Study of thermal boundary layer in pulsatile flows

By D. Kah, T. Ewan AND M. Ihme

1. Motivation and objectives

Fully integrated reactive simulations in internal combustion (IC) engines have become a critical target for the automotive industry, where CFD has an increasing impact in the decision process for the design of new prototypes. Although a substantial level of maturity has been reached for simulations, there are some fields where the potential of CFD can still be leveraged to improve engine energetic and environmental efficiencies. One of these fields concerns thermal losses due to heat transfer at the wall between a hot combustion chamber and the cool surroundings. Given thermodynamic conditions, it is extremely difficult, to monitor the heat loss experimentally. CFD represents a possible way to get a reliable prediction of heat loss.

Besides, at the same bulk and surrounding conditions, the experimental study conducted in Kearney et al. (2001) concluded that the heat loss magnitude could vary by a factor of two from steady to pulsatile bulk flows. A pulsatile flow is characterized by steep variations of the intensity of its velocity, strong enough to possibly inverse its sign. This kind of flow is typically found in cylinders of internal-combustion engines, where successive sequences of compressions and expansions occur. The identification of key parameters that control the heat loss intensity can substantially improve the engine efficiency.

However, although thermal boundary layers are well understood in steady state flows, their characterization in transient flows is only at its early phase Costamagna et al. (2003) have identified some quasi-coherent structures in these types of flows. The objective of this study is to set up an efficient numerical framework for the study of the thermal boundary layer in a model cylinder of an internal combustion engine.

The remainder of this report is organized as follows. A compressible formulation is first presented, as well as the ALE formalism used to treat the moving geometry due to the compression/expansion phase. Since the Mach number is typically small in these flows, Section 3 presents a first attempt to build a low-Mach solver accounting for variations in density due to temperature variation. Section 4 presents verification results for dilatational flows. Finally, a summary of the accomplishments and associated perspectives is given in the concluding section.

2. Compressible modeling framework

2.1. Modeling framework and hypotheses

The flow field is described by the solution of the Navier-Stokes equations:

\[
\begin{align*}
\partial_t \rho + \partial_j (\rho u_j) &= 0, \\
\partial_t (\rho u_i) + \partial_j (\rho u_j u_i) &= -\partial_x P + \partial_x \sigma_{ij}, \\
\partial_t (\rho e) + \partial_j (\rho u_j e) + P \partial_x (u_j) &= \sigma_{ij} \partial_x (u_i) + \partial_x (\lambda \partial_x T) - \partial_x (q) - \dot{\mathcal{Q}}_{\text{ih}}.
\end{align*}
\]
where $P$ denotes the pressure, $e$ denotes the gas internal energy, $\lambda$ is the thermal conductivity, and $T$ the temperature. The quantity $q$ represents the multispecies enthalpy diffusion flux, and the term $q_{\Delta h}$ is a volumetric heat source term. Since a homogeneous mixture and no heat source term are considered, the terms $q$ and $q_{\Delta h}$ are not considered in the present study. The viscous stress tensor $\tau_{ij}$ and the strain rate tensor $S_{ij}$ are written respectively as:

$$
\tau_{ij} = 2\mu \left[ S_{ij} - \frac{1}{3} \delta_{ij} \partial_x (u_k) \right], \quad S_{ij} = \frac{1}{2} \left[ \partial_{x_j} (u_i) + \partial_{x_i} (u_j) \right].
$$

(2.2)

The relationship between the thermodynamic quantities is prescribed by the ideal gas law,

$$
\rho = \frac{P}{rT},
$$

(2.3)

where $r = R/W$, where $R$ denotes the perfect gas constant, and $W$ the mixture molecular mass.

System (2.1) describes the gas dynamics in a fixed domain, i.e., experiencing no changes. However, in our case of interest, the domain is compressed and expanded. Therefore, the model has to be formulated in this moving reference frame. The ALE formulation consists in writing the conservation equations Eq. (2.1) in a different reference frame than the usually fixed one referred to by the spatial variable $x$: therefore, in the present study, the grid reference frame is introduced and is referred to by the spatial variable $\chi$.

Deriving the corresponding equations starts by applying the Reynolds transport theorem to an arbitrary volume $V = V(\phi(\chi, t))$ whose boundary $S = \partial V$ moves with the mesh velocity $v_{\chi}$:

$$
\partial_t |_{\chi} \int_V f(x, t) \, d^3x = \int_V \partial_t f(x, t) \, d^3x + \int_S f(x, t) v_{\chi} \cdot n \, d^2x.
$$

(2.4)

Combining Eq. (2.4) with the Reynolds transport theorem applied to a material volume $V$ leads to:

$$
\partial_t |_{\chi} \int_V f(x, t) \, d^3x + \int_S f(x, t) (u - v_{\chi}) \cdot n \, d^2x = \frac{d}{dt} \int_V f(x, t) \, dV,
$$

(2.5)

and furthermore, applying dynamic relations to the right-hand side of Eq. (2.5):

$$
\partial_t |_{\chi} \int_{V_\chi(\chi, 0)} f(x, t) \, d^3x + \int_S f(x, t) (u - v_{\chi}) \cdot n \, d^2x = \int_V S_f(x, t) \, d^3\chi + \int_S \sigma d^2x,
$$

(2.6)

where $S_f$ and $\sigma$ are, respectively, the volumetric and surface source term for $f$. Finally, applying the Gauss theorem to the surface integral, and the change of variable $x = \Phi(\chi, t)$ at a given time, leads to:

$$
\int_{V_\chi(\chi, 0)} \partial_t J f(x, t)|_{\chi} + J \nabla_{\chi} \cdot (f(x, t)(u - v_{\chi})) \, d^3\chi = \int_{V_\chi(\chi, 0)} J \nabla_{\chi} \cdot \sigma + JS_f \, d^3\chi.
$$

(2.7)

The change of variables leads to the change of the differential volume $d^3x = det(\nabla_{\chi} x) d^3\chi$, where the quantity $det(\nabla_{\chi} x) = J$, which is the dilatation rate, verifies the relation:
\[ \partial_t J = J \nabla_x \cdot v_X. \]

Rewriting system (2.1) in the grid reference frame leads to:

\[
\begin{align*}
\partial_t J \rho + J \partial_{x_j} (\rho w_j) &= 0, \\
\partial_t (J \rho u_i) + J \partial_{x_j} (\rho w_j u_i) &= -J \partial_{x_j} P + J \partial_{x_j} \tau_{ij}, \\
\partial_t (J \rho e) + J \partial_{x_j} (\rho w_j e) + J P \partial_{x_j} (u_j) &= \tau_{ij} \partial_{x_j} (u_i) + J \partial_{x_j} (\lambda J \partial_{x_j} T),
\end{align*}
\]

(2.8)

where the velocity \( w_j \) is defined as \( u_j - v_{X,j} \).

### 2.2. Numerical framework

The base code used in this work, can be used in either compressible or low-Mach formulation. It relies on a structured grid approach. In order to ensure boundedness of the scalars, the spatial solver is based on a high-order accurate finite difference WENO 3 scheme. This property is important for reactive flows. A spatial staggered-variable formulation is employed in the code, meaning that discrete velocity values are located at cell faces, while discrete pressure, density, and scalar values are located at cell centers. Relative to a collocated formulation, staggering has the advantage that derivative stencils are more localized in space. This localization contributes to the accuracy of the stencil.

For example, pressure gradients are needed in the code at the cell faces where discrete velocity values exist. In a staggered formulation, a second-order pressure gradient can be created at a cell face using discrete pressure values that are only one cell apart. In a collocated formulation, a second-order gradient would require discrete pressure values that span three cells. In the compressible formulation, time integration is done with a second-order, 5-stage explicit Runge-Kutta scheme (Stanescu & Habashi 1998).

### 2.3. Numerical configuration

#### 2.3.1. Geometry

The geometry considered is a simple cylinder section with a moving wall that has the role of a piston in an IC engine. The stroke is 9.65 cm long and the radius is 4.3 cm. The piston speed corresponds to a regime of 3000 rpm, and the compression ratio (CR) is 9. These features correspond to the engine of Alharbi & Sick (2010) that provides experimental results for an eventual validation of the simulation. Figure 1 represents the configuration geometry. In the longitudinal direction, one face is the moving wall, and the other is the cylinder head. Non-slip conditions are applied to both of them. Neumann conditions are applied for the scalars. Periodic conditions are applied in the azimuthal direction, Dirichlet conditions for the temperature \( T = 400 \text{ K} \), and non-slip conditions are applied on the cylinder wall. Finally, a symmetry condition is applied at the centerline.

The cylinder is initialized with a homogeneous mixture of air with a temperature of 400 K, and a density of 1 kg m\(^{-3}\).

#### 2.3.2. Results

To obtain a first idea of the flow, and the boundary layer, a first mesh is considered that allows results to be obtained in a short amount of time while being refined enough to see small-scale variations. The mesh consists of 128 cells in the longitudinal direction \( x \), 256 cells in the radial direction denoted by \( y \) here, and 8 cells in the azimuthal direction denoted by \( z \) here, for an angular section of 45 degrees. In order to capture the structure
of the wall-boundary layers, the mesh is refined close to the wall, with a length reaching 30 \( \mu \)m in the y-direction. At these conditions, the Reynolds number based on the mean velocity is about 25,000, and the Mach number based on the highest piston velocity is lower than 0.03. A sinusoidal profile is imposed for the piston velocity, with its maximum velocity set by the compression ratio and the engine speed. The engine speed is such that a cycle takes 20 ms. A cycle is defined as a sequence of compression and expansion phases.

Computations are run at \( CFL = 1 \) (the CFL condition includes convection, viscosity, and diffusion). The first interesting question is to evaluate if convergence has been reached in the sense of similarity in the flow structure at the same cycle instant for one cycle to another. Some insight is given by Figures 2 and 3 showing the temperature profile along the radial direction for different cycles (cycle 2, 4, 6, 8). Figure 2 corresponds to the location \( x = 1 \) cm, and Figure 3 to the location \( x = 8 \) cm. The solution is axisymmetric, so the profiles are given at one given angle.

The chosen instant is 2 ms, which is in the beginning of the compression phase. Whereas before six cycles the solution is highly fluctuating, starting cycle 8, some similarity is seen with previous cycles. But some development in the radial flow is present, so that the results cannot be considered to have converged yet. Therefore, a higher number of cycles has to be run to reach a converge state. Nevertheless, these results suggest that this number will be around ten.

Moreover, the decaying profile of the mean radial temperature comes from the fact that, given the initial conditions, the cylinder loses energy from one cycle to another due to heat loss at the wall. Since the isothermal wall is at constant temperature, a steady state may occur where energy gain and loss through the wall cancel out during a cycle. Whether and when this steady state is reached may be interesting information provided by the simulation.

Finally let us notice the similar trend of the boundary layers at the two x-locations. While temperature decays in both cases, this decaying is more pronounced at \( x = 8 \) cm. At the same time, the bulk temperature is higher at \( x = 8 \) cm. In this case, about 30 given points are included in the boundary layer. Refinement is conducted to assess the characteristic wall distance for this case. Temperature variations along the radius are more intense when going further from the piston. Analyzing the underlying mechanism is the objective of further studies.
Two-dimensional flow results are provided in order to assess the onset of turbulence. Figures 4, 5 and 6 display temperature, longitudinal, radial velocities for instants 2 ms, and 8 ms. The x-scaling corresponds to the chamber size at the instant of display. At 2 ms, the compression phase has just started, so that the chamber is almost fully extended, whereas at 8 ms, the compression phase is almost completed so that the chamber is close to its minimum extent. The results displayed are obtained for cycle 8. While the thermal boundary layer is clearly visible at 2 ms, it is hardly noticeable at 8 ms. Moreover, the occurrence of a secondary flow of cooled air along the piston can be seen at 2 ms. This flow is characterized by a descending vortex located at $y = 2.7$ cm and visible at Figure 5(b) and Figure 6(b). Along with this vortex, the velocity is characterized by small fluctuations, specially at 8 ms. The study of the resulting vorticity field will give some quantitative information about these turbulent structures and be complementary to the study conducted by Costamagna et al. (2003).

Compressible Navier-Stokes equations solve for every waves of the problem, including the pressure waves. However, since the Mach number is low (0.03), pressure waves are not supposed to significantly modify the flow. This is actually seen on the results where the impact of waves traveling at the sound speed are not of first order. Therefore, a low-Mach formulation is relevant in this case, as it could release the acoustic constraints on the time step and potentially provide faster computations. Therefore, the rest of this report is devoted to the design of an accurate low-Mach formulation for this problem.
3. Low-Mach formulation: first attempt for a dedicated solver

3.1. Modeling framework for low-Mach solver

The significance of acoustic effects in a flow are in part a function of the local Mach number, and in part a function of other factors such as the time history of the background pressure. When acoustic effects do not strongly influence a flow, a useful simplification can be applied to the above governing equations. This simplification begins with a non-dimensionalization of the Navier-Stokes equations. We introduce the following
non-dimensional quantities:

$$\rho = \frac{\rho}{\rho_\infty}, \quad p = \frac{P}{P_\infty}, \quad u = \frac{u}{u_\infty}, \quad T = \frac{T}{T_\infty}, \quad \mu = \frac{\mu}{\mu_\infty}, \quad \lambda = \frac{\lambda}{\lambda_\infty}, \quad x = \frac{x}{x_\infty}, \quad t = \frac{t}{x_\infty/u_\infty}, \quad e = \frac{e}{P_\infty/\rho_\infty},$$

(3.1)

where quantities with the subscript $\infty$ denote characteristic values of the configuration.

The nondimensional Navier-Stokes equations are:

$$\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -Ma^{-2} \frac{\partial}{\partial x_i} (P),$$

$$\frac{\partial}{\partial t} (\rho e) + \frac{\partial}{\partial x_j} (\rho e u_j) = -P_0 \frac{\partial}{\partial x_j} u_j + \frac{\gamma}{(\gamma - 1) Pr_\infty Re_\infty} \frac{\partial}{\partial x_i} (\lambda \frac{\partial}{\partial x_i} T).$$

(3.2)

In the low-Mach limit, i.e., when $Ma \to 0$, the low-Mach Navier-Stokes system becomes:

$$\frac{\partial}{\partial t} (\rho) + \frac{\partial}{\partial x_j} (\rho u_j) = 0,$$

$$\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -Ma^{-2} \frac{\partial}{\partial x_i} (P),$$

$$\frac{\partial}{\partial t} (\rho e) + \frac{\partial}{\partial x_j} (\rho e u_j) = -P_0 \frac{\partial}{\partial x_j} u_j + \frac{\gamma}{(\gamma - 1) Pr_\infty Re_\infty} \frac{\partial}{\partial x_i} (\lambda \frac{\partial}{\partial x_i} T),$$

(3.3)

where the terms involving powers of $Ma$ have been neglected. The resulting set of equations has two differences from the initial one. First, viscous effects are negligible in the internal energy equation. Then, the gradient of the pressure $P_0$ becomes zero. This means that the time scale considered for this derivation is much larger than the time scale of the acoustic waves so that a pressure equilibrium can form. Following the derivation done by Muller (1998), another term appears in the momentum equation, $P_1$, which is the term of second order in $Ma$ of the expansion of the pressure term. Although the term $P_1$ is usually referred to as the dynamic pressure, it has no physical meaning per se. This term is computed from the continuity equation through a Poisson equation. Once computed, the velocity field is finally updated to satisfy the continuity equation. For a thorough derivation of the low-Mach equations, the reader is referred to Muller (1998).

The homogenous pressure $P_0$ verifies the equation of state (EOS) $P_0(t) = \rho(t, x)rT(t, x)$, where the density and the temperature may still depend on space.

At this point, the low-Mach formulation Eq. (3.3) and the EOS consists of four equations, but involves five unknowns: $\rho, u, e, P_0, P_1$. In the particular case of an incompressible flow, an unknown is removed. But in the general case, solving this system implies making assumptions that are not necessarily consistent with the physics of the problem. In what follows, the numerics are described and the procedure to implement system (3.3) in the existing numerical framework is outlined.

### 3.2. Numerical algorithm

#### 3.2.1. Existing algorithm

The low-Mach formulation of the code relies on a semi-implicit, finite-difference type time advancement, and an iterative predictor-corrector updating scheme (see Figure 7). After computing thermodynamic properties at the beginning of the time loop, scalars are advanced and then the density is evaluated from the equation of state. The momentum equation is advanced omitting the pressure gradient term. The velocity value obtained from this incomplete momentum equation is the predictor value. Then applying
the corrector dependence of the dynamic pressure in the continuity equation leads to a Poisson equation for the pressure. Once the pressure field is evaluated, the corrector velocity value is computed. If the corrector has converged, the next time step is considered. Otherwise, another predictor-corrector iteration is performed. The index $k$ refers to the current number of sub-iterations.

Scalars (internal energy in our case) and momentum are advanced in time using a second-order accurate Crank-Nicolson scheme. In the context of this scheme, the flux terms need to be computed at the mid-point in time. Additionally, the code uses time staggering. The discrete density and scalar solutions that are solved for are offset in time from the discrete velocity solutions by a half-step. Similarly to the spatial staggering, this staggering is performed to decrease the width of a discretization stencil. Specifically, the offset of velocity places the velocity solution at the temporal mid-point of the scalar transport equation timestep. The velocity solution can therefore be directly used in a scalar transport equation with second-order accuracy. Similarly, the density solutions that are used in the momentum equations are most accurate when located at the mid-point of the momentum time step.

As implemented, the source of density variation can stem only from heat release in a reactive flow. Since the interest of this work concerns non-reactive flows, the density remains constant. Hence, this framework needs to be extended in order to account for dilatational flows.

### 3.2.2. Algorithm extension to non-reactive low-Mach flow solver

In order to account for a dilatational flow, a temperature update needs to be implemented. This takes three changes in the algorithm shown in Figure 7. First, the internal energy equation of system (3.3) is implemented and advanced. The time resolution of the implemented internal energy, similar to the one used for the scalar, reads:

$$\frac{\partial}{\partial t} (\rho e_i) = \frac{\rho_{i,k}^{n+3/2} (\delta e_i) + \rho_{i,k}^{n+3/2} e_{i,k}^{n+3/2}}{\Delta t} - \frac{(\rho_{i,k}^{n+1/2} e_{i,k}^{n+1/2})}{\Delta t} + O(\Delta t^2), \text{where } \Delta t = t^{n+1} - t^n,$$

(3.4)
Here the updated term is decomposed into a correction added to the value of the current sub-iteration $k$. One can notice that the density is considered constant. Although not true, since the flow is not considered incompressible, the fact that the system made of Eq. (3.3) and the EOS is ill-posed makes it necessary to involve this kind of assumption. Then, once the internal energy is updated, the thermodynamic pressure $P_0$ is rescaled to account for temperature changes and to ensure total mass conservation. Finally, the local density is computed from the equation of state.

Some caution is needed to correctly take into account cell volume change effects due to changes in geometry when computing the fluxes in the context of the Crank-Nicolson scheme.

### 3.3. Results for compression cases

The dilatational flow solver is compared against results from the compressible flow used as a benchmark for two cases: an adiabatic and an isothermal sequence of cylinder compression/expansion. Since the piston moves at a velocity much below the speed of sound, a high level of similarity is expected between the results of the low-Mach and compressible formulations. The configuration is illustrated in Figure 1 is considered, with the following dimensions: $L = 10$ cm, $R = 2.5$ cm, and $\theta = 12.5$ degrees, and the discretization: $nx = 32, ny = 128, nz = 1$. The compression ratio is 10. As in Section 2.3.2, a sinusoidal profile is assumed for the piston, where the maximum velocity is defined to ensure a cylinder speed of 3333 revolutions per minute (rpm). At this speed, a cycle take 18 ms. The initial mixture is air with a density of 1 kg/m$^3$ and a temperature of $T = 500$ K.

#### 3.3.1. Adiabatic compression

This first test consists of an adiabatic compression. The objective is to test the ALE formulation of the dilatational solver. Velocity profiles are displayed in Figure 8 at two instants (6 ms and 8 ms) of the compression phase where the compression rate becomes significant ($CR = 2.5$ and $CR = 5$, respectively). The longitudinal velocity profile is displayed in the top part of Figure 8 while the radial velocity is displayed in the bottom part of Figure 8. Moreover, two spatial axial locations are considered. The first one is located in the piston vicinity (1 mm) and the corresponding results are the one experiencing the largest amount of fluctuations. The second is located downstream of the piston (5 mm) with the corresponding results experiencing the least amount of fluctuations. The excellent level of comparison with results with the compressible formulation verifies the ALE implementation of the adiabatic low-Mach formulation.

#### 3.3.2. Isothermal compression

In this case, a Dirichlet boundary condition is set for the wall temperature at $y = R$: $T = 400$ K. The objective is to assess the accuracy of the density and temperature fields provided by the dilatational solver in the presence of heat losses. Figure 9 shows the comparison between temperature profiles, while Figure 10 shows the comparison between velocity profiles. As before, the compressible-formulation results serve as reference. For the sake of graphic legibility, profiles are displayed at one axial location only, 1 mm from the piston. Contrary to the adiabatic case, large discrepancies occur, much larger than an acceptable level of difference due to the difference between the two formalisms and the numerics. The temperature front moving from the wall to the centerline is significantly more advanced in the low-Mach case, whereas values of temperature far from the isothermal region do not match. Both phenomena are caused by the inaccurate treatment of the local density in Eq. (3.4). Since the effect of heat conduction is not accounted for
in the density change after the energy update, the rescale of the thermodynamic pressure is erroneous. Hence, the updated density based on this pressure becomes inaccurate as well. Since the time rate of change of the local density is the quantity that contributes the most to the residual of the Poisson equation, errors on the density reflect on the velocity field, which explains the discrepancies observed in Figure 10.

These results point to the need to consider thermal effects in the update of density. The adopted strategy is explained in the next section.

4. An accurate solver for dilatational flow with moving geometry

4.1. Corrected model

As highlighted in the previous section, the impact of heat flux on the density requires consideration. A relation between density and heat flux can be directly isolated from the density and energy equations of Eq. (3.2), which are rewritten in terms of substantial variables (McMurtry et al. 1986) as

\[
\begin{align*}
D_t (\rho) + \rho \partial_{x_j} (u_j) &= 0, \\
\rho D_t (e) &= -P_0 \partial_{x_j} u_j + \frac{\gamma}{(\gamma - 1)} \partial_{x_j} (\lambda \partial_{x_j} T).
\end{align*}
\] (4.1)
Thermal boundary layers in pulsatile flows

Figure 9. Temperature profiles. Solid line: compressible. Dotted line: low-mach. (a): results at $t = 6$ ms. (b): results at 8 ms.

Figure 10. Velocity profiles. Solid line: compressible. Dotted line: low-mach. (a, c): results at $t = 6$ ms. (b, d): results at 8 ms. (a, b): longitudinal velocity. (c, d): radial velocity.

Here $\gamma$ is assumed constant for the sake of simplicity, but without loss of generality (since the formulation can be extended to the general case of temperature-dependent heat capacities), expressing the energy in terms of the pressure and the density in the
energy equation, and combining it with the continuity equation leads to:

\[
\partial_{x_j}(u_j) = \frac{1}{\gamma P_0} \left[ (\gamma - 1) \partial_{x_j}(\lambda \partial_{x_j} T) - D_t P_0 \right].
\] (4.2)

The time rate of change of \( P_0 \) is evaluated by integrating Eq. (4.2) over the computational domain. Using the Gauss theorem and the fact that \( P_0 \) is homogeneous, the equation

\[
D_t P_0 \int d^3 x = -\gamma P_0 \int u d^2 x + (\gamma - 1) \int (\lambda \partial_{x_j} T) d^2 x
\] (4.3)

is obtained, where the first term on the right-hand side represents the domain volume rate of change due to moving surfaces, and the second term is the integrated heat-flux over the non-adiabatic walls. This equation is the fifth equation of the dilatational system. Then the velocity divergence in the continuity equation is substituted in Eq. (4.2).

4.2. Numerics

This subsection briefly describes the challenges and solutions for implementing the above model. Two equations need to be implemented: the pressure and the continuity equation. So far, the density equation is implemented only to compute the residual term of the Poisson equation. Therefore, a complete implementation of the continuity equation as an evolution equation is performed. Since a staggered grid is used, computing \( u_j \partial_{x_j} \rho \) is not natural in this context. Therefore, the following equation is implemented:

\[
\partial_t (\rho) + \rho \partial_{x_j}(u_j) + \partial_{x_j}(\rho u_j) - \rho \partial_{x_j}(u_j) = 0,
\] (4.4)

where the convection operator is computed through the conservation convection term and the actual velocity divergence that are naturally computed in this context.

The spatial discretization of the term \( \partial_{x_j}(\rho u_j) \) needs extra caution in order not to generate spurious local extrema. Using a WENO 3 scheme that is dispersive does not prevent this to happen. Indeed, fluxes at the cell interface are computed from the data of density from both sides of this interface, and some spurious oscillations might occur. In the context of a compressible formulation, these oscillations are damped by acoustic waves and thus are not a big concern. But in a context where no acoustics is involved, preventing these instabilities from occurring is a critical property that the numerics need to fulfill.

The solution considered here is to compute the flux of the term \( \partial_{x_j}(\rho u_j) \) using an upwind method in combination with a flux-splitting strategy as described in Bouchut et al. (2003). The resulting numerical method becomes then a combination of Finite Differences and Finite Volumes. The spatial discretization is first order in space. Its extension to second-order following Bouchut et al. (2003) does not create any issue. However, extra caution is needed when computing high order fluxes for density and momentum on staggered grids as some instabilities may occur as explained in Kah (2010).

This scheme is tested first using an explicit Crank-Nicolson time solver. Enhancements in both space and time solver will be provided in the near future. In order to compute the pressure evolution, the integrated heat transfer through the domain surface and the time rate of change of the volume are implemented.
4.3. Results

In order to highlight improvements from the first attempt, tests with this new flow solver are conducted for the isothermal compression with the configuration and conditions described in section 3.3. As before, the solver performance is assessed by comparisons of radial profiles with results from the compressible formulation for one cycle. Figures 11, 12, 13 show results for temperature, longitudinal and radial velocity, respectively. Results are displayed at four different instants: $t = 6$ ms, $t = 8$ ms, $t = 11$ ms, $t = 15$ ms. The two first instants lie in the compression phase, whereas the last two correspond to the expansion phase. Finally, these radial profiles are monitored at 1 mm from the piston, where the flow experiences a high amount of fluctuations. Comparison of temperature profile in Figure 11 shows a much improved level of agreement between the new solver and the compressible results during the compressible phase than with the previous low-Mach solver. During the expansion phase, differences become more significant. The boundary layer is more diffused than in the compressible case. However, this behavior was expected since spatial order is lower for the dilatable solver than for the compressible one. Moreover, the level of agreement of the bulk flow temperature far from the boundary layer is excellent. These results clearly validate the above developments on the treatment of the density and the pressure.

Concerning longitudinal velocity, Figure 12 shows an improved level of agreement at
$t = 6$ ms. Differences occur at $t = 8$ ms as the maximum velocity value for the new dilatable solver is located slightly above that for the one for the compressible solver. This may be due to the set of differences between the two solvers: compressibility, space and time solvers, and numerical accuracy. Further investigation of these differences will be conducted. The quality of comparison with the compressible results is on a complete different level from that with the previous low-Mach solver. One can also note that all the trends in the profile are captures by the new solver. Nevertheless, some questions are raised by the comparison close to the centerline during the expansion phase, where peaks are observed in the compressible results, specially at $t = 11$ ms. These peaks will be the object of further investigation as no obvious physical reason justifies them.

Comparison of radial velocities is provided in Figure 13. The conclusions are similar to those drawn for the longitudinal velocity. One may point out again that the peak for the compressible results observed at $t = 11$ ms has no obvious justification.

Finally, comparisons of integrated quantities are provided in Figure 14: thermodynamic (or volume-averaged in the compressible case) pressure and volume-averaged temperature. Good agreement on these quantities is critical since they monitor the total energy and pressure in the cylinder for simulations that are conducted under the same conditions. These results provide relative errors against compressible results for the averaged
Figure 13. Radial profiles at 1 mm from the piston. Solid line: compressible. Dotted line: low-mach. Dashed line: new solver. Results at $t = 6$ (a), 8 (b), 11 (c), 15 (d) ms.

Figure 14. Relative errors. Dashed black line: error of the low-mach solver. Solid red line: error of the new dilatable solver. (a): averaged pressure. (b): averaged temperature.

pressure (Figure 14a) and temperature (Figure 14b). A drastic improvement is observed. Indeed, at the end of the cycle, the pressure error drops from $0.78\%$ with the old solver to $0.178\%$ with the new one, dividing the error by 4.5. For the temperature, the error drops from $0.78\%$ to $0.11\%$, reducing the error by a factor 7. Since these quantities depends on the value of the heat loss, that itself depend on the local values of the temperature at the wall, improvements are expected to be made by increasing numerical order.
5. Conclusion

This paper presents the numerical framework for the study of the thermal boundary layer in a cylinder that experiences phases of compression/expansion. The aim is to reproduce the flow conditions in an internal combustion engine in order to characterize the boundary layer structure and then to predict the amount of heat loss. First results with a compressible formulation have brought some insight in terms of converge computation, turbulence appearance and boundary layer structure. However, the time-efficiency of such simulations is restricted by acoustic waves that turn out not to have a first-order impact on the flow. Therefore, a low-Mach solver for dilatational flows has been designed and implemented in the software that was used. A verification test has been performed and results were compared to results given by the compressible formulation, considered as the reference in this context. The improved level of approximation validates the strategy adopted for the dilatable flow solver and paves the way for future developments.

Among the developments expected, the increase of numerical order in space will improve results limiting dissipation. Real fluid thermodynamics are to be considered in order to obtain reliable comparisons with experimental results. Eventually, an acceptable mesh refinement is to be determined in order to access the fully resolved boundary layer topology and the vorticity field in order to highlight the mechanism that controls the heat loss in an engine.

Acknowledgments

The authors acknowledge computing and visualization time on the Certainty cluster at Stanford (MRI-R2 NSF Award 0960306).

REFERENCES


