

where V is the total volume of the computational domain and v is the y -component of the fluid three-velocity, i.e. $v \equiv u^y$.

Dissipative properties of the numerical solution are monitored by consideration of two complimentary strategies. The time evolution of the total kinetic energy in the system, found through the equation ([Radice & Rezzolla 2012](#)),

$$E_k(t) = \rho W(W - 1), \quad (7.5)$$

is used to study any energy losses over time that arise from numerical and/or physical dissipation. This technique was employed by [San & Kara \(2015\)](#) to study the dissipative properties of different numerical schemes in the context of the classical Euler equations. Any loss in kinetic energy found from the use of Equation (7.5) will correspond to the total energy loss, i.e. numerical dissipation plus physical dissipation (if $\eta \neq 0$). If physical dissipation is not resolved then any dissipation comes as a result of numerical dissipation only. This result enables the direct analysis of the cumulative effect numerical and physical dissipation will have on the solution over time in a manner similar to what was achieved by [Tsoutsanis et al. \(2015\)](#) in the non-relativistic context.

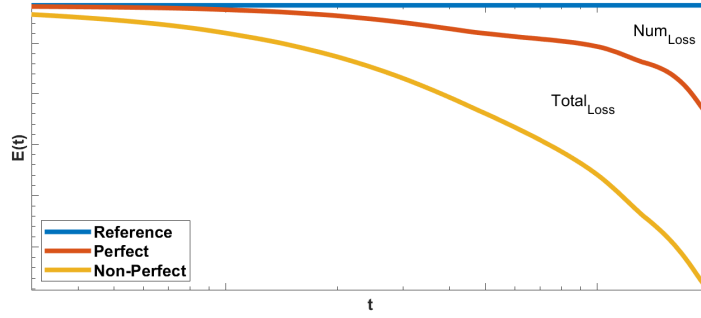


Figure 7.1: Example of the time evolution of kinetic energy computed from Equation (7.5). The area between the blue and red line (Num_{Loss}) corresponds to energy lost due to numerical dissipation. The area between the blue yellow line ($\text{Total}_{\text{Loss}}$) corresponds to the total energy lost due to numerical and physical dissipation.

Figure 7.1 depicts an example of kinetic energy loss over time. The blue *reference* line corresponds to the maximum value of E_k at $t = 0$, hence if no dissipation whatsoever occurs during the numerical simulation then the kinetic energy would be constant over time. Equation (7.5) is computed using the initial conditions; therefore the reference solution represents an idealised, fictive, case. The *red* and *yellow* lines provide an example of what the decay of kinetic energy could potentially be over time when a perfect ($\eta = 0$) and non-perfect ($\eta \neq 0$) fluid are used, respectively. This analysis is only used at early simulation times until $t = 2$, at which time the instability is still experiencing its linear growth phase. To quantify the volume-averaged amount of kinetic energy lost during this time due to numerical dissipation the expression

$$\text{Num}_{\text{Loss}} = \frac{1}{V} \int_{t=0}^2 (E_{\text{Reference}} - E_{\text{Perfect}}) dt \quad (7.6)$$

is used which corresponds to the region in Figure 7.1 that displays Num_{Loss} which is the area between the blue and red line — this is only applicable when a perfect fluid ($\eta = 0$) is used. When a non-perfect fluid is used, the loss in kinetic energy is due to both physical and numerical dissipation which is defined as $\text{Total}_{\text{Loss}}$ and given by

$$E_{\text{Loss}} = \int_{t=0}^2 (E_{\text{Reference}} - E_{\text{Non-Perfect}}) dt. \quad (7.7)$$

To find the kinetic energy lost due to physical dissipation alone, i.e. the region between the red and yellow line from Figure 7.1, the $\text{Viscous}_{\text{Loss}}$ is calculated,

$$\text{Viscous}_{\text{Loss}} \approx E_{\text{Loss}} - \text{Num}_{\text{Loss}}. \quad (7.8)$$

Equation (7.8) can only be used as an approximation because the numerical methods used for the evaluation of the non-perfect equations will also add numerical viscosity to

the solution.

To quantify the small-scale structures appearing in the developed flow profile, the density power spectrum is computed which is applied to the rest-mass density field at selected flow times in the same manner as [Radice & Rezzolla \(2012\)](#). This method was originally proposed by [Beckwith & Stone \(2011\)](#) to estimate the inherent numerical viscosity of a code. Smaller-scale structures can be smeared out by dissipative schemes so by quantifying to what extent they appear in the solution provides insight into a scheme's ability at resolving complex, intricate flow structures. The one-dimensional integrated power spectra for a given quantity, in this case the rest-mass density $\rho(x, y)$, is defined as

$$P_\rho(k) = \int_{-1}^1 |\hat{\rho}(k, y)|^2 dy, \quad (7.9)$$

where k is the wavenumber and

$$\hat{\rho}(k, y) = \int_{-1/2}^{1/2} \rho(x, y) e^{-2\pi i k x} dx \quad (7.10)$$

is the one-dimensional Fourier transform of ρ which is computed using the Fast Fourier Transform (FFT) algorithm in Matlab. Finally, following the aforementioned previous authors, the density power spectrum is normalised according to

$$\sum_{k=1}^{NX/2} P_\rho(k) = 1. \quad (7.11)$$

To compliment the quantitative analysis, contour plots of the rest-mass density field are also shown where appropriate to provide qualitative insight into the behavior of the numerical solution.

7.2 The Inviscid Case

The inviscid case is considered first whereby $\eta = 0$. An initial grid resolution study was performed in which the RMS value of the velocity y -component from Equation (7.4) was used to determine the growth of the instability. The inset of Figure 7.2 reports the v_{RMS} using the WENO3 scheme and HLL Riemann solver for different grid resolutions. It is evident that the coarser grid resolutions produce a smaller instability growth whereas the two finest resolutions provide similar v_{RMS} profiles for the linear growth stage, i.e. until time $t \approx 2.5$. From this simple analysis it is concluded that the grid resolution of 512×1024 provides adequate spatial (grid) resolution which can be used as a base grid for subsequent analysis. The main result in Figure 7.2 conveys the v_{RMS} at the grid resolution 512×1024 using all the numerical schemes described so far. Considering again the linear growth

stage, the higher-order MP-WENO schemes show the best agreement with one another whilst also reporting the steepest growth whereas the WENO3 and MUSCL2 (which is included as a low-order comparative) schemes report successively shallow growth profiles.

The rest-mass density contours at time $t = 4$, i.e. when the linear growth stage has finished, are found in Figure 7.4. Perhaps the most striking difference between the reported snapshots is the occurrence of secondary flow features for the higher-order MP-WENO5 and MP-WENO7 schemes in Figures 7.4c and 7.4d, respectively. [Radice & Rezzolla \(2012\)](#) attributed these secondary flow features to the numerical schemes themselves yet they were reported only for full-wave Riemann solvers; here the HLL Riemann solver is used which does not resolve the intermediate contact discontinuity in the Riemann fan. The lower-order MUSCL2 and WENO3 schemes in Figures 7.4a and 7.4b, respectively, lack the appearance of any secondary flow feature. Subtle differences are present in the amount of vortex rotation present yet their qualitative features appear starkly different from the disorganised and intricate flow structure that develops for the higher-order schemes.

In an attempt to quantify the flow structures of the instability in the inviscid case from Figure 7.4, the power spectra for the rest-mass density profiles is computed according to Equations (7.9), (7.10), and (7.11). The rest-mass density power spectra for all considered numerical schemes is found in Figure 7.3. A clear difference in the computed profiles can be seen which indicates the higher-order schemes possess a lower numerical dissipation characteristic than their lower-order counterparts. This dissipation is however purely numerical and arises due to the numerical scheme itself. The MP-WENO7 scheme emerges as the least dissipative numerical scheme whereas the MUSCL2 scheme appears as the most dissipative. Figure 7.3 then supports the rest-mass density contours in Figure 7.4 which demonstrate how the higher-order schemes are capable of resolving smaller, complex flow features in the inviscid case where $\eta = 0$. Whether the secondary flow features can be deemed physical or not remains to be concluded and requires the inclusion of physical viscosity for this to be realised ([Radice & Rezzolla 2012](#)). Furthermore, this finding also offers a glimpse into what role the numerical viscosity plays regarding the initial growth of the instability. From Figure 7.2 it was seen that the high-order schemes reported a steeper increase in the v_{RMS} value, similarly the inset showed how a finer grid resolution predicted a steeper growth of the v_{RMS} . It is then possible to conclude that the high-order schemes predict an increased growth-rate compared to the lower-order schemes using the same grid which can be attributed to the amount of numerical dissipation present within the numerical scheme.

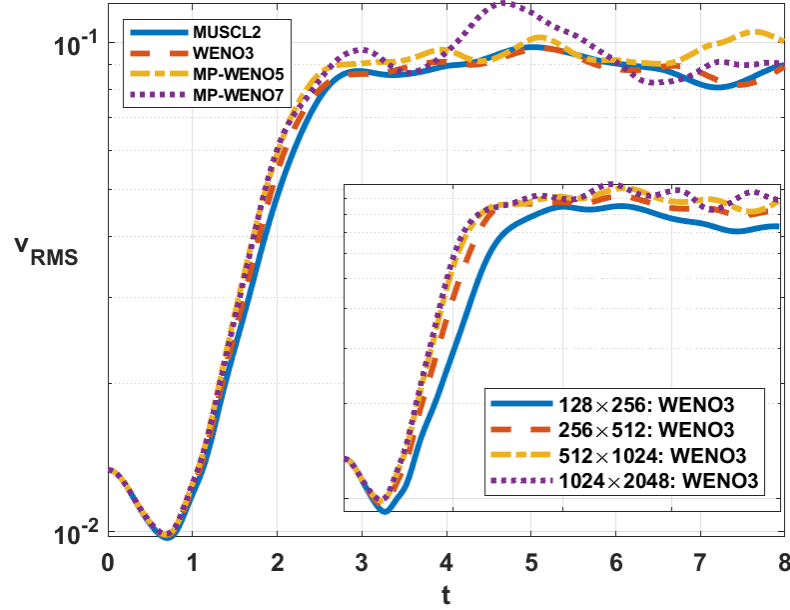


Figure 7.2: Evolution of the v_{RMS} for all numerical schemes when $\eta = 0$ using a grid resolution of 512×1024 . The inset shows the effect of grid refinement when the WENO3 spatial reconstruction method is used. The HLL Riemann solver is used throughout.

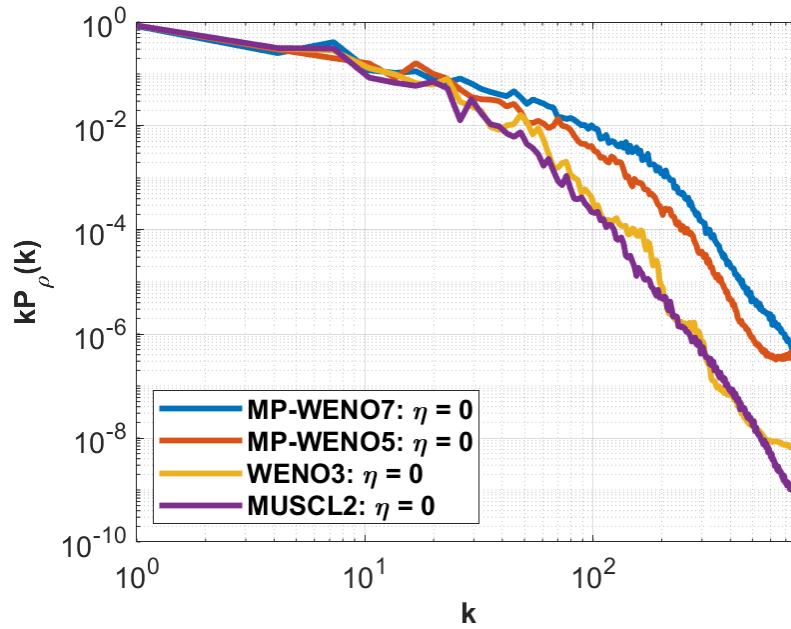


Figure 7.3: The rest-mass density power spectrum at $t = 4$ when $\eta = 0$ for all numerical schemes considered using a grid resolution of 512×1024 .

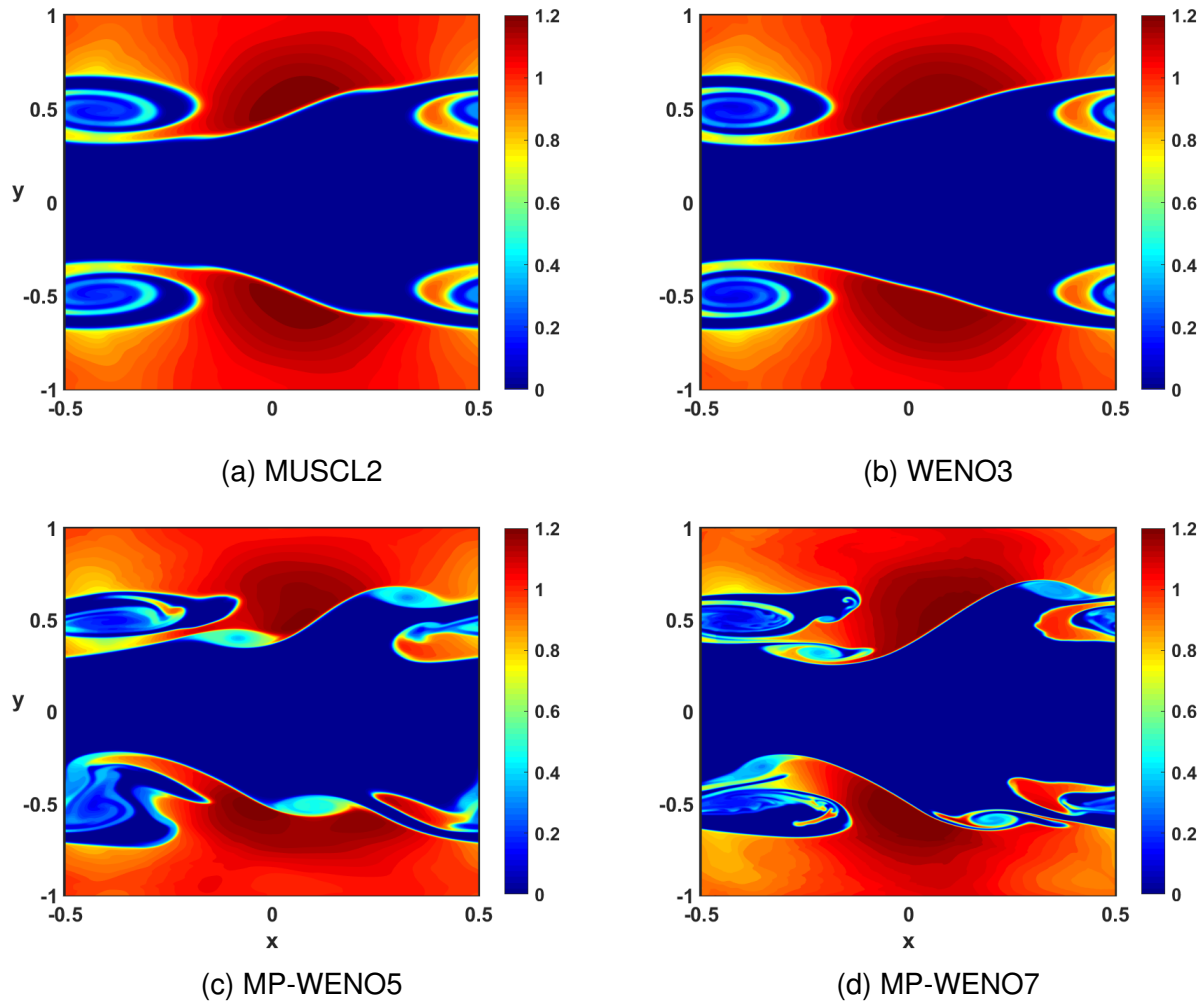


Figure 7.4: Rest-mass density contours at $t = 4$ for the MUSCL2 (7.4a), WENO3 (7.4b), MP-WENO5 (7.4c) and MP-WENO7 (7.4d) numerical schemes. For all cases the HLL approximate Riemann solver was used with a grid resolution of 512×1024 .

7.3 Influence of Shear Viscosity

The influence of the numerical solution in the face of physically resolved shear viscosity is considered. The numerical simulations performed in Chapter 7.2 are repeated for a shear viscosity coefficient of $\eta = 10^{-3}$. This enables a direct comparison of how the numerical solutions may differ when high-order methods are used in the presence and absence of physically resolved viscosity. Furthermore, it was noted by [Takamoto & Inutsuka \(2011\)](#) that a converged solution of the instability growth-rate is not achievable in the perfect fluid case. This is due to the triggering and subsequent interference of grid-scale perturbations in the solution. [Radice & Rezzolla \(2012\)](#) demonstrated how small-scale perturbations appeared to be a function of the grid resolution as well as the numerical scheme; in other

words, a function also of the numerical viscosity present. If however, physical viscosity is included then any grid-scale perturbations can be effectively smoothed out. Figure 7.5 exemplifies a converged growth when physical viscosity is resolved. Using a constant resolution of 512×1024 , the evolution of v_{RMS} is reported for a fixed value of $\eta = 10^{-3}$ for all numerical schemes. A good agreement is found between all WENO-based schemes with the MUSCL2 scheme falling marginally short of the former predictions. The inset of Figure 7.5 features the same evolution but for the MP-WENO7 scheme using $\eta = 10^{-3}$ and a varying grid resolution. Interestingly, by using the high-order scheme, convergence in the growth is achieved for the viscous case at all grid resolutions.

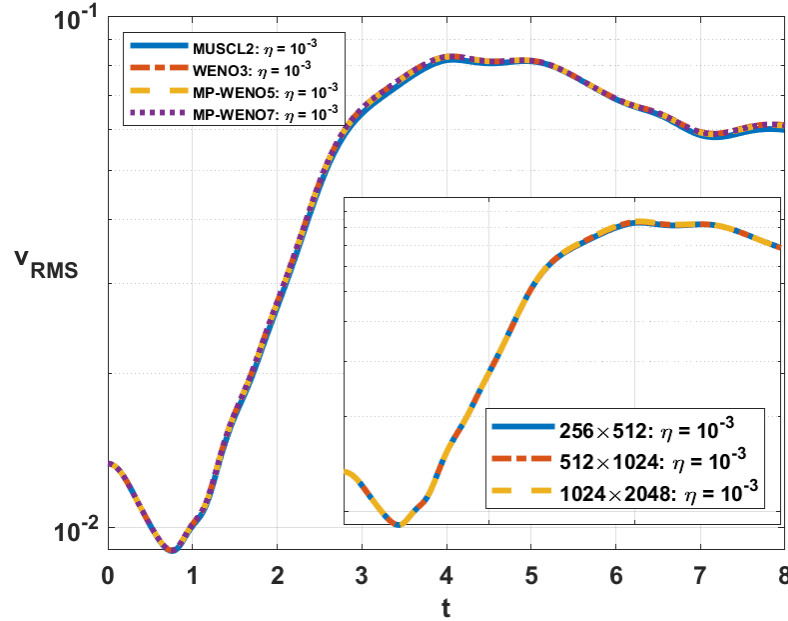


Figure 7.5: Evolution of the v_{RMS} for all numerical schemes when $\eta = 10^{-3}$, the inset shows the affect of grid refinement when the MP-WENO7 spatial reconstruction method is used also for $\eta = 10^{-3}$. The HLL Riemann solver is used throughout.

Having seen that the growth of the instability is similar when physical viscosity is resolved at $\eta = 10^{-3}$, the rest-mass density contours should be considered to glimpse any qualitative features arising in the developed flow profile. Figure 7.6 shows the rest-mass density contours at $t = 4$ for all numerical schemes considered at a grid resolution of 512×1024 . It is immediately apparent that differences manifest in the flow profile are not obviously discernible by eye. Instead, given previous insight found from comparing the instability growth in the viscous case, it appears the numerical solutions have converged regardless of the numerical schemes deployed, at least for when $\eta = 10^{-3}$ and at the time directly after the linear growth stage of the instability has finished.

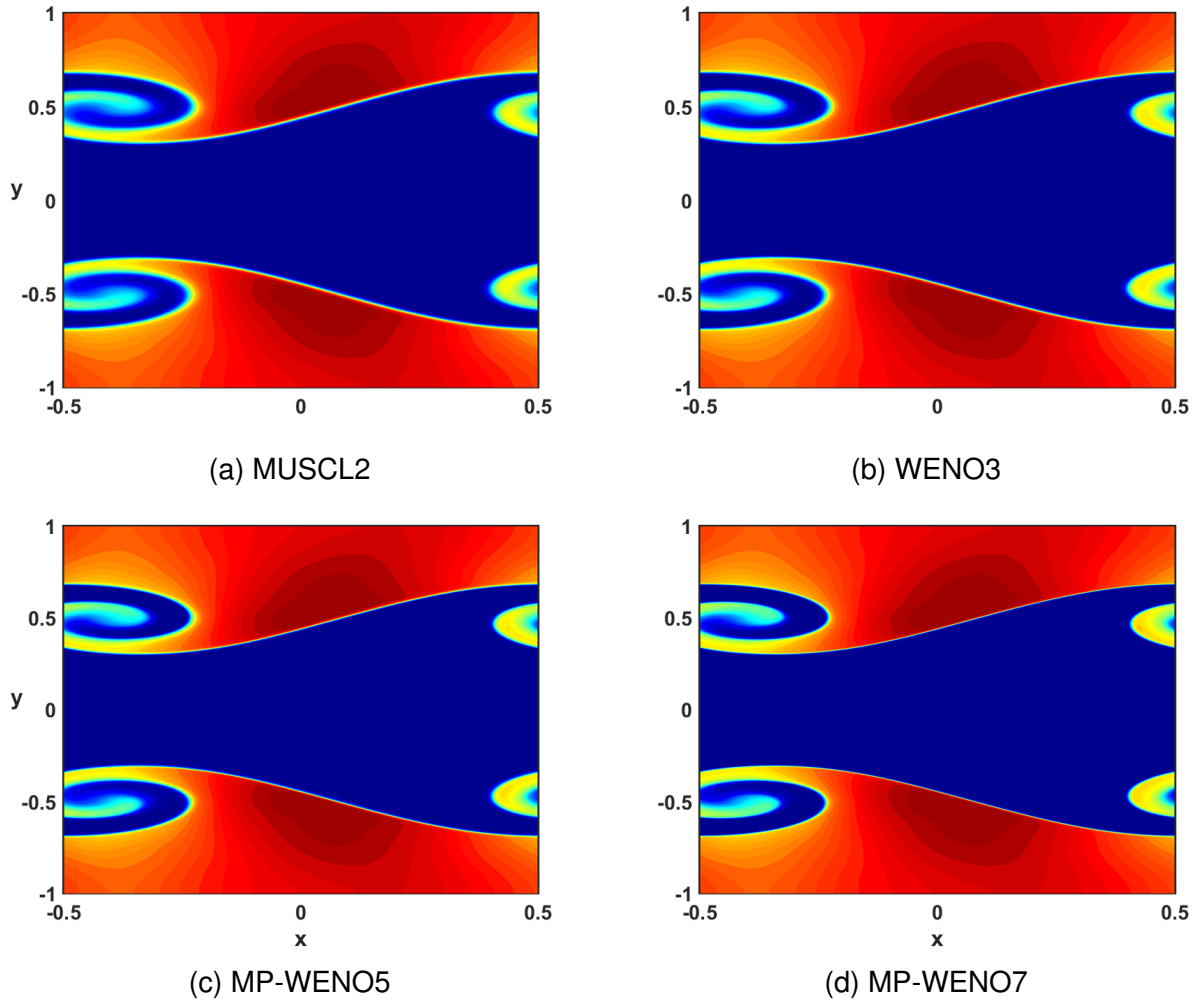


Figure 7.6: Rest-mass density contours at $t = 4$ for the MUSCL2 (7.6a), WENO3 (7.6b), MP-WENO5 (7.6c) and MP-WENO7 (7.6d) numerical schemes at $\eta = 10^{-3}$. For all cases the HLL approximate Riemann solver was used with a grid resolution of 512×1024 .

To provide quantitative comparison between the results presented thus far in Chapter 7.3, the time evolution of the total kinetic energy is considered from Equation (7.5). Figure 7.7 presents the time histories of the kinetic energy for all spatial reconstruction schemes when $\eta = 0$ and $\eta = 10^{-3}$. The figures on the left-hand-side show the decay of kinetic energy using the two specified values of η for the coarse (Figure 7.7a), medium (Figure 7.7c) and fine (Figure 7.7e) resolutions. By considering the results when $\eta = 0$, any decay in the kinetic energy comes from purely numerical dissipation which is a function of the grid resolution and the numerical schemes used. As the grid becomes finer the decay due to numerical dissipation lessens however for the coarser simulations a clear difference in kinetic energy decay can be found between the numerical schemes. The second-order MUSCL scheme is shown to be the most dissipative whereas the MP-WENO7 scheme is

the least, reaffirming the conclusions drawn from Figure 7.3 in Chapter 7.2. When physical dissipation is included in the simulation, the profile of the kinetic energy decay begins to converge as the grid resolution increases.

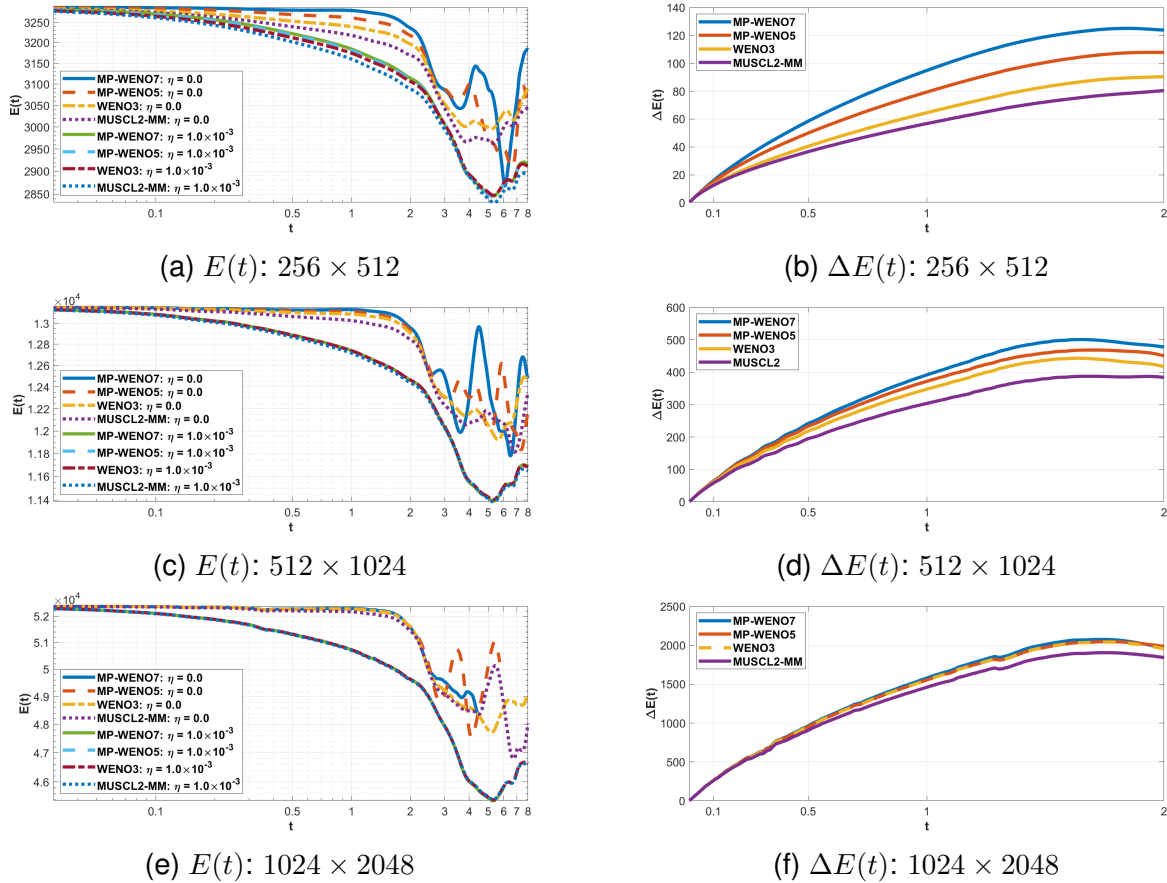


Figure 7.7: Time evolution of the kinetic energy defined in Equation (7.5) for when $\eta = 0$ and $\eta = 10^{-3}$ (left figures) using different grid resolutions: coarse–fine (top–bottom) and all spatial reconstruction schemes. The right-hand-side column shows the difference in kinetic energy whereby the solution when $\eta = 10^{-3}$ is subtracted from the solution when $\eta = 0$ for the time up until $t = 2$.

The figures on the right-hand-side of Figure 7.7 report the difference between the decay of kinetic energy for the case in which $\eta = 0$ and $\eta = 10^{-3}$. This difference is defined as the decay found when $\eta = 10^{-3}$ is subtracted from the decay when $\eta = 0$. In other words, it represent the additional decay in kinetic energy occurring when physical viscosity is included. It can be clearly seen from Figures 7.7b and 7.7d that the amount of additional decay provided by the inclusion of physical viscosity actually increases when a high-order scheme is used relative to the lower-order ones — it does not however mean that the overall kinetic energy decay is greater. The likely explanation for this result resides in

the fact that the higher-order scheme is able to provide a more accurate solution at the non-dissipative stage of the governing equations, the solution of which is then fed directly into the dissipative stage and used as the initial conditions at each time-step. Considering Figures 7.7c and 7.7d, although the higher-order schemes show a greater influence from the presence of physical viscosity, the overall decay of kinetic energy finds reasonable agreement, indicative that the physical viscosity still dominates the numerical viscosity at this specified value of η .

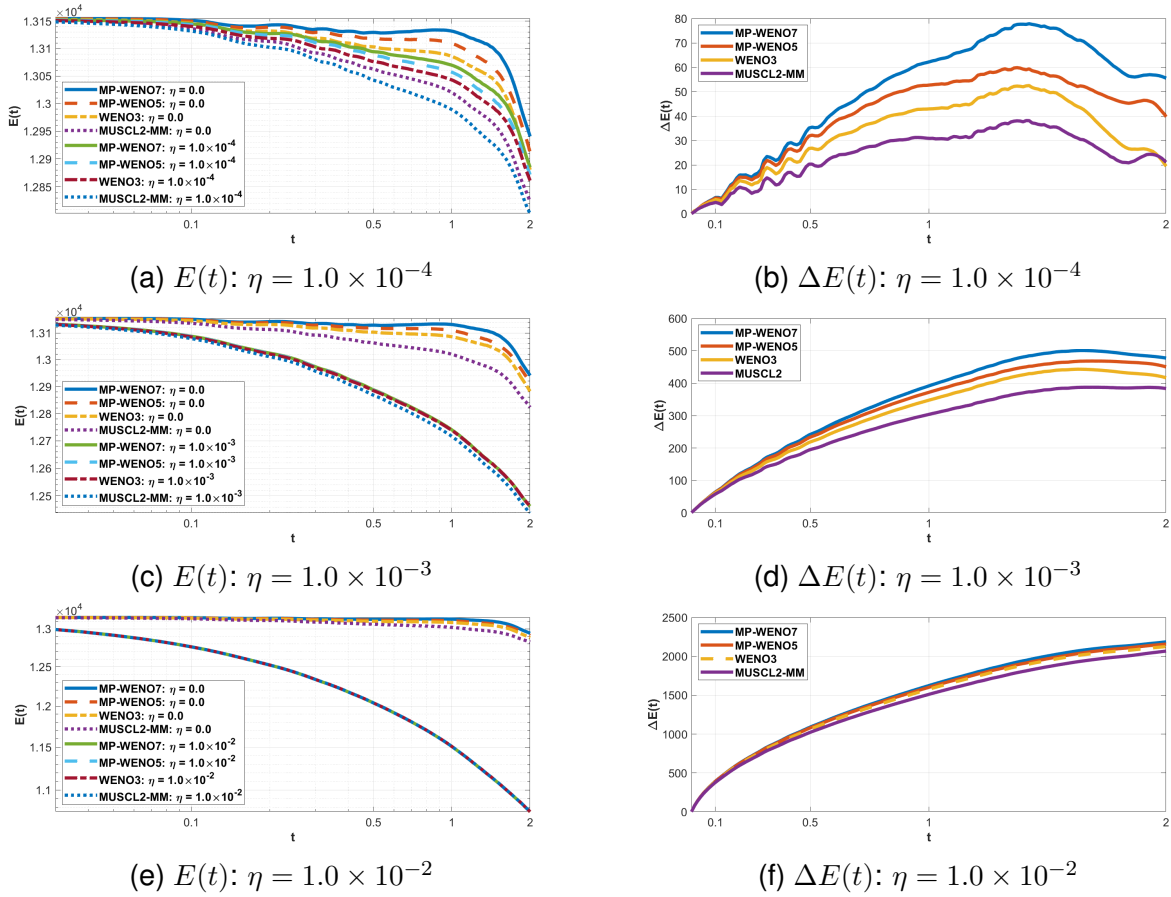


Figure 7.8: Time evolution of the kinetic energy defined in Equation (7.5) for when $\eta = 0, 10^{-4}, 10^{-3}$ and $\eta = 10^{-2}$ (left figures) using the medium grid resolution all spatial reconstruction schemes. The right-hand-side column shows the difference in kinetic energy whereby the viscous solution is subtracted from the solution when $\eta = 0$ for the time up until $t = 2$.

Further to the above analysis, a case in which a fixed grid yet varying η is considered to test the above conclusion further. A range of η is chosen such that at its smallest value, $\eta = 10^{-4}$, the physical viscosity has little affect on the solution and at its largest, $\eta = 10^{-2}$, the physical viscosity dominates the solution. Figure 7.8, in a similar spirit to Figure 7.7,

displays the evolution of kinetic energy and the corresponding differences between the viscous and inviscid solution. In the case of $\eta = 10^{-4}$ it is clear from Figures 7.8a and 7.8b that the prescribed η has a smaller impact on the overall decay of kinetic energy compared to when $\eta = 10^{-2}$ in Figures 7.8e and 7.8f.

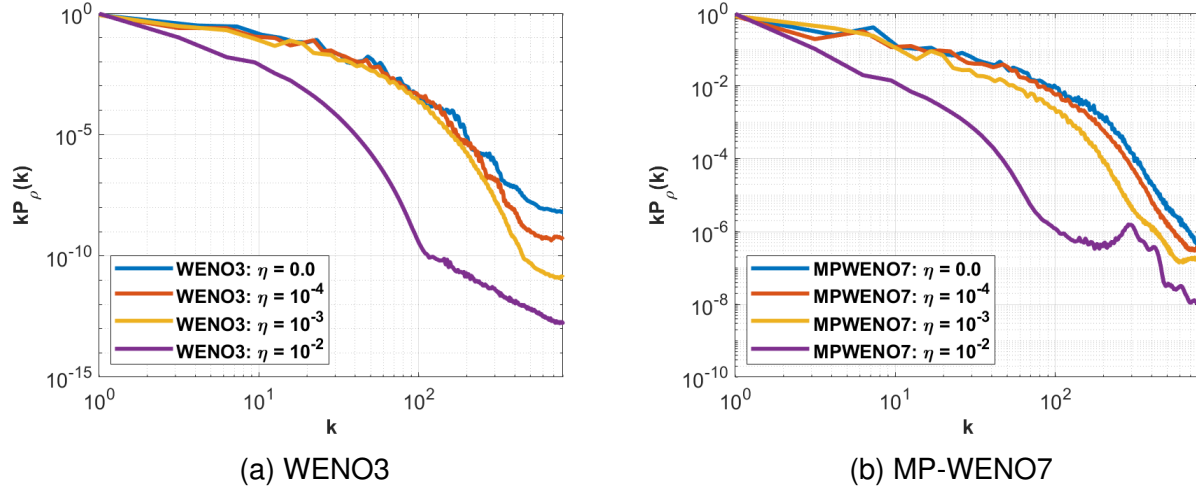


Figure 7.9: The rest-mass density power spectrum at $t = 4$ for varying η using the WENO3 (7.9a) and MP-WENO7 scheme (7.9b) at a grid resolution of 512×1024 .

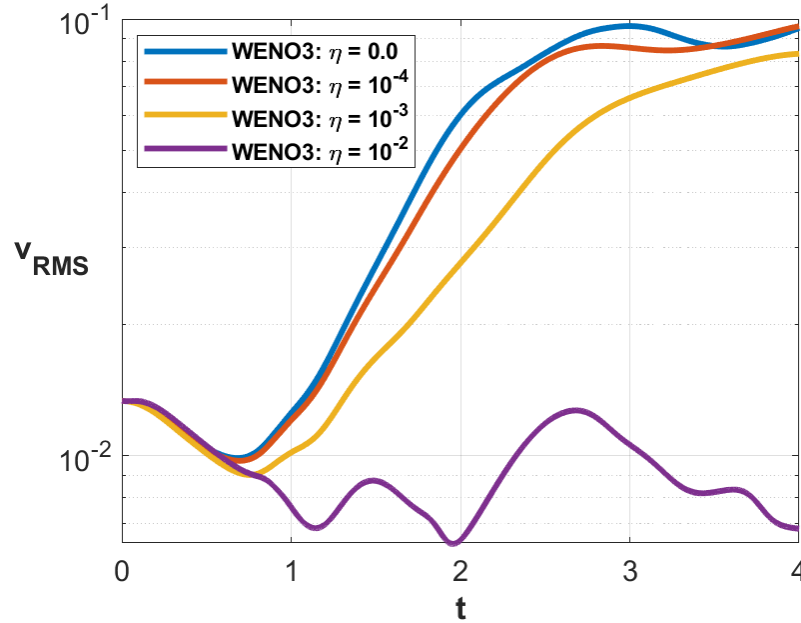


Figure 7.10: Evolution of the v_{RMS} for the WENO3 scheme at varying value of η .

The density power spectrum for the case in which the WENO3 and MP-WENO7 schemes are used for different values of η is shown in Figure 7.9 for the grid resolution of 512×1024 . Considering the WENO3 results in Figure 7.9a, an overall lower value of the power spectra is found when compared to the MP-WENO7 results shown in Figure 7.9b. This is indicative of the higher overall dissipation present in WENO3 solutions. Since it was previously seen that the time evolution of the kinetic energy in the dissipative case from Figure 7.8 is similar between different numerical schemes, it is suggested that this overall difference is still due to numerical dissipation. Note also how at the higher wave numbers a greater difference is seen for the WENO3 scheme than for the MP-WENO7. The result obtained using $\eta = 10^{-2}$ is clearly dominated by viscous dissipation in both cases and follows a largely different trend to its less viscous counterparts.

As a final note, the time evolution of the v_{RMS} is reported in Figure 7.10 for the WENO3 scheme at varying values of η in it is clear that the viscosity actively reduces the instability growth as η increases.

7.4 On Estimating Numerical Viscosity using Physical Viscosity

In a recent study, [Obergaullinger & Aloy \(2020\)](#) postulated that due to the resemblance in nature of physical and numerical viscosity, scaling relations to estimate the effective numerical viscosity of a code could be determined by establishing relationships from results achieved both with and without the inclusion of physical viscosity. A relationship was found by tracking the growth of the non-relativistic Kelvin–Helmholtz instability over (early) time. Differences between a solution at a very fine resolution and those at a coarser resolution or with physical viscosity present revealed a pattern that indicated numerical and physical viscosity both had a similar effect on the solution in terms of the instability growth. From this they were able to conclude a scaling correlation between physical and numerical viscosity. With this in mind, an investigation is conducted to evaluate this general hypothesis: numerical viscosity is capable of effectively mimicking physical viscosity. Following on from this, an attempt is made to measure the numerical viscosity present in the solution using physically resolved viscosity.

The time evolution of the kinetic energy according to Equation (7.5) is used as the measurement parameter in this investigation, as opposed to [Obergaullinger & Aloy \(2020\)](#) who made use of the instability growth. If numerical viscosity does truly mimic physical viscosity then one could argue that there should exist a correlation between the amount of energy dissipated through either mechanism—this is basic premise of the current investigation. Three simulations are performed when $\eta = 0$, i.e. only numerical dissipation is present, using grid resolutions of 512×1024 , 256×512 and 128×256 . A further three simulations are performed when $\eta \neq 0$ using a finer grid resolution of 1024×2048 , i.e. both

numerical dissipation (although to a lesser extent) and physical dissipation are present. This method is applied to simulations using the WENO3, MP-WENO5 and MP-WENO7 spatial reconstruction schemes. A trial and error procedure then ensues to fathom an agreeable value of η using the fine resolution run to match the decay of kinetic energy in the simulations where $\eta = 0$ at the three coarser grid resolutions. In other words, a physical viscosity is prescribed such that the resulting decay of kinetic energy matches that of a solution where physical viscosity is not present but achieved using a coarser resolution. Figure 7.11 depicts the attempts to match the decay of kinetic energy using the fine resolution and non-zero η with the results obtained at coarser resolutions and $\eta = 0$. A perfect matching is not always attainable, especially when a match is sought for the coarsest grid resolution. Note that the results have been scaled through consideration of the grid resolution, therefore these results only serve to match the trend of the kinetic energies and not the specific values.

7.4.1 Numerical Viscosity Estimations for WENO Reconstruction

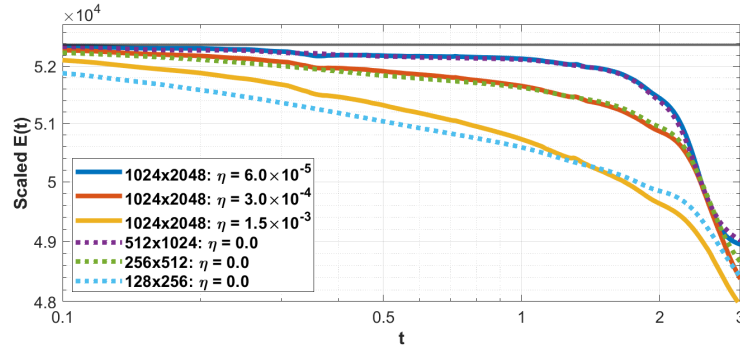
Using the raw data corresponding to the data presented in Figure 7.11, the volume-average total kinetic energy loss is calculated for each case using Equation (7.7), this quantity is hereafter referred to as E_{Loss} . This step enables the kinetic energies to be directly compared and since the loss rates are considered to be approximately equal then so should the volume average of total kinetic energy loss. These findings are displayed in Tables 7.1, 7.2 and 7.3 for the WENO3, MP-WENO5 and MP-WENO7 schemes, respectively. A power-law relation is found of the form $y = ax^b$ by considering $x \equiv NX$ as the free variable, and when $x \equiv \eta$. Therefore, a scaling relation providing an approximate value of kinetic energy loss using either NX or η as an input is found. The scaling relation is found using a simple power-law of the form

$$E_{\text{Loss}} \approx ax^b \quad (7.12)$$

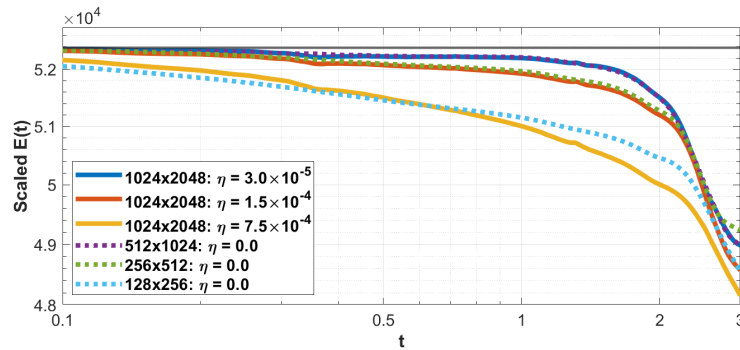
where either $x \equiv \eta$ or $x \equiv NX$ and have coefficients denoted with subscripts 1 and 2 (see Tables 7.4 and 7.5), respectively. Equating the left-hand-side of Equation (7.12) gives rise to a formula relating η and NX such that

$$\eta \approx \left(\frac{a_2}{a_1} NX^{b_2} \right)^{\frac{1}{b_1}}, \quad (7.13)$$

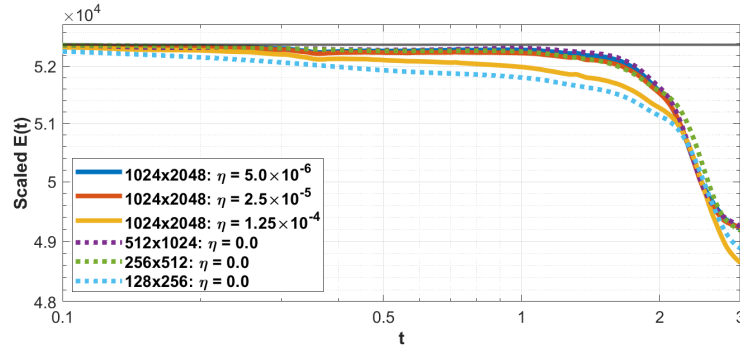
where the coefficients are problem- and scheme-dependent. Figure 7.12 depicts this relationship and how it could be used to predict the inherent numerical viscosity of the schemes for this problem. As expected, the plot shows that a finer grid resolution results in a smaller numerical viscosity and vice versa. The same figure also includes the relationship found when the MP-WENO5 and MP-WENO7 schemes are studied. The coefficients for Equation (7.13) for the WENO3, MP-WENO5 and MP-WENO7 schemes are found in Tables 7.4 and 7.5.



(a) WENO3



(b) MP-WENO5



(c) MP-WENO7

Figure 7.11: Time-evolution of the total kinetic energy in the system at different grid resolutions and different values of η for the WENO3 (7.11a), MP-WENO5 (7.11b) and MP-WENO7 (7.11c) schemes.

7.4.2 Numerical Reynolds Number

Given the estimated value of numerical viscosity found from the technique presented in Chapter 7.4.1, consideration is given into how the flow dynamics may contrast. Rest-mass density contours at $t = 2$ are shown using the corresponding values of η and grid resolution for the WENO3, MP-WENO5 and MP-WENO7 schemes in Figures 7.15, 7.17 and

Δx	η	E_{Loss}
1/1024	6×10^{-5}	6.50×10^2
1/1024	3×10^{-4}	1.51×10^3
1/1024	1.5×10^{-3}	3.19×10^3
1/512	0	6.80×10^2
1/256	0	1.53×10^3
1/128	0	3.40×10^3

Table 7.1: Volume-averaged total energy losses for the cases using the WENO3 scheme.

Δx	η	E_{Loss}
1/1024	3.0×10^{-5}	5.35×10^2
1/1024	1.5×10^{-5}	1.02×10^2
1/1024	7.5×10^{-5}	2.68×10^3
1/512	0	5.32×10^2
1/256	0	9.40×10^2
1/128	0	2.39×10^3

Table 7.2: Volume-averaged total energy losses for the cases using the MP-WENO5 scheme.

Δx	η	E_{Loss}
1/1024	5.0×10^{-6}	3.77×10^2
1/1024	2.5×10^{-5}	4.76×10^2
1/1024	1.25×10^{-4}	9.10×10^2
1/512	0	3.77×10^2
1/256	0	4.49×10^2
1/128	0	1.24×10^3

Table 7.3: Volume-averaged total energy losses for the cases using the MP-WENO7 scheme.

WENO3		MP-WENO5		MP-WENO7	
a_1	b_1	a_1	b_1	a_1	b_1
7.3×10^4	0.47	9.9×10^4	0.51	2.5×10^4	0.36

Table 7.4: Coefficients for $x \equiv \eta$ using Equation (7.12).

WENO3		MP-WENO5		MP-WENO7	
a_2	b_2	a_2	b_2	a_2	b_2
8.1×10^5	-1.1	8.6×10^5	-1.2	7.1×10^4	-0.87

Table 7.5: Coefficients for $x \equiv \text{NX}$ using Equation (7.12).

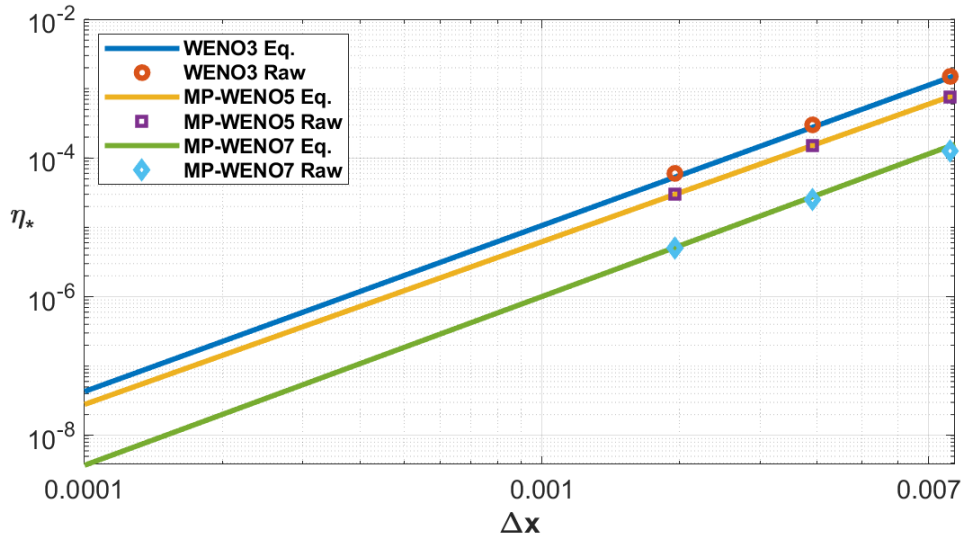


Figure 7.12: The predicted numerical viscosity as a function of grid spacing computed from Equation (7.13) for the WENO3, MP-WENO5 and MP-WENO7 schemes.

7.18, respectively. In the case of the WENO3 scheme in Figure 7.15, differences between the numerical solution are not clearly visible at first glance. Although the instability has not yet had time to fully develop, differences are apparent between the solutions upon close inspection. Looking at the rest-mass density power spectrum in Figure 7.14b, a complementary picture can be built that highlights an important difference between numerical and physical viscosity. The first two pairs considered in Figure 7.15a and 7.15d and Figures 7.15b and 7.15e do not contain any obvious differences immediately discernible by eye. The final pair in Figures 7.15c and 7.15f on the other hand show a clear difference. This is reflected in the rest-mass density power spectrum where the latter pair demonstrates less agreement, particularly at the large scales. This indicates that although the overall dissipation of kinetic energy is deemed to be approximately the same for solutions at a fine grid resolution and a prescribed physical viscosity compared with those with no physical viscosity at coarser resolutions, a correlation between the actual flow structure is not always achievable. This highlights the different roles numerical and viscosity play on the evolution of the instability. This serves as evidence that the contributions from numerical viscosity due to the grid resolution and numerical scheme do not have the same overall effect on the solution. Furthermore, it also indicates that a lower bound for the applicability of predicting the numerical viscosity through Equation (7.13) with any degree of confidence and that the formula is best applied to the two higher resolutions considered. In addition, Figure 7.14 reports the v_{RMS} evolution for the aforementioned η and grid resolution cases when the WENO3 and MP-WENO5 scheme is employed. At early time until $t = 2$ the relevant pairs demonstrate good agreement, an indication that the technique applied by [Obergaullinger & Aloy \(2020\)](#) applies here also.

Considering the rest-mass density contours for the MP-WENO5 and MP-WENO7 schemes in Figures 7.17 and 7.18, it is clear that secondary flow structures appear throughout the solutions, sometimes even when physical viscosity is resolved. Figure 7.14b provides the rest-mass density power spectrum for the MP-WENO5 case. Disparity between the spectra is even more apparent when compared to what was found for the WENO3 case. As indicated by previous authors, in particular by [Radice & Rezzolla \(2012\)](#), these secondary flow features are not expected to appear from the initial conditions adopted in this article but arise as a consequence of the numerical schemes themselves. In these cases the HLL approximate Riemann solver is employed for the numerical flux computations, hence a Riemann solver capable of resolving the intermediate contact discontinuity within the Riemann fan is not used. Therefore, the secondary flow features shown here arise due to the high-order spatial reconstruction. What remains to be seen is whether these secondary structures are genuine flow features captured by the numerical scheme or if they are artificial and purely a numerical artefact.

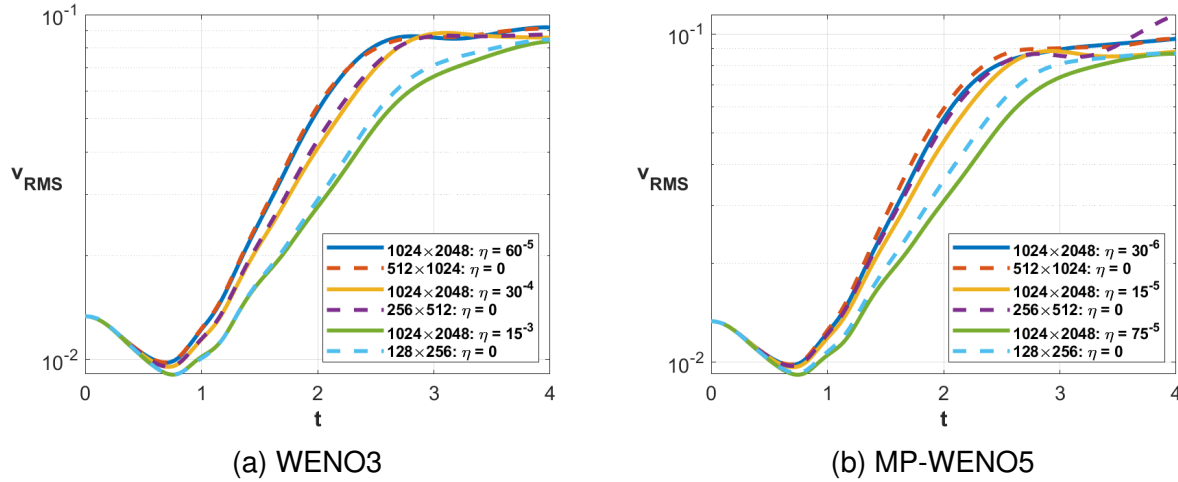


Figure 7.13: Evolution of the v_{RMS} for different grid resolutions and values of η for the WENO3 (7.13a) and MP-WENO5 (7.13b) schemes.

Given an estimated value of numerical viscosity it then becomes possible to compute a numerical Reynolds number in each computational cell from

$$Re_* = \frac{|u^x| \Delta x}{\eta_*}, \quad (7.14)$$

where η_* denotes the numerical viscosity. In general the velocity should be measured in an appropriate reference frame. In this work the velocity of our computational frame is used because the total momentum of the system remains zero in the frame. Contours of the numerical cell Reynolds number can be seen in Figure 7.16 based on the findings for the WENO3 scheme. Looking at the contour gradation in the shear region, it is found that the numerical cell Reynolds number approaches unity, in other words, the numerical cell

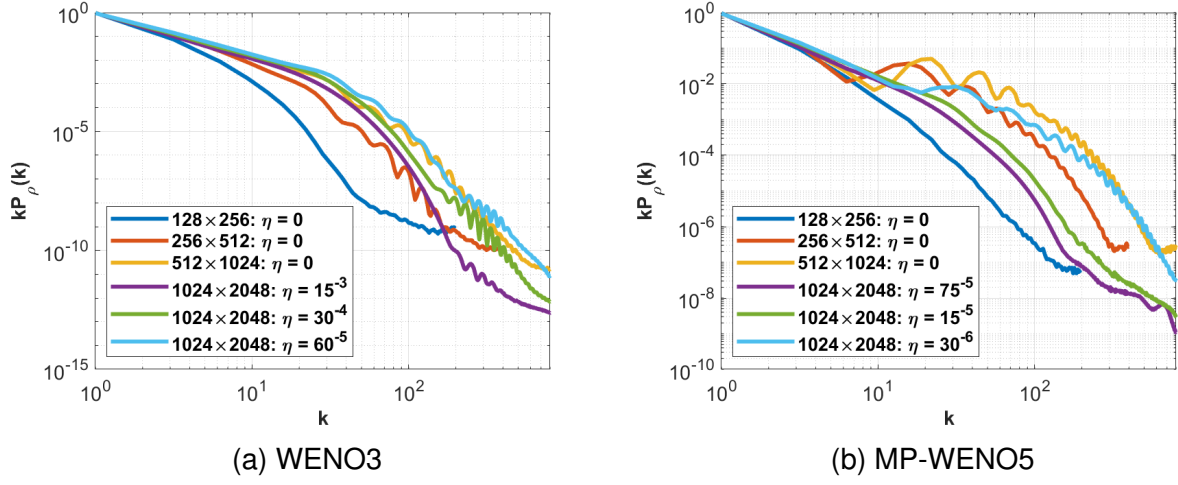


Figure 7.14: Rest-mass density power spectra for different grid resolutions and values of η for the WENO3 (7.14a) and MP-WENO5 (7.14b) schemes.

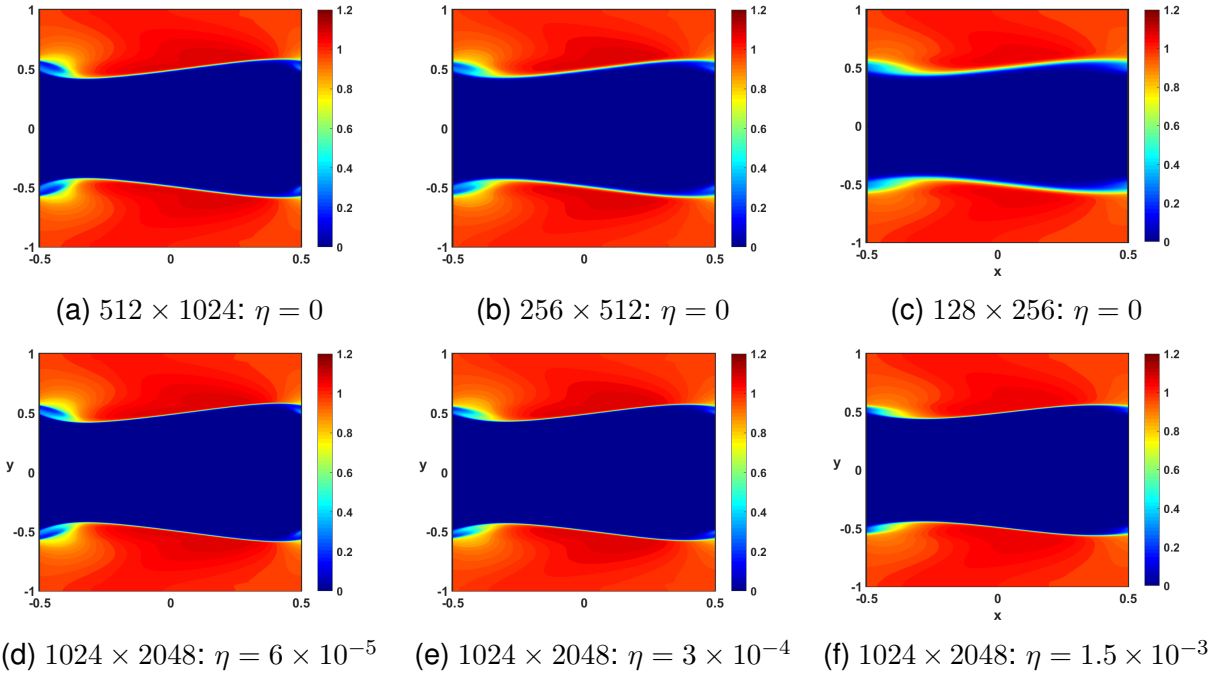


Figure 7.15: Rest-mass density contours at $t = 2$ for the WENO3 scheme at different resolutions and values of η .

Reynolds number tends to unity at Δx . Since in a numerical calculation the spatial scale comparable to Δx cannot be fully described it is therefore logical that the numerical cell Reynolds number approaches unity.

To understand whether these secondary flow features are physical or not, the MP-WENO5

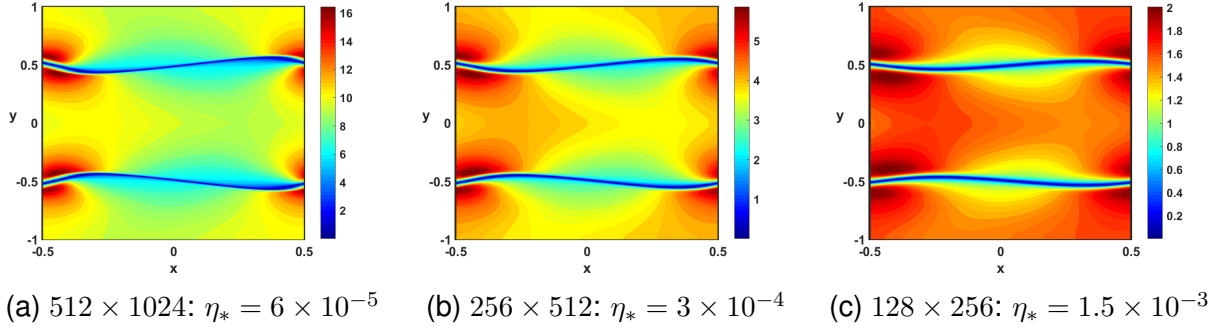


Figure 7.16: Numerical cell Reynolds number contours at $t = 2$ for the WENO3 scheme at different spatial resolutions.

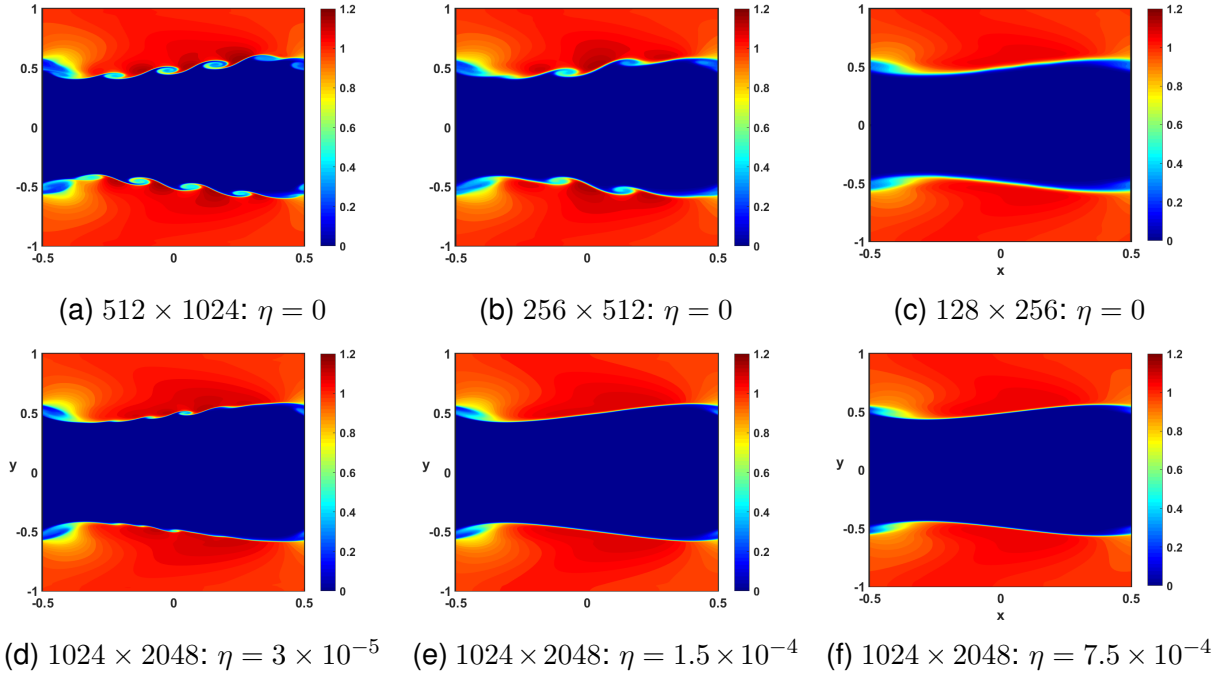


Figure 7.17: Rest-mass density contours at $t = 2$ for the MP-WENO5 schemes at different resolutions and values of η .

scheme is considered for further analysis. Figure 7.17b presents a solution when physical viscosity is included yet secondary flow features are still present. Firstly the grid resolution is increased by a factor of two whilst maintaining the level of physical viscosity. This finding is presented in Figure 7.19b in which a reduction in the presence of secondary vortices is seen, although upon close inspection they have not been completely eliminated. This indicates that the secondary features are non-physical since as the grid resolution increases a more accurate numerical solution should also be witnessed. To further confirm this the test is also performed using a larger value of η at the very fine resolution which

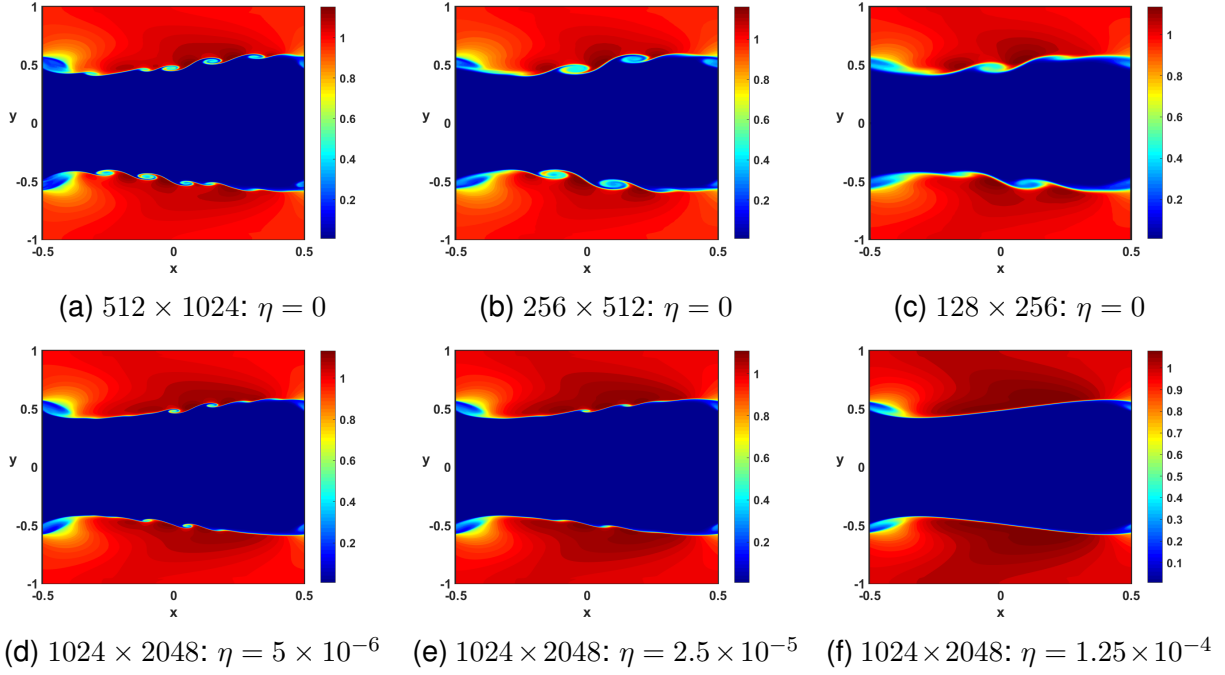


Figure 7.18: Rest-mass density contours at $t = 2$ for the MP-WENO7 schemes at different resolutions and values of η .

is shown in Figure 7.19c, which shows a definite absence of secondary flow structures. For the results in which secondary flow instabilities are not observed it is found that the numerical Reynolds number at the cell size is on the order of unity surrounding the shear layer, $|u|\Delta x/\eta_* \sim 1$. This means that physical viscosity is comparable to the numerical viscosity at any scale described in the simulation, hence grid-scale perturbations that result in secondary flow features are absent. In this numerical experiment, the important conclusion is made that to obtain a physically correct solution one should include physical viscosity and increase the resolution so that the physical Reynolds number at the cell size surrounding shear flow features should remain on the order of unity.

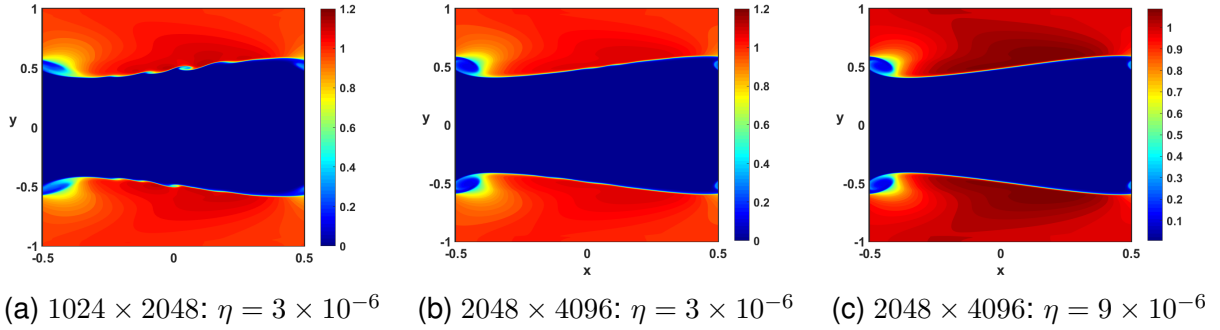


Figure 7.19: Rest-mass density contours at $t = 2$ for the MP-WENO5 scheme at high resolutions and finite values of η .

7.5 An Accuracy and Cost Evaluation of High-Order Schemes with Physical Viscosity

One such motivation behind the usage of high-order methods resides in the computational requirements necessary to resolve accurate flow physics, i.e. computational time and memory considerations. In other words, it is advantageous to apply numerical methods with the smallest truncation error and largest discretisation to reduce the computational burden otherwise imposed from extremely fine grid resolutions ([Rezzolla & Zanotti 2013](#)).

To evaluate this claim, the relativistic Kelvin–Helmholtz instability test case is performed in which the solutions obtained using a fine grid and low-order spatial reconstruction method are compared with solutions using a coarser resolution and a high-order spatial reconstruction method. It has been shown by other authors who considered the relativistic Kelvin–Helmholtz instability in the context of non-dissipative RHD and RMHD that grid-scale numerical perturbations can grow, particularly when a full-wave Riemann solver is used, see e.g. [Mignone et al. \(2009\)](#), [Beckwith & Stone \(2011\)](#), [Radice & Rezzolla \(2012\)](#). This feature means that the growth of the instability may not converge in the non-dissipative case ([Takamoto & Inutsuka 2011](#)). Therefore, to provide a more consistent evaluation two considerations are made. Firstly, the HLL approximate Riemann solver is used throughout which is known to be more dissipative than its full-wave counterparts, but is less susceptible to the development of numerical grid-scale perturbations. Secondly, to further ensure that such numerical perturbations remain absent in the numerical solution, the schemes/grid resolutions are compared with a fixed η throughout. This means that achieving a converged solution across the different numerical schemes is more likely when these factors are considered. Although the solutions obtained at a coarse resolution will not always be able to capture the flow structures on the smallest scale due to the constraints imposed by the grid, the larger structures which are still dependent on the prescribed shear viscosity and inherent numerical viscosity are hoped to be replicated.

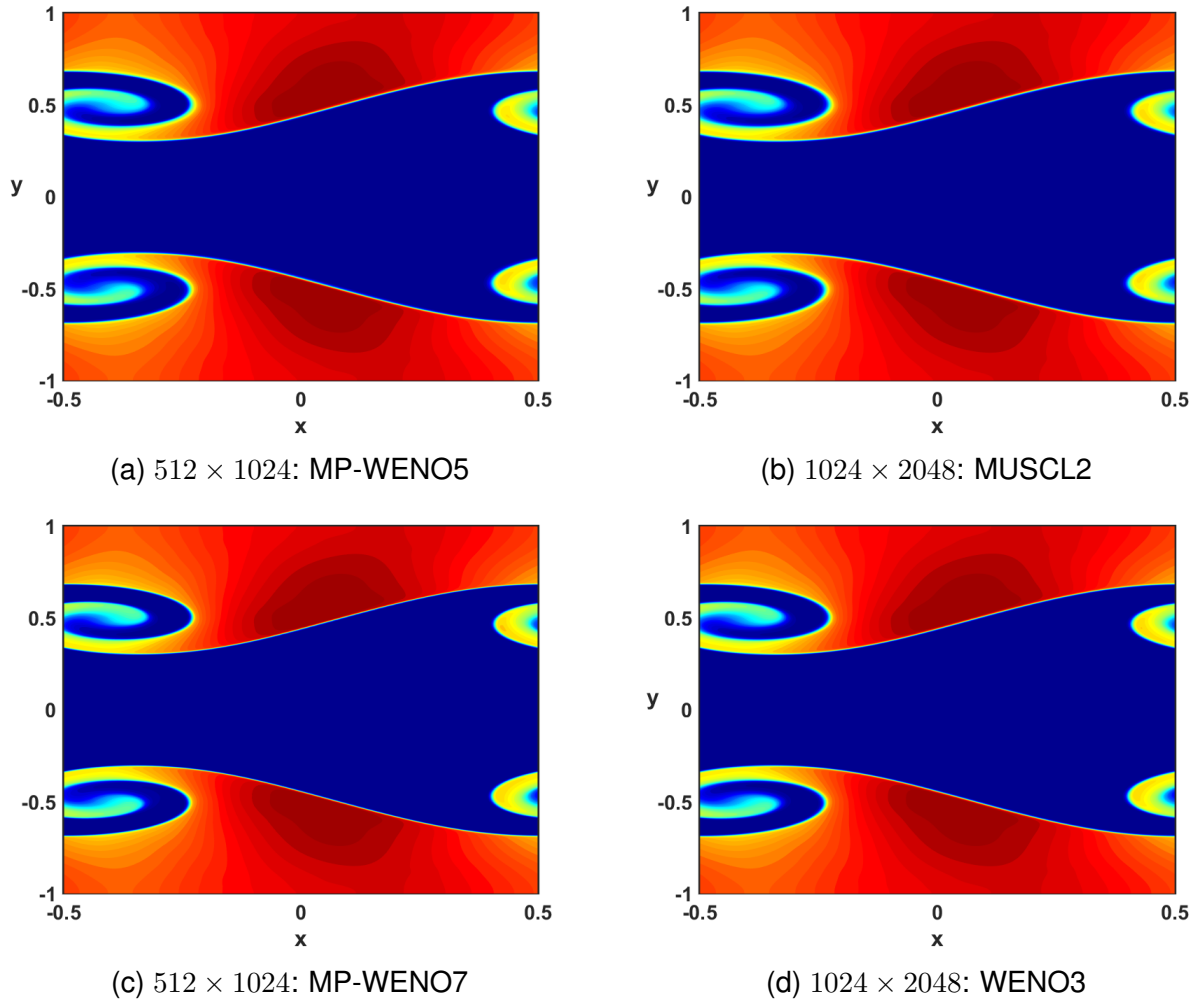


Figure 7.20: Rest-mass density contours of the relativistic Kelvin–Helmholtz instability at $t = 4$. The solutions are obtained using the HLL approximate Riemann solver and a shear viscosity parameter of $\eta = 10^{-3}$. The solutions using a coarse grid resolution of 512×1024 and a higher-order scheme (left) are seen in Figures 7.20a and 7.20c whilst the results obtained with a finer grid resolution and lower-order scheme (right) are seen in Figures 7.20b and 7.20d.

Figure 7.20 shows the rest-mass density contours of the relativistic Kelvin–Helmholtz instability at $t = 4$ using the HLL approximate Riemann solver and a shear viscosity parameter of $\eta = 10^{-3}$. At first glance, the results demonstrate good qualitative agreement when the high-order, coarse resolution solutions (Figures 7.20a, 7.20c) are compared with the low-order, fine resolution solutions (Figures 7.20b, 7.20d). Figure 7.22a shows the time evolution of v_{RMS} for which no difference discernible by eye can be seen between the growth of the instability between the solutions obtained. The instability growth at an even coarser resolution of 256×512 at high-order is comparable with the findings of solutions

using resolution 512×1024 using lower-order schemes for which little difference can be seen between results. Similarly, the inset shows this evolution of those cases presented in Figure 7.20, demonstrating that negligible difference is found between results in terms of the instability growth.

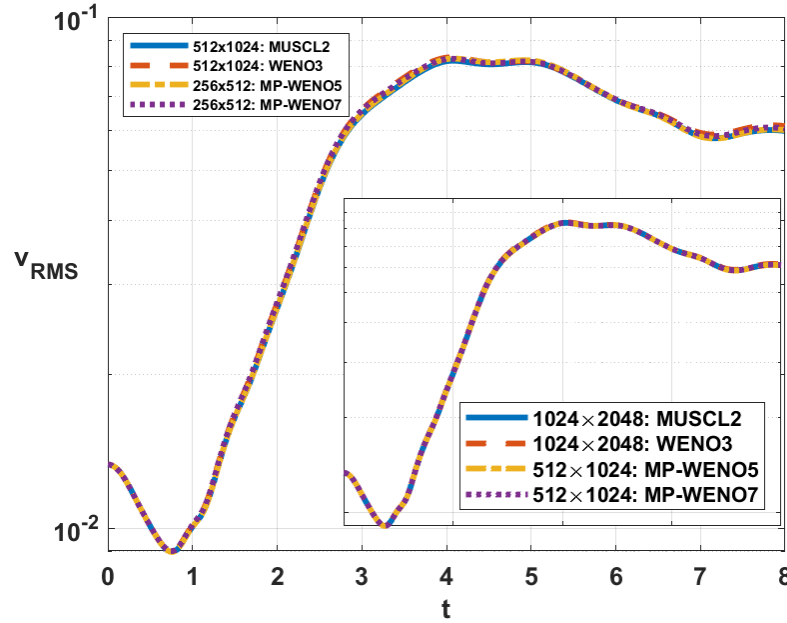


Figure 7.21: Evolution of the v_{RMS} different numerical schemes at different grid resolutions using $\eta = 10^{-3}$.

To further extend this analysis the power spectrum of the rest-mass density for the instability at $t = 4$ is shown in Figures 7.23a and 7.23b. Naturally, the results obtained with the finer resolution are able to capture smaller-scale features when compared with the coarser resolutions. Therefore, it is only possible here to make general overall comparisons between schemes at inertial, intermediate, wave numbers. The results obtained for the range of numerical schemes differ marginally in their rest-mass density power spectrum however they do report consistent power spectra across the different grid resolutions. Interestingly, the MP-WENO7 scheme at the lower grid resolutions reports the least dissipation in this range. These findings offer a positive insight into the performance of high-order schemes at coarser resolutions in terms of their quantitative accuracy when contrasted with lower-order schemes at finer grid resolutions. To further support this finding, the additional volume-average kinetic energy losses due to the inclusion of physical viscosity are shown in 7.22b, which is computed from Equation (7.8) until $t = 2$. It's clear that the amount of energy loss considered here converges for all schemes at finer grid resolutions. What is interesting is the rate of conversion of each individual numerical scheme. It can be seen that the higher-order schemes converge to the ultimate value

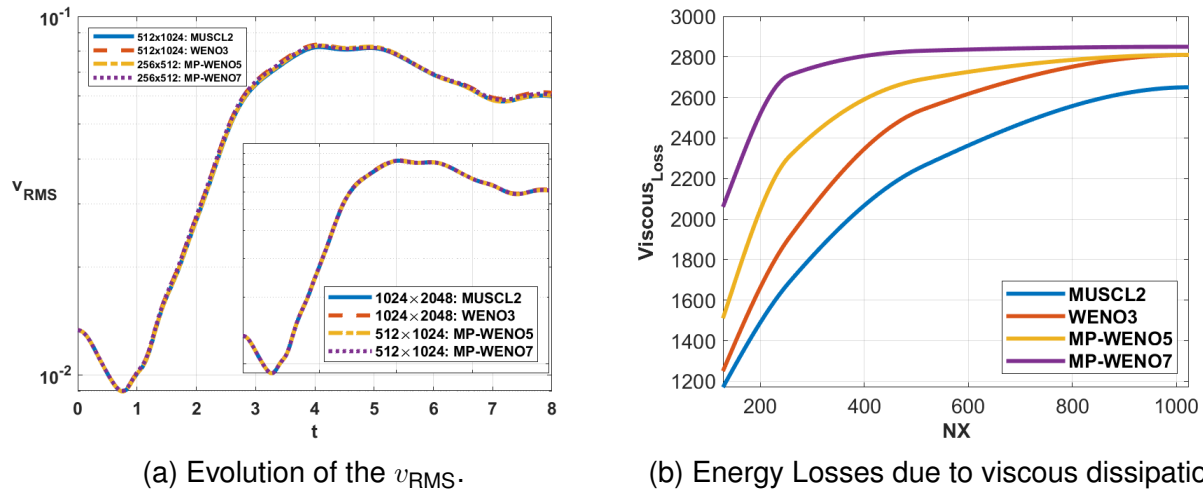


Figure 7.22: Evolution of the v_{RMS} and the volume-averaged kinetic energy loss contribution arising from the inclusion of viscosity for different numerical schemes at different grid resolutions using $\eta = 10^{-3}$.

of kinetic energy loss at a faster rate with respect to the grid resolution, meaning that the physical viscosity contribution is solved more accurately at coarser resolution when a high-order scheme is used. This supports the conjecture that due to the more accurate solutions achieved during the perfect fluid step when a high-order scheme is used which subsequently feeds into the dissipative part, a more accurate overall solution can be attained.

Finally, the computational timings for the numerical schemes and grid resolutions used are considered. Table 7.6 provides the normalised CPU timings with respect to the computational time required by the MUSCL2 scheme at the finer resolution. The MUSCL2 scheme is found to be approximately 20% less computationally expensive than the MUSCL2 scheme at the finest grid resolution however for the base/practical grid their difference is only marginal. Most notably is the computational saving found when the higher-order scheme is used on a coarser grid. When the MP-WENO5 scheme is used on the coarser grid, the computational burden is only about 17% of that required by the MUSCL2 scheme at the finer resolution. Furthermore, the MP-WENO7 requires about 18% of that time. The computational savings found by deploying the higher-order scheme on a coarser mesh are relatively substantial and could therefore contribute to the decision of which numerical scheme and grid size is suitable for certain applications.

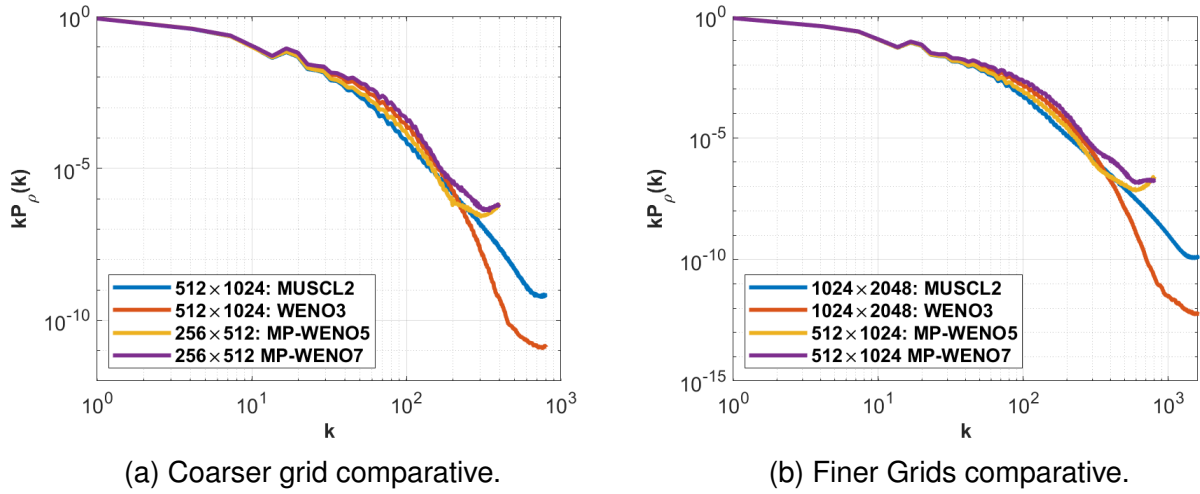


Figure 7.23: The rest-mass density power spectrum at $t = 4$ using high-order schemes at coarse grid resolutions and low-order schemes at fine grid resolutions.

Grid Size	Scheme	Normalised CPU Time
1024×2048	MUSCL2	1
1024×2048	WENO3	0.798
512×1024	MP-WENO5	0.168
512×1024	MP-WENO7	0.187
512×1024	MUSCL2	1
512×1024	WENO3	0.984
256×512	MP-WENO5	0.169
256×512	MP-WENO7	0.178

Table 7.6: Normalised CPU time with respect to the MUSCL2 scheme for the high vs low-order schemes presented in this section.

7.6 Conclusions

Numerical solutions to the non-perfect (dissipative) RHD equations using a causal theory of physical dissipation have been presented using a finite volume high-resolution shock-capturing (HRSC) Godunov-type framework. The kinetic energy decay is employed as a flow diagnostic to provide an estimation of the numerical dissipation of high-order schemes based on results obtained both with and without physically resolved dissipation for three orders of WENO reconstruction. Based on the premise that numerical viscosity can effectively mimic physical viscosity an empirical relationship between the grid spacing Δx and the numerical shear viscosity η_* for the WENO3, MP-WENO5, and MP-WENO7 scheme were established. This technique is problem- and scheme-dependant and requires a trial and error procedure to fathom the correct relationships between the dissi-

pation of kinetic energy arising from numerical and physical dissipation. To perform the same analysis using a different strategy for evaluating the numerical flux, such as the HLLC approximate Riemann solver, which is known to produce less numerical dissipation than the HLL approximate Riemann solver, is recommended as a future work.

It has been found that for the higher-order numerical schemes, namely the MP-WENO5 and MP-WENO7, the secondary flow features were present even when physical shear viscosity was resolved. To understand whether these flow features were physical attributes of the flow physics or purely numerical artefacts, two further tests were performed. Firstly, the same simulations with physical viscosity, albeit a small amount, were re-performed using a very fine grid resolution. A reduction in the size of the secondary flow features was then visible, indicating that the secondary flow features were likely caused by grid-scale perturbations. Secondly, using the very fine grid resolution, a further increase in the amount of physical shear viscosity η by a factor of three was made. From this experiment, it was found that the secondary features vanished, giving credence to the theory that such features are numerical artefacts and that physically resolved viscosity is required to achieve a converged solution. It has also been found that in order to ensure a converged solution of the relativistic Kelvin–Helmholtz instability, one should seed the numerical solution with enough physical shear viscosity such that the physical cell Reynolds number at the shear layer approaches unity.

High-order methods may offer the prospect to save computational time, because it is possible to use a higher-order scheme at a coarser spatial resolution and still achieve global flow-field properties as when a low-order scheme at a finer resolution is used. This hypothesis was tested using the relativistic Kelvin–Helmholtz instability when a value of physical viscosity was prescribed such that a converged solution could be achieved, i.e., the physical viscosity would be enough to damp any artificial grid-scale perturbations. It has been found that the high-order methods used herein offered this performance advantage if the macroscopic properties of the flow are what an investigator is placing emphasis on and resulted in an approximate computational saving of 80%. Additional test problems such as those involving shock-waves are recommended to be further investigated as a future work.

Part IV

Closing Remarks

Chapter 8

Thesis Summary

8.1 General Conclusions

In this thesis a two-dimensional relativistic hydrodynamics (RHD) CFD solver was developed using a finite volume, Godunov-type, HRSC framework. High-order WENO schemes, WENO3, MP-WENO5 and MP-WENO7 were implemented alongside the HLL, HLLC and rotated-hybrid HLL/HLLC approximate Riemann solvers and advanced forward in time using a third-order TVD Runge–Kutta method. The rotated-hybrid HLL/HLLC Riemann solver has been shown to alleviate the Carbuncle problem found when the standalone HLLC Riemann solver is used for multi-dimensional problems involving strong shocks. This was demonstrated via two numerical test cases (odd–even decoupling and the relativistic double Mach reflection) and two astrophysical test cases (the relativistic Richtmyer–Meshkov instability and a relativistic jet). For all cases the Carbuncle problem was absent when the proposed rotated-hybrid HLL/HLLC Riemann solver was employed, hence improving the robustness of the numerical computations.

Included in conjunction with the aforementioned HRSC framework was a causal solution to the dissipative RHD equations via the use Israel–Stewart theory for extended irreversible thermodynamics (EIT). This development permits the use of high-order numerical methods because the governing equations were split into a perfect and non-perfect part, the latter containing an advection and diffusion component to be treated separately. Upon satisfactory one- and two-dimensional numerical benchmark test results, an investigation then ensued into the performance of high-order numerical methods with the inclusion of physical viscosity using the relativistic Kelvin–Helmholtz instability. This investigation found that high-order methods could be potentially be very useful due to their lower numerical viscosity and ultimately provided a more accurate solution to the Israel–Stewart equations which uses values obtained from the perfect fluid step as initial conditions for their solution. An attempt was made to quantify the amount of numerical viscosity present in the simulation based on the volume-averaged kinetic energy decay however this method is problem- and scheme-dependent and requires a cumbersome trial and error procedure

to fathom the correct coefficients needed to successfully relate numerical viscosity to the grid size via a power-law. Insight was provided regarding the appearance of secondary flow features during the initial relativistic Kelvin–Helmholtz instability evolution. The analysis gives credence to the notion that any secondary flow features appearing at early simulations times are non-physical and arise due to the numerics rather than the flow physics itself.

8.2 Summary of Achievements

The achievements presented in this thesis can be summarised as:

- A two-dimensional finite volume, Godunov-type, HRSC CFD was developed for relativistic hydrodynamics *ab initio*, containing a selection of numerical methods such as high-order WENO schemes, HLL and HLLC approximate Riemann solvers and a third-order TVD Runge–Kutta method.
- A rotated-hybrid HLL/HLLC approximate Riemann solver was proposed, tested and concluded to improve the robustness of numerical solutions containing strong shock waves and alleviate the notorious Carbuncle phenomenon.
- Physical dissipation was incorporated into the HRSC framework following a causal description of non-perfect fluids through use of Israel–Stewart theory.
- Performed investigations into the behaviour of high-order numerical methods when used in conjunction with physical viscosity using the relativistic Kelvin–Helmholtz instability, highlighting potential advantages of their use
- Attempted to measure numerical dissipation using knowledge of prescribed physical viscosity coefficients and outline a new methodology for doing so.

8.3 Future Work

The quest for high-fidelity solutions to the equations of relativistic hydrodynamics and indeed, relativistic magnetohydrodynamics, is far from over. As far as the relativistic CFD solver presented in this thesis is concerned, an extension and subsequent parallelisation would be a productive next step. By extending the current implementations to higher dimensions alongside scalability measures, a better representation of the high-energy phenomena can be achieved.

It would be interesting to evaluate the performance of the rotated-hybrid Riemann solver for the MHD case since the HLLD Riemann solver initially developed by [Miyoshi & Kusano \(2005\)](#) could also be susceptible to Carbuncle-related instabilities.

For the dissipative RHD case it would be worthwhile to consider higher-order terms in the Israel–Stewart formulation and apply it to initial conditions of interest to the study of the QGP. If higher-order methods are truly beneficial for measuring the viscosity of this matter through numerical simulations then they should be tested under appropriate conditions too. Finding an estimated amount of numerical viscosity within a simulation is also a big challenge for the future and should be given due consideration if accurate measures of the QGP viscosity are sought through the use of numerical calculations.

Bibliography

- Akamatsu, Y., ichiro Inutsuka, S., Nonaka, C. & Takamoto, M. (2014), 'A new scheme of causal viscous hydrodynamics for relativistic heavy-ion collisions: A Riemann solver for quark-gluon plasma', *Journal of Computational Physics* **256**, 34–54.
- Baier, R. & Romatschke, P. (2007), 'Causal viscous hydrodynamics for central heavy-ion collisions', *European Physical Journal C* **51**(3), 677–687.
- Baiotti, L., Giacomazzo, B. & Rezzolla, L. (2009), 'Accurate evolutions of inspiralling neutron-star binaries: Assessment of the truncation error', *Classical and Quantum Gravity* **26**(11).
- Balsara, D. S. & Shu, C. W. (2000), 'Monotonicity preserving weighted essentially non-oscillatory schemes with increasingly high order of accuracy', *Journal of Computational Physics* **160**(2), 405–452.
- Banyuls, F., Font, J. A., Ibanez, J. M., Marti, J. M. & Miralles, J. A. (1997), 'Numerical $\{3 + 1\}$ General Relativistic Hydrodynamics: A Local Characteristic Approach', *The Astrophysical Journal* **476**(1), 221–231.
- Baym, G., Friman, B. L., Blaizot, J. P., Soyeur, M. & Czyz, W. (1983), 'Hydrodynamics of ultra-relativistic heavy ion collisions', *Nuclear Physics, Section A* pp. 541–570.
- Beckwith, K. & Stone, J. M. (2011), 'a Second-Order Godunov Method for Multi-Dimensional Relativistic Magnetohydrodynamics', *The Astrophysical Journal Supplement Series* **193**(1), 6.
- Bodo, G., Mignone, A. & Rosner, R. (2004), 'Kelvin-Helmholtz instability for relativistic fluids', *Physical Review E - Statistical Physics, Plasmas, Fluids, and Related Interdisciplinary Topics* **70**(3), 4.
- Boris, J. P. & Book, D. L. (1973), 'Flux-corrected transport. I. SHASTA, a fluid transport algorithm that works', *Journal of Computational Physics* **11**(1), 38–69.
- Bucciantini, N. & Del Zanna, L. (2006), 'Local Kelvin-Helmholtz instability and synchrotron modulation in Pulsar Wind Nebulae', *Astronomy and Astrophysics* **454**(2), 393–400.

- Bugner, M., Dietrich, T., Bernuzzi, S., Weyhausen, A. & Brügmann, B. (2016), 'Solving 3D relativistic hydrodynamical problems with weighted essentially nonoscillatory discontinuous Galerkin methods', *Physical Review D* **94**(8), 1–23.
- Colella, P. & Woodward, P. R. (1984), 'The Piecewise Parabolic Method (PPM) for gas-dynamical simulations', *Journal of Computational Physics* **54**(1), 174–201.
- Davis, S. F. (1984), 'A rotationally biased upwind difference scheme for the euler equations', *Journal of Computational Physics* **56**, 65–92.
- Del Zanna, L. & Bucciantini, N. (2002), 'An efficient shock-capturing central-type scheme for multidimensional relativistic flows', *Astronomy & Astrophysics* **390**, 1177–1186.
- Del Zanna, L., Chandra, V., Inghirami, G., Rolando, V., Beraudo, A., De Pace, A., Pagliara, G., Drago, A. & Becattini, F. (2013), 'Relativistic viscous hydrodynamics for heavy-ion collisions with ECHO-QGP', *European Physical Journal C* **73**(8), 1–26.
- Dolezal, A. & Wong, S. (1995), 'Relativistic Hydrodynamics and Essentially Non-Oscillatory Shock Capturing Schemes', *Journal of Computational Physics* **120**, 266–277.
- Donat, R., Font, J. A., Ibáñez, J. M. & Marquina, A. (1998), 'A Flux-Split Algorithm Applied to Relativistic Flows', *Journal of Computational Physics* **146**(1), 58–81.
- Eckart, C. (1940), 'The thermodynamics of irreversible processes. III. Relativistic theory of the simple fluid', *Physical Review* **58**, 919–924.
- Einfeldt, B. (1988), 'On Godunov-Type Methods for Gas Dynamics', *SIAM Journal on Numerical Analysis* **25**(2), 294–318.
- Eulderink, F. & Mellema, G. (1995), 'General relativistic hydrodynamics with a Roe solver', *Astronomy & Astrophysics Supplement Series* **110**, 587–623.
- Ferrari, A., Trussoni, E. & Zaninetti, L. (1978), 'Relativistic Kelvin-Helmholtz Instabilities In Extragalactic Radio-Sources', *Astronomy and Astrophysics* **64**, 43–52.
- Font, J. A. (2000), 'Numerical hydrodynamics in general relativity', *Living Reviews in Relativity* **3**.
- Font, J. A., Ibáñez, J. M., Marquina, A. & Martí, J. M. (1994), 'Multidimensional relativistic hydrodynamics: Characteristic fields and modern high-resolution shock-capturing schemes', *Astronomy and Astrophysics* **282**, 304–314.
- Godunov, S. (1959), 'A difference method for numerical calculation of discontinuous solutions of the equations of hydrodynamics', *Matematicheskii Sbornik* **47**(3), 271–306.
- Harten, A. (1983), 'High Resolution Schemes for Hyperbolic Conservation Laws', *Journal of Computational Physics* **49**, 357–393.

- Harten, A., Engquist, B., Osher, S. & Chakravarthy, S. (1987), 'Uniformly High Order Accurate Essentially Non-oscillatory Schemes, III', *Journal of Computational Physics* **71**, 231–303.
- Heinz, U., Shen, C. & Song, H. (2012), 'The viscosity of quark-gluon plasma at RHIC and the LHC', *AIP Conference Proceedings* **1441**, 766–770.
- Hiscock, W. & Lindblom, L. (1983), 'Stability and Causality in Dissipative Relativistic Fluids', *Annals of Physics* **151**(2).
- Inoue, T. (2012), 'Richtmyer-meshkov-type instability of a current sheet in a relativistically magnetized plasma', *Astrophysical Journal* **760**(1), 0–4.
- Israel, W. & Stewart, J. M. (1979), 'Transient relativistic thermodynamics and kinetic theory', *Annals of Physics* **118**(2), 341–372.
- Jameson, A., Schmidt, W. & Turkel, E. (1981), Numerical solution of the Euler equations by finite volume methods using Runge Kutta time stepping schemes, in '14th Fluid and Plasma Dynamics Conference, AIAA-81-1259'.
- Jiang, G. S. & Shu, C. W. (1996), 'Efficient implementation of weighted WENO schemes', *Journal of Computational Physics* **126**(126), 202–228.
- Kane, J., Drake, R. & Remington, B. (1999), 'An evaluation of the Richtmyer-Meshkov instability in supernova remnant formation', *The Astrophysical Journal* **20**(1), 335–340.
- Karpenko, I., Huovinen, P. & Bleicher, M. (2014), 'A 3 + 1 dimensional viscous hydrodynamic code for relativistic heavy ion collisions', *Computer Physics Communications* **185**(11), 3016–3027.
- Kim, S. D., Lee, B. J., Lee, H. J. & Jeung, I. S. (2009), 'Robust HLLC Riemann solver with weighted average flux scheme for strong shock', *Journal of Computational Physics* **228**(20), 7634–7642.
- Kim, S. D., Lee, B. J., Lee, H. J., Jeung, I. S. & Choi, J.-Y. (2010), 'Realization of contact resolving approximate Riemann solvers for strong shock and expansion flows', *International Journal for Numerical Methods in Fluids* **62**, 1107–1133.
- Landau, L. & Lifshitz, E. (1987), *Fluid Mechanics*.
- Levy, D. W., Powell, K. G. & van Leer, B. (1993), 'Use of a rotated Riemann solver for the two-dimensional Euler equations', *Journal of Computational Physics* **106**, 201–214.
- Liou, M. S. (2000), 'Mass Flux Schemes and Connection to Shock Instability', *Journal of Computational Physics* **160**(2), 623–648.
- Liu, X.-D., Osher, S. & Chan, T. (1994), 'Weighted Essentially Non-oscillatory Schemes', *Journal of Computational Physics* **115**, 200–212.

- Maccormack, R. W. (2011), 'The Carbuncle CFD Problem', *49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition* (January), 1–13.
- Martí, J. M., Ibáñez, J. M. & Miralles, J. A. (1991), 'Numerical relativistic hydrodynamics: Local characteristic approach', *Physical Review D* **43**(12), 3794–3801.
- Martí, J. M. & Müller, E. (1996), 'Extension of the piecewise parabolic method to one-dimensional relativistic hydrodynamics', *Journal of Computational Physics* **123**, 1–14.
- Martí, J. M. & Müller, E. (2003), 'Numerical Hydrodynamics in Special Relativity', *Living Reviews in Relativity* pp. 1–100.
- Martí, J. M. & Müller, E. (2015), 'Grid-based Methods in Relativistic Hydrodynamics and Magnetohydrodynamics', *Living Reviews in Computational Astrophysics* **1**(1), 182.
- Martí, J. M., Müller, E., Font, J. A., Ibáñez, J. M. & Marquina, A. (1997), 'Morphology and Dynamics of Relativistic Jets', *The Astrophysical Journal* **479**(1), 151–163.
- Martí, J. & Müller, E. (1994), 'The Analytical Solution of the Riemann Problem in Relativistic Hydrodynamics', *Journal of Fluid Mechanics* **258**, 317–333.
- Matsumoto, J. & Masada, Y. (2013), 'Two-dimensional numerical study for rayleigh-taylor and richtmyer-meshkov instabilities in relativistic jets', *Astrophysical Journal Letters* **772**(1).
- Meshkov, E. E. (1972), 'Instability of the interface of two gases accelerated by a shock wave', *Fluid Dynamics* **4**(5), 101–104.
- Mignone, A. & Bodo, G. (2005), 'An HLLC Riemann solver for relativistic flows - I. Hydrodynamics', *Monthly Notices of the Royal Astronomical Society* **364**, 126–136.
- Mignone, A. & Bodo, G. (2006), 'An HLLC Riemann solver for relativistic flows - II. Magnetohydrodynamics', *Monthly Notices of the Royal Astronomical Society* **368**(3), 1040–1054.
- Mignone, A., Plewa, T. & Bodo, G. (2005), 'The Piecewise Parabolic Method for Multidimensional Relativistic Fluid Dynamics', *The Astrophysical Journal Supplement Series* **160**(1), 199–219.
- Mignone, A., Ugliano, M. & Bodo, G. (2009), 'A five-wave Harten-Lax-van Leer Riemann solver for relativistic magnetohydrodynamics', *Monthly Notices of the Royal Astronomical Society* **393**(4), 1141–1156.
- Miyoshi, T. & Kusano, K. (2005), 'A multi-state HLL approximate Riemann solver for ideal magnetohydrodynamics', *Journal of Computational Physics* **208**(1), 315–344.

- Mohseni, F., Mendoza, M., Succi, S. & Herrmann, H. J. (2014), ‘Relativistic effects on the Richtmyer-Meshkov instability’, *Physical Review D - Particles, Fields, Gravitation and Cosmology* **90**(12), 1–6.
- Mourão Roque, V. R. & Lugones, G. (2018), ‘Assessing the role of the Kelvin-Helmholtz instability at the QCD cosmological transition’, *Journal of Cosmology and Astroparticle Physics* **JCAP03**(051).
- Muronga, A. (2007), ‘Relativistic dynamics of nonideal fluids: Viscous and heat-conducting fluids. I. General aspects and 3+1 formulation for nuclear collisions’, *Physical Review C - Nuclear Physics* **76**(1), 1–20.
- Nishikawa, H. & Kitamura, K. (2008), ‘Very simple, carbuncle-free, boundary-layer-resolving, rotated-hybrid Riemann solvers’, *Journal of Computational Physics* **227**(4), 2560–2581.
- Núñez-de la Rosa, J. & Munz, C. D. (2018), ‘Hybrid DG/FV schemes for magnetohydrodynamics and relativistic hydrodynamics’, *Computer Physics Communications* **222**, 113–135.
- Nunez-de la Rosa, J. & Munz, C. D. (2016), ‘XTROEM-FV: A new code for computational astrophysics based on very high order finite-volume methods - II. Relativistic hydro- and magnetohydrodynamics’, *Monthly Notices of the Royal Astronomical Society* **460**(1), 535–559.
- Obergaulinger, M. & Aloy, M.-Á. (2020), Numerical viscosity in simulations of the two-dimensional Kelvin-Helmholtz instability.
URL: [arxiv:2001.01927](https://arxiv.org/abs/2001.01927)
- Okamoto, K., Akamatsu, Y. & Nonaka, C. (2016), ‘A new relativistic hydrodynamics code for high-energy heavy-ion collisions’, *European Physical Journal C* **76**(10).
- Okamoto, K. & Nonaka, C. (2017), ‘A new relativistic viscous hydrodynamics code and its application to the Kelvin–Helmholtz instability in high-energy heavy-ion collisions’, *European Physical Journal C* **77**(6), 1–14.
- Pareschi, L. & Russo, G. (2005), ‘Implicit–Explicit Runge–Kutta Schemes and Applications to Hyperbolic Systems with Relaxation’, *Journal of Scientific Computing* **25**(1/2).
- Pons, J. A., Martí, J. M. & Müller, E. (2000), ‘The exact solution of the Riemann problem with non-zero tangential velocities in relativistic hydrodynamics’, *Journal of Fluid Mechanics* **422**, 125–139.
- Quirk, J. J. (1994), ‘A contribution to the great Riemann solver debate’, *International Journal for Numerical Methods in Fluids* **18**, 555–574.

- Radice, D. & Rezzolla, L. (2011), 'Discontinuous Galerkin methods for general-relativistic hydrodynamics: Formulation and application to spherically symmetric spacetimes', *Physical Review D - Particles, Fields, Gravitation and Cosmology* **84**(2), 1–26.
- Radice, D. & Rezzolla, L. (2012), 'THC : a new high-order finite-difference high-resolution shock-capturing code for special-relativistic hydrodynamics', *Astronomy & Astrophysics* **547**(A26).
- Radice, D. & Rezzolla, L. (2013), 'Universality and Intermittency in Relativistic Turbulent Flows of a Hot Plasma', *The Astrophysical Journal* **766**(1), L10.
- Rana, Z. A., Thornber, B. & Drikakis, D. (2011), 'Transverse jet injection into a supersonic turbulent cross-flow', *Physics of Fluids* **23**.
- Rembiasz, T., Obergaulinger, M., Cerdá-Durán, P., Aloy, M.-Á. & Müller, E. (2017), 'On the Measurements of Numerical Viscosity and Resistivity in Eulerian MHD Codes', *The Astrophysical Journal Supplement Series* **230**(18), 32.
- Ren, Y. X. (2003), 'A robust shock-capturing scheme based on rotated Riemann solvers', *Computers and Fluids* **32**, 1379–1403.
- Rezzolla, L. & Zanotti, O. (2001), 'An improved exact Riemann solver for relativistic hydrodynamics', *Journal of Fluid Mechanics* **449**, 395–411.
- Rezzolla, L. & Zanotti, O. (2013), *Relativistic Hydrodynamics*, first edn, Oxford University Press.
- Richtmyer, R. D. (1960), 'Taylor instability in shock acceleration of compressible fluids', *Communications on Pure and Applied Mathematics* **13**(2), 297–319.
- Roe, P. (1986), 'Characteristic-Based Schemes for the Euler Equations', *Annual Review of Fluid Mechanics* **18**(1), 337–365.
- Roe, P. L. (1981), 'Approximate Riemann solvers, parameter vectors, and difference schemes', *Journal of Computational Physics* **43**, 357–372.
- Romatschke, P. (2010), 'New Developments in Relativistic Viscous Hydrodynamics', *International Journal of Modern Physics E* **19**(01), 1–53.
- Romatschke, P. & Romatschke, U. (2007), 'Viscosity information from relativistic nuclear collisions: How perfect is the fluid observed at RHIC?', *Physical Review Letters* **99**(17), 1–4.
- Roy, V. & Chaudhuri, A. K. (2012), '2 + 1 dimensional hydrodynamics including bulk viscosity: A systematic study', *Physical Review C - Nuclear Physics* **85**(2), 1–13.

- Rusanov, V. V. (1961), 'The calculation of the interaction of non-stationary shock waves and obstacles', *USSR Computational Mathematics and Mathematical Physics* **1**(2), 267–279.
- San, O. & Kara, K. (2015), 'Evaluation of Riemann flux solvers for WENO reconstruction schemes: Kelvin-Helmholtz instability', *Computers and Fluids* **117**, 24–41.
- Schenke, B., Jeon, S. & Gale, C. (2010), '(3+1)D hydrodynamic simulation of relativistic heavy-ion collisions', *Physical Review C - Nuclear Physics* **82**(1), 1–14.
- Schenke, B., Jeon, S. & Gale, C. (2011), 'Elliptic and triangular flow in event-by-event D=3+1 viscous hydrodynamics', *Physical Review Letters* **106**(4), 1–4.
- Schneider, V., Katscher, U., Rischke, D., Waldhauser, B., Maruhn, J. & Munz, C.-D. (1993), 'New Algorithms For Ultra-Relativistic Numerical Hydrodynamics', *Journal of Computational Physics* **105**, 92–107.
- Shu, C.-w. (1997), Essentially Non-Oscillatory and Weighted Essentially Non-Oscillatory Schemes for Hyperbolic Conservation Laws Operated by Universities Space Research Association, Technical Report 97-65.
- Shu, C. W. (2003), 'High-order finite difference and finite volume WENO schemes and discontinuous galerkin methods for CFD', *International Journal of Computational Fluid Dynamics* **17**(2), 107–118.
- Shu, C.-W. (2009), 'High Order Weighted Essentially Nonoscillatory Schemes for Convection Dominated Problems', *SIAM Review* **51**(1), 82–126.
- Shu, C.-W. & Osher, S. (1988), 'Efficient Implementation of Essentially Non-Oscillatory Shock-Capturing Schemes', *Journal of Computational Physics* **77**, 439–471.
- Simon, S. & Mandal, J. C. (2019), 'A simple cure for numerical shock instability in the HLLC Riemann solver', *Journal of Computational Physics* **378**, 477–496.
- Suresh, A. & Huynh, H. T. (1997), 'Accurate monotonicity-preserving schemes with Runge-Kutta time stepping', *Journal of Computational Physics* **136**(1), 83–99.
- Svärd, M. & Mishra, S. (2011), 'Implicit-explicit schemes for flow equations with stiff source terms', *Journal of Computational and Applied Mathematics* **235**(6), 1564–1577.
- Takamoto, M. & Inutsuka, S.-i. (2011), 'A fast numerical scheme for causal relativistic hydrodynamics with dissipation', *Journal of Computational Physics* **230**(18), 7002–7017.
- Taub, A. H. (1948), 'Relativistic Rankine-Hugoniot equations', *Physical Review* **74**(3), 328–334.

- Tchekhovskoy, A., McKinney, J. C. & Narayan, R. (2007), 'wham: A WENO-based general relativistic numerical scheme - I. Hydrodynamics', *Monthly Notices of the Royal Astronomical Society* **379**(2), 469–497.
- Tian, C. & Chen, Y. (2016), 'Numerical Simulations of Kelvin–Helmholtz Instability: a Two-Dimensional Parametric Study', *The Astrophysical Journal* **824**(60).
- Titarev, V. A. & Toro, E. F. (2004), 'Finite-volume WENO schemes for three-dimensional conservation laws', *Journal of Computational Physics* **201**(1), 238–260.
- Toro, E. (2009), *Riemann Solvers and Numerical Methods for Fluid Dynamics*, third edn, Springer-Verlag.
- Toro, E. F., Spruce, M. & Speares, W. (1994), 'Restoration of the contact surface in the HLL-Riemann solver', *Shock Waves* **4**, 25–34.
- Townsend, J. F., Könözy, L. & Jenkins, K. W. (2020), 'On the Development of a Rotated-Hybrid HLL/HLLC Approximate Riemann Solver for Relativistic Hydrodynamics', *Monthly Notices of the Royal Astronomical Society* **496**, 2493–2505.
- Trachenko, K. & Brazhkin, V. V. (2020), 'Minimal quantum viscosity from fundamental physical constants', *Science Advances* **6**(17), eaba3747.
- Tsoutsanis, P. (2019), 'Stencil selection algorithms for WENO schemes on unstructured meshes', *Journal of Computational Physics: X* **4**, 100037.
- Tsoutsanis, P., Kokkinakis, I. W., Könözy, L., Drikakis, D., Williams, R. J. R. & Youngs, D. L. (2015), 'Comparison of structured- and unstructured-grid, compressible and incompressible methods using the vortex pairing problem', *Computer Methods in Applied Mechanics and Engineering* **293**, 207–231.
- van Leer, B. (1979), 'Towards the Ultimate Conservative Difference Scheme. V. A Second-Order Sequel to Godunov's Method', *Journal of Computational Physics* **32**(1), 101–136.
- Wang, P., Abel, T. & Zhang, W. (2008), 'Relativistic hydrodynamic flows using spatial and temporal adaptive structured mesh refinement', *The Astrophysical Journal Supplement Series* **176**, 467–483.
- Woodward, P. R. & Colella, P. (1984), 'The Numerical Simulation of Two-Dimensional Fluid Flow with Strong Shocks', *Journal of Computational Physics* **54**, 115–173.
- Zanotti, O. & Dumbser, M. (2015), 'High order numerical simulations of the Richtmyer-Meshkov instability in a relativistic fluid', *Physics of Fluids* **27**(7).
- Zhang, W. & MacFadyen, A. I. (2006), 'RAM: A Relativistic Adaptive Mesh Refinement Hydrodynamics Code', *The Astrophysical Journal Supplement Series* **164**, 255–279.

-
- Zhang, W., MacFadyen, A. & Wang, P. (2009), 'Three-Dimensional Relativistic Magnetohydrodynamic Simulations of the Kelvin-Helmholtz Instability: Magnetic Field Amplification By a Turbulent Dynamo', *The Astrophysical Journal* **692**(1), L40–L44.
- Zrake, J. & MacFadyen, A. (2013), 'Spectral and Intermittency Properties of Relativistic Turbulence', *The Astrophysical Journal Letters* **763**.