High-Order Central ENO Finite-Volume Scheme for Ideal MHD

A. Susanto[a], L. Ivan[a,*], H. De Sterck[a], C. P. T. Groth[b]

[a]*Department of Applied Mathematics, University of Waterloo, Waterloo, Ontario, Canada N2L 3G1*
[b]*University of Toronto Institute for Aerospace Studies, Toronto, Ontario, Canada M3H 5T6*

Abstract

A high-order accurate finite-volume scheme for the compressible ideal magnetohydrodynamics (MHD) equations is proposed. The high-order MHD scheme is based on a central essentially non-oscillatory (CENO) method combined with the generalized Lagrange multiplier divergence cleaning method for MHD. The CENO method uses *k*-exact multidimensional reconstruction together with a monotonicity procedure that switches from a high-order reconstruction to a limited low-order reconstruction in regions of discontinuous or under-resolved solution content. Both reconstructions are performed on central stencils, and the switching procedure is based on a smoothness indicator. The proposed high-order accurate MHD scheme can be used on general polygonal grids. A highly sophisticated parallel implementation of the scheme is described that is fourth-order accurate on two-dimensional dynamically-adaptive body-fitted structured grids. The hierarchical multi-block body-fitted grid permits grid lines to conform to curved boundaries. High-order accuracy is maintained at curved domain boundaries by employing high-order spline representations and constraints at the Gauss quadrature points for flux integration. Detailed numerical results demonstrate high-order convergence for smooth flows and robustness against oscillations for problems with shocks. A new MHD extension of the well-known Shu-Osher test problem is proposed to test the ability of the high-order MHD scheme to resolve small-scale flow features in the presence of shocks. The dynamic mesh adaptation capabilities of the approach are demonstrated using adaptive time-dependent simulations of the Orszag-Tang vortex problem with high-order accuracy and unprecedented effective resolution.

Keywords: Magnetohydrodynamics, CENO, ENO, High-Order Schemes, Adaptive Mesh Refinement, GLM

1. Introduction

This paper proposes a high-order accurate finite-volume (FV) scheme for the compressible ideal magnetohydrodynamics (MHD) equations. The high-order MHD scheme is based on the central essentially non-oscillatory (CENO) method that was introduced for the compressible Euler equations by Ivan and Groth [1], and has since been extended to the Navier-Stokes equations [2, 3, 4]. The CENO method uses Barth's *k*-exact reconstruction mechanism [5] to obtain high-order solution accuracy in combination with a monotonicity procedure that switches between a high-order reconstruction and a limited low-order reconstruction. Both reconstructions are performed on central stencils, and the switching is based on a smoothness indicator [1]. The hybrid CENO approach is combined in this paper with the generalized Lagrange multiplier (GLM) divergence cleaning method for MHD that was proposed by Dedner et al. [6] to obtain a FV MHD scheme that is high-order accurate in smooth flow regions and robust against spurious oscillations at discontinuities. The proposed high-order accurate MHD scheme has several desirable properties. First, it is suitable for general polygonal grids because Barth's k-exact polynomial reconstruction procedure is inherently multi-dimensional and can be used on general stencils that do not need to be grid-aligned. Second, the scheme can in principle be implemented with arbitrary order. Third, it can be used directly on block-adaptive grids, which can pose significant challenges to MHD schemes due to the $\nabla \cdot \vec{B}$ constraint. And fourth, high-order

*Corresponding Author
Email addresses: asusanto@uwaterloo.ca (A. Susanto), livan@math.uwaterloo.ca (L. Ivan), hdesterck@math.uwaterloo.ca (H. De Sterck), groth@utias.utoronto.ca (C. P. T. Groth)

accuracy is maintained at curved domain boundaries of the body-fitted mesh by employing accurate spline representations for the boundaries that enable high-order accurate surface flux computations. These features constitute significant new developments in high-order finite-volume schemes for MHD. The particular implementation of the proposed scheme that we present in this paper is a highly sophisticated fourth-order accurate parallel MHD method on two-dimensional (2D) dynamically-adaptive multi-block body-fitted quadrilateral grids. Our adaptive 2D body-fitted structured grids are composed of blocks with quadrilateral cells that are organized in a rectangular structure as on a Cartesian grid and, thus, the grid blocks are sometimes also referred to as logically Cartesian. The body-fitted meshes in our work can have grid lines conforming to curved boundary surfaces and stretching of the grid lines is permitted to allow for anisotropic mesh spacing.

Development of high-order numerical methods for MHD is an active area of research. Just like for other nonlinear hyperbolic systems, spurious oscillations at shocks are a major challenge in MHD, but an additional significant challenge in MHD is that the high-order numerical method needs to handle the $\nabla \cdot \vec{B}$ constraint in a proper way. Indeed, it is well-known that simply extending conservation law methods for the Euler equations to the MHD hyperbolic system does not work, since $\nabla \cdot \vec{B}$ may grow in an uncontrolled fashion (beyond truncation error levels), which may result in unphysical forces and numerical instability [7, 8]. A variety of approaches have been proposed to remedy this issue. One option is to employ an elliptic correction scheme, called the "Hodge Projection", which essentially projects a vector field onto its solenoidal part [7, 9]. While the elliptic correction scheme maintains solenoidality up to machine accuracy (in the chosen discretization), it requires a Poisson equation to be solved at each hyperbolic step. This is not natural in a 'hyperbolic' simulation code and can be inconvenient in terms of implementation, especially in parallel since the discrete Poisson potential variables are tightly coupled across the whole computational domain. As an alternative, Powell [8] proposed a divergence control method that only attempts to approximately satisfy the divergence constraint. In Powell's approach, the ideal MHD system is rewritten into its symmetrizable and Galilean-invariant form through the introduction of source terms proportional to the divergence of the magnetic field. This modification maintains the hyperbolic character of the MHD equations, but comes at the cost of conservation, and may lead to incorrect jumps for problems with strong discontinuities [10]. For this reason, this approach has lost some of its initial popularity. A third method to control $\nabla \cdot \vec{B}$ is the class of schemes that fall under the category of 'constrained transport' methods, which preserve the solenoidality of the magnetic field through staggered spatial discretizations [11]. The normal components of the magnetic field are stored on cell faces, and in every time-step the field is updated in such a way that $\nabla \cdot \vec{B}$ remains zero up to machine accuracy (in the chosen discretization). This approach, however, requires the magnetic field variables to be treated differently from the fluid variables, which may be inconvenient for implementation. The approach is attractive from a physics point of view and is straightforward to derive and implement for second-order accurate codes on regular Cartesian grids. It can be extended with second-order accuracy to logically Cartesian grids and to triangular or tetrahedral unstructured grids [12, 13], but extensions beyond second order [14] and to general polygonal grids are far from trivial. In particular, interpolation and restriction of the magnetic field at resolution changes on block-adaptive grids need to be treated very carefully and sophisticated approaches have been developed for this [14, 15, 16].

More recently, Dedner et al. [6] proposed the GLM-based divergence cleaning technique. Through the introduction of a new transport variable, the divergence error is convected out of the domain, while keeping the hyperbolicity of the system intact. Unlike the Powell source term method, conservation in all physical variables is maintained. And unlike approaches following the constrained transport methodology, there is no need to stagger the grid, or to place the magnetic fields at locations different from those where the fluid variables are located. The GLM approach can easily and naturally be applied on general polygonal grids, and there is no need to integrate complicated source terms involving flow variable derivatives as in Powell's approach. A particular version of GLM-MHD (the so-called 'purely hyperbolic' correction) results in numerical solutions that satisfy the $\nabla \cdot \vec{B}$ constraint up to machine accuracy (in the chosen discretization) for stationary problems, and this property can also be obtained for time-dependent solutions [17]. The GLM-MHD approach thus provides an attractive alternative to the more commonly established ways of divergence control, because it is effective in controlling divergence error, is simple to implement, preserves conservation, and can easily be applied on general grids. We choose GLM for our MHD scheme because, combined with the CENO method, it leads to a high-order MHD scheme that can be applied on general grids (including our adaptive multi-block body-fitted structured grid) and naturally handles resolution changes on block-adaptive grids.

Adaptive mesh refinement has proven to be very effective for treating problems with disparate length scales, providing the required spatial resolution while minimizing memory and storage requirements. Recently, Groth and co-researchers [18, 19, 20, 21, 22] have developed a flexible block-based adaptive mesh refinement (AMR) scheme allowing automatic solution-directed mesh adaptation on multi-block body-fitted meshes consisting of two-dimensional quadrilateral and three-dimensional hexahedral computational cells. We have implemented our new high-order MHD scheme that combines CENO and GLM into a hierarchical quadtree block-based AMR procedure for multi-block body-fitted quadrilateral mesh that is based on this previous work [18, 19, 20, 22]. This block-based approach has been shown to enable efficient and scalable parallel implementations for a variety of flow problems, as well as to allow for local refinement of body-fitted mesh with anisotropic stretching. The latter aids in the treatment of complex flow geometry and flows with thin boundary, shear, and mixing layers and/or discontinuities and shocks. Extensions of the block-based body-fitted AMR approach for embedded boundaries not aligned with the mesh [23] and with an anisotropic refinement strategy [24] are also possible and have been developed.

In recent years, various high-order schemes have been proposed for the MHD system. Many recent developments employ discontinuous Galerkin (DG) finite element methods [25, 26, 27, 28], and others are based on essentially non-oscillatory (ENO) FV schemes and on weighted ENO (WENO) FV schemes [13, 29, 30, 31, 32, 33, 34]. Most of these high-order approaches were only described and implemented for regular Cartesian grids. Our high-order MHD scheme uses a different approach. As already mentioned, it is based on Barth's *k*-exact reconstruction procedure [5], which uses a least-squares approach on overdetermined stencils to compute polynomial reconstruction coefficients, in a multi-dimensional way that can handle general polygonal grids. In order to control spurious oscillations at shocks, we use the CENO monotonicity procedure that was introduced by Ivan and Groth [1] for the Euler equations, and has since been extended to the Navier-Stokes equations [2, 3, 4]. Our implementation of this CENO monotonicity procedure switches between an unlimited piecewise cubic reconstruction (fourth-order accurate) and a limited piecewise-linear reconstruction (second-order accurate), with the switching based on the smoothness indicator introduced in [1]. Note that the scheme we describe can in principle be implemented with arbitrary order, but fourth-order accuracy is a suitable practical choice for the numerical results to be presented in this paper. The smoothness indicator is computed in each cell to determine whether the flow is locally smooth and well-resolved. For cells containing non-smooth or under-resolved solution content, the unlimited *k*-exact reconstruction is switched to limited piecewise linear reconstruction. The smoothness indicator can also be used directly to formulate a criterion for AMR. The CENO scheme is called central because both the high-order and the low-order stencils are central with respect to the cell. The method is called an ENO method because it satisfies the ENO property [35]:

$$TV(u^{n+1}) = TV(u^n) + O(\Delta x^{k+1}), \tag{1}$$

where u^n denotes a solution variable u at time level n, Δx is the grid spacing, k is the order of polynomial reconstruction, and TV stands for total variation. The ENO property allows the presence of small spurious oscillations that have a magnitude on the order of the truncation error, but it does not allow $O(1)$ Gibbs-like oscillations at discontinuities [35]. It is important to note that the CENO method proposed by Ivan and Groth [1] does not choose between asymmetric stencils as most other methods do that try to enforce the ENO property, but instead uses a hybrid approach that chooses between high-order and low-order central reconstructions. Note that Harten and Chakravarthy [36] also proposed a technique on Cartesian grids to obtain an ENO reconstruction using central stencils by hybridizing a high-order reconstruction with a first-order formulation, and this served as an inspiration for the CENO approach of [1, 2]. The fixed stencil used during the CENO reconstruction procedure avoids the complexity of considering multiple non-central stencil configurations that characterizes traditional ENO schemes. Note also that the CENO method of Ivan and Groth is not a central ENO method in the sense of Nessyahu and Tadmor's staggered mesh philosophy [37], but it uses non-staggered central stencils of different order. We note that our limited low-order least-squares scheme for MHD with GLM divergence cleaning is similar to the discretization proposed by Yalim et al. in [17] (implemented on unstructured grids), and our high-order method is a high-order extension of this approach that combines least-squares reconstruction with GLM. Our CENO-GLM high-order MHD scheme thus provides an alternative to high-order DG and ENO/WENO methods for MHD, and is attractive because it can naturally be applied on general grids. Also, while some high-order MHD schemes based on DG and WENO have been proposed for unstructured meshes [25, 32], and several others can potentially be (non-trivially) extended to general geometry

[28, 31], none has yet presented a way to deal with curved boundaries. Our CENO implementation for MHD has full capabilities to treat curved domain boundaries with high-order accuracy. Also, few existing high-order MHD codes are parallel or adaptive, and most of them are limited to Cartesian grids. Our new high-order MHD scheme significantly advances the state-of-the-art of high-order finite-volume MHD schemes because it is suitable for general and adaptive grids. This is demonstrated by our parallel dynamically block-adaptive implementation of the new high-order scheme with high-order accurate curved boundaries, which constitutes a significant advance over existing high-order MHD codes.

This paper is organized as follows. The ideal MHD equations and the GLM formulation are described in Sect. 2. In Sect. 3 we give a detailed description of our high-order MHD scheme, which is obtained by combining the CENO method with GLM divergence cleaning. Section 4 describes detailed numerical results that demonstrate high-order convergence for smooth flows, and robustness against oscillations for Riemann problems and other flows with shocks. In particular, we also present a new MHD extension of the well-known Shu-Osher test problem [38] to test the ability of our high-order MHD scheme to resolve small-scale flow features in the presence of shocks. Finally, we demonstrate the dynamic mesh refinement capabilities of our implementation using adaptive time-dependent simulations of the Orszag-Tang vortex problem [39] with high-order accuracy and unprecedented effective resolution. Concluding remarks are presented in Sect. 5.

2. Ideal MHD

2.1. Ideal MHD Equations

The ideal MHD system is described by the following equations in conservation form:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0, \tag{2}$$

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot \left(\rho \vec{v}\vec{v} + \vec{I}(p + \frac{\vec{B} \cdot \vec{B}}{2}) - \vec{B}\vec{B} \right) = 0, \tag{3}$$

$$\frac{\partial \vec{B}}{\partial t} + \nabla \cdot (\vec{v}\vec{B} - \vec{B}\vec{v}) = 0, \tag{4}$$

$$\frac{\partial e}{\partial t} + \nabla \cdot \left((e + p + \frac{\vec{B} \cdot \vec{B}}{2})\vec{v} - (\vec{v} \cdot \vec{B})\vec{B} \right) = 0. \tag{5}$$

Equations 2 to 5 are supplemented with a solenoidality condition for the magnetic field,

$$\nabla \cdot \vec{B} = 0. \tag{6}$$

The conserved quantities of the ideal MHD equation system are the density, ρ, the momentum, $\rho\vec{v}$ (with $\vec{v}$ being the velocity), the magnetic field, $\vec{B}$, and the energy, e. The plasma pressure, p, is given by the equation of state for a perfect gas

$$p = (\gamma - 1)\left(e - \frac{1}{2}\rho|\vec{v}|^2 - \frac{1}{2}|\vec{B}|^2 \right) \tag{7}$$

where γ is the adiabatic index. We use $\gamma = 5/3$ in our numerical tests except where noted.

The ideal MHD equations can also be written in quasi-linear form, which, in one dimension, is given as

$$\frac{\partial \mathbf{U}}{\partial t} + \mathbb{A}(\mathbf{U}) \cdot \frac{\partial \mathbf{U}}{\partial x} = 0, \tag{8}$$

where $\mathbf{U} = (\rho, \rho\vec{v}, \vec{B}, e)^T$. The matrix $\mathbb{A}$ can then be used to determine the eigenvalues of the system, which are given as follows:

$$\lambda_{1,2} = v_x \pm c_{f_x}, \tag{9}$$

$$\lambda_{3,4} = v_x \pm c_{A_x}, \tag{10}$$

$$\lambda_{5,6} = v_x \pm c_{s_x}, \tag{11}$$

$$\lambda_7 = v_x, \tag{12}$$

$$\lambda_8 = 0, \tag{13}$$

where the three different wave speeds are: the fast magnetosonic wave speed, c_{f_x}, the Alfvén wave speed, c_{A_x}, and the slow magnetosonic wave speed, c_{s_x}, defined as

$$c_{f_x}^2 = \frac{1}{2}\left(\frac{\gamma p + B^2}{\rho} + \sqrt{\left(\frac{\gamma p + B^2}{\rho}\right)^2 - 4\frac{\gamma p B_x^2}{\rho^2}}\right), \tag{14}$$

$$c_{A_x}^2 = \frac{B_x^2}{\rho}, \tag{15}$$

$$c_{s_x}^2 = \frac{1}{2}\left(\frac{\gamma p + B^2}{\rho} - \sqrt{\left(\frac{\gamma p + B^2}{\rho}\right)^2 - 4\frac{\gamma p B_x^2}{\rho^2}}\right). \tag{16}$$

2.2. GLM Control of the $\nabla \cdot \vec{B}$ Constraint

The GLM-MHD formulation can be described as follows. Following a similar approach as for the Maxwell equations [40], the divergence constraint (Eq. 6) can be coupled with the induction equation through the introduction of a new potential variable, ψ [6]. The equations describing the evolution of the magnetic field, Eq. 4 and Eq. 6, are then replaced with the following equations

$$\frac{\partial \vec{B}}{\partial t} + \nabla \cdot (\vec{v}\vec{B} - \vec{B}\vec{v}) + \nabla\psi = 0, \tag{17}$$

$$\mathbb{D}(\psi) + \nabla \cdot \vec{B} = 0. \tag{18}$$

Different choices of the operator $\mathbb{D}(\psi)$ determine whether the corrections are of so-called elliptic, parabolic or hyperbolic type. Dedner et al. [6] found that the hybrid hyperbolic and parabolic correction scheme provides the best balance of accuracy and stability, while at the same time keeping the system conservative in the physical variables. The hybrid hyperbolic-parabolic correction scheme defines $\mathbb{D}(\psi)$ as

$$\mathbb{D}(\psi) = \frac{1}{c_h^2}\frac{\partial \psi}{\partial t} + \frac{1}{c_p^2}\psi. \tag{19}$$

With $\mathbb{D}(\psi)$ as given by Eq. 19, the induction equation and the divergence constraint equation can be rewritten as

$$\frac{\partial \psi}{\partial t} + c_h^2 \nabla \cdot \vec{B} = -\frac{c_h^2}{c_p^2}\psi. \tag{20}$$

As can be seen from these equations, the system is still conservative except for the evolution equation of ψ, which is not a physical variable. This preservation of conservation for physical variables is the main advantage of the GLM method over the Powell method that was proposed earlier to approximately satisfy the divergence constraint [41]. Replacing the zero eigenvalue (Eq. 13), two new eigenvalues arise in the GLM-MHD formulation, which are $\pm c_h$. The coefficients c_p and c_h control the amount of diffusion in ψ and the advection speed, respectively. The 'purely hyperbolic' correction can be obtained by taking c_p to infinity (no diffusion). Following [6], we choose these two coefficients to be related in our numerical simulations through the following expression

$$c_r = \frac{c_p^2}{c_h}, \tag{21}$$

with c_r chosen to be 0.18. The coefficient c_h, then, determines how fast the divergence of the magnetic field is advected out of the domain, whereas c_p controls its dissipation. Therefore, to ensure that the error is advected as fast as possible, it is desirable to set c_h as high as possible. However, because the two new eigenvalues have magnitude c_h, it is also important to set c_h small enough so that it will not affect the time-step criterion of the simulation. Thus, c_h is often chosen to be the largest of all MHD eigenvalues in the whole domain over all cell interfaces (i, j), which can be written as

$$c_h = \max_{i,j} \left(|v_n| + c_{f_n} \right), \tag{22}$$

where v_n and c_{f_n} are the plasma velocity and the fast magnetosonic wave speed (Eq. 14) in the direction normal to the interfaces (i, j).

2.3. Boundary Condition Treatment of ψ at Inflow and Outflow Boundaries

The choice of c_h as given by Eq. 22 ensures that no eigenvalue will exceed the largest physical eigenvalue in the domain, while at the same time, it guarantees that the divergence error will be advected out of the simulation domain with the fastest physical wave speed in the flow solution. Since the two additional eigenvalues are $\pm c_h$ regardless of the actual plasma velocities and wave speeds, eigenvalues of both signs will always exist at all cell interfaces. This means that treatment similar to subsonic inlet and outlet boundary conditions (see [42]) is always required for inflow and outflow boundary conditions. Since the waves with eigenvalues $\pm c_h$ only carry changes in the normal magnetic field and ψ [6], only these two variables need to be taken into account at boundaries to accommodate these waves. For example, consider superfast inflow boundary conditions and assume without loss of generality that $v_n > 0$. Since the inflow velocity is faster than the fast magnetosonic wave, all the MHD eigenvalues are positive (information travels into the computational domain). However, for GLM-MHD one cannot just prescribe all variables, because one of the eigenvalues, $-c_h$, is necessarily negative, even when the flow is superfast at the inflow boundary. One of either ψ or the normal magnetic field has to be extrapolated from the interior solution, and because the inflow magnetic field is prescribed at the boundary, it is ψ that has to be extrapolated from the interior. The same logic applies to superfast outflow. Assume again that $v_n > 0$. Without GLM, all of the variables would just be extrapolated from the inside of the domain, since all eigenvalues are positive, hence no information is propagating into the domain. However, due to the negative eigenvalue $-c_h$, ψ needs to be prescribed at the outflow boundary. A suitable choice for ψ is to set it to zero at superfast outflow boundaries. (This is consistent with Yalim et al. [17], who set ψ to a constant at the superfast outlet boundaries.) In our experience, the specification of boundary conditions has proven to be important in properly applying the GLM method. We emphasize the proper treatment of inflow and outflow boundary conditions here because this was not fully explained in [6]. Several MHD test cases with superfast inflow and outflow boundary conditions [43] will be investigated in what follows to assess the accuracy and stability of the scheme we propose in the next section.

3. High-Order CENO Scheme for Ideal MHD with GLM Divergence Cleaning

In this section we give a detailed description of the proposed high-order CENO scheme for MHD, which is obtained by combining Ivan and Groth's CENO approach with GLM divergence cleaning. We first describe the high-order FV framework, followed by discussions on Barth's k-exact reconstruction and the CENO reconstruction selection process using the CENO smoothness indicator. Numerical flux calculation and source term integration for our high-order MHD CENO method are described next, followed by a discussion on how our MHD CENO implementation obtains high-order accuracy at curved boundaries.

3.1. High-Order Finite-Volume Formulation

Consider hyperbolic conservation law

$$\frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot \vec{\mathbf{F}} = \mathbf{S}, \tag{23}$$

where $\mathbf{U}$ is the vector of conserved variables, $\vec{\mathbf{F}}$ consists of the flux terms of the system, and $\mathbf{S}$ is a source term vector. As mentioned before, our MHD CENO scheme can be applied on general grids with polygonal cells, but for

simplicity we give the formulas for the case of quadrilateral cells in a body-fitted structured grid. For a quadrilateral cell (i, j), the semi-discrete FV form of Eq. 23 is given as

$$\frac{d\overline{\mathbf{U}}_{i,j}}{dt} = -\frac{1}{A_{i,j}} \sum_{l=1}^{4} \sum_{m=1}^{N_g} (\omega \vec{\mathbf{F}}_{num} \cdot \vec{n} \Delta l)_{i,j,l,m} + \overline{\mathbf{S}}_{i,j}, \tag{24}$$

where $\overline{\mathbf{U}}_{i,j}$ is the numerical approximation of the average value of $\mathbf{U}$ in cell (i, j), $\vec{\mathbf{F}}_{num}$ is the numerical flux function, $A_{i,j}$ is the area of the computational cell (i, j), N_g is the number of Gauss quadrature points on each cell face and ω is the associated Gauss quadrature weight to each of the Gauss points. The actual number of flux quadrature points, N_g, depends on the order of solution reconstruction, with two Gauss quadrature points per face for third- and fourth-order accurate schemes, but only one Gauss quadrature point per face for second-order or lower [1]. The high-order accurate calculation of the average source term, $\overline{\mathbf{S}}_{i,j}$, requires the accurate integration of $\mathbf{S}$ and is discussed in Sect. 3.4. The form given by Eq. 24 separates the spatial and temporal discretizations, which essentially reduces the system of partial differential equations (PDEs) to a system of ordinary differential equations (ODEs) in time for each cell. The order of the polynomial reconstruction then determines the spatial accuracy of the solution by providing more accurate approximations of the solution values at the Gauss quadrature points for flux calculation. In general, an order-k polynomial reconstruction provides an order-$(k + 1)$ accurate spatial discretization for smooth problems. We use standard explicit second-order and fourth-order Runge-Kutta methods [42] to integrate the ODE system in time for the second-order and fourth-order accurate spatial discretizations to be compared in our time-dependent numerical test problems. For steady-state simulations, we use a five-stage optimally smoothing method regardless of the solution accuracy [44].

3.2. k-Exact Piecewise Polynomial Reconstruction

The spatial order of accuracy of the CENO FV scheme is determined by the order of the polynomial function used to reconstruct the solution. Following Barth [5], the variation of a solution variable, u, at any location within the quadrilateral computational cell (i, j), assumes the form

$$u^k_{i,j}(\vec{X}) = \sum_{\substack{p_1=0 \\ (p_1+p_2\leq k)}}^{k} \sum_{p_2=0}^{k} (x - \overline{x}_{i,j})^{p_1} (y - \overline{y}_{i,j})^{p_2} D^k_{p_1 p_2}, \tag{25}$$

where k is the order of the polynomial function, $\vec{X} = (x, y)$ are the coordinates at which the solution is sought, $(\overline{x}_{i,j}, \overline{y}_{i,j})$ are the coordinates of the centroid of cell (i, j), and $D^k_{p_1 p_2}$ are high-order polynomial coefficients that will need to be determined for each of the primitive variables for every cell, based on a set of cell averages, $\overline{u}_{\gamma,\delta}$, in the neighbourhood of cell (i, j). For the test cases presented in this paper, linear $(k = 1)$ and cubic $(k = 3)$ reconstructions are chosen to obtain second- and fourth-order accurate schemes. The monotonicity-preserving procedure, which is discussed in Sect. 3.3, reduces k to 1 and applies limiters in regions of the flow that are deemed under-resolved or to contain discontinuities.

The coefficients $D^k_{p_1 p_2}$ are determined by solving an overdetermined system of linear equations in a least-squares sense, fitting the reconstruction polynomial to the solution averages for cell (i, j) and for its neighbouring cells in the stencil of cell (i, j). For a k^{th}-order polynomial, the number of coefficients $D^k_{p_1 p_2}$ is given by $\mathcal{N}_D = \frac{(k+1)(k+2)}{2}$ [1, 2, 4]. Thus, there are 3 coefficients to be determined for $k = 1$ or linear reconstruction, 6 coefficients for $k = 2$ or quadratic reconstruction, and 10 coefficients for $k = 3$ or cubic reconstruction. Following the requirements imposed by Barth [5], it is important that these coefficients are determined in such a way that the following conditions are satisfied:

- **Conservation of the mean.** The average of the reconstructed polynomial function over cell (i, j) should recover exactly the cell-averaged value $\overline{u}_{i,j}$:

$$\overline{u}_{i,j} = \frac{1}{A_{i,j}} \iint_{\mathcal{A}_{i,j}} u^k_{i,j}(\vec{X})\, dA. \tag{26}$$

- k**-exactness.** The reconstructed polynomial function should be able to reconstruct polynomials of degree k or less exactly [5], or

$$u^k_{i,j}(\vec{X}) - u_{\text{exact}}(\vec{X}) = O(\Delta x^{k+1}). \tag{27}$$

- **Compact support.** The reconstructed polynomial function should depend only on average values within a relatively small neighbourhood [5]. Only the cell-averaged data within the supporting stencil is used for reconstruction purposes.

In theory, $\mathcal{N}_D$ determines the minimum size of the supporting stencil, but in practice more neighbours are included to make the reconstruction more robust for complicated and stretched meshes [1, 2, 4]. On our 2D body-fitted structured grid blocks, first-degree neighbours are included for $k = 0$ and $k = 1$ reconstruction stencils (a total of 8 neighbours), and first- and second-degree neighbours are included for $k = 2$ and $k = 3$ reconstruction stencils (a total of 24 neighbours).

Consider reconstruction for cell (i, j). In the reconstruction step an overdetermined system $\mathbb{A}\mathbf{D} - \mathbf{B} = 0$ is solved in the least-squares sense, together with the constraint of Eq. 26, which is imposed exactly. Here, $\mathbf{D}$ is the array of polynomial coefficients, $D^k_{p_1 p_2}$, and the equations $\mathbb{A}\mathbf{D} - \mathbf{B} = 0$ are given by

$$(\mathbb{A}\mathbf{D} - \mathbf{B})_{\gamma,\delta} = \left(\frac{1}{A_{\gamma,\delta}} \iint_{\mathcal{A}_{\gamma,\delta}} u^k_{i,j}(\vec{X})\, dA \right) - \overline{u}_{\gamma,\delta} = 0. \tag{28}$$

There is one equation for each cell (γ, δ) in the stencil of cell (i, j). Each equation matches the actual cell average $\overline{u}_{\gamma,\delta}$ in cell (γ, δ) with the average over cell (γ, δ) of the reconstructed polynomial $u^k_{i,j}(\vec{X})$ for cell (i, j). Equation 26 is enforced analytically by replacing $u^k_{i,j}$ with Eq. 25 and expressing the first coefficient, D^k_{00}, as a function of the other $M = \mathcal{N}_D - 1$ polynomial unknowns as

$$D^k_{00} = \overline{u}_{i,j} - \sum_{\substack{p_1=0 \\ (p_1+p_2\neq 0)}}^{k} \sum_{p_2=0}^{k} D^k_{p_1 p_2} \left(\overline{x^{p_1} y^{p_2}} \right)_{i,j} , \tag{29}$$

where the geometric moment $\left(\overline{x^{p_1} y^{p_2}} \right)_{i,j}$ of powers (p_1, p_2) is given by

$$\left(\overline{x^{p_1} y^{p_2}} \right)_{i,j} = \frac{1}{A_{i,j}} \iint_{\mathcal{A}_{i,j}} \left(x - \overline{x}_{i,j} \right)^{p_1} \left(y - \overline{y}_{i,j} \right)^{p_2} \, dA \, . \tag{30}$$

Substituting $u^k_{i,j}$ from Eq. 25 in Eq. 28 and using Eq. 29 for D^k_{00} the following overdetermined linear system for the M unknowns is obtained

$$\begin{bmatrix} L_1 \\ L_2 \\ \vdots \\ L_J \\ \vdots \\ L_{N_n} \end{bmatrix}_{M \times N_n} \begin{pmatrix} D^k_{01} \\ D^k_{02} \\ \vdots \\ D_{p_1 p_2} \\ \vdots \\ D^k_{k0} \end{pmatrix}_{N_n \times 1} - \begin{pmatrix} w_1(\overline{u}_1 - \overline{u}_I) \\ w_2(\overline{u}_2 - \overline{u}_I) \\ \vdots \\ w_J(\overline{u}_J - \overline{u}_I) \\ \vdots \\ w_{N_n}(\overline{u}_{N_n} - \overline{u}_I) \end{pmatrix}_{M \times 1} = \begin{pmatrix} 0 \\ 0 \\ \vdots \\ 0 \\ \vdots \\ 0 \end{pmatrix}_{M \times 1} \tag{31}$$

where a unique index $J = (\gamma, \delta)$ has been assigned to each of the N_n neighbours in the supporting reconstruction stencil and the index $I = (i, j)$ denotes the cell having the solution reconstructed. The generic row L_J of the matrix $\mathbb{A}$ for a neighbouring cell J is given by

$$L_J = \begin{pmatrix} w_J \left(\widehat{x^0 y^1} \right)_{IJ} & w_J \left(\widehat{x^0 y^2} \right)_{IJ} & \dots & w_J \left(\widehat{x^{p_1} y^{p_2}} \right)_{IJ} & \dots & w_J \left(\widehat{x^k y^0} \right)_{IJ} \end{pmatrix} , \tag{32}$$

in which w_J is a geometric weight specific to each neighbour J which serves the purpose of improving the locality of

the reconstruction, becoming especially important for stretched meshes with boundary curvature [45]. (In essence, equations corresponding to close-by neighbour cells in the reconstruction stencil get larger weights in the least-squares solution than neighbour cells that are further away.) The matrix coefficients $\left(\widehat{x^{p_1}y^{p_2}}\right)_{IJ}$ for the pair of I and J cells have the expression

$$\left(\widehat{x^{p_1}y^{p_2}}\right)_{IJ} = \left(\frac{1}{A_{\gamma,\delta}} \iint_{\mathcal{A}_{\gamma,\delta}} \left(x - \overline{x}_{i,j}\right)^{p_1} \left(y - \overline{y}_{i,j}\right)^{p_2} \, dA\right) - \left(\overline{x^{p_1}y^{p_2}}\right)_{i,j} \ , \tag{33}$$

where the quantities $\left(\widehat{x^{p_1}y^{p_2}}\right)_{IJ}$ depend only on the geometry of I and J cells, and involve a monomial integration that can be computed by applying quadrature rules. An efficient way to calculate the geometric moments $\left(\widehat{x^{p_1}y^{p_2}}\right)_{IJ}$ using only the $\left(\overline{x^{p_1}y^{p_2}}\right)$ moments is described in [2].

QR factorization or multiplication with the pseudo-inverse of $\mathbb{A}$ can be used to determine the solution of Eq. 31, as described in more detail in [1, 2, 4]. The complete solution of the constrained least-squares problems is then obtained by calculating D_{00} using Eq. 29. In each time step, the constrained least-squares reconstruction problem is solved for each cell and for each primitive variable. Matrix $\mathbb{A}$ depends completely on the geometry and is the same for all least-squares problems in a given cell (i, j) (i.e., for each solution variable) and for all time steps, so it can be precomputed and stored for computational efficiency (see [1, 2, 4] for details). As explained in Sect. 3.5, one-sided stencils and additional constraints on the least-squares solution are used to handle boundary conditions with high-order accuracy at curved boundaries.

3.3. CENO Smoothness Indicator to Enforce Monotonicity

The CENO method controls monotonicity throughout the computational domain by selecting a limited linear reconstruction in cells where the flow is deemed to be nonsmooth or under-resolved, and a high-order k-exact reconstruction elsewhere. The limited linear reconstruction is based on k-exact reconstruction with $k = 1$ combined with the standard Venkatakrishnan limiter, see [2, 46]. To estimate whether the flow in cell (i, j) is under-resolved or non-smooth, a variable $\mathcal{S}$, the smoothness indicator, is computed [1]:

$$\mathcal{S} = \frac{\alpha \, c_s}{\max(1 - \alpha, \epsilon)}, \tag{34}$$

where α is given by

$$\alpha = 1 - \frac{\sum_{\gamma} \sum_{\delta} \left(u^k_{\gamma,\delta}(\vec{X}_{\gamma,\delta}) - u^k_{i,j}(\vec{X}_{\gamma,\delta})\right)^2}{\sum_{\gamma} \sum_{\delta} \left(u^k_{\gamma,\delta}(\vec{X}_{\gamma,\delta}) - \overline{u}_{i,j}\right)^2} \tag{35}$$

and $c_s = (\mathcal{N}_{SOS} - \mathcal{N}_D)/(\mathcal{N}_D - 1)$ is a positive constant. Here, $\mathcal{N}_{SOS}$ stands for 'size of stencil' used for reconstruction, $\mathcal{N}_D$ stands for 'degrees of freedom' and denotes the number of unknown polynomial coefficients, and ϵ is introduced to avoid division by zero (we use $\epsilon = 10^{-8}$). Further, γ and δ denote the indices of the neighbouring cells to the cell (i, j) that are part of its reconstruction stencil, and $\vec{X}_{\gamma,\delta}$ is the centroid of cell (γ, δ). (Note that the stencil used for computing the smoothness indicator can also be chosen smaller than the reconstruction stencil. In our numerical results, we compute the smoothness indicator associated with each primitive solution variable in cell (i, j) using a stencil with nine cells, i.e., the cell (i, j) and its eight first-degree neighbours.) The parameter α basically measures how accurately centroidal solution values of neighbouring cells can be reproduced using the reconstruction for cell (i, j). The range of α is $-\infty < \alpha \leq 1$: for smooth variation, the second term of the right-hand side of Eq. 35 tends to be close to zero and α is very close to one; for cells close to a discontinuity or with an under-resolved feature, the magnitude of α tends away from one and it can also become negative. The range of the smoothness indicator $\mathcal{S}$ is $-c_s < \mathcal{S} < c_s/\epsilon$: for smooth variation ($\alpha$ very close to one), $\mathcal{S}$ is large; for nonsmooth or under-resolved features (α away from one), $\mathcal{S}$ is small. The smoothness indicator $\mathcal{S}$ is then compared with a cutoff value $\mathcal{S}_C$: when $\mathcal{S} > \mathcal{S}_C$ the solution is deemed locally smooth and the high-order reconstruction is used, and for $\mathcal{S} \leq \mathcal{S}_C$ the solution is locally nonsmooth or under-resolved, and the limited low-order reconstruction is used. We also use $\mathcal{S}$ in our adaptive

procedure to refine regions where the solution is nonsmooth or under-resolved. A potential disadvantage of this approach is that it is not fully parameter-free. However, we have found it easy to pick suitable values of $\mathcal{S}_C$ based on the range recommended in [2] for the numerical tests shown in Sect. 4. Additionally, it is worth emphasizing that a single value $\mathcal{S}_C$ is selected and applied to all solution variables and all mesh resolutions used for solving a particular problem. The form of the smoothness indicator is inspired by the definition of multiple-correlation coefficients and least-squares goodness-of-fit testing; see [2] for a more detailed discussion with further motivation for the approach. As is shown by extensive testing for the Euler and Navier-Stokes equations in [1, 2, 3, 4] and is further confirmed by the numerical MHD tests presented below, the CENO approach with smoothness indicator $\mathcal{S}$ is robust in terms of providing high-order accurate numerical approximations while avoiding spurious oscillations.

One more element has to be added to the approach in order to get good results for problems with uniform regions. In uniform regions, the formula for α in Eq. 35 may lead to $0/0$ in the second term of the right-hand side, rendering the smoothness indicator unpredictable. It has been observed before [47] in a related context that it is desirable to eliminate the effect of switching mechanisms altogether in nearly uniform regions, and just use high-order reconstruction. To do so, we define the newly proposed parameter

$$\xi_{i,j} = \sqrt{\sum_{\substack{p_1=0 \\ (0<p_1+p_2\le k)}}^{k} \sum_{p_2=0}^{k} (D^k_{p_1 p_2})^2 (A_{i,j})^{p_1+p_2}}, \tag{36}$$

which measures the variability of solution variable u in cell (i, j). (It takes into account all the derivatives at the centroid of cell (i, j).) When $\xi_{i,j}$ is smaller than a threshold value (low variability), high-order reconstruction is always used, and only when $\xi_{i,j}$ is greater than the threshold the smoothness indicator is computed and the CENO switching mechanism is activated. In particular, the smoothness indicator for the solution variable u is evaluated in cell (i, j) when

$$\xi_{i,j} > \epsilon_A + \epsilon_R \overline{u}_{i,j}, \tag{37}$$

where ϵ_A and ϵ_R represent absolute and relative variability thresholds, chosen to be 10^{-5} for the simulations performed in this paper.

3.4. Numerical Flux Function and High-Order Accurate Source Term Integration

In this subsection we discuss numerical flux computation and high-order accurate treatment of the GLM source term (in Eq. 20) for the high-order MHD CENO scheme. We use the Lax-Friedrichs numerical flux function for the implementation of the proposed high-order MHD CENO scheme. Following Dedner et al. [6], the equations for B_x and ψ are decoupled from the rest of the system, so the Lax-Friedrichs numerical fluxes are applied only to the other seven variables. The fluxes at the interfaces for B_x and ψ are calculated by having these variables assume the following values at the cell interfaces [6]:

$$B_{x,m} = \frac{1}{2}(B_{x,r} + B_{x,l}) - \frac{1}{2\,c_h}(\psi_r - \psi_l), \tag{38}$$

$$\psi_m = \frac{1}{2}(\psi_r + \psi_l) - \frac{c_h}{2}(B_{x,r} - B_{x,l}), \tag{39}$$

where the subscripts l and r denote the left and right reconstructed states at cell interfaces and c_h is the global maximum of $|v_x| + c_{f_x}$ at cell interfaces. These values are substituted directly into the exact flux formulas for the B_x and ψ equations. In a multi-dimensional setting, B_x is effectively B_n, which is the magnetic field component normal to the interface. These ψ_m and $B_{n,m}$ values are also used for flux calculation purposes of the other seven variables, which uses the Lax-Friedrichs numerical flux with local values of $|v_x| + c_{f_x}$ as the largest wave speed that determines the size of the numerical dissipation.

As an alternative, one can also apply the standard Lax-Friedrichs flux directly to the full system with nine variables, without decoupling the 2×2 system. One can expect this to be more diffusive since in this case c_h (the global maximum of $|v_x| + c_{f_x}$) determines the numerical diffusion, but we have not found much difference with the

decoupled approach when trying this for our numerical tests. Nevertheless, in the numerical results presented below we use the decoupled approach. Other flux functions such as Roe and HLLE can also be considered. Wheatley et al. [48] compared flux functions for high-order DG methods, and found that using more accurate Riemann solvers improves results in some cases (e.g., at shocks), but does often not make much difference in smooth regions of the flow. This is also expected for CENO since the intercellular solution jumps diminish in size as the reconstruction order increases, but a detailed investigation of this for the CENO MHD scheme is beyond the scope of this paper.

The ideal (non-modified) MHD system (Eq. 2 - Eq. 5) is a hyperbolic system of equations, so it easily fits within the CENO framework. However, the GLM-MHD formulation adds a source term to the ψ-update equation (Eq. 20), and it is important to ensure that this source term is integrated with high-order accuracy. Dedner et al. [6] suggest that source terms be incorporated separately using an operator splitting approach. By solving the resulting ODE exactly, the source term integration step can be made unconditionally stable (thus, adding no extra time step restriction to the hyperbolic system). In Sect. 17.5 of [49], LeVeque explains that such source term treatment leads to an order of accuracy that is at most second-order accurate in time. This makes the operator-splitting technique for handling the source term not desirable for high-order purposes. Another mechanism to treat the source term is by integrating it directly. Though some types of source terms might pose a challenge, the source term of the hyperbolic-parabolic GLM-MHD equation system involves only the variable ψ, so integrating it with high-order accuracy turns out to be straightforward. Indeed, integrating Eq. 20 over cell (i, j) gives

$$\frac{d}{dt}\left(\iint_{\mathcal{A}_{i,j}} \psi \, dA\right) = -c_h^2\left(\iint_{\mathcal{A}_{i,j}} \nabla \cdot \vec{B} \, dA\right) - \frac{c_h^2}{c_p^2}\left(\iint_{\mathcal{A}_{i,j}} \psi \, dA\right), \tag{40}$$

which directly leads to the discrete equation

$$\frac{d\overline{\psi}_{i,j}}{dt} = -\frac{1}{A_{i,j}} \sum_{l=1}^{4} \sum_{m=1}^{N_g} (\omega \vec{\mathbf{f}}_{num} \cdot \vec{n} \Delta l)_{i,j,l,m} - \frac{c_h^2}{c_p^2} \overline{\psi}_{i,j}, \tag{41}$$

with $\vec{\mathbf{f}}_{num}$ the numerical flux function for Eq. 20. This is a high-order discretization of Eq. 20 as long as the fluxes are computed with high-order accuracy. In summary, discretizing the source term of Eq. 20 with high-order accuracy is easy because integrating ψ in the source term of Eq. 20 directly leads to the average value $\overline{\psi}_{i,j}$ in cell (i, j), which is one of the primary variables in the code stored in cell (i, j). The source term in discrete Eq. 41 can in principle influence the stability bound and allowable time-step for the ODE system, but a straightforward analysis shows that it does not limit the allowable time-step beyond the usual hyperbolic CFL condition for the values of c_h and c_p and the grid resolutions we use in our test problems. This was confirmed in all our numerical experiments.

3.5. High-Order Accuracy at Curved Boundaries

In our CENO MHD implementation, two general mechanisms are available to prescribe boundary conditions. The first mechanism uses ghost cells. Every grid block in our hierarchical block-adaptive body-fitted quadrilateral grid framework is equipped with three or four layers of ghost cells. In the numerical results to be presented in Sect. 4, we compare second-order results with fourth-order results. The second-order simulations employ three layers of ghost cells for each block, and the fourth-order results employ four layers of ghost cells for each block. All blocks have the same size, and the parallelization strategy distributes blocks over parallel MPI processes as uniformly as possible (with typically multiple blocks per MPI process and one MPI process per CPU core), resulting in adequate load balancing. The ghost cells enable the message passing that parallelizes the code. They are also used in the adaptivity mechanism to transfer information between coarse and fine blocks, as is explained in Sect. 4.3. Note that the number of ghost cell layers is one greater than the number required to enable reconstruction in the first ghost cell layer; this additional ghost cell layer is necessary for computing the smoothness indicator in the first layer of ghost cells (which determines whether the high-order or low-order reconstruction is used there) [2]. The ghost cells can also be used to impose boundary conditions at the domain boundaries in standard ways. All second-order simulations use ghost cells to impose boundary conditions. Ghost cells are also used to impose boundary conditions for our fourth-order tests in certain cases, for example in the case of periodic boundary conditions. However, for

high-order accuracy near curved boundaries, a more accurate second mechanism for boundary conditions is needed.

The second boundary condition mechanism relies on accurate representation of the curved boundaries with high-order piecewise polynomial splines, on the use near boundaries of one-sided stencils that only contain cells within the computational domain, and on additional constraints imposed on the least-squares reconstruction problem at the Gauss points [50]. It is also important to compute the geometric data such as cell areas, centroid locations, etc. to the same order of accuracy as that of the interior scheme [2, 4]. We represent curved boundaries with piecewise polynomial splines of an order consistent with that of the FV numerical scheme, which allows us to locate Gauss quadrature points and compute flux integrals with high accuracy. One-sided reconstruction stencils are used for the first and second layer of cells in the computational domain at the boundaries, and constraints are added to the least-squares reconstruction of the cells in the first layer to accurately impose certain types of boundary conditions on the curved boundaries at the Gauss points. When ghost cells are not used, the one-sided reconstructed values at the Gauss points are directly plugged into the exact MHD flux functions to obtain the numerical flux. For variables to be left free at the boundaries (extrapolation from the computational domain), no additional constraints are necessary. For variables to be imposed at the boundaries, the appropriate constraints are added at the Gauss points used in the flux integration.

More generally, our framework accepts Robin boundary conditions (linear combination of Dirichlet and Neumann), and also linear relations among variables which form a coupling constraint for a set of reconstructed variables [2]. These coupling constraints can be used to impose wall conditions at curved boundaries with high accuracy. This has been explained for Euler flows in [1, 2], and we extend it here to perfectly conducting walls in MHD problems. For perfectly conducting walls, we impose that $\vec{B}\cdot\vec{n}=0$ and $\vec{v}\cdot\vec{n}=0$ in each Gauss quadrature point. Let (n_x^g, n_y^g) be the normal vector in Gauss quadrature point g of cell (i, j), and let (x_g, y_g) be its coordinates. Then, using the polynomial expansion of Eq. 25, the conditions $\vec{B}\cdot\vec{n}=0$ and $\vec{v}\cdot\vec{n}=0$ at the Gauss point can be expressed as

$$\sum_{\substack{p_1=0\\(p_1+p_2\le k)}}^{k}\sum_{p_2=0}^{k}(x_g-\overline{x}_{i,j})^{p_1}(y_g-\overline{y}_{i,j})^{p_2}n_x^g\,(D^k_{p_1p_2})_{B_x}+\sum_{\substack{p_1=0\\(p_1+p_2\le k)}}^{k}\sum_{p_2=0}^{k}(x_g-\overline{x}_{i,j})^{p_1}(y_g-\overline{y}_{i,j})^{p_2}n_y^g\,(D^k_{p_1p_2})_{B_y}=0, \tag{42}$$

and

$$\sum_{\substack{p_1=0\\(p_1+p_2\le k)}}^{k}\sum_{p_2=0}^{k}(x_g-\overline{x}_{i,j})^{p_1}(y_g-\overline{y}_{i,j})^{p_2}n_x^g\,(D^k_{p_1p_2})_{u}+\sum_{\substack{p_1=0\\(p_1+p_2\le k)}}^{k}\sum_{p_2=0}^{k}(x_g-\overline{x}_{i,j})^{p_1}(y_g-\overline{y}_{i,j})^{p_2}n_y^g\,(D^k_{p_1p_2})_{v}=0 \tag{43}$$

with $(D^k_{p_1p_2})_{B_x}$ the polynomial coefficients for the B_x magnetic field component, and similar for the B_y, u and v vector components. To impose $\vec{B}\cdot\vec{n}=0$ in the reconstruction, we solve the least-squares reconstruction problems for the B_x and B_y polynomials together, with the additional constraints of Eq. 42 for each Gauss point. Similarly, the least-squares reconstruction problems for u and v are solved together to impose $\vec{v}\cdot\vec{n}=0$, with the additional constraints of Eq. 43. For full implementation details, see [2].

4. Numerical Results

In this section we present numerical results that demonstrate high-order convergence for smooth flows and robustness against oscillations for flows with shocks. We present four continuous test problems followed by two problems with discontinuities, including a new MHD extension of the well-known Shu-Osher test problem [38]. Finally, we demonstrate the dynamic AMR capabilities of our implementation using adaptive time-dependent simulations of the Orszag-Tang vortex problem [39] with high-order accuracy and unprecedented effective resolution.

4.1. Continuous Problems

We first present two smooth test problems on Cartesian grids, which are the rotated Alfvén problem from [10], and the magnetostatic problem from [25]. We then present two continuous test problems on body-fitted multi-block structured grids with non-rectangular cells and curved boundaries: the rotating radial outflow problem and the expanding tube problem from [43].

To quantify the accuracy of the numerical solution, the errors are measured in the L_1, L_2, and L_∞ norms:

$$L_1 = |E|_1 = \frac{1}{A_T} \sum_{i,j} \iint_{\mathcal{A}_{i,j}} \left| u^k_{i,j}(x,y) - u_{exact}(x,y) \right| \, dA, \tag{44}$$

$$L_2 = |E|_2 = \sqrt{\frac{1}{A_T} \sum_{i,j} \iint_{\mathcal{A}_{i,j}} \left[u^k_{i,j}(x,y) - u_{exact}(x,y) \right]^2 \, dA}, \tag{45}$$

$$L_\infty = |E|_\infty = \max_{i,j} \left(\frac{1}{A_{i,j}} \iint_{\mathcal{A}_{i,j}} \left| u^k_{i,j}(x,y) - u_{exact}(x,y) \right| \, dA \right), \tag{46}$$

where A_T is the total area of the computational domain. The integrals are evaluated with high-order accurate Gaussian quadrature, see [2] for details. In most of our numerical tests, we compare convergence for four numerical methods: fourth-order CENO, fourth-order unlimited k-exact reconstruction, second-order CENO, and second-order unlimited k-exact reconstruction. The CENO methods switch between the k-exact reconstruction and the limited piecewise linear reconstruction based on the smoothness indicator. We use a smoothness indicator cut-off value $\mathcal{S}_C = 800$ except where noted.

4.1.1. Rotated Alfvén Travelling Wave Propagation

The circularized Alfvén wave problem from [10] represents analytical solutions of the MHD equations for arbitrary amplitudes. The wave propagates with an angle of $\alpha = 30°$ with respect to a Cartesian grid, and assumes the initial conditions (as in [10]): $\rho = 1$, $v_\parallel = 0$, $p = 0.1$, $B_\parallel = 1$, $v_\perp = B_\perp = 0.1 \sin(2\pi(x\cos(\alpha) + y\sin(\alpha))$, and $v_z = B_z = 0.1 \cos(2\pi(x\cos(\alpha) + y\sin(\alpha)]$. The parallel velocity, $v_\parallel$, is set to zero, which corresponds to the travelling wave test case. The perpendicular and parallel directions are defined with respect to the direction of wave propagation. These initial conditions give an Alfvén speed of 1, which corresponds to a transit period of 1. The computational domain is set to be periodic (using ghost cells), with ranges [0,1/cos(α)] for x, and [0,1/sin(α)] for y. As in [10], the number of cells in the x-direction is equal to the number of cells in the y-direction, which corresponds to a ratio of $1/\sqrt{3}$ between Δx and Δy. The simulations are run for 5 transit periods (or up to $t = 5$). Density and other scalar variables are expected to be constant throughout the simulation since they are not perturbed by the Alfvén wave, and the errors for these variables are much smaller than for $\vec{v}$ and $\vec{B}$, which is consistent with the finding of Toth [10]. Thus, only the accuracy of the $\vec{v}$ and $\vec{B}$ fields were assessed for convergence studies.

As can be seen from Fig. 1, the expected order of convergence is achieved for the x-direction magnetic field, at least in the asymptotic limit. For the sake of brevity, only the results of the x-direction magnetic field are shown, but the other variables behave in a similar manner. The effect of the CENO monotonicity-preserving reconstruction switching procedure ($\mathcal{S}_C = 800$) can be seen: a "transition" regime occurs where the mesh is not fine enough and the smooth flow features are not sufficiently resolved. This transition regime does not occur for the fourth-order method because it sufficiently resolves the flow already with low resolution, see also [2, 4]. The high-order scheme represents significant savings in the number of computational cells required for some specific level of accuracy: a 64-by-64 grid resolution was sufficient for the fourth-order scheme to obtain a smaller error than the limited second-order scheme on a 384-by-384 grid.

4.1.2. Two-Dimensional Magnetostatic Problem

We next consider the magnetostatic problem from [25]. The exact solution of this stationary problem is known: $\rho = 1, v_x = 0, v_y = 0, v_z = 0, B_x = -\cos(\pi x)e^{-\pi y}, B_y = \sin(\pi x)e^{-\pi y}, B_z = 0, p = 19.84\,(\gamma - 1), \psi = 0$. Following Warburton et al. [25], this exact solution is used as the initial condition for the simulation, and the error at steady-state is a measure of the deviation of the numerical solution from the exact solution. The second-order methods use ghost cells to impose boundary conditions: $\vec{v}$ and $\vec{B}$ are imposed in the ghost cells (accurate average values of the exact solution, obtained by numerical quadrature), and ρ, p and ψ are extrapolated to the ghost cells (the average values are linearly extrapolated). The fourth-order methods use one-sided reconstruction, with the exact values of $\vec{v}$ and $\vec{B}$ imposed at the Gauss points using constraints, and ρ, p and ψ are left free at the Gauss points. Figure 2 shows how the error norm of B_x converges to zero as a function of grid size with the expected order. The fourth-order

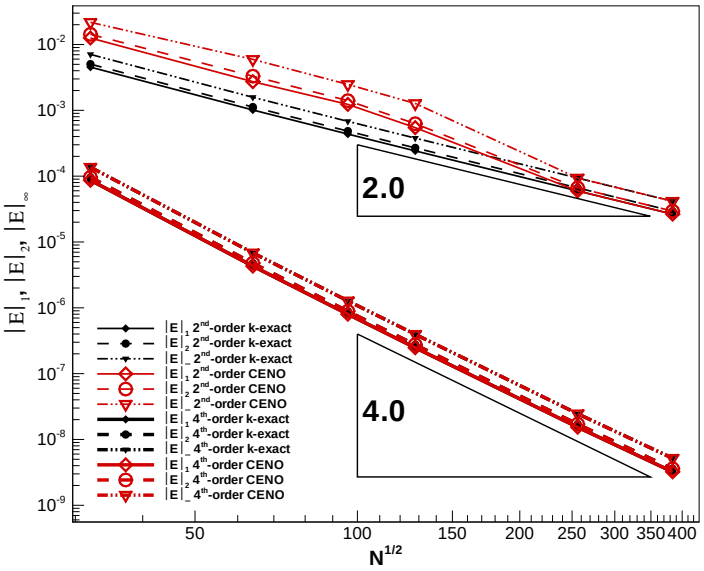


Figure 1: The L_1-, L_2-, and L_∞-norm errors for the magnetic field in the x-direction for the rotated Alfvén wave problem, calculated at $t = 5$ (five transit periods). N is the total number of grid cells. The solution is compared with the initial conditions to compute the error. The error converges to zero with the expected order of accuracy in the asymptotic limit. A transition region is observed for the second-order CENO scheme, consistent with the findings of Ivan and Groth [2, 4].

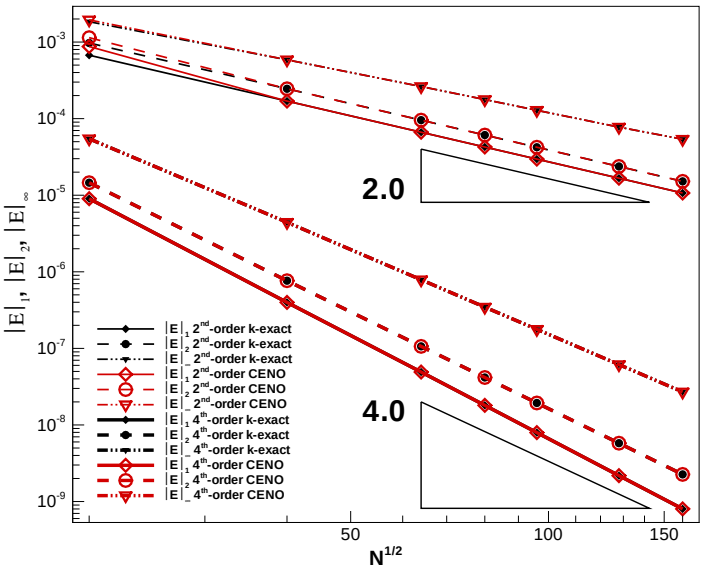


Figure 2: The L_1-, L_2-, and L_∞-norm errors for the magnetic field in the x-direction for the magnetostatic problem. The steady-state solution is compared with the initial conditions to compute the error. The error converges to zero with the expected order of accuracy in the asymptotic limit.

scheme requires much fewer computational cells to achieve a specified level of error (in this case, the error can differ by as much as 4 orders of magnitude for the same number of cells).

4.1.3. Superfast Rotating Outflow From a Cylinder

We next consider the rotated outflow problem from [43] on a body-fitted structured grid with non-rectangular cells and curved boundaries. While the exact analytical solution is not available, several theoretical flow invariants are available [43], with which the corresponding computed quantities can be compared. We measure error in the entropy, s, and the radial magnetic field, B_r.

The problem is defined on a domain between two concentric circles, and superfast inflow conditions (normal

velocity faster than the fast magnetosonic wave speed as given in Eq. 14) are imposed at the inner circle. The domain goes from $r = 1$ to $r = 6$, and the inflow conditions imposed at the $r = 1$ boundary are $\rho = 1$, $p = 1$, $v_r = 3$, $v_\theta = 1$, and $B_r = 1$. The second-order methods use ghost cells to impose boundary conditions. At the inner boundary (inflow), ρ, p, $\vec{v}$ and $\vec{B}$ are imposed in the ghost cells using linear interpolation (to impose the desired values exactly at the domain boundary), and ψ is extrapolated linearly from the interior of the domain. At the outer boundary (outflow), ρ, p, $\vec{v}$ and $\vec{B}$ are extrapolated linearly, and ψ is set to zero using linear interpolation (to impose the desired value exactly at the domain boundary). The fourth-order methods use high-order piecewise polynomial spline representation of the curved boundaries, combined with one-sided reconstruction and constraints. At the inner boundary (inflow), ρ, p, $\vec{v}$ and $\vec{B}$ are imposed by constraints at the Gauss points, and ψ is left free. At the outer boundary (outflow), ψ is set to zero by constraints at the Gauss points, and ρ, p, $\vec{v}$ and $\vec{B}$ are left free.

The steady-state solution of the rotated outflow problem obtained with the fourth-order CENO scheme on a mesh with 80-by-80 cells can be seen in Fig. 3. The magnetic field lines are not aligned with the streamlines. The solutions obtained with second- and fourth-order CENO schemes are compared in Fig. 4. It can be seen that the errors converge to zero with the expected order of accuracy. For this problem, the second-order scheme has not reached the asymptotic regime beyond the transition region yet for the resolutions we tested, and the second-order CENO error remains above the unlimited second-order error due to ongoing switching from unlimited k-exact reconstruction to limited second-order reconstruction, especially for the error in the radial magnetic field as seen in Fig. 4(b). This is possibly due to the inability of the piecewise linear function to capture the curvature of the boundaries properly, so the switching procedure continues to see some cells close to the boundaries as under-resolved, thus limiting the reconstruction functions at these places and affecting the magnitude of the error. In contrast, for fourth-order CENO and the unlimited k-exact schemes produce the same error for resolutions above 80-by-80. It is clear that our approach can handle curved boundaries with high-order accuracy, and the fourth-order method requires significantly fewer cells than the second-order method to obtain a given error level.

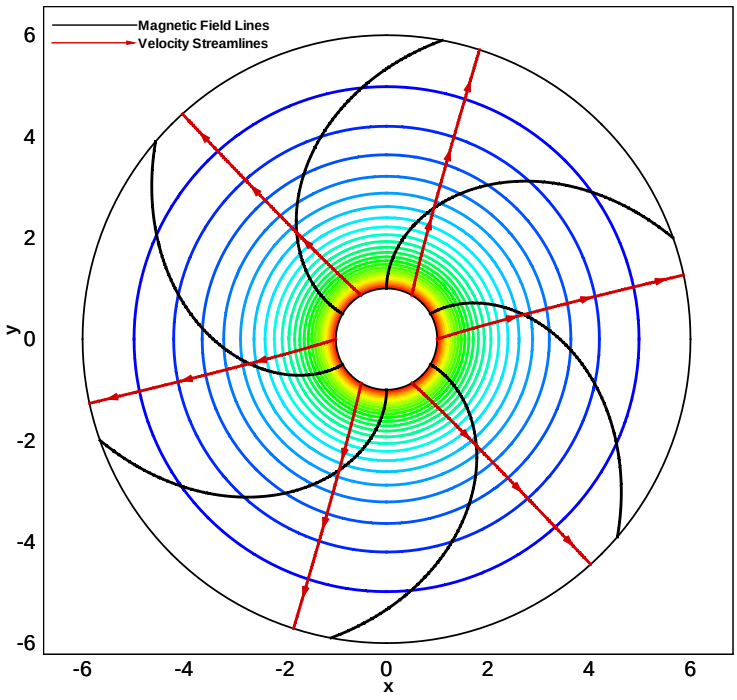
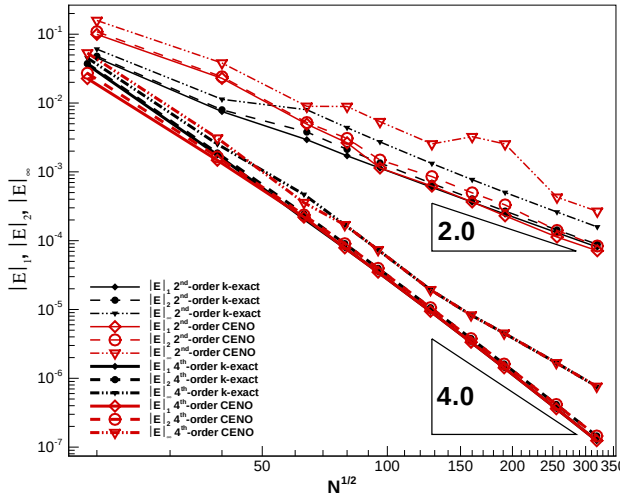
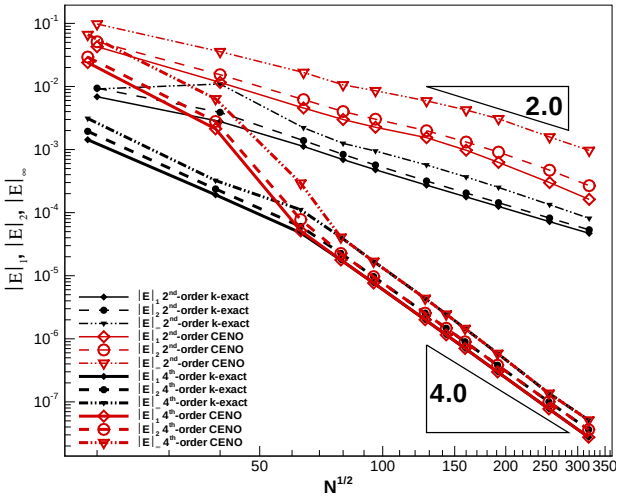


Figure 3: Density contour lines, magnetic field lines, and streamlines for the rotated outflow problem, obtained on a mesh with 80-by-80 cells. The magnetic field lines and the streamlines are not aligned. The flow is smooth throughout the entire domain, which enables high-order convergence rates. The density contour lines are equally spaced in the range (0.17,0.97).



(a) The L_1-, L_2-, and L_∞-norm errors for entropy, which is constant in the domain.



(b) The L_1-, L_2-, and L_∞-norm errors for the radial magnetic flux. While there is no analytical solution for the full magnetic field, the radial component of the magnetic field can be determined, due to the conservation of the radial magnetic flux.

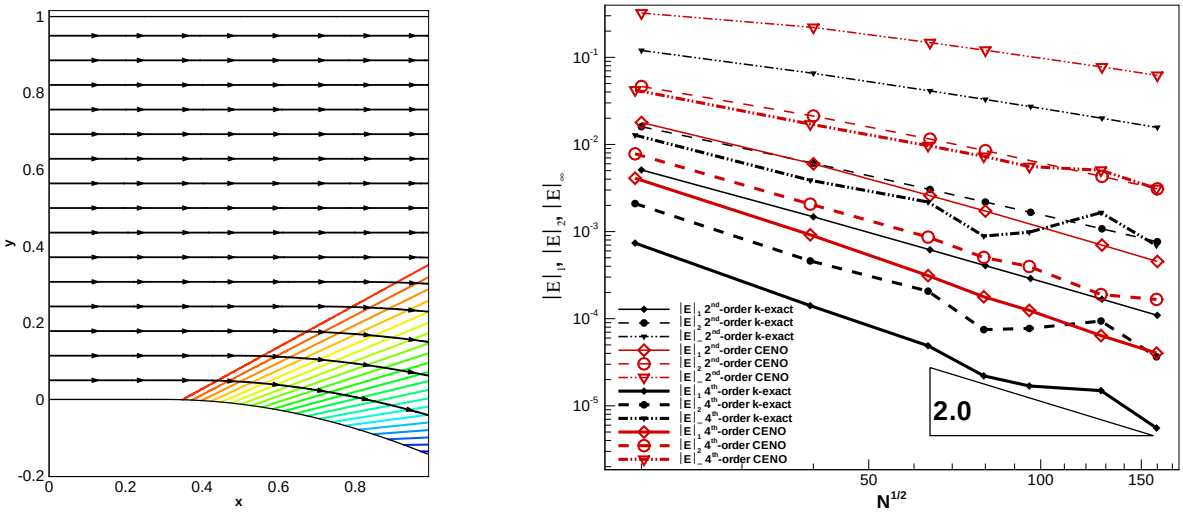
Figure 4: Convergence study for the rotated outflow problem using both unlimited k-exact reconstruction (black lines) and CENO with $\mathcal{S}_C = 800$ (red lines).

4.1.4. Expanding Tube Problem

The expanding tube problem from [43] is another continuous problem that uses a body-fitted structured grid with curved boundaries. It models plasma flow in an expanding tube, which gives rise to an MHD solution that contains a rarefaction wave with a weak discontinuity at the edge of the rarefaction (see Fig. 5). Across the weak discontinuity, the first spatial derivative of the flow variables is discontinuous. Therefore, even if fourth-order reconstruction accuracy is targeted (by employing degree-3 polynomial functions to reconstruct the solution), the solution accuracy is still limited to second-order near the weak discontinuity.

The flow is simulated on a domain with $x \in [0, 1]$, and $y \in [y_0(x), 1]$, where $y_0(x) = \cos\left(\frac{\pi}{4}(x - 0.3)\right) - 1$ for $x \in [0.3, 1]$, and zero elsewhere in the domain. The lower wall starts to curve at $x = 0.3$, giving rise to a rarefaction wave downstream of the weak discontinuity. The boundary curve follows a cosine function rather than a straight line to avoid a geometrical singularity in the boundary, which results in the rarefaction wave not converging to a single point, as can be seen from Fig. 5(a) [43]. At the $x = 0$ boundary, a uniform inflow with the following conditions is imposed: $\rho = 1$, $p = 1$, $v_x = 8$, and $B_x = 4$. These initial conditions correspond to superfast horizontal inflow conditions, with an acoustic Mach number $v_x/c = 8\sqrt{3/5}$, and Alfvénic Mach number $v_x/c_{A_x} = 2$. The second-order methods use ghost cells to impose boundary conditions. The superfast inflow and outflow boundary conditions at the left and right boundaries, respectively, are implemented as for the rotating outflow test problem of Sect. 4.1.3. For the top and bottom boundaries, standard wall boundary conditions are implemented that symmetrically copy ρ, p and ψ to the ghost cells, and mirror $\vec{v}$ and $\vec{B}$ with respect to the wall. The fourth-order methods use high-order piecewise polynomial spline representation of the wall boundaries, combined with one-sided reconstruction and constraints. For the top and bottom wall boundaries, ρ, p, ψ and the tangential components of $\vec{v}$ and $\vec{B}$ are left free, while the normal components of $\vec{v}$ and $\vec{B}$ are set to zero at the Gauss points using constraints. The high-order outflow boundary condition is handled as in the rotating outflow test problem of Sect. 4.1.3. We simply use ghost cells for the high-order inflow boundary condition, since the flow remains uniform close to the inflow boundary and ρ, p, $\vec{v}$ and $\vec{B}$ can just be imposed in all ghost cell layers, while ψ can be extrapolated linearly.

To assess the accuracy of the solution, entropy, which is one of the invariants for this flow, is measured. Figure 5(b) shows convergence analysis of the entropy error. As can be seen in this figure, second-order accuracy is achieved for the L_1-norm error of the entropy for both the second-order and the fourth-order accurate methods. While Fig. 5(b) illustrates how the weak discontinuity in the solution limits the order of accuracy, reduction in the total error is still observed when higher-order polynomial functions (the fourth-order method) are used to represent the solution. It is



(a) The expanding tube flow, solved on a 160-by-160 grid with the fourth-order CENO scheme. The contour lines are equally spaced in the range (0.45,0.95) (21 contours).

(b) Convergence study for the expanding tube problem. The entropy is compared with the entropy at the inflow to compute the error, and nearly second-order accuracy for the L_1-norm error is observed for both the second- and fourth-order methods.

Figure 5: Expanding tube flow: density contour lines and entropy convergence study. The error converges with at most second-order accuracy, due to the non-existence of higher-order derivatives across the weak discontinuity. Convergence study is performed for entropy using both unlimited k-exact reconstruction (black lines) and CENO with $\mathcal{S}_C = 800$ (red lines).

interesting to note that the log error of the CENO solution decreases linearly, whereas some zigzagging is present in the unlimited k-exact error plot. This can be explained by the fact that the flow is not fully smooth, and the weak discontinuity that exists can potentially generate spurious oscillations when monotonicity is not enforced, though the level at which these oscillations occur is apparently much smaller than the solution variation. Note also that, even at the highest attempted resolution, the convergence plots of the CENO error do not converge to those of unlimited k-exact reconstruction (as was the case for the other test cases), implying that, due to the weak discontinuity, reconstruction switching is always performed for at least a few cells.

4.2. Problems with Discontinuities

4.2.1. Rotated Brio-Wu Shock Tube Problem

The Brio-Wu shock tube problem [51] is a standard test case to demonstrate the capability of a numerical MHD scheme to handle discontinuities. The initial conditions are given by

$$(\rho, v_\perp, v_\parallel, v_z, B_\perp, B_\parallel, B_z, p, \psi) = \begin{cases} (1, 0, 0, 0, 0.75, 1, 0, 1, 0) \text{ for } x_1 < 0, \\ (0.125, 0, 0, 0, 0.75, -1, 0, 0.1, 0) \text{ for } x_1 > 0, \end{cases} \tag{47}$$

with $\gamma = 2$. Here, x_1 is the coordinate variable perpendicular to the shock, given by $x_1 = x\cos\alpha + y\sin\alpha$ with α the angle at which the shock frame of reference is rotated with respect to the x-axis (we choose α =45°). This setup is illustrated in Fig. 6.

Ghost cells are used and constant extrapolation boundary conditions are applied to all boundaries, though the top and the bottom boundaries require that the cells not only be copied to the ghost cells, but also shifted to the left or the right by one cell (similar to Fig. 10 from [10]). It is important to note that, for this boundary condition to work, the ratio between the spacing in the x-direction and the spacing in the y-direction needs to be 1, because otherwise the 45° symmetry would not translate to a (1,1) translational symmetry, see also [10, 34, 52].

The Brio-Wu problem gives rise to several types of waves and shocks: fast rarefaction waves, a contact discontinuity, a slow compound wave, and a slow shock [51]. Simulations were performed for the rotated cases using 600 cells in the x-direction, and 4 cells in the y-direction. The density plot is shown in Fig. 7, and illustrates that our method is robust with respect to spurious oscillations. The fourth-order solution has slightly sharper features than

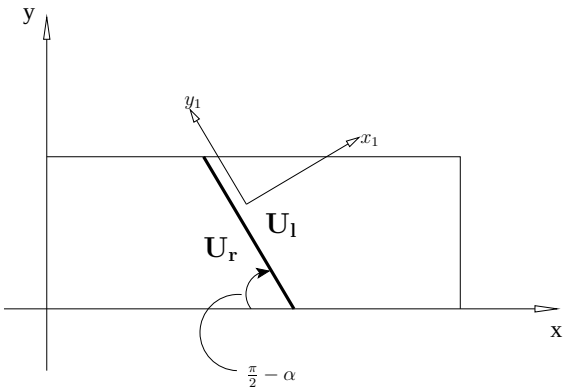


Figure 6: Setup for the rotated one-dimensional problems with discontinuities. The discontinuity is rotated 45° counterclockwise with respect to the x-axis. The solution thus exhibits a translational symmetry in the y_1-direction.

the second-order solution. All of the important wave features are captured well without spurious oscillations, except for a slight undershoot between the fast rarefaction (FR) and the slow compound wave (SM) (which is also observed in other work on high-order MHD schemes [28, 34]).

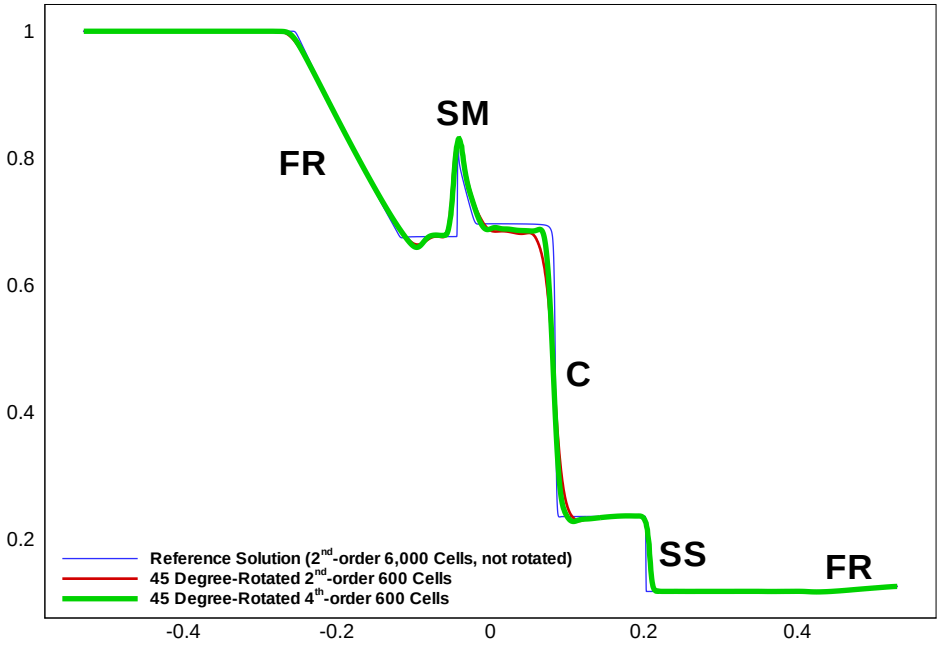


Figure 7: Comparison of the density solution of the Brio-Wu Shock Tube Problem at $t = 0.1414$, rotated at 45°. Here, FR denotes fast rarefaction, SM slow compound wave, C contact discontinuity, and SS slow shock. A cutoff value of $\mathcal{S}_C = 8,000$, is chosen for these simulations.

4.2.2. MHD Extension to Shu-Osher Shock Tube Problem

The shock tube problem proposed by Shu and Osher [38] is commonly used to test the ability of high-order numerical schemes to resolve small-scale flow features in the presence of shocks. A sinusoidal density perturbation is added downstream of a purely advecting supersonic shock wave. The interaction of the shock wave with the sinusoidal part of the density field gives rise to fast oscillations and complex flow features downstream to the shock. The Shu-Osher shock tube problem provides an excellent testbed to highlight the benefits of the improved accuracy of high-order numerical schemes, while at the same time the presence of the shock puts the robustness and stability of the schemes to test. In what follows, we develop a new MHD version of the Shu-Osher shock tube problem.

First consider a fast travelling shock wave without the sinusoidal density perturbation. The fast shock wave advects with shock speed s, satisfying the Rankine-Hugoniot condition:

$$s(\mathbf{U_r} - \mathbf{U_l}) = \mathbf{F}(\mathbf{U_r}) - \mathbf{F}(\mathbf{U_l}) \tag{48}$$

where $\mathbf{U_r}$ and $\mathbf{U_l}$ denote the state vectors of the right and left state, respectively, and $\mathbf{F}(\mathbf{U_i})$ denotes the flux evaluated at state i. The stable purely advecting fast shock that is desired for the MHD equivalent of the Shu-Osher shock tube problem needs to satisfy the following conditions in addition to the Rankine-Hugoniot condition:

1. In the shock frame, the flow should move in the direction from low pressure to high pressure to ensure that entropy increases across the shock;
2. In the shock frame, velocities normal to the shock need to be faster than the fast magnetosonic wave speed (Eq. 14) upstream, and faster than the Alfvén wave speed (Eq. 15) but slower than the fast magnetosonic wave speed downstream;
3. In the simulation frame of reference, the normal velocity downstream of the shock should be zero so the density perturbation stays intact until the shock goes through it;
4. The magnetic field normal to the shock should be continuous to ensure zero magnetic field divergence.

In the shock frame, the shock is stationary ($s = 0$), so the Rankine-Hugoniot condition (Eq. 48) simplifies to

$$\mathbf{F}(\mathbf{U_l}) = \mathbf{F}(\mathbf{U_r}). \tag{49}$$

We choose the following initial conditions that satisfy conditions 1-4 and the Rankine-Hugoniot condition:

$$(\rho, u_\perp, u_\parallel, u_z, B_\perp, B_\parallel, B_z, p, \psi) = \begin{cases} (1, 0, 0, 0, 1, 1, 0, 1, 0) \text{ for } x < 4, \\ (3.5, 5.8846, 1.1198, 0, 1, 3.6359, 0, 42.0267, 0) \text{ for } x > 4. \end{cases} \tag{50}$$

The numbers in Eq. 50 were obtained by numerically solving the MHD Rankine-Hugoniot condition, and were rounded to four decimal digits (which is sufficiently accurate for the numerical tests). These initial conditions lead to a shock that travels unperturbed to the left with a speed of 8.2385 (rounded).

Equation 50 represents the unperturbed portion of our newly proposed MHD version of the Shu-Osher shock tube problem. Similar to the Shu-Osher problem, sinusoidal perturbation is added to the downstream part of the density field (because the shock and the flow travel to the left, the sinusoidal perturbation is added to ρ_l). The initial density function is then chosen as

$$\rho_l = 1 + 0.2\sin(5x), \ \rho_r = 3.5 \tag{51}$$

and all the other variables are kept as given in Eq. 50.

As in the case of the rotated Brio-Wu problem, the initial condition given by Eq. 50 and 51 has been applied in the rotated frame of reference $x_1 - y_1$ (see Fig. 6). The boundary conditions for our simulation of this problem are as explained in Sect. 4.2.1. The left and right boundaries are taken sufficiently far from the initial discontinuity, such that they do not influence the solution. The comparison of the density profiles between the different methods is shown in Fig. 8. The benefit of the high-order method is clear: using the same number of cells, the fourth-order method captures the small-scale flow features much better than the second-order method. For all simulations performed for this section, no stability or overshoot problem were observed, which indicates that the monotonicity-preserving mechanism is doing its job properly to ensure that the method is stable in the presence of discontinuities.

4.3. Application of CENO with Dynamic Adaptive Mesh Refinement: Orszag-Tang Vortex Problem

In this section we demonstrate the dynamic AMR capabilities of our implementation using adaptive time-dependent simulations of the Orszag-Tang vortex problem [28, 31, 39] with high-order accuracy and unprecedented effective resolution. The problem is challenging because it is time-dependent and contains multiple complex and interacting discontinuities. The Orszag-Tang vortex problem is a good test for our dynamic adaptive mesh refinement and coarsening procedure.

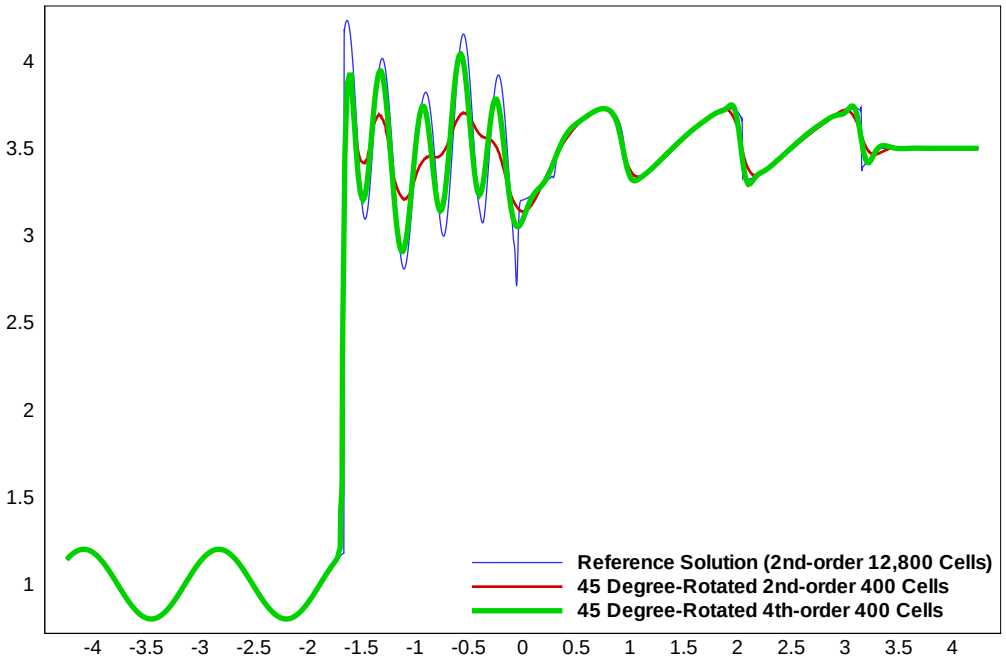


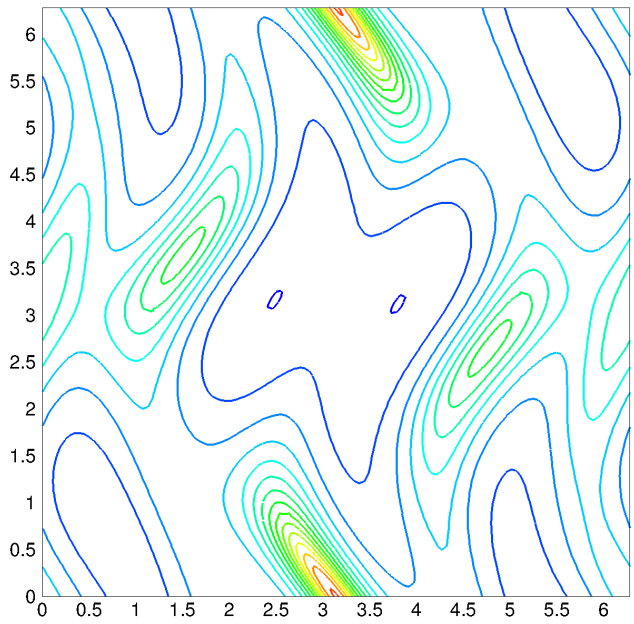
Figure 8: Comparison of the density solution of the MHD Shu-Osher problem at $t = 0.6906$, rotated at 45°. As can be observed from the figure, the fourth-order method produces results that are much closer to the non-rotated reference result in the highly oscillatory region, illustrating the benefits of high-order accuracy. A cutoff value of $\mathcal{S}_C = 80$ is used for these simulations.

We have implemented our new high-order MHD scheme that combines CENO and GLM into a hierarchical quadtree block-based AMR procedure for multi-block body-fitted quadrilateral mesh that is based on the previous work of Groth and co-workers [18, 19, 20, 22] and is extended to high-order accuracy as in [1, 4]. We give a brief summary of the approach, and details are described in [1, 2, 19]. In our hierarchical quadtree block-based AMR algorithm, mesh adaptation is accomplished by dividing and coarsening appropriate solution blocks. In regions requiring increased cell resolution, a 'parent' block is refined by dividing it into four 'children'. Each of the four quadrants or sectors of a parent block becomes a new block having the same number of cells as the parent, thereby doubling the cell resolution in the region of interest. This process can be reversed in regions that are deemed over-resolved and four children can be coarsened into a single parent block. The mesh refinement is constrained such that the grid resolution changes by at most a factor of two between adjacent blocks, and the minimum resolution is not less than that of the initial mesh. A hierarchical quadtree data structure and additional interconnects between the 'leaves' of the trees are used to keep track of mesh refinement and the connectivity between solution blocks. The hybrid CENO solution reconstruction procedure is used in conjunction with standard multigrid-type restriction and interpolation operators to evaluate the solution on all blocks created by the coarsening and division processes. Interpolation is performed with high-order accuracy by computing reconstructed polynomials for solution variables in each coarse-grid cell and integrating them over the fine-grid children cells to determine the fine-grid cell averages with high-order accuracy (see [2] for details). Restriction and interpolation are performed in such a way that conservation is maintained, but in our CENO-GLM MHD approach no special treatment is required for restricting or interpolating the cell-centred magnetic fields: restriction or interpolation may introduce errors of the order of the discretization error, and they are handled properly by the GLM mechanism for controlling $\nabla \cdot \vec{B}$.

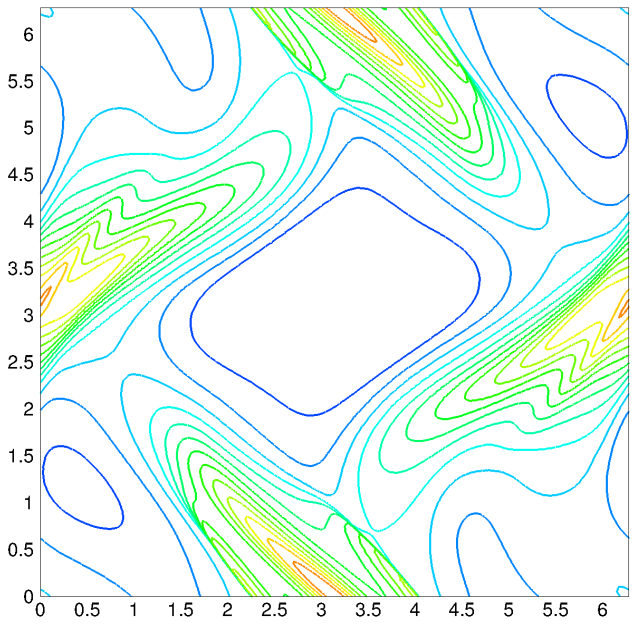
Grid refinement and coarsening are based on the maximum value of the CENO smoothness indicator over each block for the density variable. For each cell, the variable

$$\mathcal{R}_c = e^{-\frac{\max(0,\mathcal{S})}{\mathcal{S}_c}} \tag{52}$$

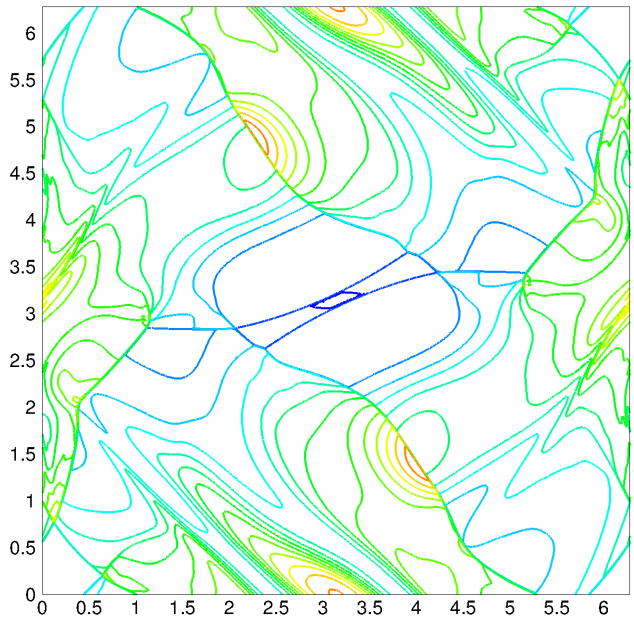
is calculated, where $\mathcal{S}$ is the value of the smoothness indicator and $\mathcal{S}_c$ is the cutoff value for the smoothness indicator. The range of $\mathcal{R}_c$ is (0,1]. The maximum $\mathcal{R}_c^B$ of all $\mathcal{R}_c$ values within a block is computed. In blocks with $\mathcal{R}_c^B$ close to



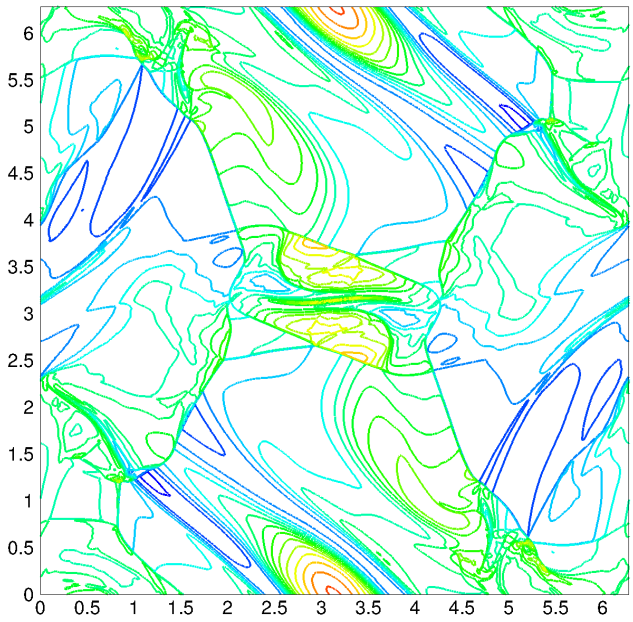
(a) Density solution at $t = 0.5$. The contour lines are equally spaced in the range (2.11,5.82) (15 contours).



(b) Density solution at $t = 1.0$. The contour lines are equally spaced in the range (1.25,6.9) (15 contours).



(c) Density solution at $t = 2.0$. The contour lines are equally spaced in the range (0.62,6.41) (15 contours).

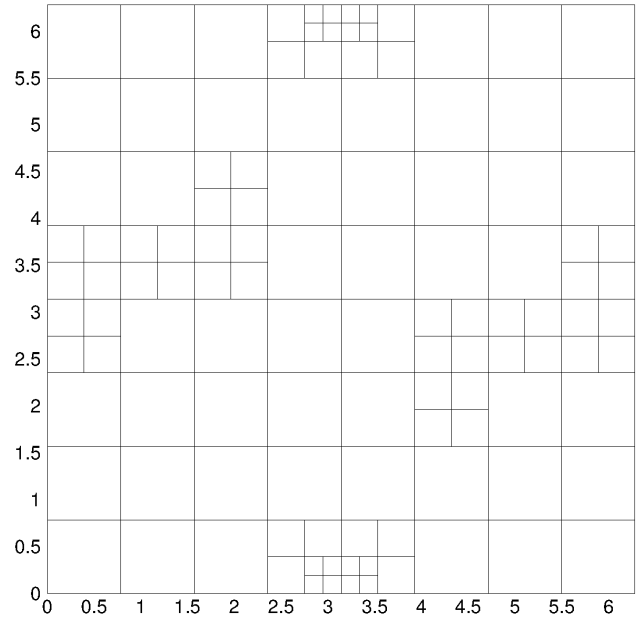


(d) Density solution at $t = 3.0$. The contour lines are equally spaced in the range (1.16,6.42) (15 contours).

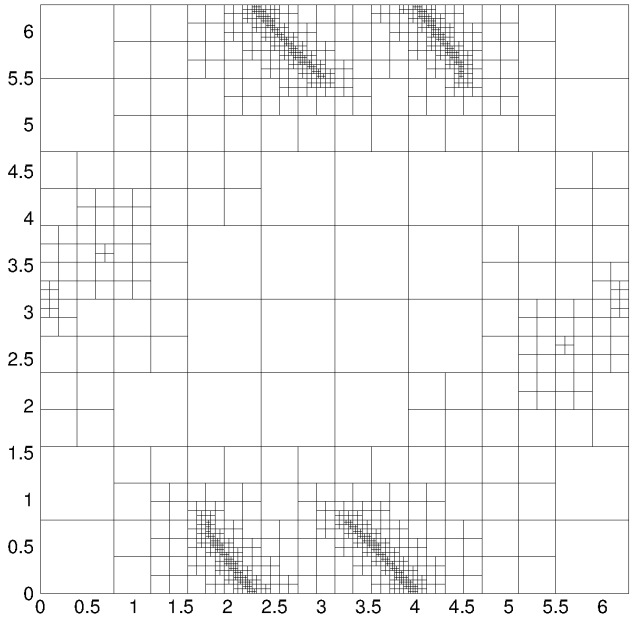
Figure 9: The evolution of density for the Orszag-Tang vortex problem at different times: $t = 0.5$, $t = 1.0$, $t = 2.0$, and $t = 3.0$. The ranges for the contour lines shown here are as in [28]. These fourth-order accurate results were obtained using dynamic grid adaption with the meshes shown in Fig. 10.

0, all cells are smooth and resolved, and blocks with $\mathcal{R}_c^B$ close to 1 have cells that are nonsmooth or under-resolved. The block-based $\mathcal{R}_c^B$ values are compared with refinement and coarsening thresholds to determine if a block should undergo refinement, or if a group of blocks should be combined for coarsening. Full details on the algorithm followed for coarsening and refinement are given in [19] and [2]. The refinement/coarsening algorithm is invoked at regular intervals during the simulation to obtain dynamic AMR.

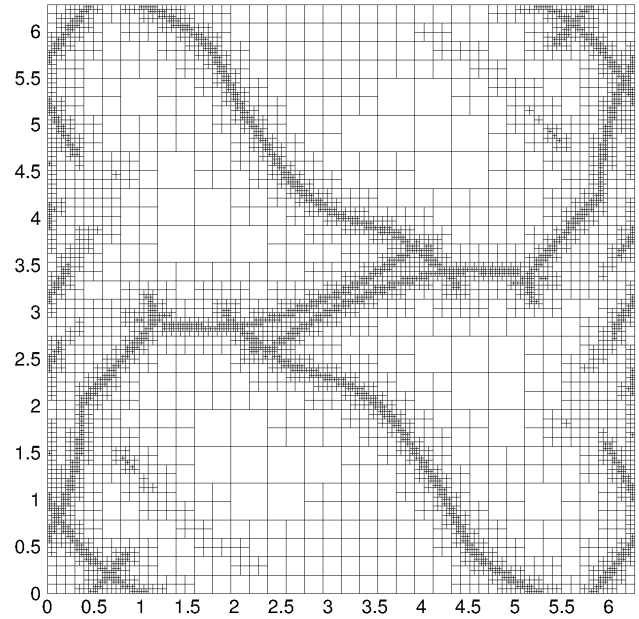
For the Orszag-Tang vortex problem, the same initial conditions and domain as in [28] are used, with $\rho = \gamma^2$, $v_x = -\sin(y)$, $v_y = \sin(x)$, $B_x = -\sin(y)$, $B_y = \sin(2x)$, and $p = \gamma$. The remaining variables (v_z, B_z, and ψ)



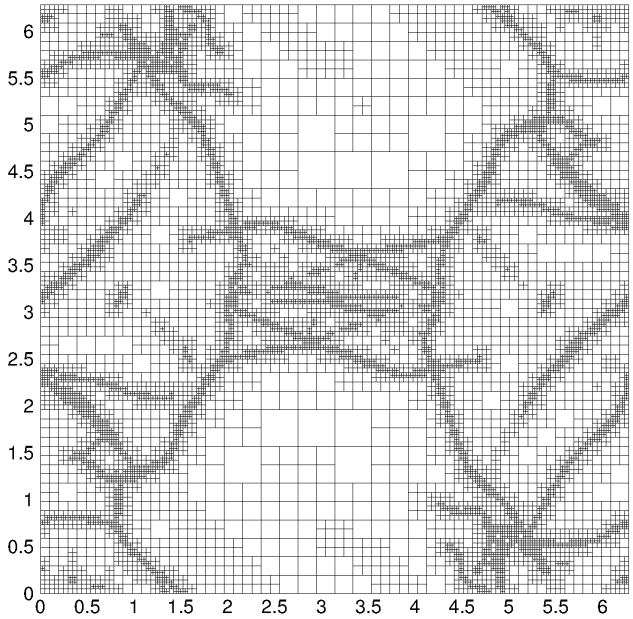
(a) AMR as applied to the Orszag-Tang vortex problem at $t = 0.5$. At this point, the mesh consists of 118 8-by-8 blocks, or 7,552 cells in total.



(b) AMR as applied to the Orszag-Tang vortex problem at $t = 1.0$. At this point, the mesh consists of 1,474 8-by-8 blocks, or 95,810 cells in total.



(c) AMR as applied to the Orszag-Tang vortex problem at $t = 2.0$. At this point, the mesh consists of 8,428 8-by-8 blocks, or 539,136 cells in total.



(d) AMR as applied to the Orszag-Tang vortex problem at $t = 3.0$. At this point, the mesh consists of 13,522 8-by-8 blocks, or 865,408 cells in total.

Figure 10: The evolution of the mesh for the simulation of Fig. 9 with adaptive refinement. Up to $t = 1.0$, the mesh is refined every 0.025 seconds, after which it is refined every 50 time steps. The lines in the figure represent the boundaries of the 8-by-8 Cartesian blocks.

are initialized to zero. The computational domain is a square with x and y values between 0 and 2π, and periodic boundary conditions (ghost cells are used). The simulation is performed with CENO cutoff tolerance $\mathcal{S}_C = 500$. The mesh is refined every 0.025 seconds up to $t = 1$. For later times, AMR is performed every 50 time steps because Δt decreases rapidly. The contour lines of the density for the Orszag-Tang vortex problem are shown at $t = 0.5$, $t = 1.0$, $t = 2.0$, and $t = 3.0$ in Fig. 9. The results show agreement with results shown in other papers [10, 28, 31, 53]. Figure 10 shows the sequence of adaptive meshes. Comparing the density contour lines shown in Fig. 9 with the way the

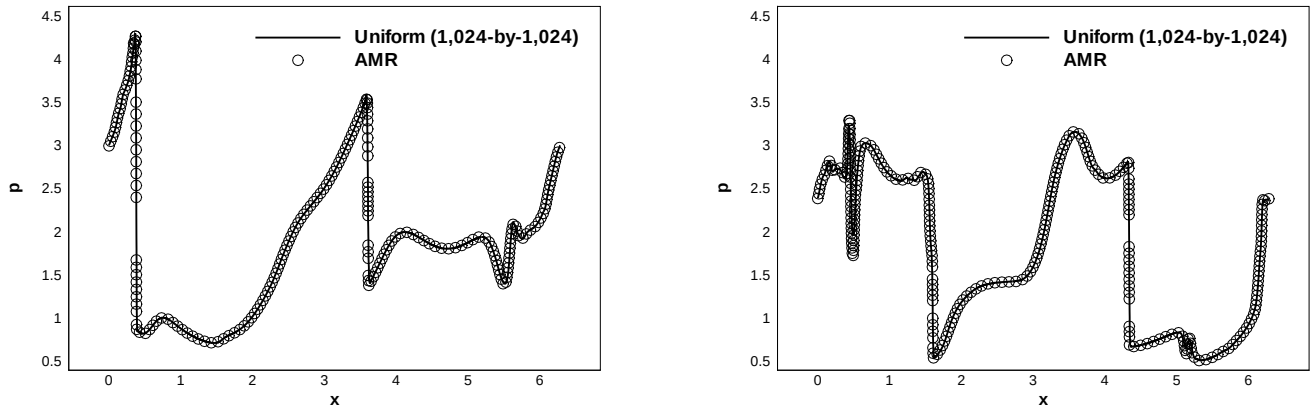


Figure 11: Pressure cuts at $y = 1.9635$ at two different times ($t = 2.0$ [left], and $t = 3.0$ [right]). High-order results obtained in combination with adaptive mesh refinement are compared with uniform high-order high-resolution results, and found to be similar. Our results agree with the results from [28] and [31].

grid is refined as shown in Fig. 10, it can be seen that the refinement closely follows the parts of the solution where interesting flow features and discontinuities occur, illustrating the effectiveness of the smoothness indicator-based refinement criterion.

Following [28] and [31], pressure distribution cuts at $t = 2.0$ and $t = 3.0$ along the line $y = 1.9635$ are shown in Fig. 11. The AMR results are compared to results on a uniform 1,024-by-1,024 mesh. The uniform mesh corresponds to the smallest cell resolution at 7 levels of refinement, while 8 levels of refinement are used in the AMR results, so that the smallest cell in the adaptive mesh (Fig. 10) corresponds to a resolution of 2,048-by-2,048 if done uniformly. From Fig. 11, it can be seen that the AMR results in general agree well with the uniform reference results. The AMR solution (as shown in Fig. 9) agrees well with the uniform reference solution (which is not shown) and with solutions shown in the literature, but our results have a much higher effective resolution than previously shown results (and they are fourth-order accurate). It is also interesting to note that, while the uniform mesh has 1,048,576 computational cells, the AMR mesh has 865,408 cells at $t = 3.0$, which is smaller than the uniform mesh, despite having twice the effective resolution at the highest level of refinement. Note also that, before $t = 3.0$, much fewer cells are used by the adaptive simulation (see Fig. 10). This illustrates the effectiveness of the CENO scheme in combination with the block-based AMR algorithm to reduce the number of required computational cells.

5. Concluding Remarks

We have proposed a high-order CENO FV scheme for ideal MHD. The scheme is based on the CENO approach that was proposed by Ivan and Groth for compressible Euler flows in [1] and uses the GLM divergence cleaning method for MHD of Dedner et al. [6]. The resulting FV MHD scheme is high-order accurate in smooth flow regions and robust against spurious oscillations at discontinuities. The proposed high-order accurate MHD scheme can be used on general polygonal grids and can deal naturally with resolution changes on hierarchical quadtree block-adaptive body-fitted grids. The proposed scheme was implemented in a highly sophisticated fourth-order accurate parallel MHD code on 2D dynamically-adaptive multi-block body-fitted structured grids, and curved boundaries are handled with high-order accuracy using high-order spline representations and constraints at the Gauss points.

Detailed numerical results were given that demonstrate high-order convergence for smooth flows, and robustness against oscillations for Riemann problems and other flows with shocks. A new MHD extension of the well-known Shu-Osher test problem [38] was proposed to test the ability of the high-order MHD scheme to resolve small-scale flow features in the presence of shocks. The dynamic AMR capabilities of our approach were demonstrated using adaptive time-dependent simulations of the Orszag-Tang vortex problem with high-order accuracy and unprecedented effective resolution.

The scheme we described can in principle be implemented with arbitrary order. It can also be extended naturally to three spatial dimensions and to unstructured grids, which are topics of future work.

Acknowledgements

This work was supported by the Canadian Space Agency and by NSERC of Canada. Computations were performed on the GPC supercomputer at the SciNet HPC Consortium and on the facilities of the Shared Hierarchical Academic Research Computing Network (SHARCNET:www.sharcnet.ca). The SciNet and SHARCNET consortia are funded by: the Canada Foundation for Innovation under the auspices of Compute/Calcul Canada; the Government of Ontario; Ontario Research Fund - Research Excellence; and the member institutions of the consortia.

References

[1] L. Ivan and C. P. T. Groth. High-order central ENO finite-volume scheme with adaptive mesh refinement. *AIAA Paper*, 2007-4323, 2007.

[2] L. Ivan. *Development of High-Order CENO Finite-Volume Schemes with Block-Based Adaptive Mesh Refinement*. PhD thesis, University of Toronto, 2011.

[3] L. Ivan and C. P. T. Groth. High-order solution adaptive central essentially non-oscillatory (CENO) method for viscous flows. *AIAA Paper*, 2011-367, 2011.

[4] L. Ivan and C. P. T. Groth. High-order solution-adaptive central essentially non-oscillatory (CENO) method for viscous flows. 2012. Submitted.

[5] T. J. Barth. Recent developments in high order k-exact reconstruction on unstructured mesh. *AIAA Paper*, 93-0668, 1993.

[6] A. Dedner, F. Kemm, D. Kroner, C-D. Munz, T. Schitzer, and M. Wesenberg. Hyberbolic divergence cleaning for the MHD equations. *Journal of Computational Physics*, 175:645–673, 2002.

[7] J. U. Brackbill and D. C. Barnes. Note: The effect of nonzero $\nabla \cdot \mathbf{B}$ on the numerical solution of the magnetohydrodynamic equations. *Journal of Computational Physics*, 35:426, 1980.

[8] K. G. Powell. An approximate Riemann solver for magnetohydrodynamics. Technical report, The University of Michigan, 1994.

[9] A. J. Chorin. A numerical method for solving incompressible viscous flow problems. *Journal of Computational Physics*, 2:12–26, 1967.

[10] G. Toth. The $\nabla \cdot \mathbf{B} = 0$ constraint in shock-capturing magnetohydrodynamics codes. *Journal of Computational Physics*, 161:605–652, 2000.

[11] C. R. Evans and J. F. Hawley. Simulation of magnetohydrodynamic flows: A constrained transport method. *Astrophysical Journal*, 332:659, 1988.

[12] H. De Sterck. Multi-dimensional upwind constrained transport on unstructured grids for 'shallow water' magnetohydrodynamics. *AIAA Paper*, 2001-2623, 2001.

[13] D. S. Balsara. Second-order accurate schemes for magnetohydrodynamics with divergence-free reconstruction. *The Astrophysical Journal Supplement Series*, 151:149–184, 2004.

[14] D. S. Balsara. Divergence-free reconstruction of magnetic fields and WENO schemes for magnetohydrodynamics. *Journal of Computational Physics*, 228:5040–5056, 2009.

[15] Dinshaw S. Balsara. Divergence-free adaptive mesh refinement for magnetohydrodynamics. *Journal of Computational Physics*, 174(2):614 – 648, 2001.

[16] G. Toth and P.L. Roe. Divergence- and curl-preserving prolongation and restriction formulas. *Journal of Computational Physics*, 180(2):736 – 750, 2002.

[17] M. S. Yalim, D. Vanden Abeele, A. Lani, T. Quintino, and H. Deconinck. A finite volume implicit time integration method for solving the equations of ideal magnetohydrodynamics for the hyperbolic divergence cleaning method. *Journal of Computational Physics*, 230:6136–6154, 2011.

[18] C. P. T. Groth and S. A. Northrup. Parallel implicit adaptive mesh refinement scheme for body-fitted multi-block mesh. Paper 2005-5333, AIAA, June 2005.

[19] J. S. Sachdev, C. P. T. Groth, and J. J. Gottlieb. A parallel solution-adaptive scheme for predicting multi-phase core flows in solid propellant rocket motors. *International Journal of Computational Fluid Dynamics*, 19(2):157–175, 2005.

[20] X. Gao and C. P. T. Groth. A parallel adaptive mesh refinement algorithm for predicting turbulent non-premixed combusting flows. *International Journal of Computational Fluid Dynamics*, 20(5):349–357, 2006.

[21] X. Gao and C. P. T. Groth. A parallel solution-adaptive method for three-dimensional turbulent non-premixed combusting flows. *Journal of Computational Physics*, 229(5):3250–3275, 2010.

[22] X. Gao, S. A. Northrup, and C. P. T. Groth. Parallel solution-adaptive method for two-dimensional non-premixed combusting flows. *Progress in Computational Fluid Dynamics*, 11(2):76–95, 2011.

[23] J. S. Sachdev and C. P. T. Groth. A mesh adjustment scheme for embedded boundaries. *Communications in Computational Physics*, 2(6):1095–1124, 2007.

[24] Z. J. Zhang and C. P. T. Groth. Parallel high-order anisotropic block-based adaptive mesh refinement finite-volume scheme. Paper 2011-3695, AIAA, June 2011.

[25] T. C. Warburton and G. E. Karniadakis. A discontinuous Galerkin method for the viscous MHD equations. *Journal of Computational Physics*, 152:608–641, 1999.

[26] F. Li and C.-W. Shu. Locally divergence-free discontinuous Galerkin methods for MHD equations. *Journal of Scientific Computing*, 22-23:413–442, 2005.

[27] C. Altmann, Thomas Belat, Michael Gutnic, Philippe Helluy, Helene Mathis, Eric Sonnendruecker, Wilfredo Angulo, and Jean-Marc Herard. A local time-stepping discontinuous Galerkin algorithm for the MHD system. In *ESAIM: PROCEEDINGS*, volume 28, pages 33–54. EDP Sciences, August 2009.

[28] F. Li, L. Xu, and S. Yakovlev. Central discontinuous Galerkin methods for ideal MHD equations with the exactly divergence-free magnetic field. *Journal of Computational Physics*, 230:4828–4847, 2011.

[29] G-S. Jiang and C. Wu. A high-order WENO finite difference scheme for the equations of ideal magnetohydrodynamics. *Journal of Computational Physics*, 150:561–594, 1999.

[30] J. Kleimann, A. Kopp, H. Fichtner, R. Grauer, and K. Germaschewski. Three-dimensional MHD high-resolution computations with CWENO employing adaptive mesh refinement. *Computer Physics Communications*, 158:47–56, 2004.

[31] J. Balbás and E. Tadmor. Non-oscillatory central schemes for one- and two-dimensional MHD equations. II: High-order semi-discrete schemes. *SIAM Journal of Scientific Computing*, 28:533–560, 2006.

[32] D. S. Balsara, T. Rumpf, M. Dumbser, and C-D. Munz. Efficient, high accuracy ADER-WENO schemes for hydrodynamics and divergence-free magnetohydrodynamics. *Journal of Computational Physics*, 228:2480–2516, 2009.

[33] A. Mignone and P. Tzeferacos. A second-order unsplit Godunov scheme for cell-centered MHD: The CTU-GLM scheme. *Journal of Computational Physics*, 229:2117–2138, 2010.

[34] A. Mignone, P. Tzeferacos, and G. Bodo. High-order conservative finite difference GLM-MHD schemes for cell-centered MHD. *Journal of Computational Physics*, 229:5896–5920, 2010.

[35] A. Harten. High resolution schemes for hyperbolic conservation laws. *Journal of Computational Physics*, 49:357–393, 1983.

[36] A. Harten and S.R. Chakravarthy. Multi-dimensional ENO schemes for general geometries. ICASE Report 91-76, 1991.

[37] H. Nessyahu and E. Tadmor. Non-oscillatory central differencing for hyperbolic conservation laws. *Journal of Computational Physics*, 87:408, 1990.

[38] C.-W. Shu and S. Osher. Efficient implementation of essentially non-oscillatory shock-capturing schemes, II. *Journal of Computational Physics*, 83:32–78, 1989.

[39] S. Orszag and C. M. Tang. Small-scale structure of two-dimensional magnetohydrodynamic turbulence. *Journal of Fluid Mechanics*, 90:129–143, 1979.

[40] C. D. Munz, P. Omnes, R. Schneider, E. Sonnendrücker, and U. Voss. Divergence correction techniques for Maxwell solvers based on a hyperbolic model. *Journal of Computational Physics*, 161:484, 2000.

[41] K. G. Powell, P. L. Roe, T. J. Linde, and D. L. De Zeeuw. A solution-adaptive upwind scheme for ideal magnetohydrodynamics. *Journal of Computational Physics*, 154:284–309, 1999.

[42] C. Hirsch. *Numerical Computation of Internal and External Flows*. John Wiley and Sons, 1990.

[43] H. De Sterck, A. Csik, D. Vanden Abeele, S. Poedts, and H. Deconinck. Stationary two-dimensional magnetohydrodynamics flows with shocks: Characteristic analysis and grid convergence study. *Journal of Computational Physics*, 166:28–62, 2001.

[44] B. van Leer, W.-T. Lee, P. L. Roe, K. G. Powell, and C.-H. Tai. Design of optimally smoothing multistage schemes for the Euler equations. *Communications in Applied Numerical Methods*, 8(10):761–769, 1992.

[45] D. J. Mavriplis. Revisiting the least-squares procedure for gradient reconstruction on unstructured meshes. Paper 2003-3986, AIAA, June 2003.

[46] V. Venkatakrishnan. On the accuracy of limiters and convergence to steady state solutions. Paper 93-0880, AIAA, January 1993.

[47] C. Michalak and C. Ollivier-Gooch. Accuracy preserving limiter for the high-order accurate solution of the Euler equations. *Journal of Computational Physics*, 228:8693–8711, 2009.

[48] V. Wheatley, H. Kumar, and P. Huguenot. On the role of Riemann solvers in discontinuous Galerkin methods for magnetohydrodynamics. *Journal of Computational Physics*, 229:660–680, 2010.

[49] R. J. LeVeque. *Finite-Volume Methods for Hyperbolic Problems*. Cambridge University Press, 2002.

[50] C. F. Ollivier-Gooch and M. Van Altena. A high-order accurate unstructured mesh finite-volume scheme for the advection-diffusion equation. *Journal of Computational Physics*, 181:729–752, 2002.

[51] M. Brio and C. C. Wu. An upwind differencing scheme for the equations of ideal magnetohydrodynamics. *Journal of Computational Physics*, 75:400, 1988.

[52] J. A. Rossmanith. An unstaggered, high-resolution constained transport method for magnetohydrodynamic flows. *SIAM Journal on Scientific Computing*, 28:1766–1797, 2006.

[53] J. Balbás, E. Tadmor, and Cheng-Chin Wu. Non-oscillatory central schemes for one- and two-dimensional MHD equations: I. *Journal of Computational Physics*, 201:261–285, 2004.