

## Development and Numerical Verification of a High-Order Unstructured Finite-Volume Framework for Tokamak Plasma Transport

Jinsu Kim<sup>a</sup>, Beomsu Kim<sup>a</sup>, Jaegon Lee<sup>b</sup>, and Yong-Su Na<sup>a\*</sup>

<sup>a</sup>Department of Nuclear Engineering, Seoul National University, Seoul, Republic of Korea

<sup>b</sup>ITER Organization, Route de Vinon-sur-Verdon, CS 90 046, 13067 St. Paul lez Durance Cedex, France

\*ysna@snu.ac.kr

\*Keywords: numerical analysis, plasma simulation

### 1. Introduction

The objective of this work is to develop a 2D tokamak edge and scrape-off-layer (SOL) transport code for transient discharge phases involving evolving magnetic configurations. Accurate prediction of the edge and SOL plasma is essential for assessing plasma-wall interaction, impurity generation and transport, separatrix conditions, and their influence on core plasma performance. In particular, plasma start-up and current ramp-up involve magnetic equilibrium evolution, and a recent full-discharge study demonstrated the importance of resolving tungsten sputtering and transport during these phases [1].

For such transient simulations, the spatial discretization should remain accurate as the magnetic equilibrium evolves without requiring the computational mesh to be aligned with each instantaneous equilibrium. In addition, a self-consistent anomalous cross-field transport model should be applied to account for changes in the plasma state. Existing 2D edge transport calculations [2, 3, 4] commonly use field-aligned structured meshes constructed from a prescribed magnetic equilibrium and apply prescribed anomalous radial transport coefficients. These approaches provide an efficient representation of strongly anisotropic transport for a specified magnetic equilibrium and have been widely used for conventional edge and divertor analysis. However, magnetic equilibrium evolution during the transition phases requires repeated mesh regeneration and conservative transfer of the plasma state between successive grids. In addition, the use of fixed transport coefficients may limit the predictive capability of transient simulations.

Recent studies have begun to tackle these two limitations. The discontinuous Galerkin method has been applied to unstructured tokamak transport [1] in combination with time-varying equilibrium, and a RANS-inspired mean-field approach [5] has been proposed to determine anomalous cross-field transport self-consistently rather than through externally prescribed coefficients.

Inspired by the previous studies above, the present paper focuses on the construction and numerical verification of the 2D tokamak edge/SOL transport solver currently under development. The solver advances the

plasma density, parallel momentum, and ion and electron energy equations using a simplified Braginskii formulation. The equations are discretized by a cell-centered finite-volume method on unstructured triangular grids. A high-order  $K$ -exact reconstruction [6] is employed to retain spatial accuracy for non-field-aligned anisotropic transport. At the same time, a CENO procedure [7] replaces the high-order polynomial with a slope-limited linear reconstruction near discontinuities to suppress unphysical oscillations. The high-order quadrature rules and the PETSc library [8] are incorporated into the same transport-solver infrastructure.

To verify the spatial discretization and the diffusion operator implemented in the transport code currently under development, an anisotropic diffusion benchmark is used to assess convergence quantitatively. Compressible Euler and ideal-MHD problems are used as auxiliary tests of the robustness of the common reconstruction and convective flux calculation in the presence of discontinuities and coupled waves. Finally, the simplified transport equations are solved on an unstructured KSTAR cross-sectional geometry to demonstrate that the verified numerical infrastructure can be applied to a tokamak-shaped domain.

The present KSTAR application is limited to a steady-state edge/SOL calculation. It does not yet include sheath boundary physics, atomic and neutral-plasma processes, drift-related transport, or a self-consistent turbulent transport closure. The remainder of this paper is organized as follows. Section 2 introduces the simplified plasma transport equations and then presents their finite-volume discretization with the CENO reconstruction procedure. Section 3 investigates the numerical verification results, and Section 4 presents the edge/SOL transport application. Finally, Section 5 summarizes the status of the code development and future considerations.

### 2. Numerical Formulation

The present transport solver evolves four conservative plasma variables, namely the density  $n$ , parallel momentum  $nu_{\parallel}$ , ion energy  $\frac{3}{2}nT_i + \frac{1}{2}m_i nu_{\parallel}^2$ , and electron energy  $\frac{3}{2}nT_e$ , in the axisymmetric R-Z plane. The governing system is based on a simplified Braginskii

formulation and consists of a continuity equation, a parallel-momentum equation, and separate ion and electron energy equations. The effective parallel heat conduction coefficients are applied based on [9]:

$$\begin{aligned}
 & \frac{\partial n}{\partial t} + \nabla \cdot (nu_{\parallel} \hat{\mathbf{b}} - D_{\perp} \nabla_{\perp} n) = 0, \\
 & \frac{\partial m_i n u_{\parallel}}{\partial t} + \nabla \cdot (m_i n u_{\parallel}^2 \hat{\mathbf{b}}) = -\hat{\mathbf{b}} \cdot \nabla (p_i + p_e) \\
 & \quad + \nabla \cdot (\eta_{\parallel} \nabla_{\parallel} u_{\parallel} + \eta_{\perp} \nabla_{\perp} (m_i n u_{\parallel})), \\
 & \frac{\partial}{\partial t} \left( \frac{3}{2} n T_i + \frac{1}{2} m_i n u_{\parallel}^2 \right) + \nabla \cdot \left( \left( \frac{5}{2} n T_i + \frac{1}{2} m_i n u_{\parallel}^2 \right) u_{\parallel} \hat{\mathbf{b}} \right) \\
 & = e n u_{\parallel} E_{\parallel} + \nabla \cdot \left( \kappa_{\parallel, i} \nabla_{\parallel} T_i + \kappa_{\perp} \nabla_{\perp} \left( \frac{3}{2} n T_i + \frac{1}{2} m_i n u_{\parallel}^2 \right) \right) \\
 & \quad + W_{ei}, \\
 & \frac{\partial}{\partial t} \left( \frac{3}{2} n T_e \right) + \nabla \cdot \left( \frac{5}{2} n T_e u_{\parallel} \hat{\mathbf{b}} \right) = -e n u_{\parallel} E_{\parallel} \\
 & \quad + \nabla \cdot \left( \kappa_{\parallel, e} \nabla_{\parallel} T_e + \kappa_{\perp} \nabla_{\perp} \left( \frac{3}{2} n T_e \right) \right) - W_{ei}, \\
 & E_{\parallel} = -\hat{\mathbf{b}} \cdot \nabla p_e, W_{ei} = \frac{3m_e n}{m_i \tau_e} (T_e - T_i), \\
 & \eta_{\parallel} = 0.64 n T_i \tau_i, \kappa_{\parallel, i} = \frac{\kappa_{SH, i} \kappa_{fl, i}}{\kappa_{SH, i} + \kappa_{fl, i}}, \kappa_{\parallel, e} = \frac{\kappa_{SH, e} \kappa_{fl, e}}{\kappa_{SH, e} + \kappa_{fl, e}}, \\
 & \kappa_{SH, i} = \frac{3.9 n T_i \tau_i}{m_i}, \kappa_{SH, e} = \frac{3.16 n T_e \tau_e}{m_e}, \\
 & \kappa_{fl, i, e} = R q n v_{th, (i, e)} T_{i, e}, R = 1.8, q = 3 \quad (1)
 \end{aligned}$$

All the numerical assessments considered in this work share a common cell-centered FV framework. The conservative variables are stored as cell-averaged values, and the CENO reconstruction scheme is employed to reconstruct pointwise primitive variables and their gradients at every quadrature point. The smoothness indicator (SI) is designed to replace the high-order reconstructed value with a slope-limited linear reconstructed one in non-smooth regions.

### 2.1. Cell-Centered FVM Discretization

The general form of a conservative equation is expressed as:

$$\frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot \mathbf{F}^c(\mathbf{U}) = \nabla \cdot \mathbf{F}^d(\mathbf{U}, \nabla \mathbf{U}) + \mathbf{S}(\mathbf{U}) \quad (2)$$

Where  $\mathbf{U}$  is the conservative variable vector,  $\mathbf{F}^c$  is the convective flux vector,  $\mathbf{F}^d$  is the diffusive flux vector, and  $\mathbf{S}$  is the volumetric source term. In the cell-centered FVM on a 2D unstructured triangle grid, the

discretization of the above equation can be shown using the cell-averaged values of each cell:

$$\bar{\mathbf{U}}_i = \frac{1}{V_i} \int_{V_i} \mathbf{U}(\mathbf{x}) dV \quad (3)$$

$$\begin{aligned}
 & \frac{\partial \bar{\mathbf{U}}_i}{\partial t} + \frac{1}{V_i} \sum_{f=1}^3 \sum_{g=1}^{N_g} \frac{\omega_g l_f}{2} \left( (\mathbf{F}^c(\mathbf{U})_{f, g} - \mathbf{F}^d(\mathbf{U}, \nabla \mathbf{U})_{f, g}) \cdot \hat{\mathbf{n}}_f \right) \\
 & = \frac{1}{V_i} \sum_{c=1}^{N_c} \omega_c \mathbf{S}(\mathbf{U})_c \quad (4)
 \end{aligned}$$

Where  $N_g$  and  $\omega_g$  is the number and weight of Gauss quadrature points on the cell face  $f$ ,  $\hat{\mathbf{n}}_f$  is the unit vector normal to the face, and  $l_f$  is the length of the cell face.  $N_c$  is the number of cell quadrature points and  $\omega_c$  is the 2D triangle quadrature weight.

Implementing a less dissipative but robust flux scheme is also an important issue in FVM. In this work, the Force- $\alpha$  scheme [10] is applied for the convective flux:

$$\begin{aligned}
 & \mathbf{F}^c(\mathbf{U}) \cdot \hat{\mathbf{n}}_f = \frac{1}{2} (\mathbf{F}(\mathbf{U})_{LLF} + \mathbf{F}(\mathbf{U})_{Richtmyer}), \\
 & \mathbf{F}(\mathbf{U})_{LLF} = \frac{1}{2} \left( \mathbf{F}(\mathbf{U}_L) + \mathbf{F}(\mathbf{U}_R) - \frac{\lambda_{\max}}{\alpha} (\mathbf{U}_R - \mathbf{U}_L) \right), \\
 & \mathbf{F}(\mathbf{U})_{Richtmyer} = \mathbf{F}(\mathbf{U}^*(\mathbf{U}_L, \mathbf{U}_R)), \\
 & \mathbf{U}^* = \frac{1}{2} \left( \mathbf{U}_L + \mathbf{U}_R - \frac{\alpha}{\lambda_{\max}} (\mathbf{F}(\mathbf{U}_R) - \mathbf{F}(\mathbf{U}_L)) \right) \quad (5)
 \end{aligned}$$

Where  $\mathbf{U}_L$  and  $\mathbf{U}_R$  are reconstructed conservative variable vectors on the left/right side of the cell face, and  $\lambda_{\max}$  is the maximum characteristic speed calculated in the face.

For the diffusive flux, the  $\alpha$ -damping scheme [11] is employed for face gradient calculation, and a simple arithmetic average of  $\mathbf{U}_L$  and  $\mathbf{U}_R$  is used for face variable determination:

$$\mathbf{U}_f = \frac{1}{2} (\mathbf{U}_L + \mathbf{U}_R),$$

$$\nabla \mathbf{U}_f = \frac{1}{2} (\nabla \mathbf{U}_L + \nabla \mathbf{U}_R) + \frac{\alpha}{L_{ref}} (\mathbf{U}_R - \mathbf{U}_L) \hat{\mathbf{n}}_f \quad (6)$$

### 2.2. High-Order CENO Reconstruction

The CENO reconstruction is applied to obtain accurate values at the quadrature points in smooth regions and to suppress unphysical oscillations in discontinuous regions. The CENO reconstruction takes two major strategies:

1)  $K^{th}$ -order polynomial reconstruction in smooth regions.

2) Slope-limited linear reconstruction when discontinuity is detected.

In the smooth region, the scalar conservative variable of the  $i^{th}$  cell  $u_i$  can be approximated by applying a Taylor-expanded polynomial:

$$u_i^K(x, y) = u_{i,c} + \sum_{m=1}^K \sum_{n=0}^m \left\{ \frac{1}{(m-n)!n!} \frac{\partial^m U}{\partial x^{m-n} \partial y^n} \right\}_{x_c, y_c} (x - x_c)^{m-n} (y - y_c)^n \quad (7)$$

Where  $(x_c, y_c)$  is the  $i^{th}$  cell's centroid position, and  $u_{i,c}$  is the pointwise value at the centroid. In theory, the extrapolated value  $u_i^K(x, y)$  and its gradient  $\nabla u_i^K(x, y)$  must be  $O(h^{K+1})$  and  $O(h^K)$  accurate in smooth regions, respectively.

The cell-averaging integral  $\frac{1}{V_i} \int_{V_i} u_i^K(x, y) dV$  should be the cell-averaged value  $\bar{u}_i$  to follow the conservation law. This can be satisfied by setting the centroid value  $u_{i,c}$  to be:

$$u_{i,c} = \bar{u}_i - \frac{1}{V_i} \sum_{m=1}^K \sum_{n=0}^m \left\{ \frac{1}{(m-n)!n!} \frac{\partial^m U}{\partial x^{m-n} \partial y^n} \right\}_{x_c, y_c} \int_{V_i} (x - x_c)^{m-n} (y - y_c)^n dV \quad (8)$$

Using (5) and (6), the reconstruction polynomial  $u_i^K(x, y)$  which satisfies cell average constraint has the form shown below:

$$\begin{aligned} & u_i^K(x, y) - \bar{u}_i \\ &= \sum_{m=1}^K \sum_{n=0}^m c_{i,mn} \{ (x - x_c)^{m-n} (y - y_c)^n - M_{i,mn} \}, \\ & M_{i,mn} = \frac{1}{V_i} \int_{V_i} (x - x_c)^{m-n} (y - y_c)^n dV, \\ & c_{i,mn} = \frac{1}{(m-n)!n!} \frac{\partial^m u}{\partial x^{m-n} \partial y^n} \Big|_{x_c, y_c} \end{aligned} \quad (9)$$

$$\begin{aligned} \rightarrow \bar{u}_i &= \frac{1}{V_i} \int_{V_i} u_i^K(x, y) dV = \bar{u}_i + \sum_{m=1}^K \sum_{n=0}^m c_{i,mn} \\ & \frac{1}{V_i} \int_{V_i} \{ (x - x_c)^{m-n} (y - y_c)^n - M_{i,mn} \} dV \end{aligned} \quad (10)$$

The reconstruction coefficients  $c_{i,mn}$  are determined from the weighted least-squares (WLSQ) problem. The WLSQ system of each cell is constructed by minimizing

the difference between cell-averaged values of neighboring cells  $\bar{u}_j$  and the values obtained from extrapolation of reconstruction polynomial  $u_i^K(x, y)$ :

$$\begin{aligned} \bar{u}_j &\simeq \bar{u}_{ij,ext} \\ &= \frac{1}{V_j} \int_{V_j} u_i^K(x, y) dV = \bar{u}_i + \sum_{m=1}^K \sum_{n=0}^m A_{ij,mn} c_{i,mn}, \\ A_{ij,mn} &= \frac{1}{V_j} \int_{V_j} (x - x_c)^{m-n} (y - y_c)^n dV - M_{i,mn} \quad (11) \\ &\rightarrow w_{ij}^\beta \sum_{m=1}^K \sum_{n=0}^m A_{ij,mn} c_{i,mn} \simeq w_{ij}^\beta (\bar{u}_j - \bar{u}_i), \end{aligned}$$

$$w_{ij}^\beta = \frac{1}{|\mathbf{x}_j - \mathbf{x}_i|^\beta}, \quad B_{ij,mn} = w_{ij}^\beta A_{ij,mn} \quad (12)$$

By applying the above relation to all neighboring cells  $V_j \in S_i$ , the reconstruction coefficients are determined from the WLSQ problem:

$$\begin{aligned} & \overbrace{\begin{pmatrix} B_{i1,10} & \cdots & B_{i1,mn} \\ \vdots & \ddots & \vdots \\ B_{iN_j,10} & \cdots & B_{iN_j,mn} \end{pmatrix}}^{\mathbf{B}_i} \overbrace{\begin{pmatrix} c_{i,10} \\ \vdots \\ c_{i,mn} \end{pmatrix}}^{\mathbf{c}_i} \simeq \overbrace{\begin{pmatrix} w_{i1}^\beta \Delta \bar{u}_{i1} \\ \vdots \\ w_{iN_j}^\beta \Delta \bar{u}_{iN_j} \end{pmatrix}}^{\mathbf{d}_i} \\ & \Delta \bar{u}_{ij} = \bar{u}_j - \bar{u}_i, \quad \mathbf{c}_i = \arg \min_c \|\mathbf{B}_i \mathbf{c} - \mathbf{d}_i\|^2 \quad (13) \end{aligned}$$

In 2D analysis, the number of reconstruction coefficients  $N_c$  of degree  $K$  equals  $\frac{K(K+3)}{2}$ . The number of neighboring cells  $N_j$  should exceed  $N_c$  to form an overdetermined WLSQ system. In the present implementation, reconstruction degrees up to  $K=6$  are supported, corresponding to  $N_c = 27$  polynomial coefficients, and  $N_j$  is set to be at least 32 in every cell.

In nearly discontinuous regions, the above reconstruction may induce unphysical oscillations. To prevent this problem and retain numerical robustness, the SI and slope-limited linear reconstruction are applied:

$$SI = \sqrt{\frac{\sum_{j=1}^3 (\bar{u}_{j,ext} - \bar{u}_j)^2}{\sum_{j=1}^3 \bar{u}_j^2 + \epsilon}} \begin{cases} \leq C \rightarrow \chi_{SI} = 0 \\ > C \rightarrow \chi_{SI} = 1 \end{cases} \quad (14)$$

$$u(x, y) = (1 - \chi_{SI}) u_i^K(x, y) + \chi_{SI} u_i^{lim}(x, y),$$

$$u_i^{lim}(x, y) = \bar{u}_i + \phi_i \nabla u \cdot (\mathbf{x} - \mathbf{x}_c) \quad (15)$$

Where  $C$  is the user-defined threshold value of SI,  $\phi_i$  is the slope limiter, and  $\nabla u$  is the linear gradient computed from compact stencil. The indicator compares the extrapolated value of face neighboring cells  $\bar{u}_{j,ext}$  with

the actual cell-averaged values  $\bar{u}_j$  while  $\epsilon$  prevents division-by-zero problems.

If the SI exceeds a certain threshold  $C$ , the reconstruction procedure converts from  $K$ -exact reconstruction to slope-limited linear reconstruction.  $C$  is set to be  $h^K$  where  $h$  is the global average mesh edge length, and Nishikawa's R3 slope limiter [12] is applied in this work.

### 3. Numerical Results

Based on the numerical framework discussed in the previous section, the numerical results are assessed here. An anisotropic diffusion problem is used to investigate numerical accuracy under a high-order reconstruction method. 2D compressible Euler and ideal-MHD problems are considered to confirm solution robustness at discontinuities.

#### 3.1. Anisotropic Diffusion Problem

In strongly magnetized plasma, the heat conduction tensor has extreme anisotropy. Under this condition, low order spatial discretization error may induce artificial dissipation and pollute solution accuracy if the magnetic field and the computational mesh are misaligned. We chose the 2D anisotropic diffusion benchmark [13] to test solution accuracy with the CENO reconstruction.

The equation is shown below:

$$\begin{aligned} \nabla \cdot (\kappa_{\parallel} \hat{\mathbf{b}}\hat{\mathbf{b}} \cdot \nabla T + \kappa_{\perp} (\hat{\mathbf{I}} - \hat{\mathbf{b}}\hat{\mathbf{b}}) \cdot \nabla T) + S(x, y) &= 0, \\ S(x, y) &= 2\pi^2 \cos(\pi x) \cos(\pi y), \\ T(x, y)_{exact} &= \psi(x, y) = \cos(\pi x) \cos(\pi y), \\ \mathbf{B}(x, y) &= \hat{\mathbf{z}} \times \nabla \psi \end{aligned} \quad (16)$$

Where  $\kappa_{\parallel}$  and  $\kappa_{\perp}$  are parallel and perpendicular heat diffusivities, respectively. This problem is solved on square  $[0, 1] \times [0, 1]$  domain with constant  $\kappa_{\perp} = 1$  and Dirichlet boundary condition.  $L_2 = \sqrt{\frac{\sum_i (E_i)^2}{N}}$ ,  $L_{\infty} = \max_i |E_i|$ ,  $E_i = (T_{i,calc} - T_{i,exact})$  were computed to quantitatively investigate convergence errors of the problem where  $T_{i,calc}$  and  $T_{i,exact}$  are calculated and exact centroid values of the  $i^{\text{th}}$  cell, respectively.

Because an anisotropic diffusion tensor makes the resulting system difficult to solve with naive linear system solvers, we investigated the PETSc library's solvers and chose GMRES as the linear system solver and the additive Schwarz method (ASM) for preconditioning.

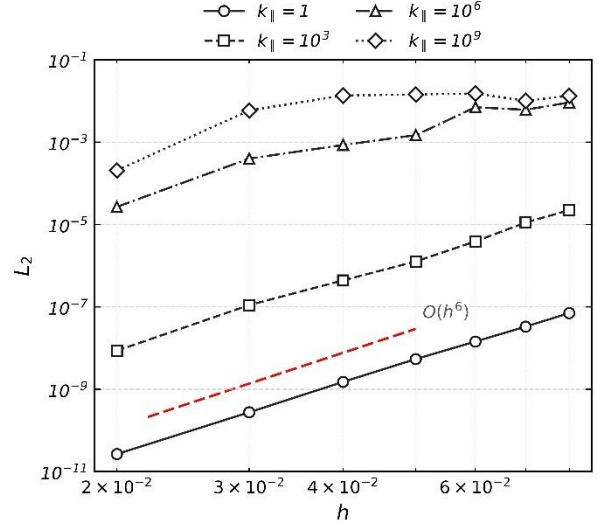


Fig. 1.  $L_2$  norm error convergence behavior under increasing transport anisotropy.

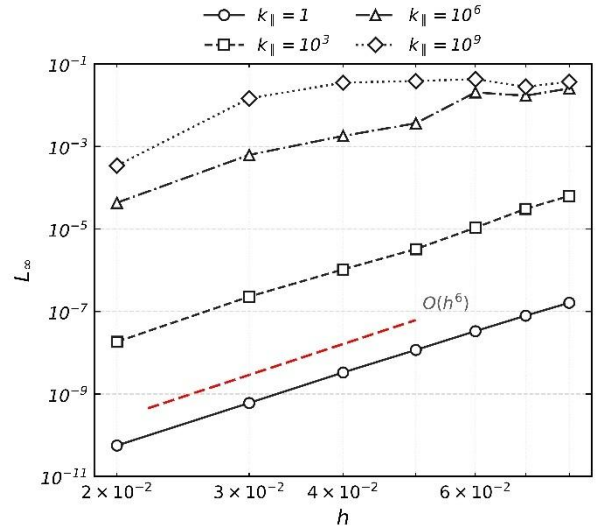


Fig. 2.  $L_{\infty}$  norm error convergence behavior under increasing transport anisotropy.

| $\kappa_{\parallel}$ | $L_2$ error<br>( $h=0.02$ ) | $L_{\infty}$ error<br>( $h=0.02$ ) | $P_{L_2}$ | $P_{L_{\infty}}$ |
|----------------------|-----------------------------|------------------------------------|-----------|------------------|
| 1                    | $2.64 \times 10^{-11}$      | $5.59 \times 10^{-11}$             | 5.82      | 5.88             |
| $10^3$               | $8.56 \times 10^{-9}$       | $1.83 \times 10^{-8}$              | 5.72      | 5.86             |
| $10^6$               | $2.67 \times 10^{-5}$       | $4.30 \times 10^{-5}$              | 5.10      | 5.46             |
| $10^9$               | $2.06 \times 10^{-4}$       | $3.42 \times 10^{-4}$              | 6.19      | 6.85             |

Table 1. Errors on the finest mesh and observed convergence orders.

Figs. 1 and 2, together with Table 1, summarize the  $L_2$  and  $L_{\infty}$  errors and the corresponding observed convergence orders for different  $\kappa_{\parallel}$ . The observed orders,  $P_{L_2}$  and  $P_{L_{\infty}}$ , were obtained from the slopes of the  $L_2$  and  $L_{\infty}$  error curves, respectively, assuming  $E(h) \propto h^p$ . To estimate the logarithmic convergence behavior, the slopes  $P_{L_2}$  and  $P_{L_{\infty}}$  were evaluated by least-squares fitting  $\log(L_{2,\infty})$  versus  $\log(h)$  using the three finest meshes ( $h=0.04, 0.03, 0.02$ ).

For  $\kappa_{\parallel} = 1$  and  $\kappa_{\parallel} = 10^3$ , the observed convergence orders are close to the theoretical 6th-order convergence. As the anisotropy increases, non-monotonic behavior appears on coarse meshes and recovers near-6<sup>th</sup>-order convergence on the finest three meshes, while the  $\kappa_{\parallel} = 10^9$  case remains under-resolved until the finest resolutions. Although the fitted slopes for  $\kappa_{\parallel} = 10^9$  exceed 6, they represent a transition from an error plateau to a resolved regime rather than a higher formal order. These results show that stronger anisotropy requires finer meshes to recover the designed high-order convergence, while the present non-field-aligned discretization remains convergent even under extreme anisotropy.

### 3.2. Hydrodynamic Shock

2D cylindrical explosion problem [14] is considered to investigate the capability of dealing with discontinuous regions. This problem estimates a smooth transition between high-order reconstruction in smooth regions and limited reconstruction in discontinuous regions, which is crucial for the robustness of the global solution.

This problem solves the 2D compressible Euler equation on square  $[-1, 1] \times [-1, 1]$  domain with the following initial condition:

$$\begin{aligned} \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) &= 0, \\ \frac{\partial \rho \mathbf{V}}{\partial t} + \nabla \cdot (\rho \mathbf{V} \mathbf{V} + p \mathbf{\tilde{I}}) &= 0, \\ \frac{\partial E}{\partial t} + \nabla \cdot ((E + p) \mathbf{V}) &= 0 \end{aligned} \quad (17)$$

$$\rho = \begin{cases} 1 & R < 0.4 \\ 0.125 & R \geq 0.4 \end{cases}$$

$$E = \begin{cases} 2.5 & R < 0.4 \\ 0.25 & R \geq 0.4 \end{cases}$$

$$u = v = 0, E = \frac{p}{\gamma - 1} + \frac{1}{2} \rho V^2, \gamma = 1.4$$

$$R = \sqrt{x^2 + y^2}, \quad \mathbf{V} = u \hat{\mathbf{x}} + v \hat{\mathbf{y}} \quad (18)$$

Due to initial discontinuities, rarefaction waves form inward as the calculation progresses, while contact discontinuities and outward-propagating shocks occur outward. Therefore, smooth expansion regions and discontinuities are formed simultaneously within a single calculation, making it suitable for evaluating both the high-order accuracy and limiting behavior of the CENO reconstruction.

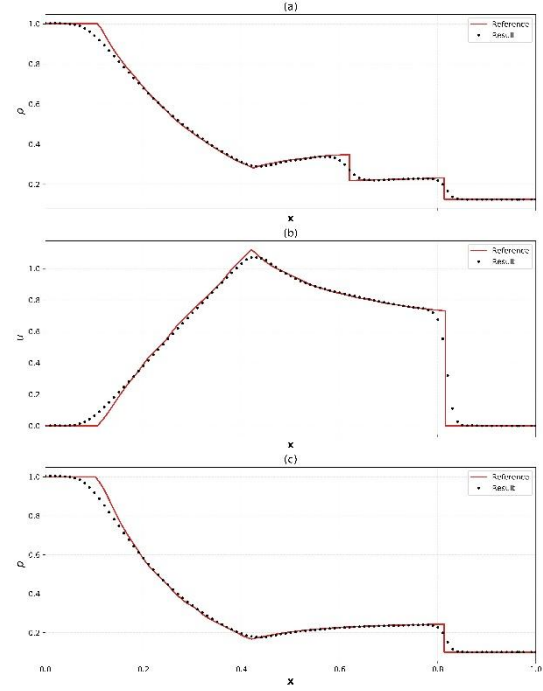


Fig. 3. Comparison of the result (black dotted line) with the reference solution (red line) for the 2D cylindrical explosion problem: (a) density, (b) velocity magnitude, and (c) pressure.

Fig. 3 compares the calculated density, velocity magnitude, and pressure profiles obtained using 23,650 cells ( $h=0.02$ ) at  $t=0.25$  with the reference solution. The calculation results reproduce the overall wave structure, consisting of rarefaction, contact discontinuity, and shock, as well as the respective propagation locations.

However, results show that shocks are broadly captured in all three variables. Also, although the location of the maximum velocity magnitude aligns well with reference, the peak value is slightly lower and rounded. This difference stems from limited linear reconstruction in the strong-gradient region, which degrades the solution accuracy in discontinuous regions. Nevertheless, no non-physical oscillations are observed around the discontinuities in density, velocity, or pressure.

### 3.3. Ideal-MHD Rotor Problem

Further evaluation is required to determine whether the current framework can account for more complex hyperbolic systems coupled with magnetic fields. The ideal-MHD rotor problem [15] is considered a well-known benchmark test for ideal-MHD. In the rotor problem, a rotating fluid in the  $[0, 1] \times [0, 1]$  domain twists the initial uniform magnetic field, generating strong magnetic tension. Therefore, we can evaluate the robustness of the reconstruction scheme under conditions where strong coupling exists among velocity, pressure, and the magnetic field, as well as in the presence of hydrodynamic discontinuities.

The governing equations and initial conditions are shown below:

$$\begin{aligned}\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) &= 0, \\ \frac{\partial \rho \mathbf{V}}{\partial t} + \nabla \cdot \left( \rho \mathbf{V} \mathbf{V} + \left( p + \frac{B^2}{2} \right) \tilde{\mathbf{I}} - \mathbf{B} \mathbf{B} \right) &= 0, \\ \frac{\partial E}{\partial t} + \nabla \cdot \left( \left( E + p + \frac{B^2}{2} \right) \mathbf{V} - (\mathbf{V} \cdot \mathbf{B}) \mathbf{B} \right) &= 0, \\ \frac{\partial \mathbf{B}}{\partial t} + \nabla \cdot (\mathbf{V} \mathbf{B} - \mathbf{B} \mathbf{V} + \psi \tilde{\mathbf{I}}) &= 0, \\ \frac{\partial \psi}{\partial t} + c_h^2 \nabla \cdot \mathbf{B} &= -\sigma_\psi \psi, \\ c_h &= \kappa_h \max_f (|\mathbf{V} \cdot \hat{\mathbf{n}}_f| + c_f), \sigma_\psi = \alpha \frac{c_h}{h_{min}}, \\ c_f &= \sqrt{\frac{1}{2} (a^2 + b^2 + \sqrt{(a^2 + b^2)^2 - 4a^2 b_n^2})}, \\ a^2 &= \frac{\gamma p}{\rho}, b^2 = \frac{B_x^2 + B_y^2}{\rho}, b_n^2 = \frac{(\mathbf{B} \cdot \hat{\mathbf{n}}_f)^2}{\rho}. \quad (19)\end{aligned}$$

To control numerical errors associated with the divergence-free constraint, the generalized Lagrange multiplier (GLM) formulation [16] is employed by introducing an auxiliary scalar  $\psi$ .

$$\begin{aligned}\rho &= \begin{cases} 10 & r \leq r_0 \\ 1 + 9f(r) & r_0 < r < r_1 \\ 1 & r \geq r_1 \end{cases} \\ u &= \begin{cases} -2(y - 0.5)/r_0 & r \leq r_0 \\ -2f(r)(y - 0.5)/r_0 & r_0 < r < r_1 \\ 0 & r \geq r_1 \end{cases} \\ v &= \begin{cases} 2(x - 0.5)/r_0 & r \leq r_0 \\ 2f(r)(x - 0.5)/r_0 & r_0 < r < r_1 \\ 0 & r \geq r_1 \end{cases}\end{aligned}$$

$$E = \frac{p}{\gamma - 1} + \frac{1}{2} \rho V^2 + \frac{B^2}{2},$$

$$\gamma = 1.4, \kappa_h = 1, \alpha = 0.1,$$

$$p = 1, B_x = \frac{5}{\sqrt{4\pi}}, B_y = 0,$$

$$r_0 = 0.1, r_1 = 0.115, f(r) = \frac{r_1 - r}{r_1 - r_0}$$

$$r = \sqrt{(x - 0.5)^2 + (y - 0.5)^2} \quad (20)$$

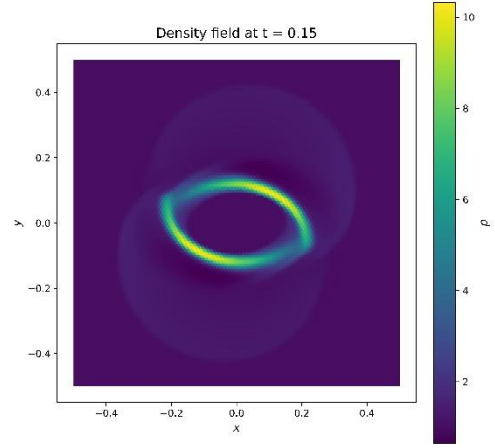


Fig. 4. Density contour of the ideal-MHD rotor problem.

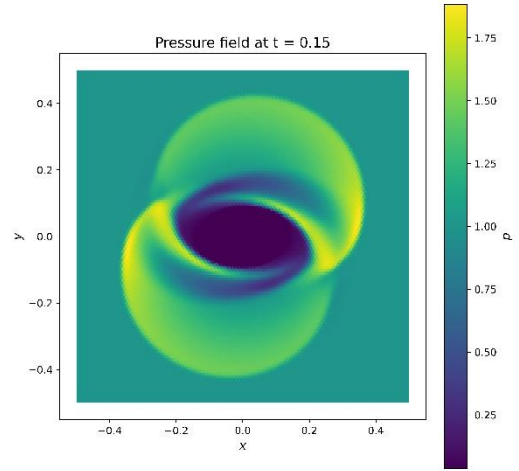


Fig. 5. Pressure contour of the ideal-MHD rotor problem.

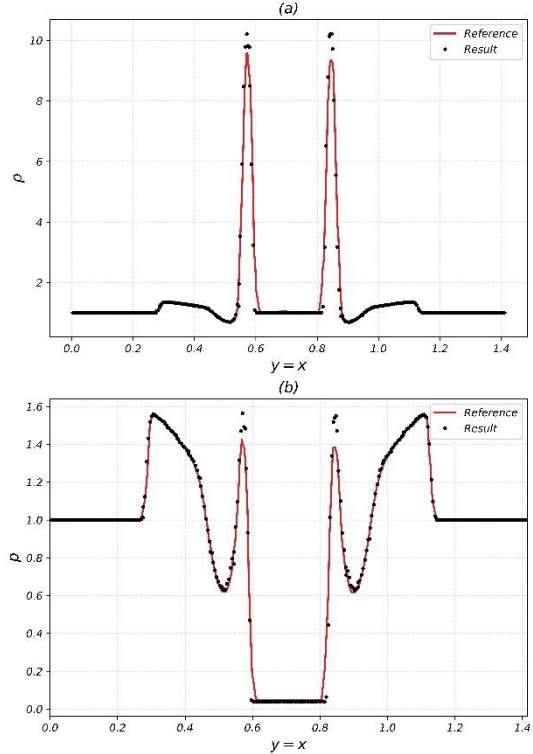


Fig. 6. Comparison of (a) density and (b) pressure results (black dotted line) with the reference solution (red line) along  $y=x$ .

23,662 cells ( $h = 0.01$ ) are used in this calculation, and the results at  $t=0.15$  are compared with a reference solution computed on a  $400 \times 400$  structured grid [17]. Fig. 6 shows the comparison between the reference solution and the calculated density and pressure profiles extracted along the  $y=x$  direction. Overall, the numerical results closely follow the reference profiles. In the peak regions, however, the calculated solution exhibits slightly higher and sharper peaks than the reference, particularly in the density profile. Since the reference profile is also obtained from a finite-resolution numerical calculation, these local differences do not necessarily indicate an overshoot of the present solution.

In steep-gradient regions, the sampled numerical values appear to be locally scattered around the reference curve. The present profiles are constructed by directly extracting cell-averaged values from unstructured cells closest to the  $y=x$  line, rather than interpolating the solution exactly onto the sampling line. Therefore, these variations may arise from the current sampling strategy. Nevertheless, no oscillatory behavior is observed around the sharp gradients, while the smooth portions of both density and pressure remain in good agreement with the reference solution.

#### 4. Preliminary Tokamak Transport Application

In this section, we performed preliminary calculations to investigate whether the current framework can be applied to plasma transport modeling in actual tokamak geometry. The purpose of these calculations is to assess whether the currently implemented numerical schemes operate consistently in an unstructured KSTAR tokamak geometry.

The simplified Braginskii equations introduced in Section 2 were used in these calculations on the magnetic equilibrium of KSTAR shot 25460 at  $t = 7.3$  s, with a characteristic mesh size of  $h = 0.02$  m. Atomic-reaction terms, neutral-plasma interaction, drift-related transport, detailed sheath boundary condition, and a self-consistent anomalous transport closure are outside the scope of the present numerical demonstration.

At the core boundary, all the variables were held fixed throughout the simulation. At the wall boundary, the flux condition was selected according to the angle  $\theta$  between the outward wall-normal vector and the local magnetic-field direction. The  $u_{\parallel}$  based flux was used for nearly tangential field incidence, whereas the sound-speed-based outflow condition was applied otherwise:

$$n_{core} = 10^{19} \text{ m}^{-3}, u_{\parallel} = 0, T_i = T_e = 50 \text{ eV},$$

$$D_{\perp} = \eta_{\perp} = \kappa_{\perp} = 0.1 \text{ m}^2/\text{s},$$

$$q_{i,e}|_{wall} = \gamma_{i,e} n T_{i,e} \Gamma_{wall},$$

$$\Gamma_{wall} = \begin{cases} |u_{\parallel} \hat{\mathbf{b}} \cdot \hat{\mathbf{n}}_{wall}| & 85^\circ < \theta < 95^\circ \\ |C_s \hat{\mathbf{b}} \cdot \hat{\mathbf{n}}_{wall}| & \text{else} \end{cases} \quad (21)$$

At the wall, extrapolated values were used for all variables in the outflow-flux calculation.

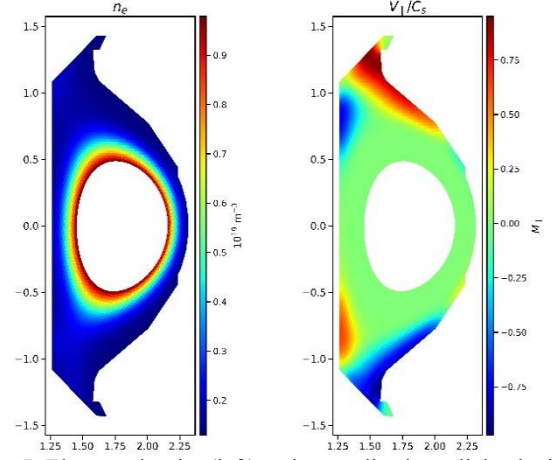


Fig. 7. Electron density (left) and normalized parallel velocity (right) in the preliminary tokamak transport calculation.

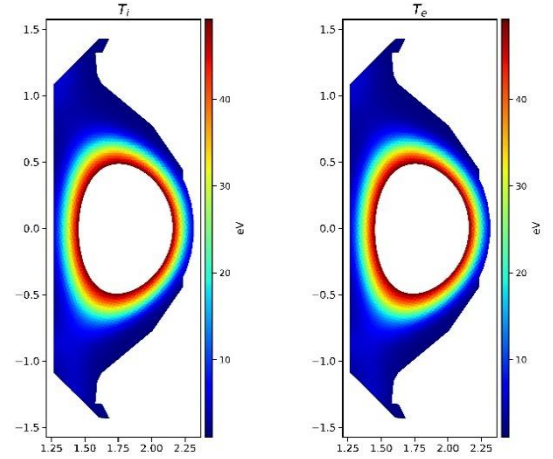


Fig. 8. Ion temperature (left) and electron temperature (right) in the preliminary tokamak transport calculation.

Figs. 7 and 8 show the two-dimensional distributions of the four calculated primitive variables. The results show that a solution was stably formed throughout the computational domain, which was filled with unstructured meshes and featured complex wall geometry. In addition, the spatial variation induced by the imposed boundary conditions and transport operators was observable. Unlike the Cartesian test geometry used in the benchmark problems discussed in the previous section, it was confirmed that the same reconstruction and residual evaluation procedures could be applied to curved boundaries and to unstructured meshes.

The results are not intended to evaluate the predictive accuracy of the transport model. Nevertheless, these calculations demonstrate that the current numerical framework provides a basic 2D transport modeling infrastructure for incorporating more complete tokamak physics modules in future work.

#### 5. Summary and Future Work

We developed the 2D unstructured-grid plasma transport framework to flexibly respond to time-varying magnetic configurations and complex tokamak geometries. The CENO reconstruction and flux schemes were applied to obtain high-order accuracy in smooth regions and to suppress numerical instability in discontinuous regions. Additionally, a PETSc-based parallel linear solver and preconditioner were employed to solve algebraic systems generated from strong anisotropic diffusion.

In the anisotropic diffusion benchmark, it was found that numerical error on the same mesh increased with increasing parallel diffusivity, and that a finer mesh at a certain level was required under extreme anisotropy. In the compressible Euler explosion and ideal-MHD rotor benchmarks, the major locations and morphologies of shock, contact discontinuity, and coupled MHD wave structures generally matched the reference solutions. The Euler explosion test exhibited limited shock smearing associated with the limited linear reconstruction, whereas the ideal-MHD rotor test retained sharp compression structures without non-physical oscillations.

Finally, we applied simplified Braginskii equations to the KSTAR geometry. This result did not show detailed edge/SOL physics simulations or transport model validation, but it demonstrates that the spatial discretization and solution procedures verified in numerical benchmarks can be applied to tokamak plasma transport calculations.

We are currently incorporating atomic-reaction terms and a RANS-inspired mean-field anomalous-transport closure [5]. This work will provide greater predictive power for modeling plasma-wall interactions and SOL/edge physics. Also, by combining dedicated core transport closure with evolving magnetic equilibrium in the future, we intend to expand the scope of application to limiter-to-diverted transitions and two-dimensional whole-cross-section transport calculations.

## REFERENCES

- [1] M. Scotto d'Abusco *et al.*, "Core-Edge 2D fluid modeling of full tokamak discharge with varying magnetic equilibrium: From WEST start-up to ramp-down," *Nuclear Fusion*, vol. 62, no. 8, p. 086002, May. 2022.
- [2] J. M. Park, "Integrated simulation of tokamak H-mode operation by a two-dimensional plasma transport code in coupled core-edge pedestal-SOL region," Ph.D. Thesis, 2005.
- [3] D.-K. Kim and S. H. Hong, "Fluid-particle hybrid simulation on the transports of plasma, recycling neutrals, and carbon impurities in the Korea superconducting tokamak advanced research divertor region," *Physics of Plasmas*, vol. 12, no. 6, May 2005.
- [4] J.-W. Park *et al.*, "Simulation of tokamak sol and divertor region including heat flux mitigation by gas puffing," *Journal of the Korean Physical Society*, vol. 61, no. 3, pp. 387–396, Aug. 2012.
- [5] H. Bufferand *et al.*, "Hierarchy of turbulent transport models with the SOLEDGE3X code," *Nuclear Fusion*, vol. 66, no. 6, p. 066029, Jun. 2026.
- [6] T. Barth and P. Frederickson, "Higher order solution of the Euler equations on unstructured grids using quadratic reconstruction," *28th Aerospace Sciences Meeting*, Jan. 1990. doi:10.2514/6.1990-13
- [7] M. R. Charest, C. P. Groth, and P. Q. Gauthier, "A high-order Central ENO finite-volume scheme for three-dimensional low-speed viscous flows on unstructured mesh," *Communications in Computational Physics*, vol. 17, no. 3, pp. 615–656, 2015.
- [8] S. Balay *et al.*, *PETSC/TAO users manual revision 3.24*, Sep. 2025.
- [9] N. M. Li *et al.*, "Simulations of divertor heat flux width using transport code with cross-field drifts under the bout++ framework," *AIP Advances*, vol. 10, no. 1, Jan. 2020.
- [10] E. F. Toro, B. Saggiorato, S. Tokareva, and A. Hidalgo, "Low-dissipation centred schemes for hyperbolic equations in conservative and non-conservative form," *Journal of Computational Physics*, vol. 416, p. 109545, Sep. 2020.
- [11] H. Nishikawa, "Beyond interface gradient: A general principle for constructing diffusion schemes," *40th Fluid Dynamics Conference and Exhibit*, Jun. 2010.
- [12] H. Nishikawa, "New unstructured-grid limiter functions," *AIAA SCITECH 2022 Forum*, Jan. 2022.
- [13] C. R. Sovinec *et al.*, "Nonlinear magnetohydrodynamics simulation using high-order finite elements," *Journal of Computational Physics*, vol. 195, no. 1, pp. 355–386, Mar. 2004.
- [14] E. F. Toro, *Riemann Solvers and Numerical Methods for Fluid Dynamics: A Practical Introduction*. Berlin, Heidelberg: Springer Berlin Heidelberg, 2009.
- [15] D. S. Balsara and D. S. Spicer, "A staggered mesh algorithm using high order Godunov fluxes to ensure solenoidal magnetic fields in magnetohydrodynamic simulations," *Journal of Computational Physics*, vol. 149, no. 2, pp. 270–292, 1999.
- [16] A. Dedner *et al.*, "Hyperbolic divergence cleaning for the MHD equations," *Journal of Computational Physics*, vol. 175, no. 2, pp. 645–673, Jan. 2002.
- [17] E. N. Avdeeva and V. V. Lukin, "Divergence-free finite-difference method for 2D ideal MHD," *Journal of Physics: Conference Series*, vol. 1336, no. 1, p. 012026, Nov. 2019.