Commun. Comput. Phys.
doi: 10.4208/cicp.160811.270212a

Vol. **13**, No. 2, pp. 479-501
February 2013

Conservative Residual Distribution Method for Viscous Double Cone Flows in Thermochemical Nonequilibrium

Andrea Lani[1,*], Marco Panesi[2] and Herman Deconinck[1]

[1] *Von Karman Institute for Fluid Dynamics, Waterloosesteenweg 72, 1640, Sint Genesius Rode, Belgium.*
[2] *Institute for Computational Engineering and Sciences, University of Texas, Austin, Texas 78712, USA.*

Received 16 August 2011; Accepted (in revised version) 27 February 2012

Available online 28 June 2012

Abstract. A multi-dimensionally upwind conservative Residual Distribution algorithm for simulating viscous axisymmetric hypersonic flows in thermo-chemical nonequilibrium on unstructured grids is presented and validated in the case of the complex flowfield over a double cone configuration. The resulting numerical discretization combines a state-of-the-art nonlinear quasi-monotone second order blended scheme for distributing the convective residual and a standard Galerkin formulation for the diffusive residual. The physical source terms are upwinded together with the convective fluxes. Numerical results show an excellent agreement with experimental measurements and available literature.

PACS: 47.40.Ki, 47.70.Nd, 47.11.-j, 47.11.Fg

Key words: Unstructured mesh, hypersonic flows, thermo-chemical nonequilibrium, residual distribution schemes, double cone.

1 Introduction

During the last three decades, remarkable progress has been made in the development of algorithms for the numerical simulation of complex fluid dynamic phenomena, while the concurrent growth of resources for parallel and distributed computing have pushed researchers to devote their efforts towards more challenging applications. The accurate simulation of high-enthalpy hypersonic flows in thermal and chemical nonequilibrium,

*Corresponding author. *Email addresses:* lani@vki.ac.be (A. Lani), mpanesi@ices.utexas.edu (M. Panesi), deco@vki.ac.be (H. Deconinck)

however, still remains extremely challenging from a computational point of view, particularly on unstructured grids. For this kind of application, the Finite Volume (FV) method [3, 11, 25, 31] has consolidated itself as the *de facto* standard technique and a reliable alternative to it has yet to be found.

Over the past decade, however, multi-dimensional upwind Residual Distribution (RD) schemes have proved to be an attractive alternative to the classical FV upwind approach based on one-dimensional Riemann solvers for the simulation of compressible flows [4,7,9,14,32]. The main advantages of RD schemes include (1) an outstanding shock capturing (due to the lower cross diffusion associated to a truly multi-dimensional upwinding and to the positivity property) and (2) a compact stencil for ensuring a linearity preserving resolution. The latter does not require expensive polynomial reconstructions based on enlarged stencils in order to guarantee second order accuracy and supports an easy and efficient parallelization [6, 36, 37].

Following a previous unique attempt to apply RD schemes to 2D inviscid chemical nonequilibrium high-speed flows [8], in this paper, the RD method is extended and adapted for simulating viscous flows in thermal and chemical nonequilibrium in complex 2D axisymmetric cases with a two-temperature model.

Unlike in [8], the convective terms of the equations are discretized by means of a strictly conservative formulation of the standard RD method, denominated Contour Residual Distribution (CRD) [5, 32], which is based on a redefinition of the positive system N scheme, the so-called Nc scheme. The latter conveniently does not require a specific set of variables (e.g. Roe parameter vector in [8]) for the linearization of the flux jacobian, at the price of an additional contour integration of the convective flux in each computational cell. Second order accuracy is obtained by means of a blended Nc/LDAc (Bxc) scheme [9,10], where the blending coefficient depends on a shock capturing sensor based on a user-defined flow variable. This second order scheme, which is presented to the reader in Section 3, is as compact as the first order one, which make it really suitable for a parallel solver. Since only the closest cell neighbors are needed, parallelization only requires one single layer of cells in the overlap region between contiguous processors. The diffusive fluxes are discretized with a standard Galerkin approach [6], while the source terms are upwinded together with the convective fluxes.

The resulting system of discrete equations is driven to steady state convergence by means of a fully implicit first-order accurate in time backward Euler scheme, which employs a preconditioned Generalized Minimum Residual (GMRES) algorithm [30] to solve linear systems arising from the corresponding Newton linearization. The parallel implicit numerical solver has been implemented within COOLFluiD[†], a collaborative multi-physics platform [16, 17] developed at the Von Karman Institute for Fluid Dynamics during the last ten years.

The article will be organized as follows: first, the multi-temperature thermo-chemical nonequilibrium model is reviewed; second, the residual distribution method is described

[†]Computational Object Oriented Libraries for Fluid Dynamics