

# Entropy-based viscous regularization for the multi-dimensional Euler equations in low-Mach and transonic flows

Marc O. Delchini<sup>a</sup>, Jean C. Ragusa<sup>\*,a</sup>, Ray A. Berry<sup>b</sup>

<sup>a</sup>*Department of Nuclear Engineering, Texas A&M University, College Station, TX 77843, USA*

<sup>b</sup>*Idaho National Laboratory, Idaho Falls, ID 83415, USA*

---

## Abstract

We present a new version of the entropy viscosity method, a viscous regularization technique for hyperbolic conservation laws, that is well-suited to low-Mach flows. By means of a low-Mach asymptotic study, new expressions for the entropy viscosity coefficients are derived. These definitions are valid for a wide range of Mach numbers, from subsonic flows (with very low Mach numbers) to supersonic flows, and no longer depend on an analytical expression for the entropy function. In addition, the entropy viscosity method is extended to Euler equations with variable area for nozzle flow problems. The effectiveness of the method is demonstrated using various 1-D and 2-D benchmark tests: flow in a converging-diverging nozzle; Leblanc shock tube; slow moving shock; strong shock for liquid phase; low-Mach flows around a cylinder and over a circular hump; and supersonic flow in a compression corner. Convergence studies are performed for both smooth solutions and solutions with shocks present.

*Key words:* entropy viscosity method, artificial viscosity, low-Mach regime, shock capturing, Euler equations with variable area.

---

---

<sup>\*</sup>Corresponding author

*Email addresses:* `delchmo@tamu.edu` (Marc O. Delchini), `jean.ragusa@tamu.edu` (Jean C. Ragusa), `ray.berry@inl.gov` (Ray A. Berry)

## 1. Introduction

Solving accurately compressible fluid equations in the low-Mach limit is an ongoing topic of research. In many engineering applications, compressibility effects require the solution of the compressible fluid equations in nearly incompressible regimes and/or for low-Mach flow problems. For example, such flows are encountered in aerodynamics in the study of airships. In the nuclear industry, flows are nearly in the incompressible regime but compressible effects cannot be neglected because of the intense heat source, and because of some postulated accident scenarios, and thus need to be accurately resolved. Hence, there is a strong interest to develop computational methods that can solve both compressible and incompressible flow problems.

When solving Euler equations for a wide range of Mach numbers, multiple questions must be addressed: stability, accuracy and solution convergence in the low-Mach regime. Because of the hyperbolic nature of the equations, shocks can form during transonic and supersonic flows and require the use of adequate numerical techniques to stabilize solution and correctly resolve the discontinuities. A wide range of stabilization methods are available in the literature: approximate Riemann solvers [1], flux-limiter techniques [2, 3], pressure-based viscosity methods [4], Lapidus method [5, 6, 7], and the entropy-viscosity method [8, 9], among others. These numerical methods are usually developed using simple equations of state and tested for transonic and supersonic flows where the disparity between the acoustic wave speed and the fluid speed is not excessively large and thus the Mach number is of order one. This approach, however, leads to a well-known accuracy problem in the low-Mach regime where the fluid velocity is smaller than the speed of sound by multiple orders of magnitude. The numerical dissipative terms become ill-scaled in the low-Mach regime and lead to the wrong numerical solution by changing the nature of the equations

28 solved. This behavior is well documented in the literature [10, 11, 12]. In [10], a  
 29 low-Mach asymptotic study has demonstrated convergence of the compressible  
 30 Euler equations to the incompressible ones. Many well-known stabilization tech-  
 31 niques, for instance, the Roe scheme and the SUPG technique, do not yield the  
 32 correct solution in the low-Mach regime and suitable modifications have been  
 33 proposed (see [13] for the Roe scheme and [12] for the SUPG method) to en-  
 34 sure the convergence to the correct solution while preserving the original shock  
 35 stabilization properties of these schemes. Additionally, the time step size may  
 36 be severely restricted when solving compressible fluid equations with an explicit  
 37 time discretization because of the large disparity between the fluid velocity and  
 38 the speed of sound. To avoid an excessive number of explicit time steps, time  
 39 preconditioners have been proposed and proved efficient [11]; however, because  
 40 they modify the time derivatives in the governing equations, such acceleration  
 41 techniques can only be used to obtain steady-state solutions for low-Mach flows  
 42 using explicit schemes. To avoid modifying the time derivatives, the temporal  
 43 implicit capabilities of the MOOSE multiphysics framework [14] are used. With  
 44 such a choice, low-Mach steady-state solutions can be obtained effectively while  
 45 preserving the accuracy of the transient solution; however, it requires the use of  
 46 nonlinear solvers.

47 In this paper, we employ the entropy viscosity method as a numerical sta-  
 48 bilization for the inviscid Euler equation and assess its performance in the low-  
 49 Mach regime. The entropy viscosity method is a viscous regularization technique  
 50 introduced by Guermond et al. to solve hyperbolic systems of equations and  
 51 has successfully been applied to multi-dimensional supersonic flows with various  
 52 spatial discretization schemes [15]. It is fairly straightforward to implement, can  
 53 be used with unstructured grids, and has dissipative terms that are consistent  
 54 with the entropy minimum principle. However, it has not been evaluated in the

55 low-Mach regime.

56 This paper is organized as follows: in Section 2 the current definition of the  
57 entropy viscosity method is recalled and its ill-scaled nature in the low-Mach  
58 regime is discussed. In Section 3, a new formulation of the viscosity residual  
59 is derived. This formulation no longer requires an analytical expression for the  
60 entropy function. A low-Mach asymptotic study is carried out to adapt the  
61 definition of the entropy viscosity coefficients in the incompressible limit while  
62 ensuring that the viscosity coefficients scale appropriately for all flow speeds  
63 (from low-Mach to supersonic). In Section 4, we extend the entropy viscosity  
64 method to Euler equations with variable area in order to model nozzle flows:  
65 the viscous dissipative terms are adapted so that the entropy minimum principle  
66 remains satisfied. Spatial and temporal discretizations and solution techniques  
67 are presented in Section 5. 1-D and 2-D numerical results are provided in  
68 Section 6 for a wide range of Mach numbers: liquid and gas nozzle flow problems,  
69 low-Mach flows over a cylinder and a circular bump (with Mach numbers as  
70 low as  $10^{-7}$ ), and supersonic flows in a compression corner [16]. Convergence  
71 studies are performed in 1-D in order to demonstrate the accuracy of the solution  
72 technique.

## 73 2. The Entropy Viscosity Method

### 74 2.1. Background

75 Euler equations in conservative form are given by

$$\partial_t \rho + \vec{\nabla} \cdot (\rho \vec{u}) = 0 \quad (1a)$$

76

$$\partial_t (\rho \vec{u}) + \vec{\nabla} \cdot (\rho \vec{u} \otimes \vec{u} + P \mathbb{I}) = 0 \quad (1b)$$

77

$$\partial_t (\rho E) + \vec{\nabla} \cdot [\vec{u} (\rho E + P)] = 0 \quad (1c)$$

78 where  $\rho$ ,  $\rho\vec{u}$  and  $E$  are the density, the momentum and the total specific energy,  
 79 respectively, and will be referred to as the conservative variables.  $\vec{u}$  is the fluid  
 80 velocity and its specific internal energy is denoted by  $e = E - \frac{u^2}{2}$ . An equation  
 81 of state, dependent upon  $\rho$  and  $e$ , is used to compute the pressure  $P$ . The tensor  
 82 product  $\vec{a} \otimes \vec{b}$  is such that  $(\vec{a} \otimes \vec{b})_{i,j} = a_i b_j$ . The identity tensor is denoted by  $\mathbb{I}$ .

83 Next, the entropy viscosity method [8, 9, 17, 18] applied to Eq. (1) is recalled.  
 84 The method consists of adding dissipative terms with a viscosity coefficient mod-  
 85 ulated by the entropy production; this allows for a high-order accuracy when  
 86 the solution is smooth (provided that the spatial and temporal discretizations  
 87 also are high order). The derivation of the viscous regularization (or dissipa-  
 88 tive terms) is carried out to be consistent with the entropy minimum principle;  
 89 details and proofs of the derivation can be found in [15]. The viscous regular-  
 90 ization thus obtained is valid for any equation of state as long as the physical  
 91 entropy function  $s$  is concave (or  $-s$  is a convex function) with respect to the  
 92 internal energy  $e$  and the specific volume  $1/\rho$ . The Euler equations with viscous  
 93 regularization become

$$\partial_t \rho + \vec{\nabla} \cdot (\rho \vec{u}) = \vec{\nabla} \cdot (\kappa \vec{\nabla} \rho) \quad (2a)$$

$$\partial_t (\rho \vec{u}) + \vec{\nabla} \cdot (\rho \vec{u} \otimes \vec{u} + P \mathbb{I}) = \vec{\nabla} \cdot (\mu \rho \vec{\nabla}^s \vec{u} + \kappa \vec{u} \otimes \vec{\nabla} \rho) \quad (2b)$$

$$\partial_t (\rho E) + \vec{\nabla} \cdot [\vec{u} (\rho E + P)] = \vec{\nabla} \cdot \left( \kappa \vec{\nabla} (\rho e) + \frac{1}{2} \|\vec{u}\|^2 \kappa \vec{\nabla} \rho + \rho \mu \vec{u} \vec{\nabla}^s \vec{u} \right) \quad (2c)$$

96 where  $\kappa$  and  $\mu$  are positive viscosity coefficients (in units of  $\text{length}^2/\text{time}$ ).  $\vec{\nabla}^s \vec{u}$   
 97 denotes the symmetric gradient operator and guarantees the method to be ro-  
 98 tationally invariant [15]. The viscosity coefficients are key ingredients in the  
 99 viscous regularization of Eq. (2). Other stabilization approaches have been pro-  
 100 posed in the literature, for instance, the Lapidus method [7, 5] or pressure-based  
 101 viscosity methods [4]. Here, we follow the work of Guermond et al. and define

102 the viscosity coefficients,  $\kappa$  and  $\mu$ , based on the local entropy production. These  
 103 coefficients are numerically evaluated using the local entropy residual  $R_{\text{ent}}(\vec{r}, t)$   
 104 defined in Eq. (3);  $R_{\text{ent}}(\vec{r}, t)$  is known to be peaked in shocks and vanishingly  
 105 small elsewhere [1].

$$R_{\text{ent}}(\vec{r}, t) := \partial_t s + \vec{u} \cdot \vec{\nabla} s \quad (3)$$

106 In the current version of the method, the ratio of  $\kappa$  to  $\mu$  is defined through  
 107 a numerical Prandtl number,  $\text{Pr} = \kappa/\mu$ .  $\text{Pr}$  is a user-defined parameter and  
 108 is usually taken in the range  $[0.001; 1]$ . Since the entropy residual  $R_{\text{ent}}(\vec{r}, t)$   
 109 may be extremely large in shocks, the definition of the viscosity coefficients  
 110 also includes a first-order viscosity coefficient that serves as an upper bound for  
 111 the entropy-based viscosity coefficients. The first-order viscosity coefficients,  
 112 denoted by  $\mu_{\text{max}}$  and  $\kappa_{\text{max}}$ , are chosen so that the numerical scheme becomes  
 113 equivalent to an upwind scheme when the first-order coefficients are employed.  
 114 The upwind scheme is known to be over-dissipative but guarantees monotonicity  
 115 [1]. In practice, the viscosity coefficients only saturate to the first-order viscosity  
 116 coefficients in shocks and are much smaller elsewhere, hence avoiding the over-  
 117 dissipation of the upwind method. The first-order viscosity coefficients  $\mu_{\text{max}}$   
 118 and  $\kappa_{\text{max}}$  are equal and set proportional to the largest local eigenvalue  $\|\vec{u}\| + c$ :

$$\mu_{\text{max}}(\vec{r}, t) = \kappa_{\text{max}}(\vec{r}, t) = \frac{h}{2} (\|\vec{u}(t, \vec{r})\| + c(t, \vec{r})), \quad (4)$$

119 where  $h$  denotes the local grid size (for higher than linear finite element rep-  
 120 resentations,  $h$  is defined as the ratio of the grid size to the polynomial order  
 121 of the test functions used, see Eq. 2.4 in [18]). For simplicity, the first-order  
 122 viscosity coefficients will only be referred to as  $\kappa_{\text{max}}(\vec{r}, t)$ . In practice, these

quantities are evaluated within a given cell  $K$  at quadrature points:

$$\kappa_{\max}^K(\vec{r}_q, t) = \frac{h_K}{2} \left( \|\vec{u}(t, \vec{r}_q)\| + c(t, \vec{r}_q) \right), \quad (5)$$

where  $\vec{r}_q$  denotes the position of a quadrature point. As stated earlier, the entropy viscosity coefficients, which we denote by  $\kappa_e$  and  $\mu_e$ , are set proportional to the entropy production evaluated by computing the local entropy residual  $R_{\text{ent}}$ . The definitions also include the inter-element jump  $J[s]$  of the entropy flux, allowing for the detection of discontinuities other than shocks (e.g., contact).  $\kappa_e$  and  $\mu_e$  are computed as follows

$$\mu_e^K(\vec{r}_q, t) = h_K^2 \frac{\max(|R_{\text{ent}}^K(\vec{r}_q, t)|, J^K[s](t))}{\|s - \bar{s}\|_\infty} \quad (6a)$$

$$\kappa_e^K(\vec{r}_q, t) = \text{Pr} \mu_e^K(\vec{r}_q, t), \quad (6b)$$

where  $\|\cdot\|_\infty$  and  $\bar{\cdot}$  denote the  $L_\infty$ -norm and the average operator over the entire computational domain, respectively. The definition of the entropy jump  $J[s]$  is spatial discretization-dependent and examples of definitions can be found in [18] for discontinuous Galerkin discretization. For continuous finite element methods (FEM), the jump of a given quantity is defined as the change of its normal derivative ( $\partial_n(\cdot) = \vec{n} \cdot \vec{\nabla}(\cdot)$ ) across the common face separating the two elements, and will be further referred to as the inter-element jump. We take the largest value over all faces  $f$  present on the boundary  $\partial K$  of element  $K$ :

$$J^K[s](t) = \max_{f \in \partial K} \max_{\vec{r}_q \in f} \left( \|\vec{u}(\vec{r}_q, t)\| \|\vec{\nabla} s(\vec{r}_q, t) \cdot \vec{n}(\vec{r}_q)\|_f \right), \quad (7)$$

where  $\llbracket a(\vec{r}_q) \rrbracket_f$  denotes the inter-element jump in  $a(\vec{r})$  at quadrature point  $\vec{r}_q$  on face  $f$  (the quadrature points  $\vec{r}_q$  are taken on the faces  $f$  of the element  $K$ ). With the definition given in Eq. (7), the jump is constant over each el-

142 element  $K$  of the computational domain. The denominator  $\|s - \bar{s}\|_\infty$  is used  
 143 for dimensionality purposes. Currently, there are no theoretical justifications  
 144 for choosing the denominator beyond a dimensionality argument. Finally, the  
 145 viscosity coefficients  $\mu$  and  $\kappa$  are as follows:

$$\mu(\vec{r}, t) = \min \left( \mu_e(\vec{r}, t), \mu_{\max}(\vec{r}, t) \right) \quad (8a)$$

146 and

$$\kappa(\vec{r}, t) = \min \left( \kappa_e(\vec{r}, t), \kappa_{\max}(\vec{r}, t) \right). \quad (8b)$$

147 Given these definitions, we have the following properties. In shock regions, the  
 148 entropy viscosity coefficients will experience a peak because of entropy produc-  
 149 tion and thus will saturate to the first-order viscosity. The first-order coefficients  
 150 are known to be over-dissipative and will smooth out any oscillatory behavior.  
 151 Elsewhere in the domain, entropy production will be small and the viscosity  
 152 coefficients  $\mu$  and  $\kappa$  will remain small. High-order accuracy for entropy-based  
 153 viscous stabilization has been demonstrated using several 1-D shock tube ex-  
 154 amples and various 2-D tests [8, 9, 18].

## 155 2.2. Issues in the Low-Mach Regime

156 In the low-Mach Regime, a smooth flow is known to approach the isentropic  
 157 limit, resulting in very little entropy production. Since the entropy viscosity  
 158 method is directly based on the evaluation of the local entropy production, it  
 159 is of interest to study how the entropy viscosity coefficients  $\mu_e$  and  $\kappa_e$  scale  
 160 in the low-Mach regime. In practice, the entropy residual  $R_{\text{ent}}$  will be very  
 161 small in that regime and so will be the denominator  $\|s - \bar{s}\|_\infty$ , thus making  
 162 the definition of the viscosity coefficients in Eq. (6) undetermined and likely ill-  
 163 scaled. One possible approach would consist of expanding the numerator and  
 164 denominator in terms of the Mach number and deriving its limit when the Mach



number goes to zero. Such derivation may not be straightforward, especially for general equations of state. However, this can be avoided by noting that the entropy residual  $R_{\text{ent}}$  can be recast as a function of pressure, density, velocity, and speed of sound as will be shown in Eq. (9) of Section 3.1. This alternate entropy residual definition is the basis for the low-Mach analysis carried out in this paper and possesses several advantages that are detailed next.

### 3. An All-speed Reformulation of the Entropy Viscosity Method

In this section, the entropy residual  $R_{\text{ent}}$  is recast as a function of pressure, density, velocity and speed of sound. Then, a low-Mach asymptotic study is carried out for the Euler equations with viscous regularization in order to derive an appropriate normalization parameter that is valid in the isentropic low-Mach regime as well as for transonic and supersonic flows.

#### 3.1. New Definition of the Entropy Production Residual

The first step in defining viscosity coefficients that behave well in the low-Mach limit is to recast the entropy residual in terms of thermodynamic variables. This provides physical insight on possible normalization choices that can be valid in both low-Mach and transonic flows. The alternate definition of the entropy residual, the derivation of which is given in Appendix A, is the following:

$$R_{\text{ent}}(\vec{r}, t) := \partial_t s + \vec{u} \cdot \vec{\nabla} s = \frac{Ds}{Dt} = \frac{s_e}{P_e} \left( \underbrace{\frac{DP}{Dt} - c^2 \frac{D\rho}{Dt}}_{\tilde{R}_{\text{ent}}(\vec{r}, t)} \right), \quad (9)$$

where  $\frac{D}{Dt}$  denotes the material derivative ( $\frac{D}{Dt} := \frac{\partial}{\partial t} + \vec{u} \cdot \vec{\nabla}$ ), and  $x_y$  is the standard shorthand notation for the partial derivative of  $x$  with respect to  $y$ , e.g.,  $P_e := \frac{\partial P}{\partial e}$ . For instance, when employing the ideal gas equation of state, the proportionality coefficient  $\frac{s_e}{P_e}$  simply is  $\frac{s_e}{P_e} = \frac{C_v}{P}$ . Note that the definition

187 of the speed of sound,  $c^2 = \left. \frac{\partial P}{\partial \rho} \right|_s$ , can be used to show that  $\tilde{R}_{\text{ent}}$  is zero in  
 188 isentropic flow regions (see Appendix C). The entropy residuals  $R_{\text{ent}}$  and  $\tilde{R}_{\text{ent}}$   
 189 are proportional to one another and will experience similar variations in space  
 190 and time. Thus, one may elect to employ  $\tilde{R}_{\text{ent}}$  instead of  $R_{\text{ent}}$  for the evaluation  
 191 of the local entropy residual. The new expression presents several advantages  
 192 which include:

- 193 • An analytical expression of the entropy function  $s$  is no longer needed: the  
 194 residual  $\tilde{R}_{\text{ent}}$  is evaluated using the local values of pressure, density, ve-  
 195 locity and speed of sound. Deriving an entropy function for some complex  
 196 equations of state may be difficult;
- 197 • Suitable normalizations for the residual  $\tilde{R}_{\text{ent}}$  can be devised. Examples  
 198 include the pressure itself or combinations of the density, the speed of  
 199 sound and the norm of the velocity, i.e.,  $\rho c^2$ ,  $\rho c ||\vec{u}||$  or  $\rho ||\vec{u}||^2$ .

200 Denoting the normalization of  $\tilde{R}_{\text{ent}}$  by  $\text{norm}_P$ , the entropy-based viscosity co-  
 201 efficients  $\mu_e$  and  $\kappa_e$  can be re-defined as follows:

$$\mu_e^K(\vec{r}_q, t) = h_K^2 \frac{\max \left( |\tilde{R}_{\text{ent}}^K(\vec{r}_q, t)|, J^K(t) \right)}{\text{norm}_P^\mu}, \quad (10a)$$

202 and

$$\kappa_e^K(\vec{r}_q, t) = h_K^2 \frac{\max \left( |\tilde{R}_{\text{ent}}^K(\vec{r}_q, t)|, J^K(t) \right)}{\text{norm}_P^\kappa}, \quad (10b)$$

203 where

$$J^K(t) = \max_{f \in \partial K} \max_{\vec{r}_q \in f} \left( ||\vec{u}(\vec{r}_q, t)|| \max \left( J^K[P](t), c^2(\vec{r}_q, t) ||J^K[\rho](t)|| \right) \right) \quad (10c)$$

204 Note that now the jump operator  $J^K$  acts on the variables appearing in  $\tilde{R}_{\text{ent}}$ ,  
 205 namely, pressure and density. The residual  $\tilde{R}_{\text{ent}}$  and the pressure jump will

only contribute to the viscosity coefficients in the shock region. On the other hand, the density jump will contribute in both contact and shock regions. The  $\mu$  and  $\kappa$  coefficients are kinematic viscosities (units of  $m^2/s$ ); the normalization parameters  $\text{norm}_P$  are thus in units of pressure, hence the use of the subscript  $P$ . Note also that we are not requiring the same normalization for both  $\mu_e$  and  $\kappa_e$  so the entropy viscosity coefficients can be different. The low-Mach asymptotic study presented next will determine the proper normalization.

### 3.2. Asymptotic Study in the Low-Mach Regime

The Euler equations with viscous stabilization, Eq. (6), bear some similarities with the Navier-Stokes equations in the sense that dissipative terms (containing second-order spatial derivatives) are present in both sets of equations. An abundant literature exists regarding the low-Mach asymptotic of the Navier-Stokes equations [10, 11, 12, 19]. The asymptotic study presented here is inspired by the work of Muller et al. [19] where an asymptotic derivation for the Navier-Stokes was presented. We remind the reader that the objective is to determine appropriate scaling for the entropy viscosity coefficients so that the dissipative terms remain well-scaled for two limit cases: (i) the isentropic low-Mach limit where Euler equations degenerate to an incompressible system of equations in the low-Mach limit and (ii) the non-isentropic limit with formation of shocks. The isentropic limit of the Euler equations with viscous regularization should yield incompressible fluid flow solutions in the low-Mach limit, namely, that the spatial steady-state pressure variations are of the order  $M^2$  and that the velocity satisfies the divergence constraint  $\vec{\nabla} \cdot \vec{u}_0 = 0$  [10, 11, 12]. For non-isentropic situations, shocks may form for any value of Mach number and the minimum entropy principle should still be satisfied so that numerical oscillations, if any, be controlled by the entropy viscosity method independently of the value of the Mach number. Our objective is to determine the appropriate

233 scaling for  $\text{norm}_P^\kappa$  and  $\text{norm}_P^\mu$  in these two limit cases.

The first step in the study of the limit cases (i) and (ii) is to re-write Eq. (2) in a non-dimensional manner. To do so, the following variables are introduced:

$$\begin{aligned} \rho^* &= \frac{\rho}{\rho_\infty}, \quad u^* = \frac{u}{u_\infty}, \quad P^* = \frac{P}{\rho_\infty c_\infty^2}, \quad E^* = \frac{E}{c_\infty^2}, \\ x^* &= \frac{x}{L_\infty}, \quad t^* = \frac{t}{L_\infty/u_\infty}, \quad \mu^* = \frac{\mu}{\mu_\infty}, \quad \kappa^* = \frac{\kappa}{\kappa_\infty}, \end{aligned} \quad (11)$$

234 where the subscript  $\infty$  denote the far-field or stagnation quantities and the  
 235 superscript  $*$  stands for the non-dimensional variables. The far-field reference  
 236 quantities are chosen such that the dimensionless flow quantities are of order 1.  
 237 The reference Mach number is given by

$$M_\infty = \frac{u_\infty}{c_\infty}, \quad (12)$$

238 where  $c_\infty$  is a reference value for the speed of sound. Then, the scaled Euler  
 239 equations with viscous regularization are:

$$\partial_{t^*} \rho^* + \vec{\nabla}^* \cdot (\rho^* \vec{u}^*) = \frac{1}{\text{Pé}_\infty} \vec{\nabla}^* \cdot (\kappa^* \vec{\nabla}^* \rho^*) \quad (13a)$$

$$\begin{aligned} \partial_{t^*} (\rho^* \vec{u}^*) + \vec{\nabla}^* \cdot (\rho^* \vec{u}^* \otimes \vec{u}^*) + \frac{1}{M_\infty^2} \vec{\nabla}^* P^* &= \frac{1}{\text{Re}_\infty} \vec{\nabla}^* \cdot (\rho^* \mu^* \vec{\nabla}^{s,*} \vec{u}^*) \\ &+ \frac{1}{\text{Pé}_\infty} \vec{\nabla}^* \cdot (\vec{u}^* \otimes \kappa^* \vec{\nabla}^* \rho^*) \end{aligned} \quad (13b)$$

$$\begin{aligned} \partial_{t^*} (\rho^* E^*) + \vec{\nabla}^* \cdot [\vec{u}^* (\rho^* E^* + P^*)] &= \frac{1}{\text{Pé}_\infty} \vec{\nabla}^* \cdot (\kappa^* \vec{\nabla}^* (\rho^* e^*)) \\ &+ \frac{M_\infty^2}{\text{Re}_\infty} \vec{\nabla}^* \cdot (\vec{u}^* \rho^* \mu^* \vec{\nabla}^{s,*} \vec{u}^*) + \frac{M_\infty^2}{2\text{Pé}_\infty} \vec{\nabla}^* \cdot (\kappa^* (u^*)^2 \vec{\nabla}^* \rho^*), \end{aligned} \quad (13c)$$

240 where the numerical Reynolds ( $\text{Re}_\infty$ ) and Péclet ( $\text{Pé}_\infty$ ) numbers are defined as:

$$\text{Re}_\infty = \frac{u_\infty L_\infty}{\mu_\infty} \text{ and } \text{Pé}_\infty = \frac{u_\infty L_\infty}{\kappa_\infty} . \quad (14)$$

241 Note that the Prandtl number used in the original version of the entropy vis-  
242 cosity method is simply given by

$$\text{Pr}_\infty = \text{Pé}_\infty / \text{Re}_\infty . \quad (15)$$

243 The numerical Reynolds and Péclet numbers defined in Eq. (14) are related to  
244 the entropy viscosity coefficients  $\mu_\infty$  and  $\kappa_\infty$ . Thus, once a scaling (in powers of  
245  $M_\infty$ ) is obtained for  $\text{Re}_\infty$  and  $\text{Pé}_\infty$ , the corresponding normalization parameters  
246  $\text{norm}_P^\mu$  and  $\text{norm}_P^\kappa$  will automatically be set. For brevity, the superscripts \* are  
247 omitted in the remainder of this section.

248 For simplicity, we use here the ideal gas equation of state; its non-dimensionalized  
249 expression is given by

$$P^* = (\gamma - 1) \rho^* \left( E^* - \frac{1}{2} M_\infty^2 (u^*)^2 \right) = (\gamma - 1) \rho^* e^* . \quad (16)$$

250 In the low-Mach isentropic limit, shocks cannot form and the compressible  
251 Euler equations are known to converge to the incompressible equations when the  
252 Mach number tends to zero. When adding dissipative terms, as is the case with  
253 the entropy viscosity method, the main properties of the low-Mach asymptotic  
254 limit must be preserved. We begin by expanding each variable in powers of the  
255 Mach number. As an example, the expansion for the pressure is given by:

$$P(\vec{r}, t) = P_0(\vec{r}, t) + P_1(\vec{r}, t) M_\infty + P_2(\vec{r}, t) M_\infty^2 + \dots \quad (17)$$

256 By studying the resulting momentum equations for various powers of  $M_\infty$ , it is

257 observed that the leading order and first-order pressure terms,  $P_0$  and  $P_1$ , are  
 258 spatially constant if and only if  $\text{Re}_\infty = \text{Pe}_\infty = 1$ . In this case, we have at order  
 259  $M_\infty^{-2}$ :

$$\vec{\nabla} P_0 = 0 \quad (18a)$$

260 and at order  $M_\infty^{-1}$

$$\vec{\nabla} P_1 = 0. \quad (18b)$$

261 Using the scaling  $\text{Re}_\infty = \text{Pe}_\infty = 1$ , the leading-order expressions for the conti-  
 262 nuity, momentum, and energy equations are:

$$\partial_t \rho_0 + \vec{\nabla} \cdot (\rho \vec{u})_0 = \vec{\nabla} \cdot (\kappa \vec{\nabla} \rho)_0 \quad (19a)$$

263

$$\partial_t (\rho \vec{u})_0 + \vec{\nabla} \cdot (\rho \vec{u} \otimes \vec{u})_0 + \vec{\nabla} P_2 = \vec{\nabla} \cdot (\rho \mu \vec{\nabla}^s \vec{u} + \kappa \vec{u} \otimes \vec{\nabla} \rho)_0 \quad (19b)$$

264

$$\partial_t (\rho E)_0 + \vec{\nabla} \cdot [\vec{u} (\rho E + P)]_0 = \vec{\nabla} \cdot (\kappa \vec{\nabla} (\rho e))_0 \quad (19c)$$

265 where the notation  $(fg)_0$  means that we only keep the 0<sup>th</sup>-order terms in the  
 266 product  $fg$ . The leading-order of the equation of state is given by

$$P_0 = (\gamma - 1)(\rho E)_0. \quad (20)$$

267 Using Eq. (20), the energy equation can be recast as a function of the leading-  
 268 order pressure,  $P_0$ , as follows:

$$\partial_t P_0 + \gamma \vec{\nabla} \cdot (\vec{u} P)_0 = \vec{\nabla} \cdot (\kappa \vec{\nabla} (P))_0. \quad (21)$$

269 From Eq. (18a), we infer that  $P_0$  is spatially constant. Thus, Eq. (21) becomes

$$\frac{1}{\gamma P_0} \frac{dP_0}{dt} = -\vec{\nabla} \cdot \vec{u}_0 \quad (22)$$

and, at steady state, we have

$$\vec{\nabla} \cdot \vec{u}_0 = 0. \quad (23)$$

That is, the leading-order of velocity is divergence-free. The same reasoning can be applied to the leading-order of the continuity equation (Eq. (19a)) to show that the material derivative of the density variable is zero:

$$\frac{D\rho_0}{Dt} := \partial_t \rho_0 + \vec{u}_0 \cdot \vec{\nabla} \rho_0 = 0. \quad (24)$$

Therefore, we conclude that by setting the Reynolds and Péclet numbers to one, the incompressible fluid results are retrieved in the low-Mach limit when employing the compressible Euler equations with viscous regularization terms present. In addition, the scaling of the Prandtl number can also be obtained using Eq. (15), hence clarifying the use of the numerical Prandtl in the original entropy viscosity method [8].

### 3.3. *Scaling of $Re_\infty$ and $Pé_\infty$ for non-isentropic flows*

Next, we consider the non-isentropic case. Recall that even subsonic flows can present shocks (for instance, a step initial condition in the pressure will trigger shock formation, independently of the Mach number). The non-dimensional form of the Euler equations given in Eq. (13) provides some insight on the dominant terms as a function of the Mach number. This is particular obvious in the momentum equation, Eq. (13b), where the gradient of pressure is scaled by  $1/M_\infty^2$ . In the non-isentropic case, we no longer have  $\frac{\vec{\nabla} P}{M_\infty^2} = \vec{\nabla} P_2$  and therefore the pressure gradient term may need to be stabilized by some dissipative terms of the same scaling so as to prevent spurious oscillations from forming. By inspecting the dissipative terms presents in the the momentum equation, having a dissipative term that scales as  $1/M_\infty^2$  leads to the following three options: (a)  $Re_\infty = M_\infty^2$  and  $Pé_\infty = 1$ , (b)  $Re_\infty = 1$  and  $Pé_\infty = M_\infty^2$ , or (c)

293  $\text{Re}_\infty = \text{Pé}_\infty = M_\infty^2$ . Any of these choices will also affect the stabilization of  
 294 the continuity and energy equations. For instance, using a Péclet number equal  
 295 to  $M_\infty^2$  may effectively stabilize the continuity equation in the shock region but  
 296 this may also add an excessive amount of dissipation for subsonic flows at the  
 297 location of the contact wave. Such a behavior may not be suitable for accuracy  
 298 purpose, making options (b) and (c) inappropriate. The same reasoning, left to  
 299 the reader, can be carried out for the energy equation (Eq. (13c)) and results in  
 300 the same conclusion. The remaining choice, option (a), has the proper scaling:  
 301 in this case, only the dissipation terms involving  $\vec{\nabla}^{s,*} \vec{u}^*$  scale as  $1/M_\infty^2$  since  
 302  $\text{Re}_\infty = M_\infty^2$ , leaving the regularization of the continuity equation unaffected  
 303 because  $\text{Pé}_\infty = 1$ .

#### 304 *3.4. An All-speed normalization of the entropy residual*

305 The study of the above limit cases yields two different possible scalings for  
 306 the Reynolds number:  $\text{Re}_\infty = 1$  in the low-Mach limit and  $\text{Re}_\infty = M_\infty^2$  for  
 307 non-isentropic flows, whereas the numerical Péclet number always scales as one.  
 308 In order to have a stabilization method valid for a wide range of Mach numbers,  
 309 from very low-Mach to supersonic flows, these two scalings should be combined  
 310 in a unique definition.

311 We begin with the normalization parameter  $\text{norm}_P^\kappa$ . Using the definition of  
 312 the viscosity coefficients given in Eq. (10) and the scaling of Eq. (11), it can be  
 313 shown that:

$$\kappa_\infty = \frac{\rho_\infty c_\infty^2 u_\infty L}{\text{norm}_{P,\infty}^\kappa}, \quad (25)$$

314 where  $\text{norm}_{P,\infty}$  is the reference far-field quantity for the normalization param-  
 315 eter  $\text{norm}_P$ . Substituting Eq. (25) into Eq. (14) and recalling that the numerical  
 316 Péclet number scales as unity, we obtain:

$$\text{norm}_{P,\infty}^\kappa = \text{Pé}_\infty \rho_\infty c_\infty^2 = \rho_\infty c_\infty^2. \quad (26)$$



Eq. (26) provides a proper normalization factor to define the  $\kappa$  viscosity coefficient. The derivation for  $\text{norm}_P^\mu$  is similar and yields

$$\text{norm}_P^\mu = \text{Re}_\infty \rho_\infty c_\infty^2 = \begin{cases} \rho \|\vec{u}\|^2 & \text{for non-isentropic flows} \\ \rho c^2 = \text{norm}_P^\kappa & \text{for low-Mach flows} \end{cases} . \quad (27)$$

A smooth function to transition between these two states is obtained by employing a smoothed shifted Heaviside function defined as follows:

$$\sigma(M) = \begin{cases} 0 & \text{if } M \leq M^{\text{thresh}} - a \\ 1 & \text{if } M \geq M^{\text{thresh}} + a \\ \frac{1}{2} \left( 1 + \frac{M - M^{\text{thresh}}}{a} + \frac{1}{\pi} \sin \left( \frac{\pi(M - M^{\text{thresh}})}{a} \right) \right) & \text{otherwise} \end{cases} \quad (28)$$

where  $M^{\text{thresh}}$  is a threshold Mach number value beyond which the flow is no longer considered to be low-Mach (we use  $M^{\text{thresh}} = 0.05$ ),  $M$  is the local Mach number, and the scalar  $a$  determines how rapidly the transition from  $\text{norm}_P^\mu = \rho c^2$  to  $\text{norm}_P^\mu = \rho \|\vec{u}\|^2$  occurs in the vicinity of  $M^{\text{thresh}}$  (e.g.,  $a = 0.005$ ). It is easy to verify that

$$\text{norm}_P^\mu = (1 - \sigma(M)) \rho c^2 + \sigma(M) \rho \|\vec{u}\|^2 \quad (29)$$

satisfies Eq. (27). Finally, we summarize the definition of the viscosity coefficients  $\mu$  and  $\kappa$  for completeness:

$$\kappa(\vec{r}, t) = \min \left( \mu_{\text{max}}(\vec{r}, t), \kappa_e(\vec{r}, t) \right), \quad (30a)$$

$$\mu(\vec{r}, t) = \min \left( \mu_{\text{max}}(\vec{r}, t), \mu_e(\vec{r}, t) \right), \quad (30b)$$

where the first-order viscosity is given by

$$\kappa_{\max}(\vec{r}, t) = \mu_{\max}(\vec{r}, t) = \frac{h}{2} (||\vec{u}|| + c) \quad (30c)$$

and the entropy viscosity coefficients by

$$\kappa_e(\vec{r}, t) = \frac{h^2 \max(\tilde{R}_{\text{ent}}, J)}{\rho c^2} \text{ and } \mu_e(\vec{r}, t) = \frac{h^2 \max(\tilde{R}_{\text{ent}}, J)}{\text{norm}_P^\mu} \quad (30d)$$

with the jumps given by

$$J = \max \left( ||\vec{u}|| [ [\vec{\nabla} P \cdot \vec{n}] ], ||\vec{u}|| c^2 [ [\vec{\nabla} \rho \cdot \vec{n}] ] \right) \quad (30e)$$

where  $\text{norm}_P^\kappa$  is computed from Eq. (29). The jump  $J$  is a function of the jump of pressure and density gradients across the face with respect to its normal vector  $\vec{n}$ . Then, the largest value over all faces is determined and used in the definition of the viscosity coefficients. With the definition of the viscosity coefficients  $\mu$  and  $\kappa$  proposed in Eq. (30), the dissipative terms are expected to scale appropriately for very low-Mach regimes as well for transonic and supersonic flows.

#### 4. Extension of the entropy viscosity technique to Euler equations with variable area

Fluid flows in nozzles and in pipes of varying cross-sectional area can be modeled using the variable-area variant of the Euler equations, where the conservative variables are now multiplied by the area  $A$ . In addition, these equations differ from the standard Euler equations in that the momentum equation Eq. (31b) contains a non-conservative term proportional to the area gradient. Here, the variable area is assumed to be a smooth function of space only.

$$\partial_t (\rho A) + \vec{\nabla} \cdot (\rho \vec{u} A) = 0, \quad (31a)$$

$$\partial_t (\rho \vec{u} A) + \vec{\nabla} \cdot [A (\rho \vec{u} \otimes \vec{u} + P \mathbb{I})] = P \vec{\nabla} A, \quad (31b)$$

$$\partial_t (\rho E A) + \vec{\nabla} \cdot [\vec{u} A (\rho E + P)] = 0. \quad (31c)$$

The application of the entropy viscosity method to the Euler equations with variable area is not fundamentally different to its application to the standard Euler equations. However, we need to derive the associated dissipative terms and verify that the entropy minimum principle is still satisfied. The variable-area Euler equations with viscous regularization are given below; details of the derivation are provided in Appendix B.

$$\partial_t (\rho A) + \vec{\nabla} \cdot (\rho \vec{u} A) = \vec{\nabla} \cdot (A \kappa \vec{\nabla} \rho), \quad (32a)$$

$$\partial_t (\rho \vec{u} A) + \vec{\nabla} \cdot [A (\rho \vec{u} \otimes \vec{u} + P \mathbb{I})] = P \vec{\nabla} A + \vec{\nabla} \cdot \left[ A \left( \mu \rho \vec{\nabla}^s \vec{u} + \kappa \vec{u} \otimes \vec{\nabla} \rho \right) \right], \quad (32b)$$

$$\begin{aligned} \partial_t (\rho A E) + \vec{\nabla} \cdot [\vec{u} A (\rho E + P)] = \\ \vec{\nabla} \cdot \left[ A \left( \kappa \vec{\nabla} (\rho e) + \frac{1}{2} \|\vec{u}\|^2 \kappa \vec{\nabla} \rho + \rho \mu \vec{u} \vec{\nabla}^s \vec{u} \right) \right]. \end{aligned} \quad (32c)$$

The dissipative terms are quite similar to the ones obtained for the standard Euler equations: each dissipative flux is simply multiplied by the variable area  $A$  in order to ensure conservation of the dissipative flux. When assuming a constant area, Eqs. 2 are recovered.

A low-Mach asymptotic limit of the Euler equations with variable area on the same model as in Section 3.2 will lead to the divergence constraint  $\vec{\nabla} \cdot (\vec{u} A) = 0$  that can be recast as  $\vec{\nabla} \cdot \vec{u} = -\vec{u} \cdot \vec{\nabla} A / A$ . The gradient of the area acts as a source term and will force the fluid to accelerate or decelerate, depending on its sign.

## 363 5. Discretizations and Solution Techniques

364 In this section, we briefly describe the spatial and temporal discretizations  
 365 and the solution techniques used to solve the system of equations Eq. (32). For  
 366 conciseness, we re-write the system of equations in the following form:

$$\partial_t \mathbf{U} + \vec{\nabla} \cdot \vec{\mathbf{F}}(\mathbf{U}) = \mathbf{S} + \vec{\nabla} \cdot \mathbf{D}(\mathbf{U}) \vec{\nabla} \mathbf{U} \quad (33)$$

367 where  $\mathbf{U} = [\rho A, \rho \vec{u} A, \rho E A]^T$  is the solution vector,  $\mathbf{F}$  denotes the inviscid flux

$$\vec{\mathbf{F}} \equiv \begin{bmatrix} \rho u A \\ (\rho u^2 + p) A \\ u(\rho E + P) A \end{bmatrix} \quad (34)$$

368 and  $\mathbf{S}$  is a source term that contains the non-conservative term  $P \vec{\nabla} A$ . The term  
 369  $\vec{\nabla} \cdot \mathbf{D}(\mathbf{U}) \vec{\nabla} \mathbf{U}$  stands for the artificial dissipative terms.

### 370 5.1. Spatial and Temporal Discretizations

371 The system of equations given in Eq. (33) is discretized using a continuous  
 372 Galerkin finite element method and temporal integrators available through the  
 373 MOOSE multiphysics framework [14].

#### 374 5.1.1. Continuous Finite Elements

In order to apply the continuous finite element method, Eq. (33) is multiplied  
 by a test function  $\mathbf{W}(\vec{r})$ , integrated by parts and each integral is decomposed  
 into a sum of integrals over each element  $K$  of the discrete mesh  $\Omega$ . The following

weak form is obtained:

$$\begin{aligned} \sum_K \int_K \partial_t \mathbf{U} \mathbf{W} - \sum_K \int_K \vec{\mathbf{F}}(\mathbf{U}) \cdot \vec{\nabla} \mathbf{W} + \int_{\partial\Omega} \vec{\mathbf{F}}(\mathbf{U}) \cdot \vec{n} \mathbf{W} - \sum_K \int_K \mathbf{S} \mathbf{W} \\ + \sum_K \int_K D(\mathbf{U}) \vec{\nabla} \mathbf{U} \cdot \vec{\nabla} \mathbf{W} - \int_{\partial\Omega} D(\mathbf{U}) \vec{\nabla} \mathbf{U} \cdot \vec{n} \mathbf{W} = 0. \end{aligned} \quad (35)$$

375 The integrals over the elements  $K$  are evaluated using a numerical quadrature.  
 376 The MOOSE framework provides a wide range of test functions and quadrature  
 377 rules. Linear Lagrange polynomials are employed as test functions in the re-  
 378 sults section. Second-order spatial convergence will be demonstrated for smooth  
 379 solutions.

#### 380 5.1.2. Temporal integration

381 The MOOSE framework offers both first- and second-order explicit and im-  
 382 plicit temporal integrators. In all of the numerical examples presented in Sec-  
 383 tion 6, the temporal derivative will be evaluated using the second-order, back-  
 384 ward difference temporal integrator BDF2. By considering three consecutive  
 385 solutions,  $\mathbf{U}^{n-1}$ ,  $\mathbf{U}^n$  and  $\mathbf{U}^{n+1}$ , at times  $t^{n-1}$ ,  $t^n$  and  $t^{n+1}$ , respectively, BDF2  
 386 can be expressed as:

$$\int_K \partial_t \mathbf{U} \mathbf{W} = \int_K (\omega_0 \mathbf{U}^{n+1} + \omega_1 \mathbf{U}^n + \omega_2 \mathbf{U}^{n-1}) \mathbf{W}, \quad (36)$$

with

$$\begin{aligned} \omega_0 = \frac{2\Delta t^{n+1} + \Delta t^n}{\Delta t^{n+1}(\Delta t^{n+1} + \Delta t^n)}, \quad \omega_1 = -\frac{\Delta t^{n+1} + \Delta t^n}{\Delta t^{n+1}\Delta t^n}, \\ \text{and } \omega_2 = \frac{\Delta t^{n+1}}{\Delta t^n(\Delta t^{n+1} + \Delta t^n)} \end{aligned}$$

387 where  $\Delta t^n = t^n - t^{n-1}$  and  $\Delta t^{n+1} = t^{n+1} - t^n$ .

## 388 5.2. Boundary conditions

389 Boundary conditions are implemented by performing a characteristic decom-  
 390 position to compute the appropriate flux at the boundaries. Our implementation  
 391 of the subsonic boundary conditions is inspired by the method described in [20]  
 392 and was adapted for a time implicit solver. Neumann boundary conditions are  
 393 used for all of the boundary types, except for the inlet supersonic boundary that  
 394 are strongly imposed with Dirichlet boundary conditions.

395 For each numerical solution presented in Section 6, the type of boundary con-  
 396 ditions used will be specified and taken among the following: supersonic inlet,  
 397 subsonic inlet (stagnation pressure boundary), subsonic outlet, and supersonic  
 398 outlet. The artificial diffusion coefficient  $D(\mathbf{U})$  is set to zero at the boundary of  
 399 the computational domain so that the boundary term  $\int_{\partial\Omega} D(\mathbf{U}) \vec{\nabla} \mathbf{U} \cdot \vec{n} \mathbf{W}$  stem-  
 400 ming from the integration by parts of the artificial dissipative terms in Eq. (35)  
 401 is ignored.

## 402 5.3. Solver

403 A Jacobian-free-Newton-Krylov (JFNK) method is used to solve for the so-  
 404 lution at the end of each time step. An approximate Jacobian matrix of the  
 405 discretized equations was derived and implemented. Obtaining the matrix en-  
 406 tries requires that the partial derivatives of pressure with respect to the conser-  
 407 vative variables be known (this is relatively simple for the stiffened and ideal  
 408 gas equations of state but may be more complex for general equations of state).  
 409 The contributions of the artificial dissipative terms to the Jacobian matrix are  
 410 approximated by lagging the viscosity coefficients (computing them with the  
 411 previous solution). For instance, this is shown in Eq. (37) for the dissipative  
 412 terms present in the continuity equation:

$$\frac{\partial}{\partial \mathbf{U}} \left( \kappa \vec{\nabla} \cdot \rho \vec{\nabla} W \right) \simeq \kappa \frac{\partial}{\partial \mathbf{U}} \left( \vec{\nabla} \cdot \rho \vec{\nabla} W \right), \quad (37)$$

where  $\mathbf{U}$  denotes any of the conservative variables and  $W$  denotes the component of  $\mathbf{W}$  associated with the continuity equation. In the above, we have neglected  $\frac{\partial \kappa}{\partial \mathbf{U}}$ .

## 6. Numerical Results

1-D and 2-D numerical solutions for the Euler equations with viscous regularization solved using the entropy viscosity method are presented here. Our results show that the new definitions for the viscosity coefficients are robust in the low-Mach limit as well as for transonic and supersonic flows and that shocks are appropriately resolved.

The first set of 1-D simulations consist of liquid water and steam flowing in a converging-diverging nozzle. This test is of interest for multiple reasons: (a) a steady state can be reached (some stabilization methods are known to have difficulties reaching a steady state, [2, 3]), (b) an analytical solution is available and a space-time convergence study can be performed, (c) it can be performed for liquid and gas phases, wherein the gas phase simulation presents a shock while the liquid-phase simulation has a significantly lower Mach number. Next, a 1-D shock tube test (in a straight pipe), taken from the Leblanc test-case suite [21], is performed. This test is known to be more challenging than Sod shock tubes and the fluid's Mach number varies spatially between 0 and 5. A convergence study is also performed to demonstrate convergence of the numerical solution to the exact solution. A slow moving shock is also investigated [22]. This test helps in assessing the ability of the method to damp the post-shock low frequency noise (oscillations). Finally, a strong shock for a liquid phase (Mach number around 0.1) is also performed [23].

The initial conditions for the aforementioned 1-D test cases are given in Table 1.

$\rho_{\text{left}}$	$u_{\text{left}}$	$P_{\text{left}}$	$\rho_{\text{right}}$	$u_{\text{right}}$	$P_{\text{right}}$
Leblanc shock tube (Section 6.3)					
1	0	$4 \cdot 10^{-2}$	$10^{-3}$	0	$4 \cdot 10^{-11}$
Strong shock for liquid phase (Section 6.4)					
1000	0	$10^9$	1000	0	$10^5$
Slow moving shock (Section 6.5)					
1	-0.81	1	3.86	-3.44	10.33

Table 1: Initial conditions for the 1-D test cases (density in  $kg/m^3$ , velocity in  $m/s$ , pressure in  $Pa$ ).

439 The 2-D simulations are outlined next. First, 2-D subsonic flows around a  
440 cylinder [12] and over a circular hump [24] are presented for various far-field  
441 Mach numbers (as low of  $10^{-7}$ ). Numerical results of a supersonic flow over  
442 a compression corner are provided to illustrate the ability of the new viscosity  
443 definitions to handle supersonic flows. Convergence studies are performed when  
444 analytical solutions are available.

445 For each simulation, data relative to the boundary conditions, the Courant-  
446 Friedrichs-Lewy number ( $CFL$ ), mesh and equation of state are provided. All of  
447 the numerical solutions presented are obtained using BDF2 as temporal integra-  
448 tor and linear (1-D mesh),  $\mathbb{P}_1$  (2-D triangular mesh), and  $\mathbb{Q}_1$  (2-D quadrangular  
449 mesh) finite elements. The spatial integrals are numerically computed using a  
450 second-order Gauss quadrature rule. Steady-state is detected in a transient sim-  
451 ulation by monitoring the nonlinear residual before proceeding with the Newton  
452 solves for a given time step. The ideal gas [25] or stiffened gas equations of state  
453 [26] are used; a generic expression is given in Eq. (38).

$$P = (\gamma - 1)\rho(e - q) - \gamma P_\infty \quad (38)$$

where the parameters  $\gamma$ ,  $q$ , and  $P_\infty$  are fluid-dependent and are given in Table 2.  
The ideal gas equation of state is recovered by setting  $q = P_\infty = 0$  in Eq. (38).



The entropy function for the stiffened gas equation of state is concave and given

Table 2: Stiffened Gas Equation of State parameters for steam and liquid water.

fluid	$\gamma$	$C_v$ ( $J.kg^{-1}.K^{-1}$ )	$P_\infty$ (Pa)	$q$ ( $J.kg^{-1}$ )
liquid water (Section 6.1)	2.35	1816	$10^9$	$-1167 \cdot 10^3$
steam (Section 6.2)	1.43	1040	0	$2030 \cdot 10^3$
liquid water (Section 6.4)	4.4	1000	$6 \cdot 10^8$	0

by

$$s = C_v \ln \left( \frac{P + P_\infty}{\rho^{\gamma-1}} \right),$$

where  $C_v$  is the heat capacity at constant volume.

Finally, the convergence rates are computed using the following relation

$$\text{rate}_h = \ln \left( \frac{\|U_{2h} - U_{\text{exact}}\|}{\|U_h - U_{\text{exact}}\|} \right) / \ln 2 \quad (39)$$

where  $\|\cdot\|$  denotes either the  $L_1$  or  $L_2$  norms and  $h$  is the characteristic grid

size.

#### 6.1. Liquid water in a 1-D converging-diverging nozzle

A simulation for liquid flow through a 1-D converging-diverging nozzle is performed. The variable area expression is given by  $A(x) = 1 + 0.5 \cos(2\pi x/L)$  with length  $L = 1m$ . At the inlet, the stagnation pressure and temperature are set to  $P_0 = 1MPa$  and  $T_0 = 453K$ , respectively. At the outlet, only the static pressure is specified:  $P_s = 0.5MPa$ . Initially, the liquid is at rest, the temperature is uniform and equal to the stagnation temperature and the pressure linearly decreases from the stagnation pressure inlet value to the static pressure outlet value. The stiffened gas equation of state is used to model the liquid water with the parameters provided in Table 2. Because of the low pressure

469 difference between the inlet and the outlet, the smooth initial conditions, and  
 470 the large value of  $P_\infty$  in Eq. (38), the flow remains subsonic and thus displays  
 471 no shock. A detailed derivation of the exact steady-state solution can be found  
 472 in [27]. A uniform mesh of 50 cells was used to obtain the numerical solution  
 473 and the time step size was computed using a  $CFL$  number of 750. Plots of  
 474 the Mach number, density, and pressure are given at steady state in Fig. 1 for  
 475 the numerical and exact solutions. The viscosity coefficients are also graphed  
 476 in Fig. 1d.

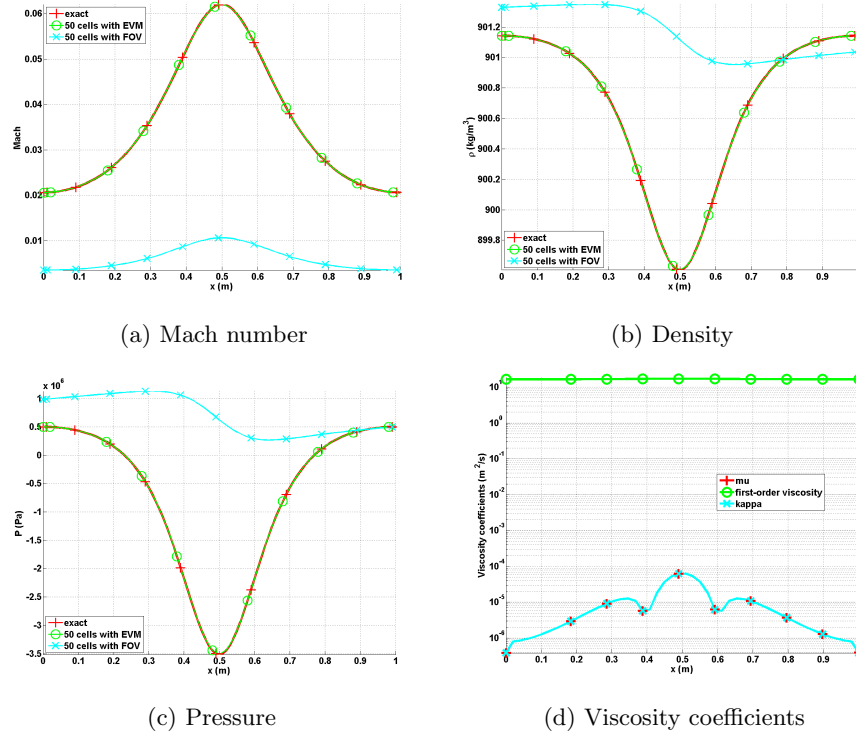


Figure 1: Steady-state solution for a liquid flowing through a 1-D converging-diverging nozzle.

477 In Fig. 1, the numerical solutions obtained using the first-order viscosity  
 478 (FOV) and the entropy viscosity method (EVM) are plotted against the exact

479 solution. The numerical solution obtained with the EVM and the exact solution  
 480 overlap, even for a fairly coarse mesh (50 cells). On the other hand, the numer-  
 481 ical solution obtained with the FOV does not give the correct steady state: this  
 482 is an illustration of the effect of ill-scaled dissipative terms in the low-Mach limit  
 483 when using the FOV. Note that the entropy viscosity coefficient is very small  
 484 compared to the first-order one (Fig. 1d): (i) the numerical solution is smooth  
 485 as shown in Fig. 1 and (ii) the flow is in a isentropic low-Mach regime A conver-  
 486 gence study was performed using the exact solution as a reference: the  $L_1$  and  
 487  $L_2$  norms of the error and the corresponding convergence rates are computed at  
 488 steady state on various uniform meshes from 4 to 256 cells. Spatial convergence  
 489 results using linear finite elements are reported in Table 3 and Table 4 for the  
 490 primitive variables: density, velocity and pressure.

Table 3:  $L_1$  norm of the error for the liquid phase in a 1-D converging-diverging nozzle at steady state.

cells	density	rate	pressure	rate	velocity	rate
4	$2.8037 \cdot 10^{-1}$	—	$8.4705 \cdot 10^5$	—	7.2737	—
8	$1.3343 \cdot 10^{-1}$	1.07	$4.7893 \cdot 10^5$	0.82	6.1493	0.24
16	$2.9373 \cdot 10^{-2}$	2.18	$1.0613 \cdot 10^5$	2.17	1.2275	2.32
32	$5.1120 \cdot 10^{-3}$	2.52	$1.8446 \cdot 10^4$	2.52	$1.8943 \cdot 10^{-1}$	2.69
64	$1.0558 \cdot 10^{-3}$	2.28	$3.7938 \cdot 10^3$	2.28	$3.7919 \cdot 10^{-2}$	2.32
128	$2.3712 \cdot 10^{-4}$	2.15	$8.4471 \cdot 10^2$	2.17	$8.5517 \cdot 10^{-3}$	2.15
256	$5.6058 \cdot 10^{-5}$	2.08	$1.9839 \cdot 10^2$	2.09	$2.0475 \cdot 10^{-3}$	2.06
512	$1.3278 \cdot 10^{-5}$	2.08	$4.6622 \cdot 10^1$	2.09	$4.9516 \cdot 10^{-4}$	2.04
1024	$3.1193 \cdot 10^{-6}$	2.08	$1.1755 \cdot 10^1$	1.99	$1.2379 \cdot 10^{-4}$	2.00

Table 4:  $L_2$  norm of the error for the liquid phase in a 1-D converging-diverging nozzle at steady state.

cells	density	rate	pressure	rate	velocity	rate
4	$3.106397 \cdot 10^{-1}$	—	$5.254445 \cdot 10^5$	—	3.288543	—
8	$7.491623 \cdot 10^{-2}$	2.05	$1.636966 \cdot 10^5$	1.68	1.823880	0.85
16	$2.079858 \cdot 10^{-2}$	1.85	$4.627338 \cdot 10^4$	1.49	$4.990605 \cdot 10^{-1}$	0.87
32	$5.329627 \cdot 10^{-3}$	1.96	$1.180287 \cdot 10^4$	1.97	$1.261018 \cdot 10^{-1}$	1.98
64	$1.341583 \cdot 10^{-3}$	1.99	$2.967104 \cdot 10^3$	1.99	$3.160914 \cdot 10^{-2}$	1.99
128	$3.359766 \cdot 10^{-4}$	1.99	$7.428087 \cdot 10^2$	1.99	$7.907499 \cdot 10^{-3}$	1.99
256	$8.403859 \cdot 10^{-5}$	1.99	$1.857861 \cdot 10^2$	1.99	$1.977292 \cdot 10^{-3}$	1.99
512	$2.10075 \cdot 10^{-5}$	2.00	$4.7024 \cdot 10^1$	1.98	$4.9516 \cdot 10^{-4}$	1.99

We note that the convergence rates measured in both the  $L_1$  and  $L_2$  norm of the error are equal to 2; the entropy viscosity method preserves the high-order accuracy of the discretization used when the numerical solution is smooth. The new definition of the entropy viscosity coefficients behaves appropriately in the low-Mach limit.

## 6.2. Steam in a 1-D converging-diverging nozzle

We use the same nozzle geometry, initial conditions and boundary conditions as in the previously example but replace liquid water with steam and use the steam parameters of the stiffened gas equation of state, Table 2. In this example, compressible effects will become dominant. The pressure difference between the inlet and outlet is large enough to accelerate the steam through the nozzle, leading to the formation of a shock in the diverging portion of the nozzle. The behavior is different from the one observed for the liquid water phase in Section 6.1 because of the liquid to gas density ratio is about 1,000. An exact solution at steady state is available for the gas phase [27]. The aim of this section is to show that when using the new definitions of the viscosity coefficients (Eq. (30)), the shock can be correctly resolved without spurious oscillations. The steady-state numerical solution, obtained using a uniform mesh

509 with 500 cells, is shown in Fig. 2. The  $CFL$  was set to 80 (a high  $CFL$  value  
510 can be used because the shock is stationary).

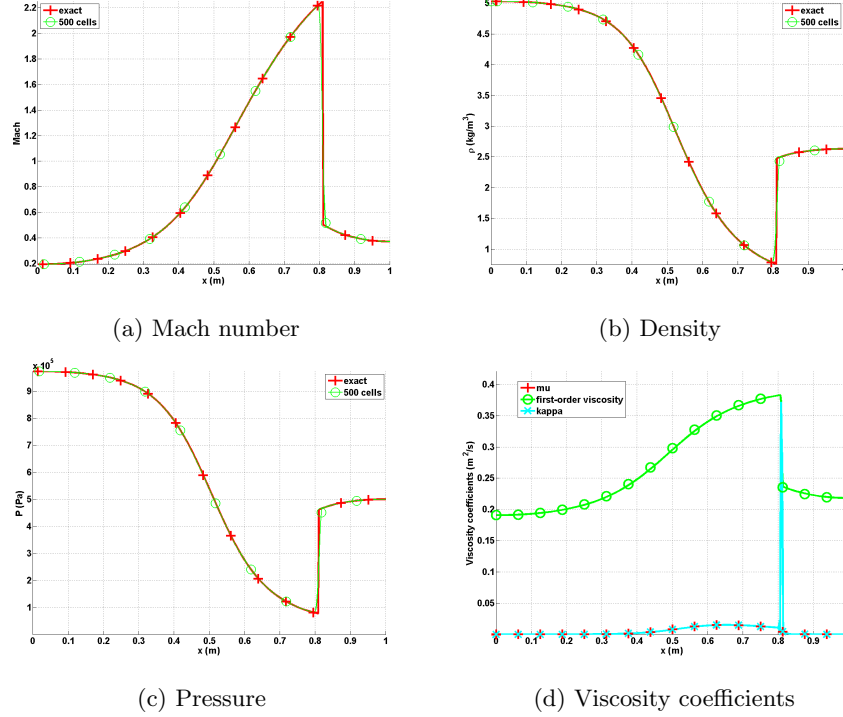


Figure 2: Steady-state solution for vapor phase flowing in a 1-D converging-diverging nozzle.

511 The steady-state solution of the density, Mach number and pressure are  
512 given in Fig. 2. The steady-state solution exhibits a shock around  $x = 0.8m$   
513 and matches the exact solution. In Fig. 2d, the first-order and entropy viscosity  
514 coefficients are plotted at steady state (on a log scale): the entropy viscosity  
515 coefficient is peaked in the shock region around  $x = 0.8m$  where it saturates to  
516 the first-order viscosity coefficient. Elsewhere, the entropy viscosity coefficient  
517 is small. In order to prove convergence of the numerical solution to the exact  
518 solution, a convergence study is performed. Because of the presence of a shock,  
519 second-order accuracy is not expected and the convergence rate of a numerical

520 solution should be 1 and  $1/2$  when measured in the  $L_1$  and  $L_2$  norms, respec-  
521 tively (see Theorem 9.3 in [28]). Results are reported in Table 5 and Table 6  
522 for the primitive variables: density, velocity and pressure. The convergence  
523 rates for the  $L_1$  and  $L_2$  norms of the error computed using Eq. (39) are in good  
agreement with the theoretical values.

Table 5:  $L_1$  norm of the error for the vapor phase in a 1-D converging-diverging nozzle at steady state.

cells	density	rate	pressure	rate	velocity	rate
5	$0.72562 \cdot 10^{-1}$	—	$1.5657 \cdot 10^5$	—	173.69	—
10	$0.4165 \cdot 10^{-1}$	0.80	$9.6741 \cdot 10^4$	0.63	120.69	0.53
20	$0.20675 \cdot 10^{-1}$	1.01	$4.9193 \cdot 10^4$	0.97	72.149	0.74
40	$0.093703 \cdot 10^{-1}$	1.14	$2.0103 \cdot 10^4$	0.73	34.716	1.06
80	$0.047328 \cdot 10^{-1}$	0.99	$1.0208 \cdot 10^4$	0.98	16.082	1.11
160	$0.023965 \cdot 10^{-2}$	0.98	$5.1969 \cdot 10^3$	0.97	7.9573	1.02
320	$0.020768 \cdot 10^{-2}$	1.03	$2.5116 \cdot 10^3$	1.05	3.7812	1.07
640	$0.0059715 \cdot 10^{-2}$	0.98	$1.2754 \cdot 10^3$	0.98	1.8353	1.04

524

Table 6:  $L_2$  norm of the error for the vapor phase in a 1-D converging-diverging nozzle at steady state.

cells	density	rate	pressure	rate	velocity	rate
5	$9.7144 \cdot 10^{-1}$	—	$2.0215 \cdot 10^5$	—	236.94	—
10	$5.9718 \cdot 10^{-1}$	0.70	$1.3024 \cdot 10^5$	0.63	166.56	0.51
20	$2.9503 \cdot 10^{-1}$	1.02	$6.6503 \cdot 10^4$	0.97	103.36	0.69
40	$1.8193 \cdot 10^{-1}$	0.69	$4.0171 \cdot 10^4$	0.73	66.374	0.64
80	$1.3366 \cdot 10^{-1}$	0.44	$2.3163 \cdot 10^4$	0.44	42.981	0.63
160	$9.6638 \cdot 10^{-2}$	0.47	$1.7263 \cdot 10^4$	0.42	31.717	0.44
320	$7.0896 \cdot 10^{-2}$	0.45	$1.2763 \cdot 10^4$	0.44	23.138	0.45
640	$5.2191 \cdot 10^{-2}$	0.44	$9.4217 \cdot 10^3$	0.44	16.910	0.45

### 525 6.3. Leblanc shock tube

526 The 1-D Leblanc shock tube is a Riemann problem designed to test the  
527 robustness and the accuracy of stabilization methods. The initial conditions  
528 are given in Table 1. The ideal gas equation of state (with  $\gamma = 5/3$ ) is used to  
529 compute the pressure. This test is computationally challenging because of the

530 large pressure ratio at the initial interface. The computational domain consists  
 531 of a 1-D straight pipe of length  $L = 9m$  with the initial interface located at  
 532  $x = 2m$ . At  $t = 0s$ , the interface is removed. The numerical solution is run  
 533 until  $t = 4s$  and the density, momentum and total energy profiles are given in  
 534 Fig. 3, along with the exact solution. The viscosity coefficients are also plotted  
 535 in Fig. 3d. These plots were run with three different uniform meshes of 800,  
 536 3200, and 6000 cells and a constant  $CFL = 1$ .

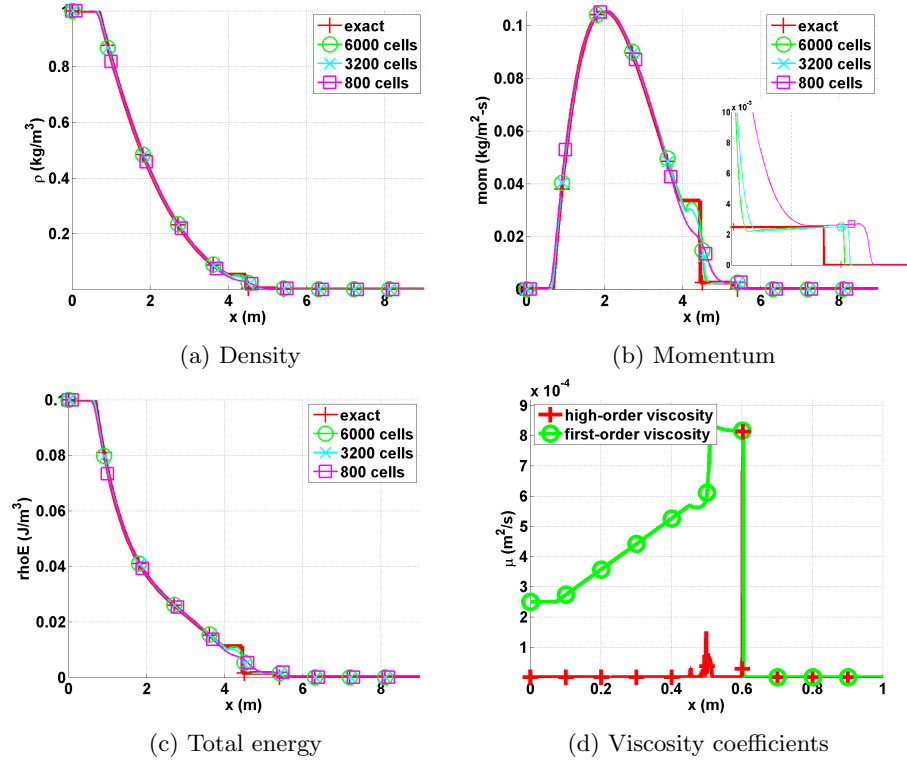


Figure 3: Exact and Numerical solutions for the 1-D Leblanc shock tube at  $t = 4s$ .

537 The density, momentum and total energy profiles are provided in Fig. 3.  
 538 In Fig. 3b, the shock region is zoomed in for better resolution: the shock is  
 539 well resolved. We also observe that the shock position computed numerically

converges to the exact position under mesh refinement. The contact wave at  $x = 4.5m$  can be seen in Fig. 3b. The entropy viscosity coefficient profile is shown in Fig. 3d and behaves as expected: it saturates to the first-order viscosity in the shock region, thus preventing oscillations from forming. At the location of the contact wave, a smaller peak is observed and is due to the presence of the jump terms in the definition of the entropy viscosity coefficient (Eq. (30)). The Mach number, not plotted, is of the order of 1.3 just before the shock and reaches a maximum value close to 5 in the contact region.

Once again, a convergence study is performed in order to prove convergence of the numerical solution to the exact solution. As in the previous example (vapor phase in the 1-D nozzle, Section 6.2), the expected convergence rates in the  $L_1$  and  $L_2$  norms are 1 and  $1/2$ , respectively. The exact solution was obtained by running a 1-D Riemann solver and used as the reference solution to compute the  $L_1$  and  $L_2$ -norms that are reported in Table 7 and Table 8 for the conservative variables: density, momentum and total energy. The convergence rates are again approaching their theoretical values.

Table 7:  $L_1$  norm of the error for the 1-D Leblanc test at  $t = 4s$ .

cells	density	rate	momentum	rate	total energy	rate
100	$1.0354722 \cdot 10^{-2}$	—	$3.5471714 \cdot 10^{-3}$	—	$1.4033046 \cdot 10^{-3}$	—
200	$7.2680512 \cdot 10^{-3}$	0.51	$2.5933119 \cdot 10^{-3}$	0.45	$9.8611746 \cdot 10^{-4}$	0.51
400	$5.0825628 \cdot 10^{-3}$	0.52	$2.0668092 \cdot 10^{-3}$	0.33	$7.7844421 \cdot 10^{-4}$	0.34
800	$3.4025056 \cdot 10^{-3}$	0.58	$1.4793838 \cdot 10^{-3}$	0.48	$5.5702549 \cdot 10^{-4}$	0.48
1600	$2.1649953 \cdot 10^{-3}$	0.65	$9.7152832 \cdot 10^{-4}$	0.61	$3.5720171 \cdot 10^{-4}$	0.64
3200	$1.2465433 \cdot 10^{-3}$	0.79	$5.5937409 \cdot 10^{-4}$	0.79	$2.0491799 \cdot 10^{-4}$	0.80
6400	$6.4476928 \cdot 10^{-4}$	0.95	$3.0244198 \cdot 10^{-4}$	0.89	$1.0914891 \cdot 10^{-4}$	0.91
12800	$3.3950948 \cdot 10^{-4}$	0.93	$1.5958118 \cdot 10^{-4}$	0.92	$5.7909794 \cdot 10^{-5}$	0.91

#### 6.4. 1-D shock tube with a liquid phase

The purpose of this test is to investigate the ability of the entropy viscosity method to stabilize a strong shock with a small Mach number [23] (this reference



Table 8:  $L_2$  norm of the error for the 1-D Leblanc test at  $t = 4s$ .

cells	density	rate	momentum	rate	total energy	rate
100	$5.7187851 \cdot 10^{-3}$	—	$1.7767236 \cdot 10^{-3}$	—	$7.6112265 \cdot 10^{-4}$	—
200	$3.8995238 \cdot 10^{-3}$	0.55	$1.4913161 \cdot 10^{-3}$	0.25	$5.5497308 \cdot 10^{-4}$	0.46
400	$2.8103526 \cdot 10^{-3}$	0.47	$1.3305301 \cdot 10^{-3}$	0.16	$4.6063172 \cdot 10^{-4}$	0.27
800	$2.1081933 \cdot 10^{-3}$	0.41	$1.1398931 \cdot 10^{-3}$	0.22	$3.7798953 \cdot 10^{-4}$	0.29
1600	$1.5731052 \cdot 10^{-3}$	0.42	$9.0394227 \cdot 10^{-4}$	0.33	$2.9584646 \cdot 10^{-4}$	0.35
3200	$1.0610667 \cdot 10^{-3}$	0.57	$6.2735595 \cdot 10^{-4}$	0.53	$2.054455 \cdot 10^{-4}$	0.53
6400	$7.3309974 \cdot 10^{-4}$	0.53	$4.4545754 \cdot 10^{-4}$	0.49	$1.4670834 \cdot 10^{-4}$	0.49
12800	$5.1020991 \cdot 10^{-4}$	0.52	$3.1266758 \cdot 10^{-4}$	0.51	$1.0299897 \cdot 10^{-5}$	0.51

is for a two-phase flow model but we are only interested in the initial conditions for the liquid phase): the Mach number in the shock region is of the order of 0.1. In this case, as explained in Section 3.2, the viscosity coefficients are required to have different order of magnitude in order to ensure the correct scaling of the dissipative terms. The purpose of this test is to validate the approach presented in Section 3.2.

The stiffened gas equation of state is used to model a liquid flow with the parameters given in Table 2. The computational domain of length  $L = 1m$  is uniformly discretized using 500 cells. The step initial conditions are given in Table 1. The simulation is run with a  $CFL = 1$  until the final time  $t_{\text{final}} = 7 \cdot 10^{-5}s$ . Results for pressure, density, velocity and the viscosity coefficients are given in Fig. 4 along with the exact solution for comparison purposes. The numerical solution is in good agreement with exact solution in Fig. 4a. The viscosity coefficients  $\mu$  and  $\kappa$  are not equal in the shock because the Mach number is of order 0.1. The viscosity coefficient  $\kappa$  saturates to the first-order viscosity in the shock region around  $x = 0.65m$  and is sufficient to stabilize the numerical scheme.

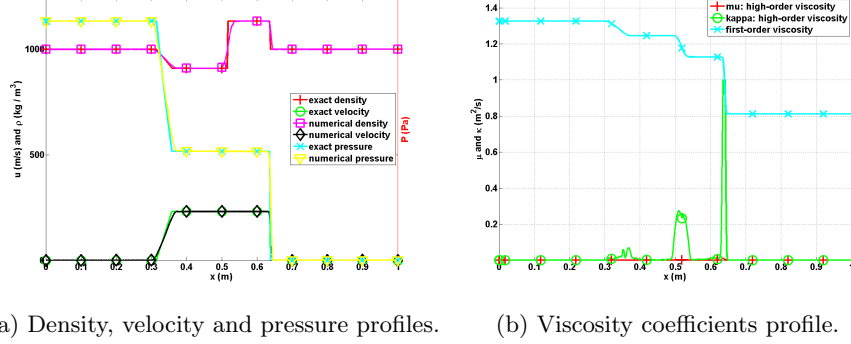


Figure 4: Numerical solution for the 1-D liquid shock tube at  $t_{\text{final}} = 7 \cdot 10^{-5} \text{ s}$ .

#### 6.5. 1-D slow moving shock

Slow moving shocks are known to produce post-shock noise of low frequency that is not damped by some numerical dissipation methods [22]. The aim of this simulation is to test the ability of the entropy viscosity method to dampen the low frequency waves. The 1-D slow moving shock consists of a shock wave moving from left to right with the initial conditions given in Table 1. The ideal gas equation of state is used with a heat capacity ratio  $\gamma = 1.4$ . In order to make the shock travel a significant distance, the final time is taken equal to  $t = 1.1 \text{ s}$ . A pressure boundary condition is used at the left boundary to let the rarefaction and contact waves exit the domain. The numerical solution, obtained with 200 equally-spaced cells, is given in Fig. 5 and is compared to the exact solution obtained from a Riemann solver. We use a  $CFL$  of 1. With this  $CFL$  value, it takes about 50 time steps for the shock to traverse one cell. The numerical results are in good agreement with the exact solution and do not display any post-shock noise. The rarefaction and contact waves are not visible on Fig. 5a since they exited the computational domain through the left pressure boundary condition earlier. As explained in [29], Godunov's type methods usually fail to resolve a slow moving shock because of the nature of the stabilization method:

594 the method scales as the eigenvalue of the appropriate field. In the case of a slow  
 595 moving shock, the dissipation added to the system is under-estimated and leads  
 596 to post-shock noise. In the case of the entropy viscosity method, the entropy  
 597 residual detects the shock position and the viscosity coefficients saturate to the  
 598 first-order viscosity values in the shock region. The main difference between a  
 599 Godunov's type method and the entropy viscosity method lies in the definition of  
 600 the first-order viscosity coefficients that are proportional to the *local maximum*  
 601 *eigenvalue*  $||\vec{u}|| + c$  and not to the eigenvalue of the characteristic field.

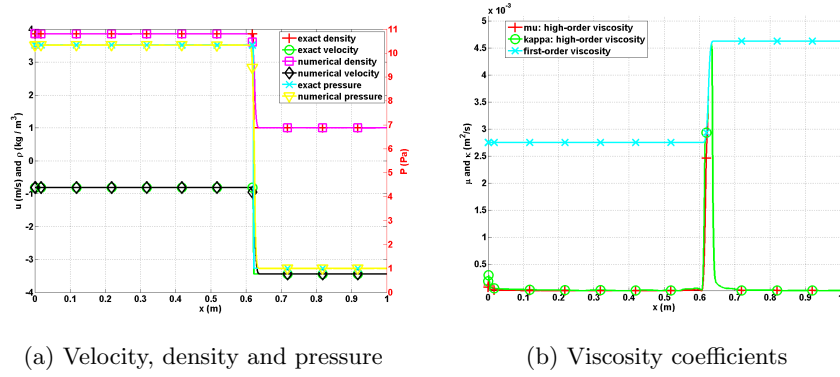


Figure 5: Slow moving shock profiles at  $t = 1.1s$ .

#### 6.6. Subsonic flow over a 2-D cylinder

603 Fluid flow over a 2-D cylinder is often used as a benchmark case to test nu-  
 604 merical schemes in the low-Mach regime [10, 11, 12]. For this test, an analytical  
 605 solution is available in the incompressible limit and is often referred to as the  
 606 potential steady-state flow solution. The main features of the potential flow are  
 607 the following:

- The solution is symmetric: the iso-Mach contour lines are used to assess the symmetry of the numerical solution;
- The steady-state velocity at the top of the cylinder is twice the incoming velocity set at the inlet;

- The spatial steady-state pressure variations are proportional to the square of inlet Mach number, i.e.,

$$\delta P = \frac{\max(P(\vec{r})) - \min(P(\vec{r}))}{\max(P(\vec{r}))} \propto M_\infty^2 \quad (40)$$

where  $\delta P$  and  $M_\infty$  denote the spatial steady-state pressure variations and the inlet Mach number, respectively.

The computational domain consists of a  $1 \times 1$  square with a circular hole of radius 0.05 in its center. A  $\mathbb{P}_1$  triangular mesh with 4008 triangular elements is employed to discretize the geometry. The ideal gas equation of state, with  $\gamma = 1.4$  is used. At the inlet, a subsonic stagnation boundary condition is used: the stagnation pressure and temperature are computed using the following relations:

$$\begin{cases} P_0 = P \left(1 + \frac{\gamma-1}{2} M^2\right)^{\frac{\gamma}{\gamma-1}} \\ T_0 = T \left(1 + \frac{\gamma-1}{2} M^2\right) \end{cases} \quad (41)$$

A static pressure boundary condition, with static pressure  $P_s = 101,325 \text{ Pa}$ , is set at the outlet boundary. The implementation of the pressure boundary conditions is based on [20]. A solid wall boundary condition is set for the top and bottom walls of the computational domain. The simulations are run until a steady state is reached (with a  $CFL$  of 40). When the residual norm (for all equations) is less than  $10^{-12}$  the steady state is considered to have been reached.

Several simulations are performed, with inlet Mach numbers  $M_{\text{inlet}}$  ranging from  $10^{-3}$  to  $10^{-7}$ , and are shown in Fig. 6. The iso-Mach contour lines are drawn using 30 equally-spaced intervals, from  $2 \times 10^{-10}$  to  $M_{\text{inlet}}$ .

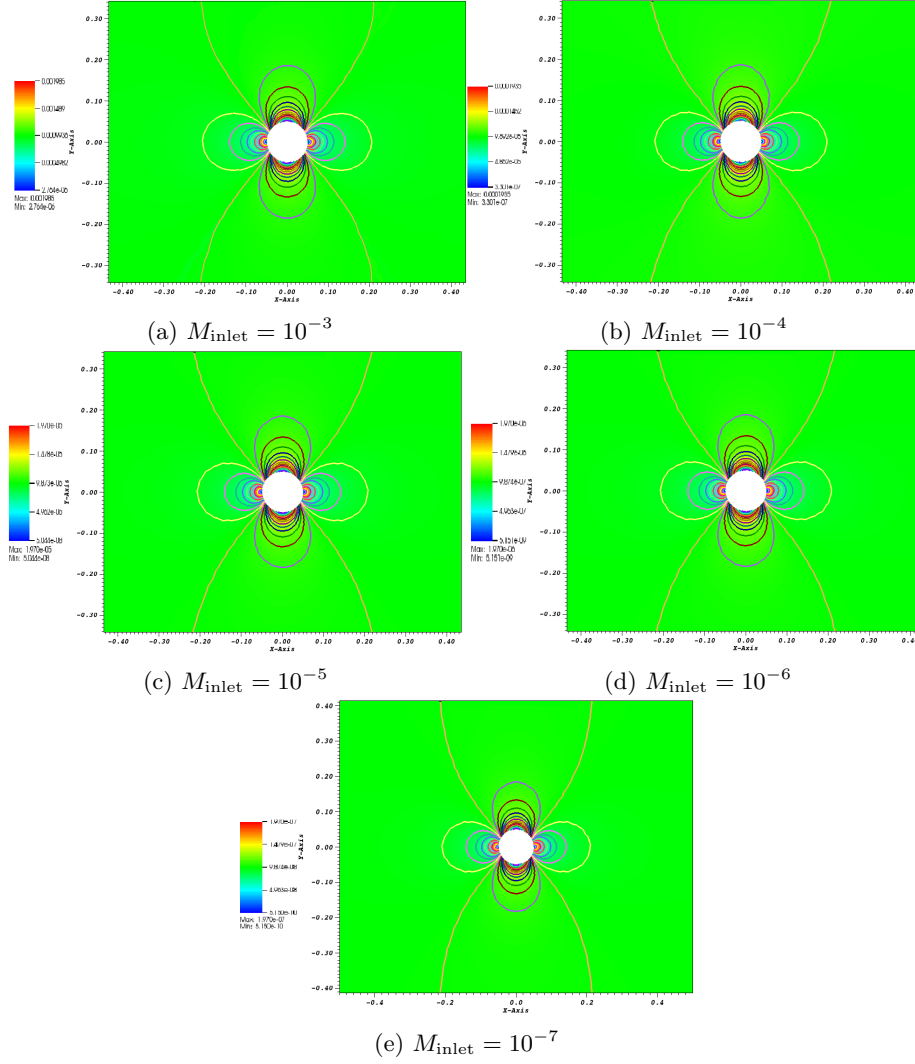


Figure 6: Iso-Mach lines for a subsonic flow over a 2-D cylinder with inlet Mach number values from ranging from  $10^{-3}$  to  $10^{-7}$  (steady-state solution).

631 The steady-state velocity at the top of the cylinder and at the inlet are  
 632 given for different Mach-number values (ranging from  $10^{-3}$  to  $10^{-7}$ ) in Table 9.  
 633 The ratio of the inlet velocity to the velocity at the top of cylinder is also  
 634 computed and is very close to the theoretical value of 2 that is expected in the  
 635 incompressible limit.

Table 9: Steady-state velocity ratio for different Mach numbers.

Mach number	inlet velocity	velocity at the top of the cylinder	ratio
$10^{-3}$	$2.348 \cdot 10^{-3}$	$1.176 \cdot 10^{-3}$	1.99
$10^{-4}$	$2.285 \cdot 10^{-4}$	$1.145 \cdot 10^{-4}$	1.99
$10^{-5}$	$2.283 \cdot 10^{-5}$	$1.144 \cdot 10^{-5}$	1.99
$10^{-6}$	$2.283 \cdot 10^{-6}$	$1.144 \cdot 10^{-6}$	1.99
$10^{-7}$	$2.283 \cdot 10^{-7}$	$1.144 \cdot 10^{-7}$	1.99

636 In Fig. 7, the spatial steady-state variations in pressure and velocity are  
 637 plotted as a function of the Mach number (on a log-log scale). The spatial  
 638 steady-state pressure variations are expected to be of the order of  $M^2$  in the  
 639 incompressible limit, which we observe. From Bernoulli's principle, this implies  
 640 that the velocity spatial variations should be of order  $M$  in the incompressible  
 641 limit, which we also observe in Fig. 7. It is known that some stabilization  
 642 methods, e.g., [10, 11, 12], can produce spatial steady-state pressure variations  
 643 with the wrong Mach-number order. Here, the entropy viscosity method yields  
 644 the correct orders in the low-Mach limit. For ease of comparison, reference lines  
 645 with slope values of 1 and 2 are also plotted.

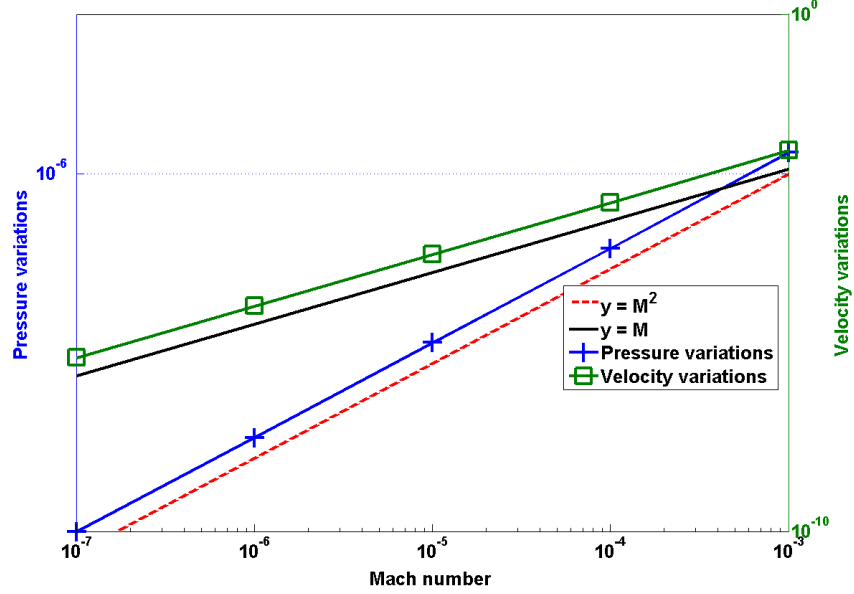


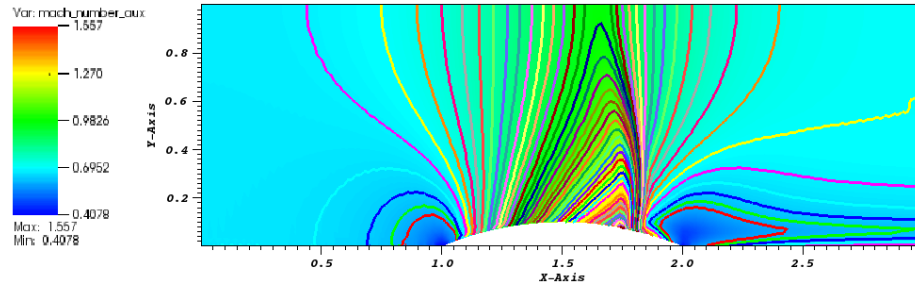
Figure 7: Log-log plot of the spatial steady-state pressure and velocity variations as a function of the far-field Mach number.

#### 6.7. Subsonic flow over a 2-D hump

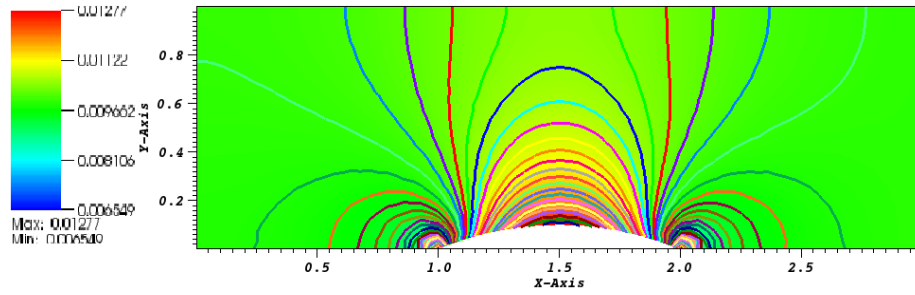
This is a another example of an internal flow configuration. It consists of a channel of height  $L = 1\text{ m}$  and length  $3L$ , with a circular bump of length  $L$  and thickness  $0.1L$ . The bump is located on the bottom wall at a distance  $L$  from the inlet. The system is initialized with an uniform pressure  $P = 101,325\text{ Pa}$  and temperature  $T = 300\text{ K}$ . The initial velocity is computed from the inlet Mach number, the pressure, the temperature and the ideal gas equation (with  $\gamma = 1.4$ ). Here,  $C_v = 717\text{ J/kg} - \text{K}$ . At the inlet, a subsonic stagnation boundary condition is used and the stagnation pressure and temperature are computed using Eq. (41). The static pressure  $P_s = 101,325\text{ Pa}$  is set at the subsonic outlet. The results are shown in Fig. 8a, Fig. 8b, Fig. 8c and Fig. 8d for the inlet Mach numbers  $M_\infty = 0.7$ ,  $M_\infty = 0.01$ ,  $M_\infty = 10^{-4}$  and  $M_\infty = 10^{-7}$ , respectively. It is expected that, for low Mach numbers, the solution does not

659 depend on the Mach number and is identical to the incompressible flow solution.  
 660 On the other hand, for a flow with  $M = 0.7$ , the compressible effects become  
 661 non negligible and a shock can form. An uniform grid of 3352  $Q_1$  elements was  
 662 used to obtain the numerical solution for Mach numbers less than and equal  
 663 to  $M_\infty = 0.01$ . A spatial mesh, once refined, was employed for the  $M_\infty = 0.7$   
 664 simulation in order to better resolve the shock. A *CFL* of 20 was employed and  
 665 the simulations were run until steady state.

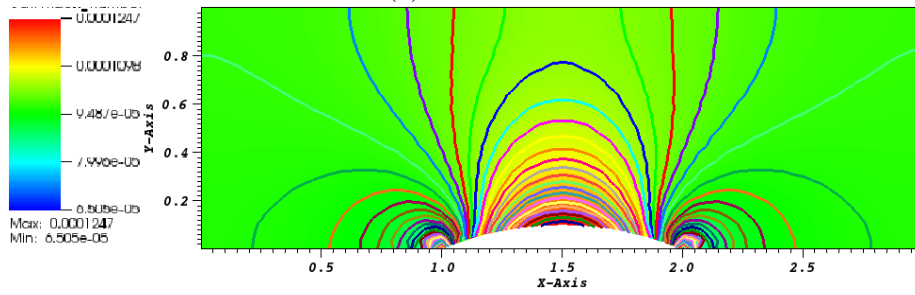




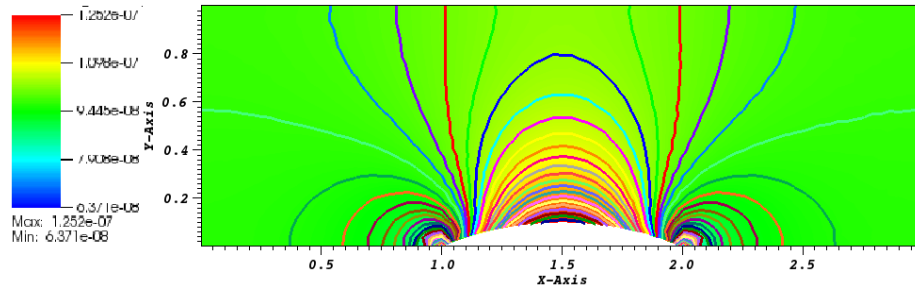
(a) Mach 0.7



(b) Mach  $10^{-2}$



(c) Mach  $10^{-5}$



(d) Mach  $10^{-7}$

Figure 8: Iso-Mach lines for a 2-D flow over a circular bump (steady-state solution).

666 The results shown in Fig. 8b, Fig. 8c and Fig. 8d correspond to the low-  
667 Mach regime. The iso-Mach lines are drawn ranging from the minimum and the  
668 maximum values (provided in each legend) using 50 equally-spaced intervals.  
669 The steady-state solution is symmetric and does not depend on the value of the  
670 inlet Mach number, as expected in the incompressible limit.

671 In Fig. 8a, the steady-state numerical solution develops a shock: the com-  
672 pressibility effects are no longer small. The iso-Mach lines are also plotted with  
673 50 intervals and range from 0.4 to 1.6. The shock is well resolved and does not  
674 display any instabilities or spurious oscillations.

#### 675 *6.8. Supersonic flow in a compression corner*

676 In this last example, we consider a supersonic flow at Mach 2.5 impinging  
677 on a corner with an angle of  $15^\circ$ . From the oblique shock theory [16], an  
678 analytical solution for this supersonic flow is available and gives the downstream-  
679 to-upstream pressure, entropy and Mach number ratios. The initial conditions  
680 are chosen to be spatially uniform: the pressure and temperature are set to  $P =$   
681  $101,325 \text{ Pa}$  and  $T = 300 \text{ K}$ , respectively. The ideal gas equation of state is used  
682 with the same parameters as in Section 6.7. The initial velocity is computed from  
683 the upstream Mach number. The inlet is supersonic and therefore, the pressure,  
684 temperature and velocity are specified using Dirichlet boundary conditions. The  
685 outlet is also supersonic and none of the characteristics enter the domain through  
686 this boundary; the values are computed by the solver.

687 The simulation is run with  $CFL = 2$  until steady state is reached. A 2-D  
688 mesh made of 16,109  $Q_1$  elements is used. The ratios for pressure, entropy and  
689 Mach number computed using the analytical (published with only two significant  
690 digits) and the numerical solutions are given in Table 10; they are in excellent  
691 agreement. The shock wave angle at steady state is also known and given by

the so-called  $\theta - \beta - M$  relation:

$$\tan \theta = 2 \cot \beta \frac{M^2 \sin^2 \beta - 1}{M^2 (\gamma + \cos^2(2\beta)) + 2}, \quad (42)$$

where  $\theta$ ,  $\beta$  and  $M$  denote the corner angle, the shock wave angle, and the upstream Mach number, respectively. For Mach 2.5 and a  $15^\circ$  corner angle, the analytical value for the shock wave angle is  $36.94^\circ$  at steady state. From Fig. 9a, the numerical value of the shock wave angle can be measured and is found to be equal to  $36.9^\circ$  and thus is in excellent agreement with the theory.

	analytical	numerical
Pressure	2.47	2.467
Mach number	0.74	0.741
Entropy	1.03	1.026

Table 10: Ratio of analytical and numerical downstream to upstream quantities for the compression corner problems (corner angle of  $15^\circ$  and inlet  $M = 2.5$  (analytical values from [16])).

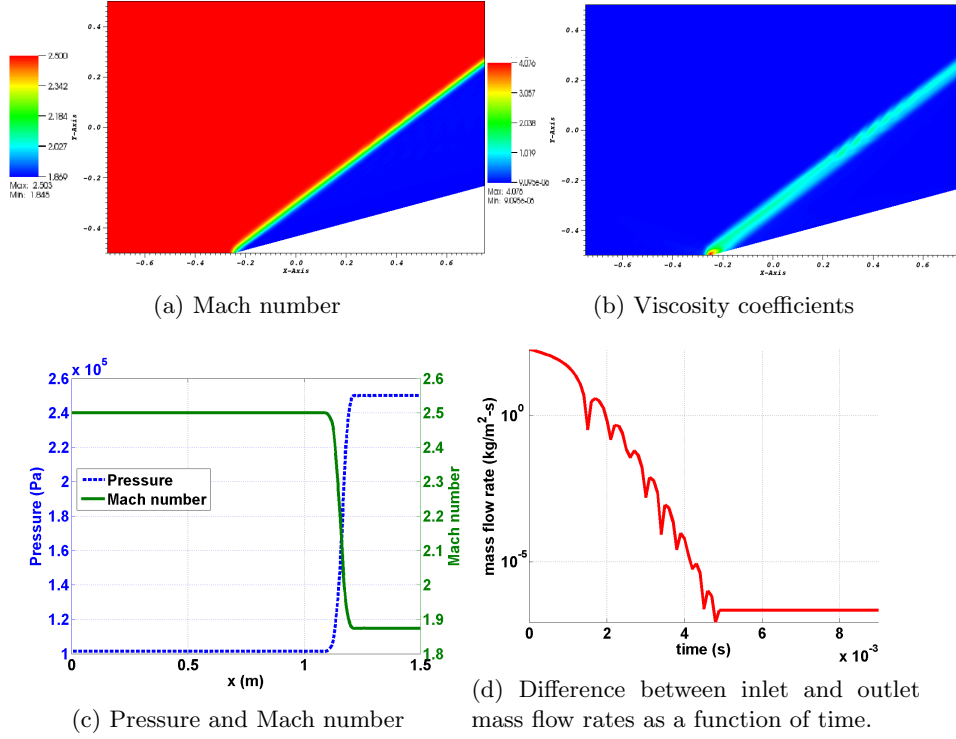


Figure 9: Steady-state solution for a flow in a 2-D compression corner.

698 The steady-state numerical solution is given in Fig. 9; the Mach number and  
 699 the viscosity coefficients are plotted in Fig. 9a and Fig. 9b, respectively. The  
 700 steady-state solution is composed of two regions of constant state separated by  
 701 an oblique shock. Fig. 9b shows that the viscosity coefficient is large in the shock  
 702 and small elsewhere, as expected. At the location of the corner ( $x = -0.25m$ ,  
 703  $y = -0.5m$ ), the viscosity coefficient is peaked because of the treatment of the  
 704 wall boundary condition: at this particular node, the normal is not well defined  
 705 and may cause some numerical errors. The 1-D graphs at  $y = 0$  for the pressure  
 706 and the Mach number are given in Fig. 9c: no spurious oscillations are observed  
 707 and the shock is well resolved. Finally, the difference between the inlet and  
 708 outlet mass flow rates is plotted in Fig. 9d and shows that a steady state has  
 709 indeed been reached.

710 The results presented in this paper demonstrate the ability of the entropy  
711 viscosity method with the new definitions of the viscosity coefficients to correctly  
712 simulate several types of flows (from very low Mach subsonic to transonic flows)  
713 without tuning parameters.

## 714 7. Conclusions

715 A new version of the entropy viscosity method that is valid for a wide range  
716 of Mach numbers has been derived and presented for the inviscid Euler equa-  
717 tions. The definition of the viscosity coefficients is now consistent with the  
718 low-Mach asymptotic limit, does not require an analytical expression for the  
719 entropy function, and is therefore applicable to a larger variety of flow regimes,  
720 from very low-Mach flows to supersonic flows. The method has also been ex-  
721 tended to Euler equation with variable area to solve nozzle flow problems. In  
722 1-D, convergence of the numerical solution to the exact solution was demon-  
723 strated by computing the convergence rates of the L1 and L2 norms for flows  
724 in a converging-diverging nozzle and in straight pipes. For smooth solutions,  
725 second-order convergence was verified; solutions with shocks converged with the  
726 expected theoretical rates of 1 ( $L_1$ -norm) and 0.5 ( $L_2$ -norm).

727 The effectiveness of the method was also demonstrated in 2-D using a se-  
728 ries of benchmark problems for both subsonic and supersonic flows in various  
729 geometries, with Mach numbers ranging from  $10^{-7}$  to 2.5. For very low-Mach  
730 flows, we numerically verified that the spatial steady-state pressure variations  
731 were proportional to the square of the Mach number, as expected in the incom-  
732 pressible limit.

733 In the future, we plan to further extend the entropy viscosity method to the  
734 seven-equation two-phase flow fluid model [20]. This two-phase flow system of  
735 equations is a good candidate for two reasons: it is unconditionally hyperbolic

736 and degenerates to the standard Euler equations when one phase disappears.

## 737 **Acknowledgments**

738 The authors (M.D. and J.R.) would like to thank Bojan Popov and Jean-Luc  
739 Guermond for many fruitful discussions. The research was carried out under  
740 the auspices the Idaho National Laboratory for the US Department of Energy.

## 741 **References**

- 742 [1] E. F. Toro, Riemann Solvers and numerical methods for fluid dynamics,  
743 2<sup>nd</sup> Edition, Springer, 1999.
- 744 [2] B. Cockburn, C. Johnson, C. Shu, E. Tadmor, Advanced numerical approx-  
745 imation of nonlinear hyperbolic equations, Lecture Notes in Mathematics  
746 1697.
- 747 [3] B. Cockburn, G. Karniadakis, C. Shu, Discontinuous galerkin methods:  
748 theory, computation and applications, Lecture Notes in Computer Science  
749 and Engineering 11.
- 750 [4] R. Lohner, Applied CFD Techniques: an Introduction based on Finite  
751 Element Methods, 2<sup>nd</sup> Edition Wiley, 2003.
- 752 [5] A. Lapidus, A detached shock calculation by second order finite differences,  
753 J. Comput. Phys. 2 (1967) 154–177.
- 754 [6] R. Lohner, K. Morgan, J. Peraire, A simple extension to multidimensional  
755 problems of the artificial viscosity due to lapidus, Commun. Numer. Meth-  
756 ods Eng. 1(14) (1985) 141–147.
- 757 [7] J. Donea, A. Huerta, Finite Element Methods for Flow Problems, Oxford  
758 University Press, 2003.

- 759 [8] J. L. Guermond, R. Pasquetti, Entropy viscosity method for nonlinear con-  
760 servation laws, *Journal of Comput. Phys* 230 (2011) 4248–4267.
- 761 [9] J. L. Guermond, R. Pasquetti, Entropy viscosity method for high-order ap-  
762 proximations of conservation laws, *Lecture Notes in Computational Science*  
763 *and Engineering* 76 (2011) 411–418.
- 764 [10] H. Guillard, C. Viozat, On the behavior of upwind schemes in the low mach  
765 number limit, *Computers & Fluids* 28 (1999) 63–86.
- 766 [11] E. Turkel, Preconditioned techniques in computational fluid dynamics,  
767 *Annu. Rev. Fluid Mech.* 31 (1999) 385–416.
- 768 [12] J. S. W. D. L, Darmofal, J. Peraire, The solution of the compressible euler  
769 equations at low mach numbers using a stabilized finite element algorithm,  
770 *Comput. Methods Appl. Mech. Engrg.* 190 (2001) 5719–5737.
- 771 [13] X.-S. Li, C.-W. Gu, An all-speed roe-type scheme and its asymptotic anal-  
772 ysis of low mach number behavior, *Journal of Computational Physics* 227  
773 (2008) 5144–5159.
- 774 [14] D. Gaston, C. Newsman, G. Hansen, D. Lebrun-Grandie, A parallel compu-  
775 tational framework for coupled systems of nonlinear equations, *Nucl. Eng.*  
776 *Design* 239 (2009) 1768–1778.
- 777 [15] J. L. Guermond, B. Popov, Viscous regularization of the euler equations  
778 and entropy principles, under review.
- 779 [16] J. D. Anderson, Modern compressible flow, in: *Guide for Verification and*  
780 *Validation in Computational Solid Mechanic.*, New York, 1982, pp. 10–  
781 2006.

- 782 [17] J. L. Guermond, R. Pasquetti, Entropy-based nonlinear viscosity for four-  
783 rier approximations of conservation laws, in: C.R. Math. Acad. Sci., Vol.  
784 326, Paris, 2008, pp. 801–806.
- 785 [18] V. Zingan, J. L. Guermond, J. Morel, B. Popov, Implementation of the  
786 entropy viscosity method with the discontinuous galerkin method, Journal  
787 of Comput. Phys 253 (2013) 479–490.
- 788 [19] B. Muller, Low-mach number asymptotes of the navier-stokes equations,  
789 Journal of Engineering Mathematics 34 (1998) 97–109.
- 790 [20] R. Berry, R. Saurel, O. LeMetayer, The discrete equation method (dem)  
791 for fully compressible, two-phase flows in ducts of spatially varying cross-  
792 section, Nuclear Engineering and Design 240 (2010) 3797–3818.
- 793 [21] R. Loubere, Validation test case suite for compressible hydrodynamics com-  
794 putation, Theoretical Division T-7 Los Alamos National Laboratory.
- 795 [22] J. J. Quirk, A contribution to the great riemann solver debate, NASA  
796 Contractor Report 191409 ICASE Report No. (1992) 92–64.
- 797 [23] R. Abgrall, R. Saurel, Discrete equations for physical and numerical  
798 compressible multiphase mixtures, Journal of Computational Physics 186  
799 (2003) 361–396.
- 800 [24] D. L. Darmofal, K. Siu, A robust multigrid algorithm for the euler equations  
801 with local preconditioning and semi-coarsening, Journal of Computational  
802 Physics 151 (1999) 728–756.
- 803 [25] P. Perrot, A to Z of Thermodynamics, Oxford University Press, 1998.
- 804 [26] O. LeMetayer, J. Massoni, R. Saurel, Elaborating equation of state for a  
805 liquid and its vapor for two-phase flow models, International Journal of  
806 Thermal Science 43 (2004) 265–276.



- 807 [27] S. LeMartelot, B. Nkonga, R. Saurel, Liquid and liquid-gas flows at all  
808 speeds., *Journal of Computational Physics* 255 (2013) 53–82.
- 809 [28] R. A. DeVore, G. G. Lorentz, *Constructive Approximation*, Springer-  
810 Verlag, 1991.
- 811 [29] T. W. Roberts, The behavior of flux difference splitting schemes near slowly  
812 moving shock waves, *Journal of Computational Physics* 90 (1990) 141–160.
- 813 [30] L. R. J., *Numerical methods for conservation laws*, Basel: Birkhauser, Read-  
814 ing, Massachusetts, 1990.

815 **A. Derivation of the entropy residual as a function of density, pres-**  
816 **sure, and speed of sound**

The entropy residual is defined as follows:

$$R_{\text{ent}}(\vec{r}, t) = \partial_t s(\vec{r}, t) + \vec{u} \cdot \vec{\nabla} s(\vec{r}, t),$$

where all variables were defined previously. This form of the entropy residual is not suitable for the low-Mach limit as explained in Section 2.1. In this appendix, we recast the entropy residual  $R_{\text{ent}}(\vec{r}, t)$  as a function of the primitive variables (pressure, velocity and density) and the speed of sound. The first step of this derivation is to use the chain rule, recalling that the entropy is a function of the internal energy  $e$  and the density  $\rho$ , yielding

$$R_{\text{ent}}(\vec{r}, t) = s_e \frac{De}{Dt} + s_\rho \frac{D\rho}{Dt},$$

817 where  $s_e$  denotes the partial derivative of  $s$  with respect to the variable  $e$ . We  
818 recall that  $\frac{D}{Dt}$  denotes the material derivative. Since the internal energy  $e$  is a  
819 function of pressure  $P$  and density  $\rho$  (through the equation of state), we use  
820 again the chain rule to re-express the previous equation as a function of the  
821 material derivatives in  $P$  and  $\rho$ :

$$\begin{aligned} R_{\text{ent}}(\vec{r}, t) &= s_e e_P \frac{DP}{Dt} + (s_e e_\rho + s_\rho) \frac{D\rho}{Dt} \\ &= s_e e_P \left( \frac{DP}{Dt} + \frac{1}{s_e e_P} (s_e e_\rho + s_\rho) \frac{D\rho}{Dt} \right) \\ &= s_e e_P \left( \frac{DP}{Dt} + \left( \frac{e_\rho}{e_P} + \frac{s_\rho}{s_e e_P} \right) \frac{D\rho}{Dt} \right). \end{aligned}$$

To prove that the term multiplying the material derivative of the density is indeed equal to the square of the speed of sound, we recall that the speed of sound is defined as the partial derivative of pressure with respect to density at

constant entropy, which can be recast as a function of the entropy as follows (see Appendix A.2 of [15]):

$$c^2 := \left. \frac{\partial P}{\partial \rho} \right|_{s=cst} = P_\rho - \frac{s_\rho}{s_e} P_e .$$

Using the following relations (see Appendix A.1 of [15])

$$P_e = \frac{1}{e_P} \text{ and } P_\rho = -\frac{e_\rho}{e_P} .$$

Substitution of these expressions into the entropy residual equation above gives Eq. (9), which is recalled below for completeness:

$$R_{\text{ent}}(\vec{r}, t) := \partial_t s + \vec{u} \cdot \vec{\nabla} s = \frac{Ds}{Dt} = \frac{s_e}{P_e} \left( \underbrace{\frac{DP}{Dt} - c^2 \frac{D\rho}{Dt}}_{\tilde{R}_{\text{ent}}(\vec{r}, t)} \right) .$$

822 **B. Derivation of the dissipative terms for the Euler equations with**  
823 **variable area using the entropy minimum principle**

824 The Euler equations (without viscous regularization) with variable area are  
825 recalled here

$$\partial_t (\rho A) + \vec{\nabla} \cdot (\rho \vec{u} A) = 0 \quad (43a)$$

$$\partial_t (\rho \vec{u} A) + \vec{\nabla} \cdot [A (\rho \vec{u} \otimes \vec{u} + P \mathbb{I})] = P \vec{\nabla} A \quad (43b)$$

$$\partial_t (\rho E A) + \vec{\nabla} \cdot [\vec{u} A (\rho E + P)] = 0. \quad (43c)$$

828 The specific entropy is a function of the density  $\rho$  and the internal energy  $e$ , i.e.,  
829  $s(e, \rho)$ . The above system of equations satisfies the minimum entropy principle  
830 [30],

$$A \rho \left( \partial_t s + \vec{u} \cdot \vec{\nabla} s \right) \geq 0. \quad (44)$$

831 The entropy function  $s$  satisfies the second law of thermodynamics,  $T ds =$   
832  $de - \frac{P}{\rho^2} d\rho$ , which implies  $s_e := T^{-1}$  and  $s_\rho := -PT^{-1}\rho^{-2}$ . One can show that  
833 [15]

$$s_e = T^{-1} \geq 0 \text{ and } P s_e + \rho^2 s_\rho = 0. \quad (45)$$

834 In order to apply the entropy viscosity method to the variable-area Euler equa-  
835 tions, dissipative terms need to be added to each equation in Eq. (43). The  
836 functional forms of these terms need to be such that the entropy residual de-  
837 rived with these terms present also satisfies the minimum entropy principle. To  
838 prove the minimum entropy principle, the extra terms appearing in the entropy  
839 residual are either recast as conservative terms or shown to be positive. The  
840 rest of this appendix presents this demonstration. Following [15], we first write  
841 the variable-area equations with dissipative terms:

$$\partial_t (\rho A) + \vec{\nabla} \cdot (\rho \vec{u} A) = \vec{\nabla} \cdot f \quad (46a)$$

$$\partial_t (\rho \vec{u} A) + \vec{\nabla} \cdot [A (\rho \vec{u} \otimes \vec{u} + P \mathbb{I})] = P \vec{\nabla} A + \vec{\nabla} \cdot g \quad (46b)$$

842

$$\partial_t (\rho E A) + \vec{\nabla} \cdot [\vec{u} A (\rho E + P)] = \vec{\nabla} \cdot (h + \vec{u} \cdot g). \quad (46c)$$

843 where  $f$ ,  $g$  and  $h$  are dissipative fluxes to be determined. Starting from the  
 844 modified system of equations given in Eq. (46), the entropy residual is derived  
 845 again. The derivation requires the following steps : express the governing laws  
 846 in terms of primitive variables  $(\rho, \vec{u}, e)$ , multiply the continuity equation by  $\rho s_\rho$   
 847 and the internal energy equation by  $s_e$ , and invoke multivariate chain rule, e.g.,  
 848  $\partial s / \partial x = s_e \partial e / \partial x + s_\rho \partial \rho / \partial x$ . These steps are similar to those used for the  
 849 standard Euler equations [15]. Some of the lengthy algebra is omitted here.  
 850 The above steps yield:

$$A \rho \left( \partial_t s + \vec{u} \cdot \vec{\nabla} s \right) = s_e \left[ \vec{\nabla} \cdot h + g : \vec{\nabla} u + \left( \frac{u^2}{2} - e \right) \vec{\nabla} \cdot f \right] + \rho s_\rho \vec{\nabla} \cdot f. \quad (47)$$

851 The next step consists of choosing a definition for each of the dissipative terms  
 852 so that the left hand-side is positive. The right hand-side of Eq. (47) can be  
 853 simplified using the relations  $g = A \mu \vec{\nabla}^s \vec{u} + f \otimes \vec{u}$  and  $h = \tilde{h} - 0.5 \|\vec{u}\|^2 f$  to give

$$A \rho \left( \partial_t s + \vec{u} \cdot \vec{\nabla} s \right) = s_e \left[ \vec{\nabla} \cdot \tilde{h} - e \vec{\nabla} \cdot f \right] + \rho s_\rho \vec{\nabla} \cdot f + A s_e \mu \vec{\nabla}^s \vec{u} : \vec{\nabla} \vec{u}. \quad (48)$$

The right hand-side is now integrated by parts:

$$\begin{aligned} A \rho \left( \partial_t s + \vec{u} \cdot \vec{\nabla} s \right) &= \vec{\nabla} \cdot \left[ s_e \tilde{h} - s_e e f + \rho s_\rho f \right] \\ &\quad - \vec{\nabla} \cdot \tilde{h} \vec{\nabla} s_e + f \cdot \vec{\nabla} (e s_e) - f \cdot \vec{\nabla} (\rho s_\rho) + A s_e \mu \vec{\nabla}^s \vec{u} : \vec{\nabla} \vec{u} \end{aligned} \quad (49)$$

854 where  $\vec{\nabla}^s$  is the symmetric gradient. The term  $A s_e \mu \vec{\nabla}^s \vec{u} : \vec{\nabla} \vec{u}$  is positive and  
 855 thus, does not need any further modification. It remains to treat the other

856 terms of the right hand-side that we now call  $rhs$ :

$$rhs = \vec{\nabla} \cdot [s_e \tilde{h} - s_e e f + \rho s_\rho f] - \tilde{h} \cdot \vec{\nabla} s_e + f \cdot \vec{\nabla} (e s_e) - f \cdot \vec{\nabla} (\rho s_\rho).$$

857 The first term in  $rhs$  is a conservative term. By carefully choosing a definition  
 858 for  $\tilde{h}$  and  $f$ , the conservative term can be expressed as a function of the entropy  
 859  $s$ . The inclusion of the variable area in the choice of the dissipative terms is also  
 860 required so that, when assuming constant area, the standard Euler equations  
 861 are recovered. The following definitions for  $\tilde{h}$  and  $f$  are chosen:

$$\tilde{h} = A\kappa \vec{\nabla}(\rho e) \text{ and } f = A\kappa \vec{\nabla} \rho,$$

862 which yields, using the chain rule,

$$rhs = \vec{\nabla} \cdot (\rho A \kappa \vec{\nabla} s) - A \kappa \underbrace{\left[ \vec{\nabla}(\rho e) \vec{\nabla} s_e - \vec{\nabla} \rho \vec{\nabla} (e s_e) + \vec{\nabla} \rho \vec{\nabla} (\rho s_\rho) \right]}_{\mathbf{Q}}$$

863 It remains to treat the term  $\mathbf{Q}$  that can be recast under a quadratic form.

864 Following [15], one obtain:

$$\begin{aligned} \mathbf{Q} &= \rho X^t \Sigma X \\ \text{with } X &= \begin{bmatrix} \vec{\nabla} \rho \\ \vec{\nabla} e \end{bmatrix} \text{ and } \Sigma = \begin{bmatrix} \rho^{-2} \partial_\rho (\rho^2 \partial_\rho s) & \partial_{\rho, es} \\ \partial_{\rho, es} & \partial_{e, es} \end{bmatrix} \end{aligned}$$

865 The matrix  $\Sigma$  is symmetric and identical to the matrix obtained in [15]. The sign  
 866 of the quadratic form can be simply determined by studying the positiveness of  
 867 the matrix  $\Sigma$ . In this particular case, it is required to prove that the matrix is  
 868 negative definite: the quadratic form is on the right hand-side and is preceded by  
 869 a negative sign. According to [15], the convexity of the opposite of the entropy

870 function, i.e.,  $-s$ , with respect to the internal energy  $e$  and the specific volume  
871  $1/\rho$  is sufficient to ensure that the matrix  $\Sigma$  is negative definite.  
872 Thus, the right hand-side of the entropy residual Eq. (47) is now either recast  
873 as conservative terms, or known to be positive. Thus, the entropy minimum  
874 principle holds.

### 875 **C. Entropy residual for isentropic flows**

876 This appendix shows that the entropy residual is zero for isentropic flows.  
 877 For convenience, we recall here the entropy residual as a function of the pressure,  
 878 density, velocity, and speed of sound:

$$\tilde{R}_{\text{ent}} = \frac{DP}{Dt} - c^2 \frac{D\rho}{Dt}. \quad (50)$$

879 Assuming an isentropic flow, pressure is only a function of density, i.e.,  $P =$   
 880  $f(\rho)$  or equivalently  $\rho = f^{-1}(P)$ . Using the definition of the speed of sound  
 881  $c^2 = \left. \frac{\partial P}{\partial \rho} \right)_s$  and the above form of the equation of state, we have

$$c^2 = \left. \frac{\partial P}{\partial \rho} \right)_s = \frac{dP}{d\rho} = \frac{df(\rho)}{d\rho}. \quad (51)$$

882 Using the chain rule, the entropy residual in Eq. (50) can be recast as follows  
 883 and proven equal to zero:

$$\tilde{R}_{\text{ent}} = \frac{df(\rho)}{d\rho} \frac{D\rho}{Dt} - c^2 \frac{D\rho}{Dt} = c^2 \frac{D\rho}{Dt} - c^2 \frac{D\rho}{Dt} = 0. \quad (52)$$