1D gas dynamic modelling of mass conservation in engine duct systems with thermal contact discontinuities

J.R. Serrano a,*, F.J. Arnau a, P. Piqueras a, A. Onorati b, G. Montenegro b

a CMT - Motores Térmicos, Universidad Politécnica de Valencia, Spain
b Dipartimento di Energia, Politecnico di Milano, Italy

Article Info
Article history:
Received 18 February 2008
Accepted 27 March 2008

Keywords:
Conservation equations
Numerical methods
Thermal contact discontinuity
Mass conservation
Time discretisation

Abstract

A detailed analysis of mass non-conservation in the proximity of thermal contact discontinuities, when solving 1-D gas dynamic flow equations with finite difference numerical methods, is carried out in this paper. A wide spectrum of finite difference numerical methods has been applied to solve such conditions. Thermal contact discontinuities are very common in current diesel engines due to back-flow in the intake valves during the valve overlap period. Every method has been shown to be incapable of correctly solving the problem raised, displaying (or revealing) a different behavior. Taking as baseline these analyses a study regarding mesh size reduction in ducts has been also performed. This solution becomes suitable since it leads to making mass conservation problems disappear. Nevertheless, most extended calculation structure in 1D gas dynamic models is not advised due to the increase of computational effort required. Thus, a new calculation structure for solving governing equations in ducts is suggested. This proposed calculation structure is based on independent time discretisation of every duct according to its CFL stability criterion. Its application to thermal contact discontinuities points out its advantages with regard to computational demand as the calculation time of every duct is adapted to its mesh size.

1. Introduction

Nowadays the challenging requirements of higher specific power, better fuel economy and increasingly stringent emission regulations [1] are demanding a significant effort from the automotive industry and researchers to develop a new generation of internal combustion engines. The optimization of gas flow through the engine intake and exhaust systems, involving the air filter and intake manifold, the EGR line, the turbocharger, the exhaust manifold, after-treatment and silencing devices, is one of the key strategies in order to improve engine performance and to reduce pollutants emission.

In this field the use of thermo-fluid dynamic simulation models is producing unquestionable success due to its ability to describe physical phenomena taking place in the engine. Thermo-fluid dynamic simulation models can be classified on the basis of their complexity as mean value or quasi-steady models, filling and emptying models, one-dimensional (1-D) wave-action models and three dimensional calculation codes [2]. Among these computer codes, the development of one-dimensional (1-D) gas dynamic numerical models has become an essential tool to assist in the design [3] and development of new engines, as they can reproduce the effects on performance caused by modifications of the engine and its ducting geometry. In recent years there has been a significant enhancement of one-dimensional gas dynamic numerical models and they have been widely used to simulate engine behaviour, from torque, power, fuel consumption [4,5] or exhaust gas

* Corresponding address: CMT - Motores Térmicos, PO Box 22012, E-46071, Valencia, Spain. Tel.: +34 963879657; fax: +34 963877659.
E-mail address: jserran@mot.upv.es (J.R. Serrano).

© 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.mcm.2008.03.015
emissions [6,7] to noise emission prediction [8–10] or load transient evolution [11,12] and have shown a suitable balance between reliability, accuracy and calculation time [13,14].

One-dimensional gas dynamic models have to solve the 1-D unsteady non-homentropic mass, momentum and energy conservation laws. These governing equations form a non-homogeneous hyperbolic system. Nowadays, highly non-homentropic flow conditions can only be solved by means of numerical methods, despite the contribution of some works with regard to the analytical solution of the governing equations system [15–17].

In most numerical models, the traditional mesh-method of characteristics (MOC) has been applied up to the mid 1980’s [18]. Although the MOC is still widely used in the resolution of boundary conditions, owing to the comprehensive and well-documented work existing on the treatment of every kind of flow boundary regions by means of a MOC-based quasi-steady approach [19], in last decades the new generation of 1-D fluid dynamic models has adopted symmetric or upwind shock-capturing numerical methods in order to solve the hyperbolic problem of conservation equations. The origin of these two important families of schemes was numerical techniques for resolution of the hyperbolic conservation laws system developed in the form of first-order methods by Courant et al. [20] and Lax [21]. On one hand, symmetric schemes apply a discretisation which is independent from inviscid flow characteristics. On the other hand, upwind schemes apply directional space discretisations according to the physical behaviour of inviscid flows.

The second-order symmetric scheme proposed by Lax & Wendroff [22] is one of the most representative symmetric schemes. The main disadvantage of this method is the Jacobian matrix evaluation. To prevent the calculation from this difficulty, Richtmyer and Morton [23] suggested a two-step variant of the Lax & Wendroff method, leading to a whole family of variants when applied to non-linear systems, represented by the MacCormack method [24].

A subgroup of upwind schemes is based on Godunov’s method [25] to solve, over each mesh interval, the local 1-D Euler equations for discontinuous neighbouring states. The second subgroup of upwind schemes is the flux vector splitting methods, proposed by Steger and Warming [26].

Finite difference schemes, with greater than first-order accuracy and constant coefficients produce local instabilities in the presence of shock waves and contact surfaces as a consequence of Godunov’s theorem [25]. These oscillations are non-physical and are known as Gibb’s phenomenon. To prevent from their generation, Boris and Book [27,28] developed the flux-corrected transport technique (FCT), whose three different steps to be solved were described by Niessner and Bulaty [29]. These techniques represent the origin of high-resolution schemes.

Harten [30] developed the total variation diminishing (TVD) concept, whereby the variation of the numerical solution is controlled in a non-linear way, so as to avoid any new extremum from appearing.

Finally, Chang [31,32] introduced a new Euler solver named the conservation element-solution element method (CE–SE), based on a different approach with regard to the resolution of fundamental conservation equations. The peculiarity of this scheme is that it does not employ any characteristic-based or flux-limiter method, but rather follows a simple approach to generate highly accurate oscillation-free solutions.

Some authors have focused their efforts on adapting some of the previously developed schemes to solve non-homogeneous problems. In that way, Briz and Giannattasio [33] suggested a CE–SE method adaptation to solve the fundamental conservation equations system considering source terms. Later, Corberán and Gascón [34–36] simplified non-homentropic conservation equations in a homogeneous system by means of considering the source terms as part of the flow terms. Besides, the duct cross-sectional area was included in the conservation variables. The Lax & Wendroff method, the Harten TVD scheme and the flux-limiting techniques of Sweby were adapted according to this new approach [34–36]. A similar approach was carried out by Liu et al. [37] so as to propose the use of flux-corrected techniques in a conservative way. Finally, Vandevoorde et al. [38] proposed a new cell-vertex TVD scheme to solve non-homoentropic conservation laws based on flux vector splitting techniques.

At the same time these approaches were being developed, the introduction of finite difference numerical methods in gas dynamic models [8,39–41] requires a continuous study of their behaviour under different flow conditions [42–45], in order to optimise the choice of every method to be used in each case.

The work reported in the present paper carries on the studies regarding the effects of numerical methods on the solution provided and faces the analysis of the behaviour of numerical methods in the proximity of thermal contact discontinuities. Such a discontinuity represents an interface of two flows of gases with different composition and/or different temperature. The unsteady, compressible gas flow in the intake and exhaust systems of internal combustion engines includes regions with varying composition and temperature. Especially critical is the thermal contact discontinuity produced in intake pipes, where this variation is mainly due to the back-flow of exhaust gas during valve overlap at low engine speed conditions that goes back again into the cylinders after a short time period. This study is developed from mass conservation problems appearing in the proximity of thermal contact discontinuities taking place close to intake valves.

The work has been carried out by means of two 1-D gas dynamic models, WAM, from the Institute “CMT-Motores Térmicos”, Universidad Politécnica de Valencia, and GASDYN, from Dipartimento di Energia, Politecnico di Milano. As a first step, the authors have checked by means of both WAM and GASDYN the behaviour of several finite difference schemes in order to describe them and attempt to clarify the possible causes of detected mass conservation problems in the proximity of thermal contact discontinuities. Then, this analysis has been extended to evaluate the influence of spatial mesh size reduction on this concrete mass conservation problem. The improvement of the results concerning mass conservation contrasts with the increase in computational effort, which would preclude the spatial mesh size reduction from being adopted as a feasible solution to the related mass conservation problem. In order to minimise the effects of spatial mesh size
reduction on the calculation time, an innovative calculation methodology based on an independent time discretisation in the ducts composing the model is described and checked in detail. In this way, the paper performs a sound solution to provide 1-D gas dynamic models with the ability of being applied in the proximity of thermal discontinuities with a satisfactory response.

2. Shock-capturing numerical methods for 1D unsteady flow

In the present work, the unsteady flow is assumed to be completely governed by compressible Navier–Stokes equations, which eventually may be reduced to Euler equations, taking into account that viscosity has a minor influence on wave motion. In addition, the propagation of pressure waves may be assumed to be planar inside elements whose length is the paramount dimension \([3]\). Under these conditions, the 1D approximation proved to be valid for those applications in which a reduced computational time is required and the geometry is not characterised by highly 3D features.

In this study the conservation equations of mass, momentum and energy are written in a strong conservative form and applied to the simulation of pipe systems, with source terms to account for the friction and the heat transfer between the gas and the walls, as follows:

\[
\frac{\partial W(x, t)}{\partial t} + \frac{\partial F(W)}{\partial x} + B(W) + C(W) = 0
\]  

where the vector \(B(W)\) and \(C(W)\) are the source term vectors. The former takes into account the variation of the cross flow section along the duct \((F)\), the latter considers the influence of gas-wall friction \((G)\), the heat transfer between the wall and the gas \((q)\) and the heat released by homogeneous chemical reactions occurring within the gas phase \((q_{re})\). Although it is possible to transport information concerning the composition of the gas \([3, 45]\), for simplicity sake the hypothesis of perfect gas has been assumed, leading to closure of the system of equations.

The final formulation of the conservation laws is a hyperbolic system of equations. Since this system cannot be solved analytically, it is solved recurring to numerical techniques.

Different numerical methods have been applied to study how mass conservation is affected when strong discontinuities in the temperature field occur. The numerical schemes applied in this work and their order of accuracy are shown in Table 1.

2.1. Analysis of the calculated results with thermal contact discontinuities

As a test case for the numerical analysis a single cylinder Diesel engine, with 550 cc swept volume and a compression ratio of 1:18 has been chosen. The engine has been simulated running at 1250 rpm with the valve timing chosen in such a way to produce a back flow in the intake system as the intake valve opens. In this way, as the intake valve opens, a contact discontinuity of hot gas starts travelling from the boundary of the intake duct towards the inlet, whereas the following part of the induction phase allows this discontinuity to travel back to the cylinder.

The intake and exhaust lines coupled to the engine are constituted by straight pipes with constant cross section area. In particular, the intake line is a 0.3 m long single duct, as observed in Fig. 1.a.

The numerical methods listed in Table 1 have been applied to the simulation of a single cylinder engine, in order to analyse different behaviour relating to mass conservation when the passage of a strong temperature contact discontinuity is involved. Due to the numerous numerical methods analysed, results have been grouped according to similarities in mass conservation behaviour and solutions from only one representative method of every group have been shown in Fig. 1, where
instantaneous temperature profiles are depicted. After the opening of the intake valve at 340 crank angle degrees, a back-flow brings hot gas from the combustion chamber into the intake pipe. This process is characterised by the presence of a strong discontinuity in the temperature field, which can be clearly seen in the temperature profile. During the following part of the induction phase the flow goes from the pipe to the cylinder, and the discontinuity is transported back to the cylinder. Since both the duration and the intensity of the back-flow are limited, the contact surface does not travel further than a few calculation nodes of the intake pipe.

In particular, methods with second-order accuracy symmetric schemes, such as the Lax & Wendroff method and the MacCormack method, are affected by numerical instabilities. An example of this aspect is the low gas temperature value, below 0 °C calculated at 2 and 6 cm from the intake valve, as it is depicted in Fig. 1.c.

The application of first order upwind methods, namely Godunov’s method, or the adoption of high resolution schemes such as TVD or FCT schemes, allows a better behaviour in terms of instabilities, balanced, however, by the addition of a first-order component which contributes to excessive diffusion. In particular, the adoption of these numerical schemes prevents the solution from being unbounded, but slightly smears the temperature discontinuity, as it can be observed in Fig. 1 Graphs b and d.

By contrast, the Random Choice Method, which is shown in Fig. 1.e, shows a particular behaviour. The solution is not affected by the presence of numerical diffusivity at all. As a matter of fact, the randomly chosen solution state allows to capture sharp gradients in the conserved fields with high resolution, exhibiting, however, a mismatch in the discontinuity position. This mismatch is contained within half of the mesh and is proportional to mesh size.
2.2. Effect of the spatial mesh size reduction on mass conservation

The impact of thermal discontinuity on mass conservation changes with the numerical scheme applied in the simulation. In order to check the ability of every finite difference numerical method to deal with this phenomenon a study of the spatial mesh size of the intake duct has been performed. Four spatial mesh sizes have been considered in this study, 5, 10, 20 and 50 mm. The results are shown in Fig. 2, where the mass conservation error calculated at every node, according to Eq. (4), has been plotted.

\[
\text{MassConservationError,}(\%) = \frac{m_i - \sum_{j=1}^{n} \frac{\Delta t_{cycle}}{m_j}}{\sum_{j=1}^{n} \frac{\Delta t_{cycle}}{m_j}} \times 100
\]

(4)

where

- \( n \) is the total number of nodes;
- \( m_i \) represents the mass of air that goes through the node \( i \);
- \( t_{cycle} \) is the cycle duration.

Whichever finite difference numerical method has been applied, noteworthy conservation mass problems have been proved close to the intake valve if a high temperature back-flow takes place, which is a common phenomenon in internal combustion engines. Nevertheless, Fig. 2 shows clearly how the mass conservation error is reduced as the mesh size is reduced for every method tested. Furthermore, the behaviour of every numerical method can be classified into first-order upwind scheme behaviour such as Godunov's method, into second-order symmetric scheme behaviour, such as the two-step Lax & Wendroff method and the MacCormack method, or into high resolution scheme behaviour, such as TVD and the two-step Lax & Wendroff method + FCT. Although the CE–SE method cannot be classified as a high resolution scheme, due to its behaviour relating mass conservation close to thermal discontinuities it can be fallen into this group, as observed in Fig. 2. The Random Choice Method and Lax–Friedrichs scheme have shown their own particular behaviour respectively.

The numerical instability of second-order symmetric schemes is also confirmed by the trend of the mass conservation error (Fig. 2 Graphs c and d), which oscillates around an average value decreasing its amplitude with the distance from the intake valve. However, second-order symmetric schemes provide better results, since to obtain a mass conservation error less than 1% along the duct the spatial mesh size can be fixed to 20 mm. Under these conditions, reducing the spatial mesh size to 10 mm results in a very accurate calculation, as shown in Fig. 2. Nevertheless, oscillation of the mean air mass flow along the nodes can be critical in order to carry out the transport of chemical species calculations, since it could lead to non-physical solutions.

Although first order upwind and high-resolution methods contribute to make instabilities disappear, they do not guarantee a completely bounded solution, also resulting in a distortion of the solution. This aspect is evident in Fig. 2 Graphs a, e, f and g, where the mass conservation error, which is remarkable in correspondence of the intake valve, decreases with the distance from the intake valve and becomes asymptotic with slight oscillations around a positive value. The Random Choice Method also shows a similar behaviour, although due to the absence of numerical diffusivity, its mass conservation error results in a flat profile along the intake duct, with the only exception being the first node, in which typical randomness of the solution can be spotted. Regarding spatial mesh size, for these types of numerical methods it needs higher reduction, at least up to 10 mm. The Lax-Friedrichs scheme tends to introduce too much numerical diffusion, hence it excessively propagates the error, which is characterised by the highest maximum value and is hardly affected by spatial mesh size reduction.

Fig. 3 only shows a summary for the most advisable methods. Fig. 3a shows the air mass going into the cylinder during a cycle for every spatial mesh size applied in the study. Fig. 3a confirms that second-order symmetric schemes are more appropriate from mass conservation point of view in case of thermal contact discontinuities. The two-step Lax & Wendroff method application results in a small variation of the air mass coming into the cylinder for every spatial mesh size, whereas with high resolution methods the air mass has a higher dependence on the spatial mesh size, and even the air mass value is unacceptable if the spatial mesh size is too coarse. It is interesting to point out that when the spatial mesh size is reduced the air mass converges to the same value. Among all the methods, on the other hand, the CE–SE method provides the worst results as the spatial mesh size grows.

It is also fundamental to take into account the computational requirements needed by numerical methods at these calculation conditions, since 1-D gas dynamics models should guarantee a suitable balance between reliability, accuracy and calculation time [13,14].

The CFL stability criterion [46] is a necessary condition so that the numerical methods are stable. This criterion requires that the information, in form of disturbances or waves, cannot travel more than the calculation makes progress, and is expressed through the equation

\[
\Delta t = \frac{\Delta x}{v c_{max}^n}
\]

(5)

where the parameter \( v \) is known as the Courant, or CFL, number, whose values belong to the interval \([0, 1]\), and \( c_{max}^n \) represents the largest wave speed present in the entire solution domain at time level \( n \).
In some numerical methods this condition is more restrictive [47]. In that way, when the mesh spatial size is reduced, the time step has to be diminished proportionally. Therefore, the computational effort significantly increases with the spatial mesh size reduction.
Generally 1-D gas dynamic codes calculate every engine component, namely ducts, plenums, cylinders, etc, with a common time-step. Hence, to ensure the stability for each modelled duct, the time step must be in accordance with the most restrictive of all the ducts. Fig. 3.b shows a comparison of computational requirements needed by different numerical methods at each spatial mesh size configuration. The average values of computational duration have been represented referring to the two-step Lax & Wendroff method computational effort using the spatial mesh size of 50 mm.

Fig. 3.b points out that high-resolution schemes are the most affected by the spatial mesh size reduction, being the TVD method the most damaged when the spatial mesh size is severely reduced, owing to the fact that this scheme is based on the Jacobian matrix resolution.

It is convenient to underline that the two-step Lax & Wendroff method with a spatial mesh size of 20 mm achieves a mass conservation error less than 1% and its computational time just slightly grows regarding 50 mm configuration. However, high-resolution schemes need to apply a spatial mesh size of 10 mm, or smaller for the specific case of the CE–SE method, to approximately reach the 1% error level. This behaviour leads to nearly double the computational time required by high resolution schemes to achieve the same accuracy than the two-step Lax & Wendroff method in the case of a 20 mm mesh size, and evidences that the two-step Lax & Wendroff method application is more advisable for mass conservation against thermal contact discontinuities resolution, although it is needed to take into account its problems with regard to temperature and mass oscillation along the nodes.

3. Calculation methodology approach based on independent time discretisation

It is clear that to reduce the spatial mesh size in a specific duct (in this case the intake duct) leads to an increase in the number of calculations per real time unit of every engine component. This fact gives rise to very high run-times to calculate a certain number of engine cycles. Nowadays it is common to adopt 1D codes for the simulation of engine transients. Therefore, since the number of cycles to be simulated is usually very large, an arbitrary mesh size reduction is not an advisable solution.

A possible solution may consist in using an independent time discretisation in ducts to be applied in 1-D gas dynamics models. This new calculation approach allows spatial mesh size reduction in specific ducts without noticeably damaging global calculation time and achieving great accuracy.

Fig. 4 shows the flowchart of the suggested approach. Independent time discretisation defines a loop which solves the fundamental conservation equations system in ducts, boundary conditions and 0-D elements connected to boundary conditions in a time-step. This loop is included inside the main loop that controls the well-running of the execution.

On one hand, the variables $t_0$ and $t_1$ are referred to each duct. In this way, $t_0$ is defined as the calculation initial time for every engine element and $t_1$ as the specific final time for each duct according to its CFL stability criterion. On the other hand, $t_{ini}$ and $t_{end}$ represent the initial and final time of the simulation time-step, respectively. The first step consists of searching the duct whose $t_1$ is maximum, which defines $t_{end}$. Once $t_{end}$ is known, the time-step is set as $t_{end} - t_{ini}$. During the time-step every engine element will be calculated once at least.

Next, the fundamental conservation equations system is solved in the duct whose $t_1$ is minimum. Furthermore, its boundary conditions are solved as well, but it must be taken into account that the rest of the engine elements are at a different calculation time. If there is any 0-D element contiguous to this duct, its thermodynamic properties are calculated from the results got from the boundary condition solution. In this case, it is considered that any other duct connected to the 0-D element keeps its mass flow and thermodynamic properties constant to the values calculated the last time the duct was solved. Finally, the new $t_1$ for the current duct is set by applying the CFL stability criterion.

These calculations are carried out as many times as required until the duct setting the maximum time-step is solved. Then, the independent time discretisation loop finishes.

Next the main loop checks whether or not the program has reached the time set as calculation end. If program ending has not been reached, the duct whose $t_1$ is maximum is searched in order to get the new time-step and the independent time discretisation loop begins again. This iterative process is carried out until the calculation end is reached.
The MOC method has been applied for the resolution of boundary conditions from the characteristics Riemann variables, $\lambda$ and $\beta$, and the entropy level \[19\]. The use of a calculation structure based on an independent time discretisation for each duct requires adaptation of the boundary conditions resolution. On this matter, two kinds of boundary conditions have to be distinguished, depending on the engine elements that the boundary condition connects.

The boundary condition between a duct discharging flow to a plenum, cylinder or any other 0-D engine element is solved from the characteristic lines, $\lambda$ and $\beta$, and the entropy level, and the output and input mass flows. There is no difference in the way the boundary condition is solved with regard to the original programme layout. However, there is a difference lying in the calculations carried out to get the thermodynamic properties in the 0-D element. Every time a contiguous duct is solved, the 0-D element has to be updated to the current time. In order to carry out the energy and mass balances in the 0-D element, every pipe end connected to the 0-D element has to be considered. This fact leads to take into account the last calculated results for every boundary condition, assuming the information is frozen since the last time they all were calculated until the current time, that is set by the last calculated boundary condition connected to the 0-D element.

Every kind of junction between ducts can be classified into another group. In this case, when the boundary condition is solved it must be considered that each duct connected to the boundary condition is at a different calculation time. This situation is represented in Fig. 5.a, which shows the resolution of the connection between two ducts. Every point represents a node of the duct; a gas velocity from right to left is assumed in the pipe.

The duct $k$ is being solved. Its calculation time is $t_{0k}$ and the aim is to advance the calculation until the time $t_{1k}$. For that, the thermo-fluid-dynamic properties of the duct $j$ are available at the calculation time $t_{0j}$.

In order to solve the boundary condition, the origin of the three characteristic lines has to be found, so that at the calculation time $t_{1k}$ they all are passing through the extreme node of the duct $k$. Due to the fact that each duct is at a different time level, two distinct time steps $\Delta t_k$ and $\Delta t_{1k}$ must be considered.
The scheme of a geometrical configuration representing Herschel–Quincke ducts is represented in Fig. 5b. This configuration presents two junctions between three ducts of different diameter as described in Fig. 5b and allows the study of phenomena related with wave interference between parallel branches. In Fig. 5c the transmission loss of the Herschel–Quincke ducts, computed with both common and independent time methodologies, is shown as a function of the ratio of the frequency $f$ to the speed of sound $a$. The result verifies the good agreement between the solution provided by both of the methodologies.

In order to check the effectiveness of the independent time discretisation calculation methodology against thermal discontinuities a pulsating flow at 25 °C discharging through two serial ducts has been analysed. The model is sketched in Fig. 6a. The duct 1 has a length of 1 m and the duct 2 has a length of 0.5 m and it is connected to a reservoir whose temperature is kept constant at 250 °C. The flow goes into the reservoir. Nevertheless, during certain time-steps there is a back-flow from the reservoir to the ducts due to the pulsating flow effect. This phenomenon has been modelled with the purpose of simulating high temperature back-flows with a simpler configuration than the engine cylinder case, which involves also variable volume and combustion feature calculations.
Table 2
Comparison of computational requirements solving the Model 1

<table>
<thead>
<tr>
<th>Mesh size (m)</th>
<th>Duct 1</th>
<th>Duct 2</th>
<th>Two-step LW</th>
<th>Two-step LW + FCT</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Time unit</td>
<td>Time unit</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Independent</td>
<td>Common</td>
<td>Independent</td>
<td>Common</td>
</tr>
<tr>
<td>0.05</td>
<td>0.96</td>
<td>1.33</td>
<td>0.05</td>
<td>0.96</td>
</tr>
<tr>
<td>0.02</td>
<td>1.31</td>
<td>2.12</td>
<td>0.05</td>
<td>1.31</td>
</tr>
<tr>
<td>0.01</td>
<td>2.46</td>
<td>4.72</td>
<td>0.05</td>
<td>2.46</td>
</tr>
</tbody>
</table>

As initial configuration, it is considered a spatial mesh size of 0.05 m for every duct. The other configurations have a suitable spatial mesh size reduction in the duct discharging flow to the reservoir (duct 2). These reductions are 0.02 m and 0.01 m.

The mass conservation error in the ducts is shown in Fig. 6.b. The distance takes its origin from the boundary condition discharging to the reservoir at 250 °C. Fig. 6.b shows results obtained from the application of the two-step Lax & Wendroff method and independent time discretisation. It is clear by comparing with the two-step Lax & Wendroff method behaviour in Fig. 2.c, that mass conservation error behaves very similar whatever calculation methodology, common or independent discretisation, is considered.

The main difference as regards the application of both calculation structures lies in the required computational effort. Table 2 shows a non-dimensional time calculation parameter for each of the methodologies studied, i.e. common and independent time discretisation. Table 2 shows also data for the case of using the two-step Lax & Wendroff method with Flux Corrected Transport scheme.

The differences regarding the computational effort between both methods are very low when the same spatial mesh size is used in both ducts and are due to different wave speed in the ducts. They are explained by slight differences in the temperature of back-flows in the duct next to the reservoir, so that the speed of sound changes in it. Therefore, since the spatial mesh size is identical, the duct with a higher speed of sound has the most restrictive CFL stability criterion. If common time discretisation is applied, this duct sets the time-step of the model. On the contrary, the use of independent time discretisation allows the duct not connected to the reservoir to be calculated by using a greater time-step, due to its minor wave speed. Therefore, this second duct is not calculated so many times and the computational effort diminishes with respect to the common time discretisation approach.

As the spatial mesh size of the duct next to the reservoir is reduced, the computational effort needed by every method increases. This fact shows the main advantage of independent time discretisation calculation methodology, due to the fact that a correct choice of spatial mesh size and numerical method to solve the fundamental conservation equations at each duct allows greater accuracy to be reached, with a significantly lower increment of computational effort.

4. Summary and conclusions

In this paper a study of mass conservation problems appearing close to thermal contact discontinuities has been carried out. The behaviour of some representative numerical methods has been analysed, in order to achieve a satisfactory solution by 1-D gas dynamic models under these conditions. Two engine simulation codes, from Universidad Politécnica de Valencia and Politecnico di Milano, have been adopted and enhanced for this study.

It has been stated that there exist severe mass conservation problems when modelling relatively common situations in internal combustion engine operation. These are the back-flows of hot gas inside the intake line during the valve overlap period. In such a way, and still missing an analytical mathematical study, spatial mesh size reduction is proposed as a feasible solution as long as the computational effort is not too high. From the results of the study, second-order accuracy schemes, which are mainly represented by the two-step Lax & Wendroff method in this study, have shown better results than high-resolution schemes and the CE–SE method regarding both mass conservation and time calculation.

To protect from an increase in computational requirement, a new calculation methodology has been suggested and developed to solve fundamental conservation equations. This methodology is based on an independent time discretisation of each duct obtained from the respective application of the CFL stability criterion. The use of the new approach leads to an improved trade-off between accuracy and computational requirements, which is mandatory in the modelling of whole internal combustion engines because of their complex duct geometry.

References


R. Courant, K.O. Friedrichs, H. Lewy, Über die partiellen Differenzengleichungen der mathematischen Physik, Mathematische Annalen 100 (1928) 32–74.