

Moving Discontinuous Galerkin Methods for Viscous Conservation Laws

Gianni Absillis*, Hong Luo*, Robert Nourgaliev**, and Patrick Greene**

Corresponding author: gabsill@ncsu.edu

* NC State University, USA.

** Lawrence Livermore National Laboratory, USA.

Abstract: Discontinuities and highly anisotropic flow features represent a considerable numerical challenge for high order discretization techniques. These challenges appear commonly in hypersonic flows, where extremely thin boundary layers and shocks are difficult to resolve, yet extremely important to represent accurately to effectively predict surface heating and shear forces. We discuss the application of Moving Discontinuous Galerkin with Interface Condition Enforcement (MDG-ICE) methods to viscous conservation laws. These methods promise to accurately represent extremely anisotropic features and ultra-thin regions with large gradients. We present a novel formulation of MDG-ICE for viscous conservation laws and contrast it with existing formulations of MDG-ICE. We validate this new approach with three test problems, designed to model the numerical challenges encountered in hypersonic flows, where without additional treatment, standard DG methods succumb to spurious oscillations.

Keywords: Numerical Algorithms, Computational Fluid Dynamics, R-adaptation, Mesh optimization.

1. Introduction

Moving Discontinuous Galerkin with Interface Condition Enforcement (MDG-ICE) methods were developed by Corrigan et al.[1] to solve conservation laws extremely accurately. The methods are a form of implicit r-adaptation, meaning that the mesh moves to more accurately (and in some cases exactly) represent the solution within the nonlinear solution procedure. This class is shared with the similar high-order implicit shock tracking (HOIST) method of Persson et al. [2]. MDG-ICE is especially well suited to problems with discontinuities and singularities; it moves the mesh to align with discontinuities allowing exact representation of the solution in the discontinuous function space. This is done without a-priori knowledge of discontinuity location and without introducing artificial dissipation such as limiters or artificial viscosity.

The prevailing approach in the current state of the practice is termed shock-capturing, where shock-like features are detected and treated with some form of additional dissipation and local refinement of the mesh (see the works by Persson and Peraire [3], Kitamura and Shima[4], and Kuzmin [5]). This approach is effective to maintain stability but locally reduces the approximation order of the solution at the shock; hence the local refinement to recover lost accuracy. For extremely strong shocks, isotropic refinement may not recover accuracy effectively enough, and anisotropic refinement presents its own class of challenges and area of research. This is where implicit r-adaptation methods represent an interesting alternative approach.

Kercher et al. [6, 7] have introduced a formulation of MDG-ICE for viscous conservation laws using a common transformation of second order diffusion terms to a system of first order equations. This formulation of MDG-ICE successfully moved the mesh to resolve boundary layers and viscous shocks, characterized by extremely thin regions with large gradients. For large systems of equations and higher dimensions, the transformation to a system of first order differential equations may drastically increase the size of the system being solved. The presented formulation also does not employ numerical fluxes

and a weak form for the viscous terms in the conservation law. We will investigate an approach that does not require the introduction of the full first order system and can continue to utilize fluxes and weak formulations, which will be more familiar to the majority of DG implementations.

2. Methodology

Consider a conservation law on the domain $\Omega_x \subset \mathbb{R}^{d_x}$ of a set of state variables $\mathbf{U} \in \mathbb{R}^m$ with convective fluxes $\mathbf{F}_j^c(\mathbf{U}) \in \mathbb{R}^{m \times d_x}$ and diffusive fluxes $\mathbf{F}_j^v(\mathbf{U}, \nabla \mathbf{U}) \in \mathbb{R}^{m \times d_x}$

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}_j^c(\mathbf{U})}{\partial x_j} - \frac{\partial \mathbf{F}_j^v(\mathbf{U}, \nabla \mathbf{U})}{\partial x_j} = 0 \quad \text{in } \Omega_x, \quad (1)$$

subject to appropriate boundary conditions on $\partial\Omega$. MDG-ICE operates on steady-state or spacetime discretized problems, ensuring that the r-adaptation is time accurate and can represent moving shocks. Reformulating the time-dependent conservation law to a space-time conservation law for time $t \in [0, T]$ can be done by the simple transformation

$$\frac{\partial \mathcal{F}_j^c}{\partial x_j} - \frac{\partial \mathcal{F}_j^v}{\partial x_j} = 0 \quad \text{in } \Omega_x \times [0, T]; \quad \mathcal{F}_j^c = \begin{bmatrix} \mathbf{F}_j^c \\ \mathbf{U} \end{bmatrix}, \quad \mathcal{F}_j^v = \begin{bmatrix} \mathbf{F}_j^v \\ 0 \end{bmatrix}. \quad (2)$$

MDG-ICE functions by simultaneously solving for the solution discretization as well as the mesh coordinates with an overdetermined system of equations comprised of the PDE discretization and additional interface condition equations. The PDE solution is represented by functions in the Sobolev space over the spacetime domain $\Omega = \Omega_x \times [0, T]$:

$$V_u = \{v \in [L^2(\Omega)]^m : \forall \mathcal{K} \in \mathcal{T}, v|_{\mathcal{K}} \in [H^1(\Omega)]^m\}, \quad (3)$$

where m is the number of state variables. The domain $\Omega \subset \mathbb{R}^d$ is broken into a tessellation $\mathcal{T}$ of non-overlapping elements $\mathcal{K}$. The formulation of Kercher et al. [6] also requires an auxiliary variable in the Sobolev space:

$$V_\sigma = \{v \in [L^2(\Omega)]^{m \times d} : \forall \mathcal{K} \in \mathcal{T}, v|_{\mathcal{K}} \in [H^1(\Omega)]^{m \times d}\}. \quad (4)$$

The product space of the functions on elements is a finite element space $\mathcal{X}$ as described by Ciarlet [8]. To each element we ascribe a reference element domain $\hat{\mathcal{K}}$ with an invertible mapping from coordinates ξ in the reference domain to the coordinates $\mathbf{x}$ in the element physical domain $\mathcal{K}$. The product space of functions in the reference elements is $\hat{\mathcal{X}}$. The coordinates of the mesh can then be represented in the product space of reference elements for an $H^1(\Omega)^d$ space: this reference space we will denote as $H^1(\hat{\Omega})$. Then the geometry discretization is in the space:

$$V_x = \{v \in [H^1(\hat{\Omega})]^d\}. \quad (5)$$

To obtain a discretization the infinite dimensional Sobolev spaces are replaced with discrete counterparts $V_{u,h}$ and $V_{\sigma,h}$: these are comprised of polynomials of up to degree N . V_x is also substituted with discrete counterparts, specifically nodal basis functions that are polynomials of up to degree N_x . If $N_x = N$ then the method is said to be *isoparametric*.

For the conservation law over the domain, we use the weak form

$$-\int_{\Omega} \mathbf{F}^c \cdot \nabla v \, d\mathbf{x} + \int_{\Gamma} \hat{\mathbf{F}}^c \cdot \llbracket v \rrbracket \, ds + \int_{\Omega} \mathbf{F}^v \cdot \nabla v \, d\mathbf{x} - \int_{\Gamma} \hat{\mathbf{F}}^v \cdot \llbracket v \rrbracket \, ds = 0 \quad \forall v \in V_u, \quad (6)$$

where $\hat{\mathbf{F}}$ represents a numerical flux that gives consistent, conservative, single-valued outputs for functions that can be multi-valued at the interfaces. The union of element interfaces is represented by Γ and the

jump operator is defined by

$$[[v]] = v \mathbf{n}^L + v \mathbf{n}^R, \quad (7)$$

with exterior unit normals for elements on either side of the interface: $\mathbf{n}^L$ for $\mathcal{K}^L$ and $\mathbf{n}^R$ for $\mathcal{K}^R$. For vectors $\mathbf{q} \in \mathbb{R}^d$

$$[[\mathbf{q}]] = \mathbf{q} \cdot \mathbf{n}^L + \mathbf{q} \cdot \mathbf{n}^R. \quad (8)$$

In addition to the conservation law over the domain, MDG-ICE incorporates an *interface condition*, which is the conservation law enforced over the element trace spaces. Consider a 0-width control volume at the interface between two elements. The flux going into one side of the control volume has to be equal to the flux going out of the other side – this is the intuition and premise for the interface condition:

$$[[\mathcal{F}]] = 0. \quad (9)$$

Here we have combined the fluxes $\mathcal{F} = \mathcal{F}^c - \mathcal{F}^v$. Note that this uses the vector definition of the jump operator (Equation 8). We construct a weak form for this interface condition by integrating with test functions in the single valued trace space corresponding to the geometry degrees of freedom: $[H^1(\hat{\Omega})]^m$, which we will denote as W .

$$\int_{\Gamma} \beta_{IC,1}(\Delta) [[\mathcal{F}]]_w ds = 0, \quad \forall w \in W, \quad (10)$$

where $\beta_{IC,1}$ is a scaling function of a characteristic distance between elements on either side of the interface Δ . We use the characteristic distance presented by Cheng et al. [9]. For two elements on either side of an interface with centroids $\mathbf{c}^L$ and $\mathbf{c}^R$ respectively, at a quadrature point $\mathbf{q}_{\Gamma}$ with unit normal exterior to the L element $\mathbf{n} = \mathbf{n}^L$ the characteristic distance is computed by:

$$\Delta = \left| (\mathbf{q}_{\Gamma} - \mathbf{c}^L) \cdot \mathbf{n} \right| + \left| (\mathbf{q}_{\Gamma} - \mathbf{c}^R) \cdot \mathbf{n} \right|. \quad (11)$$

This scaling improves the conditioning of the nonlinear optimization problem. Finally, the constitutive relations, which are a consequence of the diffusive fluxes, impose a continuity condition on $\mathbf{U}$. This is represented in the weak form:

$$\int_{\Gamma} \beta_{IC,2}(\Delta) \|\mathbf{G}\|_F [[\mathbf{U}]] \cdot \mathbf{n} w_2 ds = 0, \quad \forall w_2 \in W_2. \quad (12)$$

Here $\mathbf{G}$ is the homogeneity tensor of $\mathcal{F}^v$ (the tensor such that $\mathbf{G} \nabla \mathbf{U} = \mathcal{F}^v$), and w_2 is another test function on the space $W_2 = H^1(\hat{\Omega})$. We will refer to the formulation using the two interface conditions in Equations 10 and 12 as the “second order system with continuity” formulation.

There is another way to enforce the continuity in the second order system sense. In the formation of compact numerical fluxes through lifting operators as outlined by Arnold et al. [10], the replacement for the vector-valued (or in our case tensor-valued) test functions τ is the gradient of the test functions ∇v . Applying this same approach to the constitutive interface condition gives the weak form:

$$\int_{\Gamma} \beta_{IC,1}(\Delta) [[\mathbf{u}]] \cdot \{\mathbf{G}^T\} \nabla w ds - \int_{\Gamma^0} \beta_{IC,2}(\Delta) [[\mathcal{F}]]_w ds = 0 \quad \forall w \in W. \quad (13)$$

As independent variables we have $\mathbf{U} \in V_{u,h}$ and $\mathbf{X} \in V_{x,h}$. The objective function $\mathbf{r}$, comprised of the three weak forms (Equations 6, 10, 12), is solved using a regularized Gauss-Newton method using Levenberg-Marquardt and other regularizations detailed in [6]. Let $\mathbf{J}$ represent the sensitivities of $\mathbf{r}$ with respect to $\mathbf{U}$ and $\mathbf{X}$. Updates $\mathbf{p}$ to the solution representation and mesh are computed by

$$(\mathbf{J}^T \mathbf{J} + \mathbf{R}) \mathbf{p} = \mathbf{J}^T \mathbf{r}, \quad (14)$$

where $\mathbf{R}$ is a positive definite regularization matrix. A purely diagonal $\mathbf{R}$ would recover a Levenberg-

Marquardt method.

2.1 Regularization terms

For the Moving Discontinuous Galerkin method in particular it is beneficial to have separate regularization parameters for entries corresponding to the PDE solution field λ_u and entries corresponding to the geometric parameters λ_X . Moré [11] provides a detailed analysis of the implementation of trust-region based Levenberg-Marquardt algorithms in which the scaling of the regularization term is represented by matrices $\mathbf{D}$ with diagonal entries $D_{ii} = d_i$ to form the regularization matrix:

$$\mathbf{R} = \mathbf{D}^T \mathbf{D}. \quad (15)$$

For the entries of the diagonal scaling matrix, a choice recommended by Moré is based on the columns of the Jacobian matrix $\mathbf{J}$ and the iteration k

$$d_i^{(0)} = \|\partial_i \mathbf{r}^{(0)}\| = \mathbf{J}^{(0)} \mathbf{e}_i, \quad (16)$$

$$d_i^{(k)} = \max \left\{ d_i^{(k-1)}, \|\partial_i \mathbf{r}^{(k)}\| \right\}, \quad k \geq 1, \quad (17)$$

where $\mathbf{e}_i$ is the unit vector that is 1 at index i and zero elsewhere. In effect this scales the entry by the maximum corresponding Jacobian column norm over all nonlinear iterations. We have found, since the d_i terms get squared by $\mathbf{D}^T \mathbf{D}$, a choice of

$$d_i^{(k)} = \max \left\{ d_i^{(k-1)}, \sqrt{\|\partial_i \mathbf{r}^{(k)}\|} \right\}, \quad k \geq 1, \quad (18)$$

has less variance and has more robust convergence properties for the test problems we examined. Another option is to scale by the norm of the residual itself:

$$d_i^{(k)} = \|\mathbf{r}^{(k)}\|^{\delta/2}, \quad (19)$$

where $\delta = 1$ is used by Kercher et al. [6], Fan and Pan [12] recommend $\delta = 2$. Note again the diagonal terms get squared in the normal equations (Equations 14 and 15).

Kercher et al. [6] employ a Laplacian grid regularization that smooths the resulting mesh motion by adding a dissipative kernel to the linearized subproblem operator. The dissipative kernel is constructed by forming a stiffness matrix with a coefficient of the regularization parameter. First, let us define the bilinear form

$$a_\Delta(\mathbf{y}, \mathbf{v}) = \int_\Omega \lambda_\Delta \nabla \mathbf{y} \cdot \nabla \mathbf{v} \, d\mathbf{x}. \quad (20)$$

Then the discrete analogue forms the stiffness matrix by using the basis $\phi_j : \text{span}\{\phi_j\}_{j=0}^{N-1} = V_x$:

$$\mathbf{R}_\Delta = [a_\Delta(\phi_i, \phi_j)]_{i,j} \in \mathbb{R}^{N \times N}. \quad (21)$$

2.2 Linear advection diffusion

As a first test case we consider the following linear advection-diffusion problem

$$a \frac{du}{dx} - \mu \frac{d^2 u}{dx^2} = 0 \quad \text{on } [0, L], \quad (22)$$

$$u(0) = u_L, \quad u(L) = u_R. \quad (23)$$

Where a is the advection velocity and μ is the diffusion coefficient. Solutions to this differential equation are characterized by the non-dimensional Péclet number

$$Pe = \frac{aL}{\mu}, \quad (24)$$

which is a ratio of advection to diffusion of the scalar quantity u . If we prescribe the Dirichlet boundary conditions on the domain $x \in [0, 1]$

$$u_L = u(0) = 0, \quad u_R = u(1) = 1.0,$$

this problem will form a boundary-layer-like profile with a Péclet number of 100. This has the analytical solution

$$u_{exact} = \frac{1 - \exp(x \cdot Pe)}{1 - \exp(Pe)}. \quad (25)$$

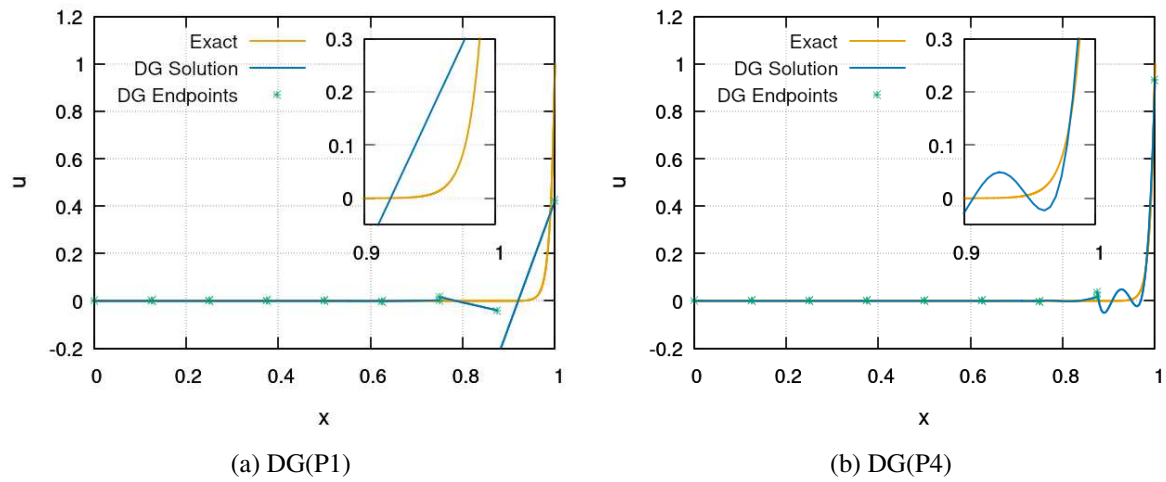


Figure 1: Standard DG solution of the boundary layer problem.

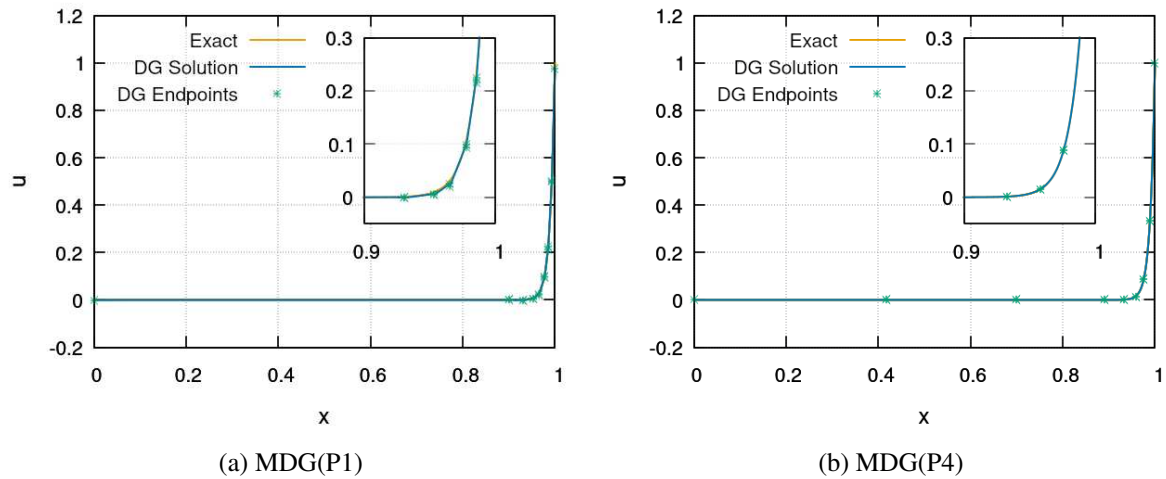


Figure 2: MDG solution of the boundary layer problem.

With a coarse 8 element mesh, there is not enough resolution to resolve the boundary layer profile without moving the mesh. Figure 1 shows how this results in spurious oscillations when using a standard discontinuous Galerkin method without artificial dissipation. This is remedied with the use of MDG-ICE

in Figure 2, where the mesh has moved into the boundary layer region to resolve the feature.

We perform a spectral convergence study, varying the polynomial order of the discretization of u (and σ for the second order formulation) over polynomial orders 1 through 5. All cases start with an initial uniform mesh of 8 elements and solution $u = 0$ (additionally, $\sigma = 0$ for the second order formulations). The geometry is linear so the coordinates are affine transformations of 9 nodes. Figure 3 compares the second order approaches to the original method by Kercher et al. [6] with 3 different splittings for the constitutive equation. The case labeled "Second Order MDG" corresponds to the case with $\beta_{IC,2} = 0$, the case labeled "Second Order MDG with Constitutive" corresponds to Equations 10 and 12 using $\beta_{IC,1} = \beta_{IC,2} = \Delta$, and the case labeled "Second Order Combined MDG" corresponds to Equation 13 with the same scaling terms.

The second order formulations perform marginally worse than the first order system approach but have half the degrees of freedom, and in the cases without the separated constitutive law, half the number of interface equations in the residual. The first order system approach tends to oscillate around the log-linear line more, generally performing better on odd polynomial orders. The choice of $G = \sqrt{\mu}$ as chosen by Kercher et al. [6], has the lowest error norms and seems least prone to solver stagnation issues among the first order system approaches. Solver stagnation reduced the performance of the other choices for G in the higher order DGP4 and DGP5 cases.

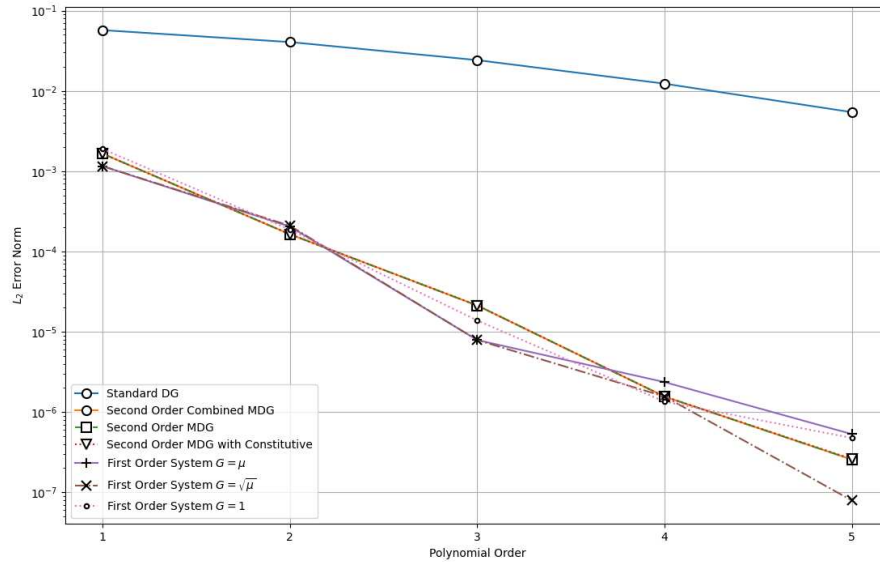


Figure 3: Convergence study of the 1D advection-diffusion problem simulating a boundary layer profile. All formulations of MDG-ICE greatly outperform standard DG (without artificial dissipation) in overall error and convergence rate. The second order formulations have marginally worse convergence rate than the first order system approach.

2.3 Spacetime viscous burger's equation

This example and the previous one are used by Kercher et al. [6] as validation cases. Consider the conservation law:

$$\frac{\partial u}{\partial t} + \frac{\partial}{\partial x} \left(\frac{1}{2} u^2 \right) - \mu \frac{\partial^2 u}{\partial x^2} = 0 \quad \text{on } x \in [0, 1], t \in [0, 1]. \quad (26)$$

subject to a free boundary condition at $x = 1$ and $t = 1$, a Dirichlet boundary condition $u = u_\infty = 0.2$ at $x = 0$, and an initial condition $u = \frac{1}{2\pi t_s} \sin(2\pi x) + u_\infty$ at $t = 0$ with a time-to-shock $t_s = 0.5$. This investigates the resolution of a viscous shock, which results in an extremely thin region where the state variable u rapidly changes. The jump is not truly discontinuous because of the introduction of the

viscosity term.

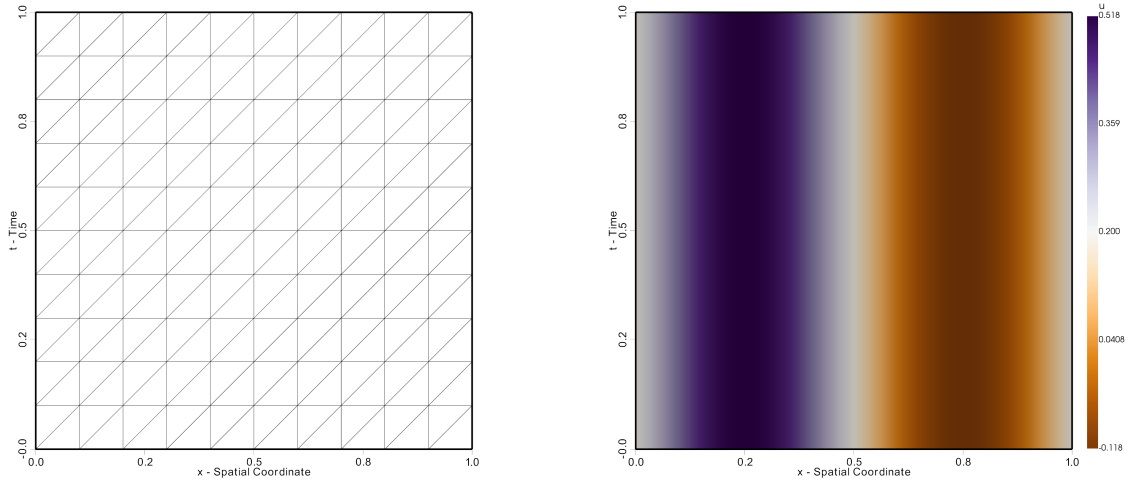
The solution field u is represented with 5th order polynomials on a mesh with linear geometry, which we notate as DG(P5/P1). No artificial diffusion is employed in the form of limiters, artificial viscosity, or other shock capturing techniques – these can distort the features resolved in the converged solution field. In addition to the weak form of the PDE and the interface conditions, we enforce one mesh quality term per element using the Target Mesh Optimization Paradigm (TMOP) developed by Knupp et al. [13]. We use the $\mu_{2,s}$ metric which optimizes for shape, with the optimal shape being an equilateral triangle. This residual needs to be scaled by a small quantity; the large magnitude of the mesh quality residuals with respect to the PDE residuals would otherwise stall convergence. The scalings chosen for each viscosity are:

- $10^{-6}(\mu_{2,s})^2$ for diffusion coefficient $\mu = 10^{-3}$,
- $10^{-8}\mu_{2,s}$ for diffusion coefficient $\mu = 5 \times 10^{-4}$,
- $10^{-10}\mu_{2,s}$ for diffusion coefficient $\mu = 10^{-4}$.

The regularization terms used are:

- $\lambda_u = 10^{-8}$ – diagonal regularization for the u degrees of freedom scaled by the column norms (see Equation 18).
- $\lambda_u = 5 \times 10^{-2}$ – diagonal regularization for the mesh degrees of freedom scaled by the square of the residual norm (Equation 19 with $\delta = 2$).
- $\lambda_\Delta = 10^{-2}$ – Laplacian grid regularization (Equations 21, and 20).

These regularization terms can be further tuned for each case independently to obtain faster convergence but for simplicity we choose to keep the same regularization for all 3 cases.



(a) Initial triangular mesh for the viscous Burgers' equation test case. The mesh has 200 linear triangle elements.

(b) Initial state of the space-time solution field u . The initial condition $u(x, 0)$ has been extrapolated in time for all t .

Figure 4: Initialization for the space-time Burgers' test problem $\mu = 10^{-3}$ case.

We generate 200 element mesh by subdividing a uniform 10×10 quadrilateral mesh into triangles using lines of positive slope in the x, t plane as shown in Figure 4. The initial solution field for the $\mu = 10^{-3}$

case is an extrapolation of the initial condition $u(x, 0)$ for all time $t \in [0, 1]$. For the lower diffusion coefficients, we employ viscosity continuation by initializing the mesh and solution field to the converged solution of the preceding case.

Figures 5 and 6 show the converged solution for the $\mu = 10^{-3}$ case. Figures 7 and 8 show the converged solution for the $\mu = 10^{-5}$ case. Finally Figures 9 and 10 show the converged solution for the most extreme viscous case we examined of $\mu = 10^{-4}$. The width of the shock feature at $t = 0.975$ is less than 0.01 units and is resolved without spurious oscillations and artificial dissipation by MDG-ICE.

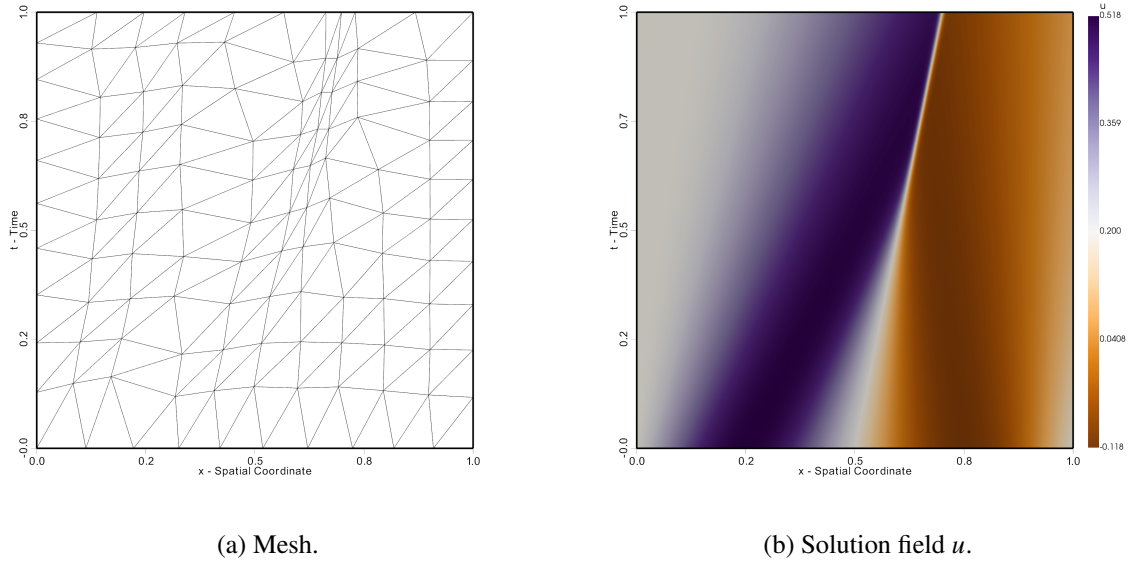


Figure 5: MDG-ICE solution and mesh for the viscous Burgers shock test problem with $\mu = 10^{-3}$.

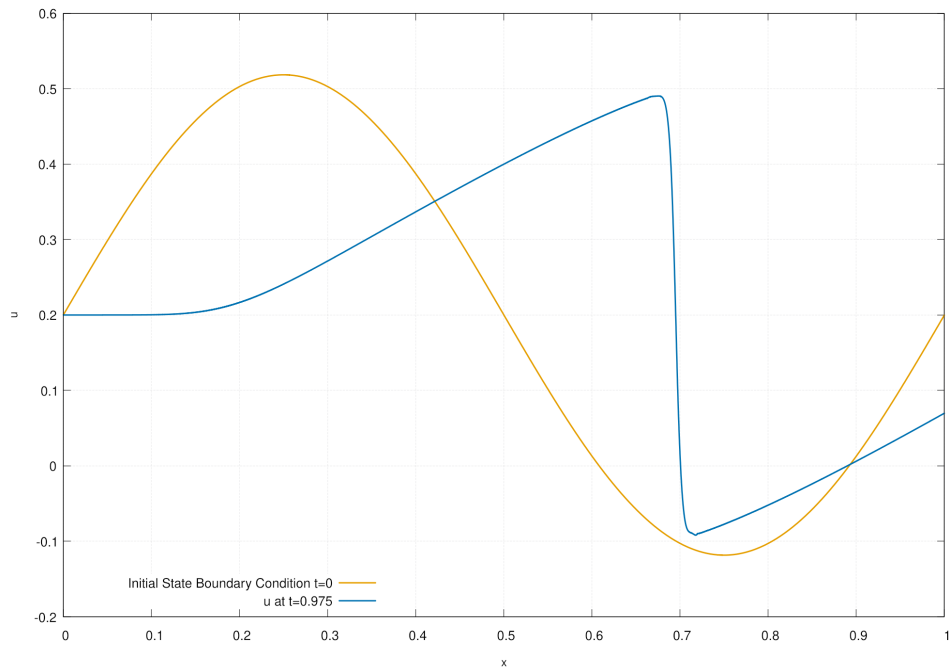
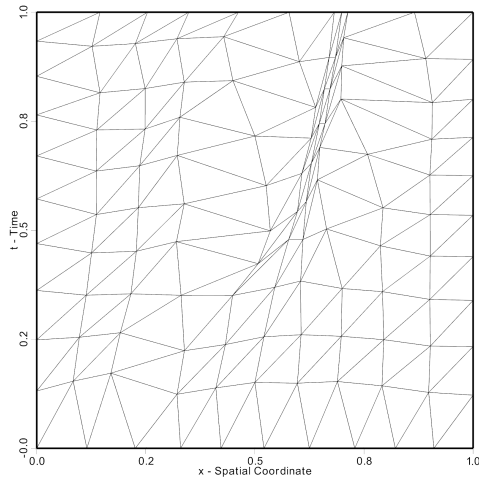
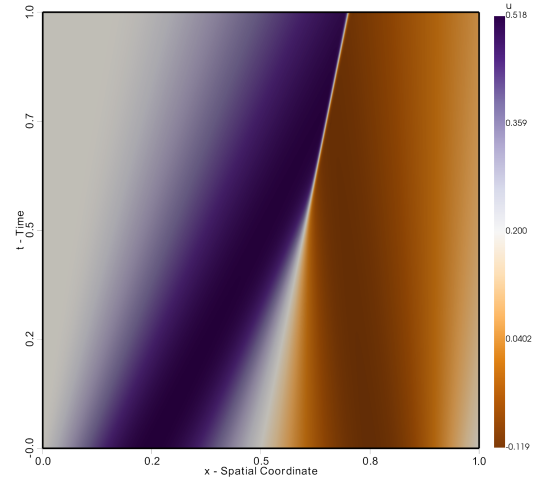


Figure 6: Cutline at $t = 0.975$ of the MDG(P5/P1) solution for the viscous Burgers shock test problem with $\mu = 10^{-3}$.



(a) Mesh.



(b) Solution field u .

Figure 7: MDG-ICE solution and mesh for the viscous Burgers shock test problem with $\mu = 5 \times 10^{-4}$.

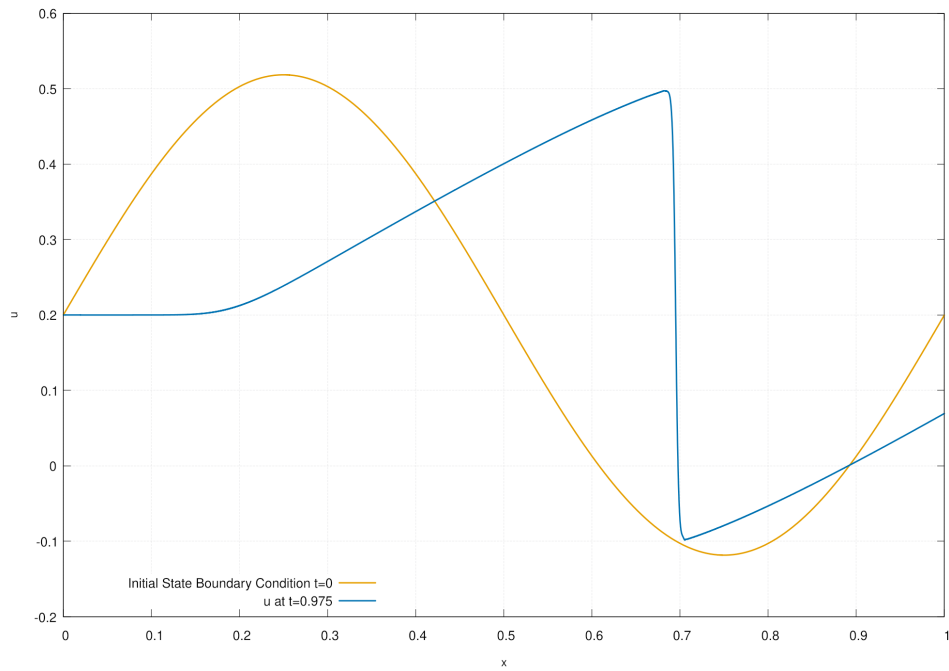


Figure 8: Cutline at $t = 0.975$ of the MDG(P5/P1) solution for the viscous Burgers shock test problem with $\mu = 5 \times 10^{-4}$.

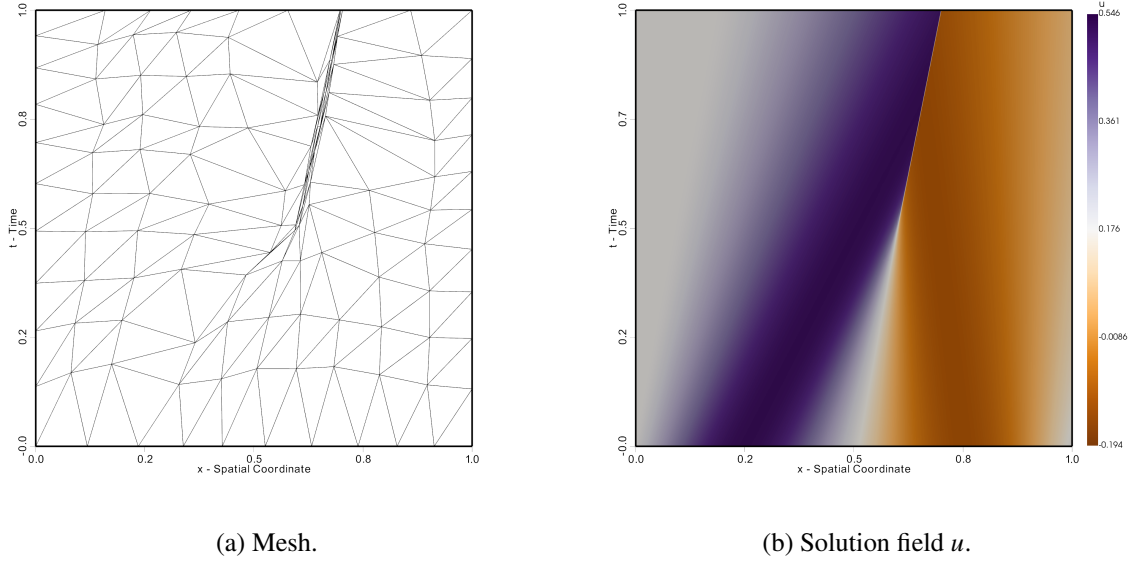


Figure 9: MDG-ICE solution and mesh for the viscous Burgers shock test problem with $\mu = 10^{-4}$.

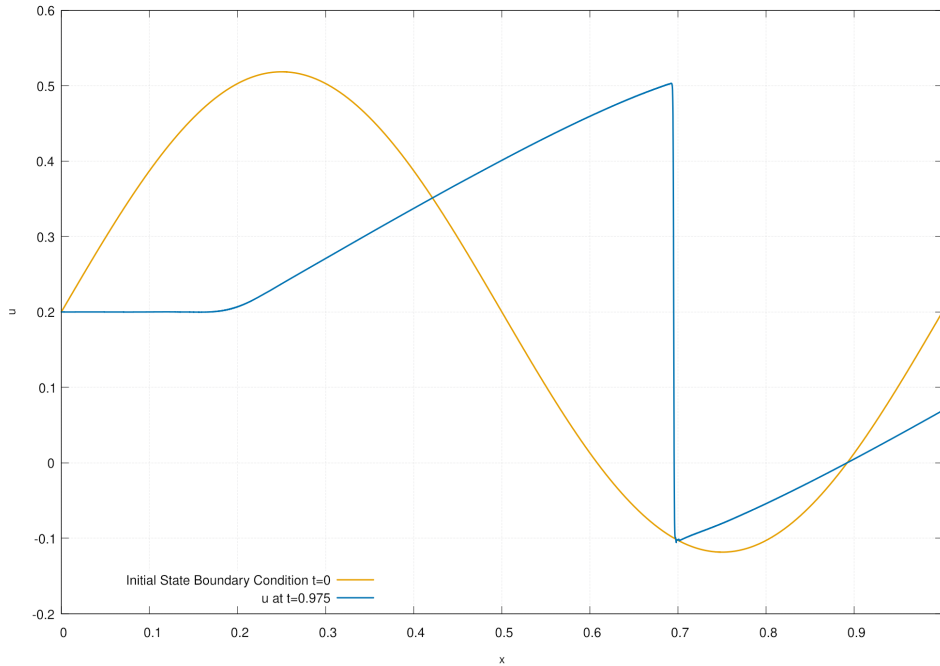


Figure 10: Cutline at $t = 0.975$ of the MDG(P5/P1) solution for the viscous Burgers shock test problem with $\mu = 10^{-4}$.

2.4 Mach 5 viscous flow around a cylinder

This test case, also presented by Kercher et al. [6], is representative of problems studied in re-entry such as the test cases presented by Nastac et al. [14]. We investigate flow with a free stream mach number of 5 around a cylinder of radius 1. This forms a bow shock wave ahead of the cylinder and a boundary layer near the surface of the cylinder. The governing equations for this are the Navier Stokes Equations, which, can be written in non-dimensional form as:

$$St_{ref} \frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}_j^c(\mathbf{U})}{\partial x_j} - \frac{\partial \mathbf{F}_j^v(\mathbf{U}, \nabla \mathbf{U})}{\partial x_j} = 0, \quad (27)$$

where U represents the conservative variables, F^c represents the convective fluxes in each coordinate direction, and F^v the viscous fluxes:

$$U = \begin{pmatrix} \rho \\ \rho u_i \\ \rho E \end{pmatrix}, \quad F_j^c = \begin{pmatrix} \rho u_j \\ \rho u_i u_j + Eu_{ref} p \delta_{ij} \\ u_j (\rho E + Eu_{ref} \Xi p) \end{pmatrix}, \quad F_j^v = \begin{pmatrix} 0 \\ \frac{1}{Re_{ref}} \tau_{ij} \\ \frac{\Xi}{Re_{ref}} (u_i \tau_{ij} + q_j) \end{pmatrix}. \quad (28)$$

The shear stress tensor and heat flux vector, respectively are given by:

$$\tau_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu \frac{\partial u_k}{\partial x_k} \delta_{ij}, \quad (29)$$

$$q_j = \frac{\mu \gamma}{Pr} \left(\frac{1}{\Xi} \frac{\partial E}{\partial x_j} - u_k \frac{\partial u_k}{\partial x_j} \right). \quad (30)$$

The dimensionless parameters, most of which are commonly used in the literature ([15], [16], [17]), used here are:

$$\begin{aligned} Re_{ref} &:= \frac{\rho_{ref} u_{ref} l_{ref}}{\mu_{ref}} && \text{Reference Reynolds Number} \\ Pr &:= \frac{c_p \mu}{\kappa} && \text{Prandtl Number} \\ Eu_{ref} &:= \frac{p_{ref}}{\rho_{ref} u_{ref}^2} && \text{Reference Euler Number} \\ Sr_{ref} &:= \frac{l_{ref}}{u_{ref} t_{ref}} && \text{Reference Strouhal Number} \\ \Xi &:= \frac{u_{ref}^2}{e_{ref}} && \text{Energy Coefficient} \\ \gamma &:= \frac{c_p}{c_v} && \text{Ratio of Specific Heats} \end{aligned} \quad (31)$$

Generally, reference parameters are chosen such that $\Xi = 1$ and can be eliminated. The total energy is the internal energy plus the kinetic energy:

$$E = e + \frac{1}{2} \Xi u_k u_k \quad (32)$$

The pressure and energy are closed by the non-dimensional equation of state:

$$p = \frac{\gamma - 1}{\Xi Eu_{ref}} \rho \left(\frac{1}{\Xi} E - \frac{1}{2} u_k u_k \right). \quad (33)$$

We omit the wake and only investigate the flow upstream of the cylinder surface to investigate steady-state effects. The flow field is characterized by the Reynolds number and Mach number. The Reynolds number is based on the free stream quantities,

$$Re = \frac{\rho_{\infty} u_{\infty} l_{ref}}{\mu_{\infty}},$$

where we choose $l_{ref} = 1$. We generate the mesh with 15 linear quadrilateral elements in the radial direction and 30 in the azimuthal direction for a total of 450 elements. The free stream boundary is a circle of radius $3 + \sqrt{2}$ with a center offset from the inner circle center by $\sqrt{2}$. The outflow planes are at an angle of 10° from horizontal on either side. We construct the mesh with a stagnation line at -45° from the positive y axis to validate that the implementation does not contain errors that cause directional dependence. A stretching factor is enforced improve the behavior of the initial shock-captured solution. The stretching factor ensures that each cell along the radial direction is 1.1 times the radial length of the previous cell.

We use the HLLC flux developed by Toro et al. [18] and for inviscid numerical fluxes and the DDG flux with interface correction as described by Danis and Yan [19] for viscous numerical fluxes.

The solution field is initialized from a low Reynolds number $Re = 100$, the mesh and mach number flow field are shown in Figures 11 and 12.

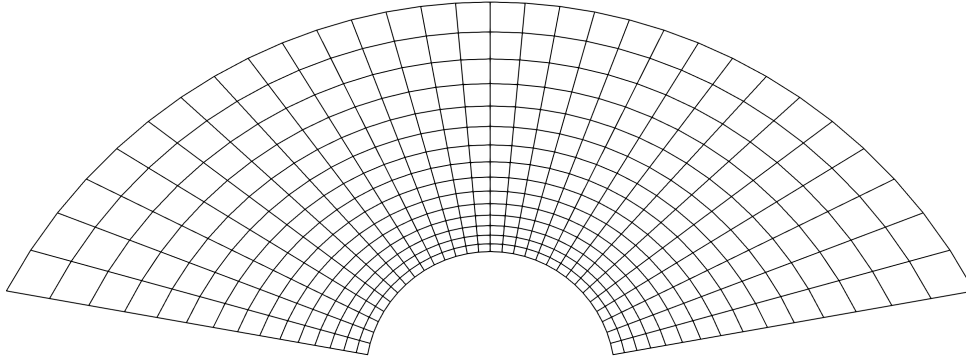


Figure 11: Initial mesh used for the Mach 5 viscous cylinder problem.

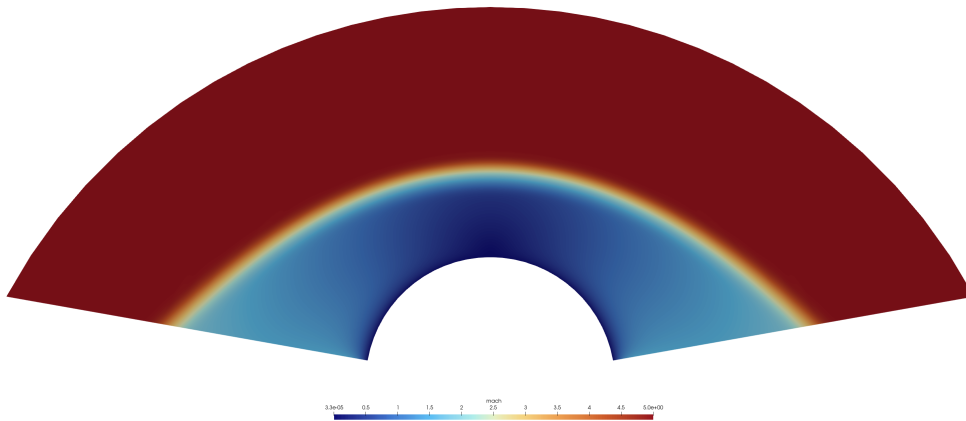


Figure 12: Initial Mach number field computed using $Re = 100$.

No limiting is used when applying MDG, and no topology modifications are made to the mesh during the solution process.

2.5 Non-dimensionalization

We choose a non-dimensionalization based on the free stream with an Euler number $Eu_{ref} = 1$. The free stream reference quantities that are set are:

$$\rho_{\infty} = 1.2754, \quad M_{\infty} = 5, \quad T_{\infty} = 273.15, \quad l_{ref} = 1.$$

From these quantities, we derive the free stream speed of sound a_{∞} using $R = 287.052\,874$ and $\gamma = 1.4$. Then we compute a free stream velocity u_{∞} from the mach number and free stream speed of sound. The

remaining reference quantities are then:

$$t_{ref} = \frac{l_{ref}}{u_{\infty}}, \quad \rho_{ref} = \rho_{\infty}, \quad u_{ref} = u_{\infty}, \quad e_{ref} = u_{\infty}^2, \\ p_{ref} = \rho_{\infty} u_{\infty}^2, \quad T_{ref} = T_{\infty}.$$

2.6 Stagnation quantities

The inviscid one-dimensional equivalent of this test problem involves flow from the freestream state passing through a normal shock and then decelerating isentropically to 0 at the surface of the cylinder, where the pressure will be the stagnation pressure from the post-shock region. Chapter 3 of Anderson [20] gives relations for these quantities that we will utilize. To begin we compute the post-shock region pressure p_2 from equation (3.57) in Anderson [20]:

$$\frac{p_2}{p_1} = 1 + \frac{2\gamma}{\gamma + 1}(M_1^2 - 1), \quad (34)$$

where M_1 is the free stream mach number. With a free stream mach number $M_1 = 5$, free stream pressure (non-dimensionalized $\hat{p}_1 \approx 0.0285$), and $\gamma = 1.4$ we compute $\hat{p}_2 \approx 0.8286$. The post-shock mach number M_2 can be computed by Equation (3.51) in Anderson [20]:

$$M_2^2 = \frac{1 + \frac{\gamma-1}{2}M_1^2}{\gamma M_1^2 - \frac{\gamma-1}{2}}. \quad (35)$$

The stagnation pressure is determined by Equation (3.31) in Anderson [20]:

$$\frac{p_0}{p} = \left(1 + \frac{\gamma-1}{2}M^2\right)^{\frac{\gamma}{\gamma-1}}. \quad (36)$$

This gives a stagnation pressure of $\hat{p}_0 \approx 0.9330$. The temperature at the boundary is an isothermal non-dimensional temperature of $\hat{T} = 2.5$. Using the free stream non-dimensional quantities to compute the gas constant $\hat{R}$, we obtain a stagnation density using the stagnation pressure, the isothermal temperature, and the gas constant with the Ideal Gas Law ($p = \rho RT$). With this we compute $\hat{\rho}_w \approx 13.0614$.

2.7 Results

The MDG method accurately resolves the viscous shock with very little overshoot and undershoot, especially along the stagnation line where the peaks of quantities of interest occur. The inviscid stagnation point properties, which are relevant given the low viscosity of the case, are accurately represented as seen in Figure 18. Figure 13 shows how the elements anisotropically adapted to resolve the viscous shock and the surface boundary layer. The pressure, temperature, mach number, and density fields shown in Figures 14, 15, 16, and 17, respectively show the resolved viscous shock and are free of numerical artifacts throughout the flow field.

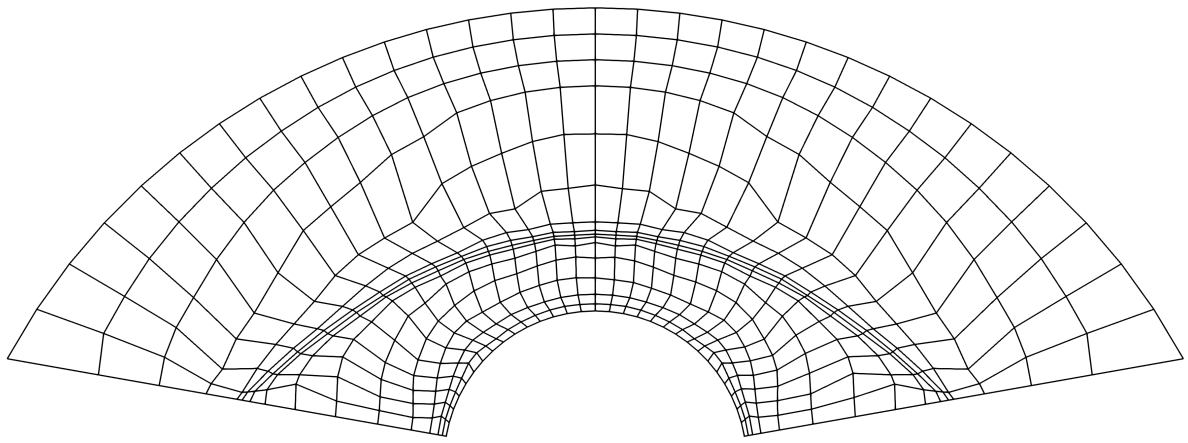


Figure 13: Final 450 element mesh for Mach 5 viscous flow around a cylinder at $Re = 10^3$.

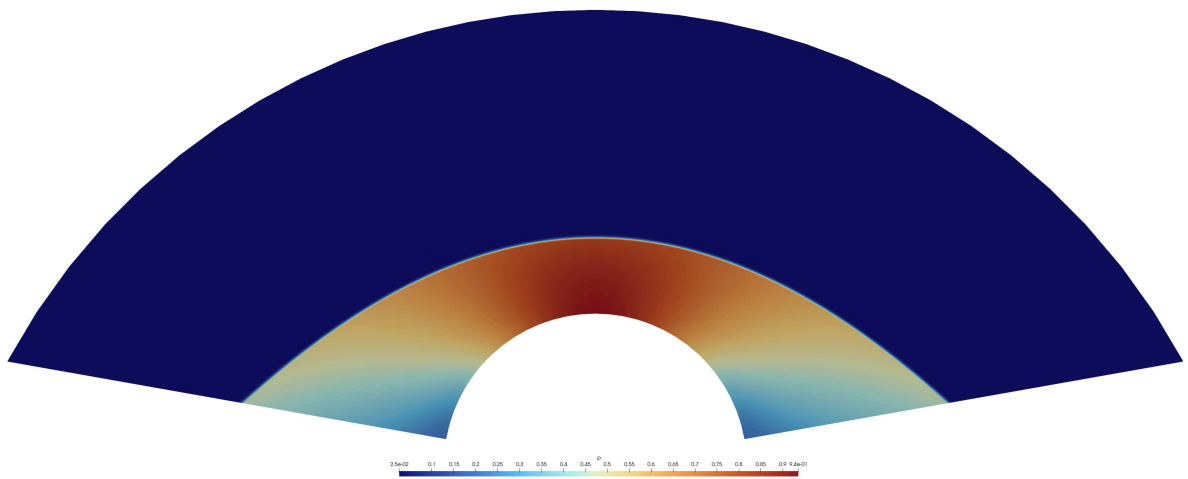


Figure 14: MDG(Q4/Q1) pressure field for Mach 5 viscous flow around a cylinder at $Re = 10^3$.

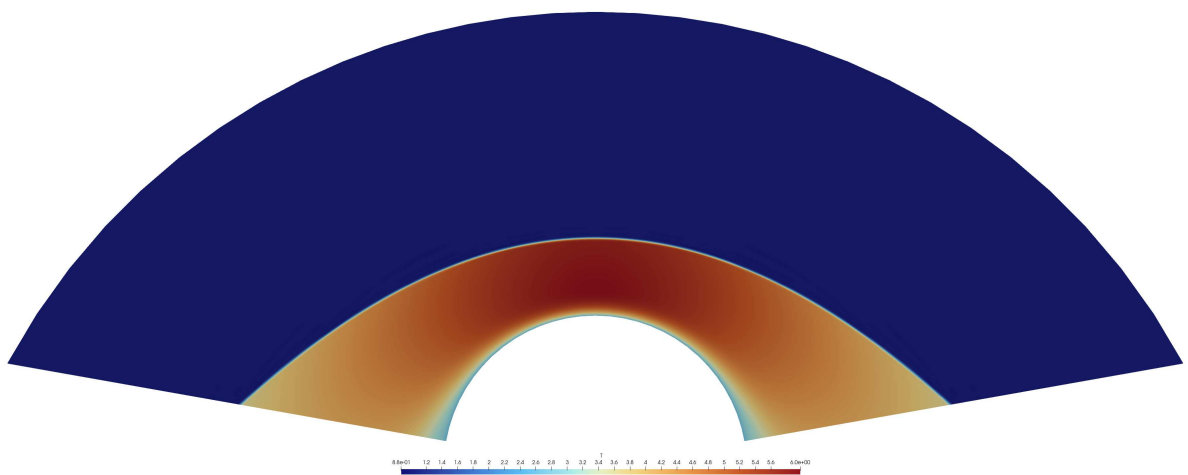


Figure 15: MDG(Q4/Q1) temperature field for Mach 5 viscous flow around a cylinder at $Re = 10^3$.

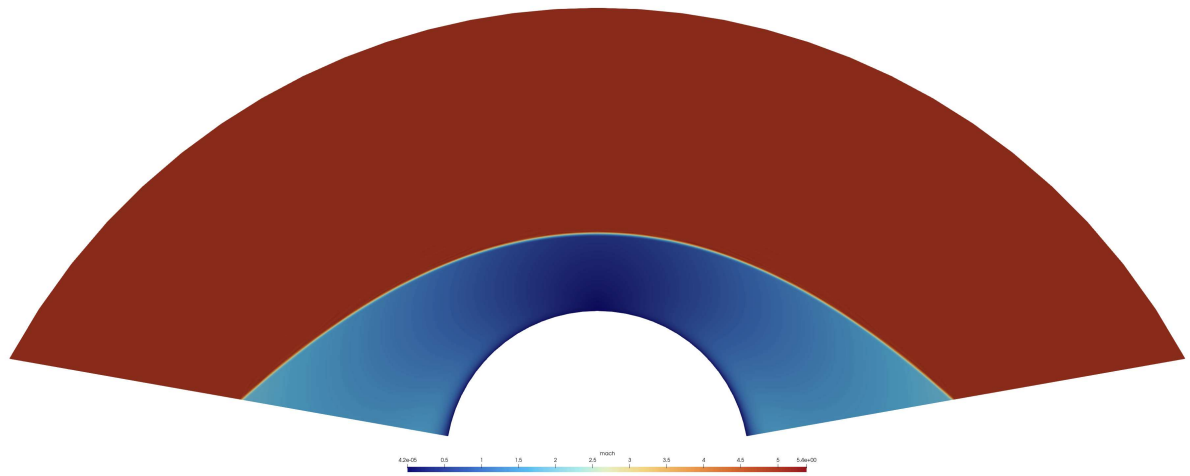


Figure 16: MDG(Q4/Q1) Mach number field for Mach 5 viscous flow around a cylinder at $Re = 10^3$.

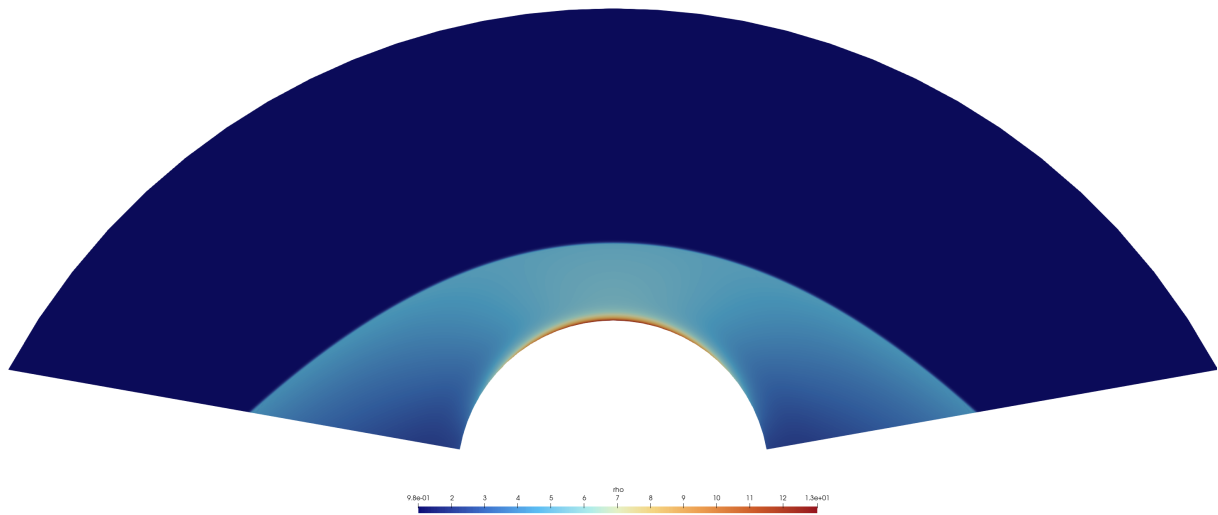
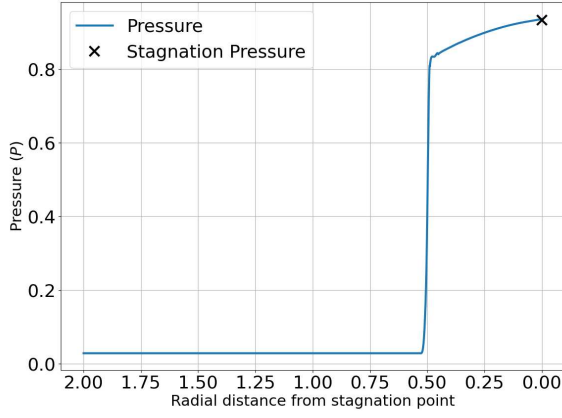
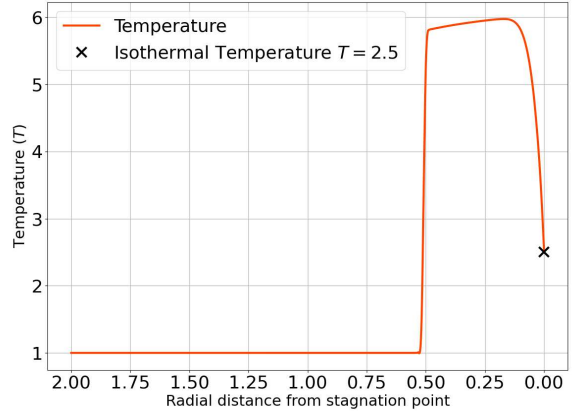


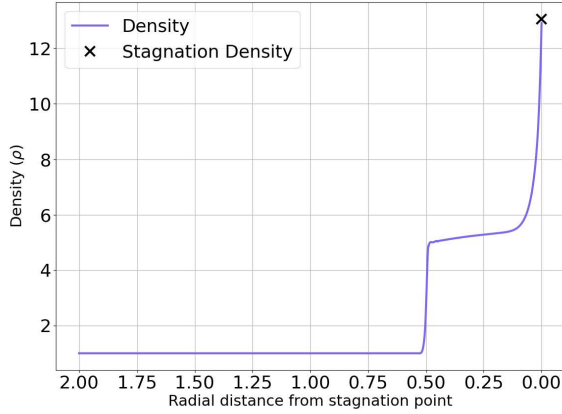
Figure 17: MDG(Q4/Q1) density field for Mach 5 viscous flow around a cylinder at $Re = 10^3$.



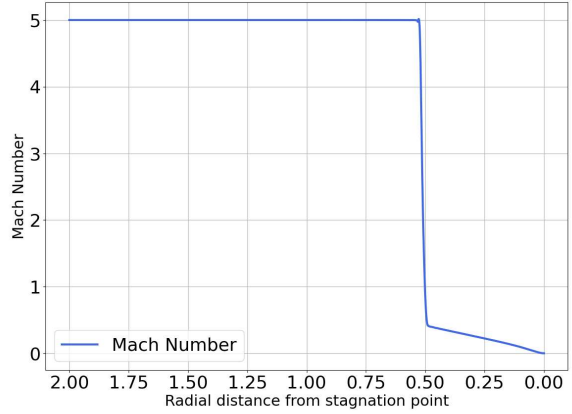
(a) Non-dimensional pressure along the stagnation line. The stagnation pressure for an inviscid analogue $p_0 \approx 0.9330$ is marked at the stagnation point.



(b) Non-dimensional temperature along the stagnation line. The isothermal boundary condition $T_w = 2.5$ is marked at the stagnation point.



(c) Non-dimensional density along the stagnation line. The density at the stagnation point computed from the stagnation pressure and isothermal temperature $\rho_w \approx 13.0614$ is marked at the stagnation point.



(d) Mach number along the stagnation line.

Figure 18: Stagnation line plots for Mach 5 flow around a viscous cylinder with $Re = 10^3$.

3. Conclusions

We present a novel formulation of the moving discontinuous Galerkin method with interface condition enforcement. The new formulation differs from previous extensions to viscous flows by avoiding representation of the auxiliary variables in the independent variables and reducing the size of the system used to enforce the constitutive equation. We also use a different continuous test functions based on the mesh geometry. This new method retains the powerful r-adaptive capabilities that accurately represent thin regions with high gradients demonstrated with previous approaches.

4. Acknowledgment

This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344 and the LLNL Academic Collaboration Team (ACT) IM Review Number LLNL-CONF-2020975.

References

- [1] Andrew Corrigan, Andrew D. Kercher, and David A. Kessler. A moving discontinuous galerkin finite element method for flows with interfaces. *International Journal for Numerical Methods in Fluids*, 89(9):362–406, 2018.
- [2] A. Shi, P.-O. Persson, and M. J. Zahr. *High-Order Implicit Shock Tracking (HOIST)*, pages 233–259. SEMA SIMAI Springer Series. Springer International Publishing, Cham, 2022.
- [3] Per-Olof Persson and Jaime Peraire. Sub-cell shock capturing for discontinuous galerkin methods. In *44th AIAA aerospace sciences meeting and exhibit*, page 112. American Institute of Aeronautics and Astronautics, 2006.
- [4] K. Kitamura and E. Shima. Towards shock-stable and accurate hypersonic heating computations: A new pressure flux for AUSM-family schemes. *Journal of Computational Physics*, 245:62–83, 2013.
- [5] D. Kuzmin. A vertex-based hierarchical slope limiter for p-adaptive discontinuous galerkin methods. *Journal of computational and applied mathematics*, 233(12):3077–3085, 2010.
- [6] Andrew D. Kercher, Andrew Corrigan, and David A. Kessler. The moving discontinuous galerkin finite element method with interface condition enforcement for compressible viscous flows. *International Journal for Numerical Methods in Fluids*, 93(5):1490–1519, 2020.
- [7] Eric J. Ching, Andrew D. Kercher, and Andrew Corrigan. The moving discontinuous galerkin method with interface condition enforcement for the simulation of hypersonic, viscous flows. *Computer Methods in Applied Mechanics and Engineering*, 427:117045, 2023.
- [8] Philippe G Ciarlet. *The finite element method for elliptic problems*. SIAM, 2002.
- [9] J. Cheng, X. Yang, X. Liu, T. Liu, and H. Luo. A direct discontinuous Galerkin method for the compressible Navier–Stokes equations on arbitrary grids. *Journal of Computational Physics*, 327:484–502, December 2016.
- [10] Douglas N. Arnold, Franco Brezzi, Bernardo Cockburn, and L. Donatella Marini. Unified analysis of discontinuous Galerkin methods for elliptic problems. *SIAM Journal on Numerical Analysis*, 39(5):1749–1779, 2002.
- [11] Jorge J Moré. The Levenberg-Marquardt algorithm: implementation and theory. In *Conference on Numerical Analysis*, Scotland, June 1977. University of Dundee.
- [12] Jinyan Fan and Jianyu Pan. Convergence properties of a self-adaptive levenberg-marquardt algorithm under local error bound condition. *Computational Optimization and Applications*, 34(1):47–62, 2006.
- [13] Patrick Knupp. Introducing the target-matrix paradigm for mesh optimization via node-movement. *Engineering with Computers*, 28:419–429, 2012.
- [14] G. Nastac, R. Tramel, and Nielsen E. Improved Heat Transfer Prediction for High-Speed Flows over Blunt Bodies using Adaptive Mixed-Element Unstructured Grids. *AIAA SCITECH 2022 Forum*, 2022.
- [15] Kleinstreuer C. *Modern Fluid Dynamics*. Fluid Mechanics and Its Applications. Springer, 1 edition, 2010.
- [16] D. C. Wilcox. *Basic Fluid Mechanics*. DCW Industries, fifth edition, 2012.
- [17] R. Nourgaliev, P. Greene, B. Weston, R. Barney, A. Anderson, S Khairallah, and J.-P. Delplanque. High-order fully implicit solver for all-speed fluid dynamics. *Shock Waves*, 29:651–689, 2018.
- [18] Eleuterio F. Toro, M. Spruce, and W. Speares. Restoration of the contact surface in the HLL-Riemann solver. *Shock Waves*, 4:25–34, 1994.
- [19] M. E. Danis and J. Yan. A new direct discontinuous Galerkin method with interface correction for two-dimensional compressible Navier-Stokes equations. *Journal of Computational Physics*, 452, March 2022. arXiv:2104.09767 [cs, math].
- [20] John D. Anderson. *Modern compressible flow: With historical perspective*. McGraw-Hill US Higher Ed USE., 4. edition, 2020.