Mathematical and Computer Modelling 54 (2011) 1747–1752

Contents lists available at ScienceDirect

Mathematical and Computer Modelling

journal homepage: www.elsevier.com/locate/mcm

Assessment of a methodology to mesh the spatial domain in the proximity of the boundary conditions for one-dimensional gas dynamic calculation

J.R. Serrano, F.J. Arnau, P. Piqueras *, M.A. Reyes-Belmonte

[a] Universidad Politécnica de Valencia, CMT-Motores Térmicos, Camino de Vera s/n, 46022 Valencia, Spain

ARTICLE INFO

Article history:
Received 14 October 2010
Received in revised form 19 November 2010
Accepted 22 November 2010

Keywords:
Gas dynamics
Time marching
Mesh
Boundary conditions

ABSTRACT

Solution of governing equations for one-dimensional compressible unsteady flow has been performed traditionally using a homogenously distributed spatial mesh. In the resulting node structure, the internal nodes are solved by applying a shock capturing finite difference numerical method whereas the solution of the end nodes, which define the boundary conditions of the pipe, is undertaken by means of the Method of Characteristics. Besides the independent solution of every method, the coupling between the information obtained by the method of characteristics and the finite difference method is key in order to reach a good accuracy in gas dynamics modeling. The classical spatial mesh could provide numerical problems leading the boundary to generate lack of mass, momentum and energy conservation because of the interpolation methodology usually applied to draw the characteristics and path lines from its departure point at calculation time to the end of the pipe during the next time-step. To deal with this undesirable behavior, in this work a modification of the traditional grid including an extra node close to the boundary is proposed in order to explore its ability to provide numerical results with higher conservation fulfillment.

1. Introduction

The work reported in the present paper deals with the solution of the boundary conditions in gas dynamics codes and the way in which the sort of spatial mesh of the 1D elements can affect the accuracy of the solution. 1D elements are those in which one of the spatial dimensions is higher than the others, so a length variable is necessary to define the element, as in pipes or ducts. Instead of the traditional mesh to discretize the 1D elements which accounts for uniformly distributed nodes, the proposal of Corberán and Gascón [1] has been developed and implemented in OpenWAM^{TM} [2,3], an open source code for gas dynamics calculation of one-dimensional compressible unsteady and non-homentropic flow developed at CMT-Motores Térmicos.

According to this proposal, for a given spatial mesh size equal to Δx an extra node is added, so that the distance from the end nodes of a 1D element to the neighboring ones is set to $\Delta x/2$ instead of Δx. As a consequence, when solving the neighboring nodes of the end of the 1D element, the mass, momentum and energy fluxes are evaluated at the extremes of their control volume which coincides with the end of the 1D element. This effect contributes to increase the accuracy of the solution obtained in the boundary conditions. The existence of contact discontinuities in the boundary conditions between 1D and 0D elements is analyzed as a validation of the proposed methodology. 0D elements are those in which

* Corresponding author. Tel.: +34 963877650; fax: +34 963877659.
E-mail address: pedpicab@mot.upv.es (P. Piqueras).

0895-7177/$ – see front matter © 2010 Elsevier Ltd. All rights reserved.
doi:10.1016/j.mcm.2010.11.073