8th. World Congress on Computational Mechanics (WCCM8)
5th European Congress on Computational Methods in Applied Sciences and Engineeering (ECCOMAS 2008)
June 30 –July 5, 2008
Venice, Italy

COUPLING RADIATION AND CONVECTION: EFFECT OF RADIATION MESH ON BOTH RESULTS AND PERFORMANCE

Guillem Colomer[1], Oriol Lehmkuhl[1,2], Ricard Borrell[1,2] and Roser Capdevila[2]

[1] Termofluids, S. L.
Magi Colet 8,
08204 Sabadell (Catalonia, Spain)

[2] Centre Tecnològic de Transferència de Calor (CTTC)
Universitat Politècnica de Catalunya (UPC)
ETSEIAT, C. Colom 11, 08222 Terrassa (Barcelona), Spain
cttc@cttc.upc.edu, http://www.cttc.upc.edu . . .

Key Words: *Radiation, Combined Problems, Computational Cost Reduction, Mesh Coarsening, Unstructured Grids, Discrete Ordinates Method*

ABSTRACT

In this work, the attention is focussed on radiative heat transfer, its modelling using the Radiative Transfer Equation (RTE), and its coupling to the general energy equation, which accounts for convection and conduction as well, via an additional source term. Normally, solving the RTE by whatever method available —discrete ordinates, discrete transfer, montecarlo. . . — requires a lot of computational power (in terms of time and memory), due to the directional nature of radiation in general, even in the case that the medium is modeled as simple as possible. On the other hand, when radiation plays only a part in the total energy transfer, a very precise solution of the RTE is not needed, and therefore some computational resources can be saved.

It is worth noting that the reduction of the computational cost, by any means, is of paramount importance when ambitious simulations of complex, multiphysics problems, relevant to the industry are performed. In situations in which radiative heat transfer plays a significant role —furnaces, combustion chambers. . . the computational demands associated to the radiative terms could become very important, depending on the modelling details. For instance, when accounting for the non gray behavior of real gases, the calculation of radiation terms can easily take the majority of the computation time (the RTE may have to be solved tens of times), unless some measures are enforced.

Specifically, our aim is to reduce such overhead in cases where combined heat transfer is studied, in order to be able to simulate real-life problems, which require fine meshes for grid-independent solutions. Therefore, the main approach we adopted to achieve this purpose is to coarsen the mesh used to account for the radiative terms, and interpolate the corresponding magnitudes. The same strategy was used before [1] for simlple cases in cartesian grids, yielding promising results, in the sense that a great reduction of control volumes was achieved. Moreover, by fine tunning the global solution algorithm, and solving the RTE only when the temperature has changed significantly, the calculation time can be decreased even further.

Encouraged by the results of the previous work, our interests are now adressed to unstructured grids. There are various procedures capable of producing a coarse mesh from a fine one:

1. we can simply form groups of cells by adjoining cells that are close together —a geometrical coarsening, where a grouping according to the length scale given by $1/\beta$ (the inverse of the extinction coefficient), can be done

2. taking into account only the algebraic matrix into which the RTE is discretized —a pure algebraic coarsening (multigrid)

3. by constructing a new, different mesh using some of the vertices of the original one, as in [2], and preserving the boundaries as much as possible to minimize interpolation-extrapolation problems in the information transfer from one mesh to the other.

There are also problems specific to unstructured grids, like the convexity problem (having control volumes with indentations, which can be produced by following the procedure 1, can severely affect convergence), that should be carefully dealt with. In order to avoid such problems, procedure 2 is preferred. The main drawback of procedure 3 is that a new mesh has to be generated.

To assess the feasibility of the above mentioned, cost reduction ideas, we have implemented a RTE solver which consists on discretizing both the angular domain —using either the Discrete Ordinates Method (DOM) or a finite volume discretization of the unit sphere, and the spatial domain —using a finite volume, unstructured mesh with arbitrary cell shapes. Given the nature of the spatial mesh, explicit solvers for the RTE [3], which perform the best, are not used, since their implementation is too involved for complex geometries. Instead, parallel iterative solvers (GMRES and BICGSTAB), better suited to unstructured grids, are employed on a low cost PC cluster. These solvers are also used reliably in a variety of problems [4].

In the present study, several situations are considered, for which the relative importance of the different terms in the energy equation ranges appropriately. Both transparent and non transparent media are taken into account, for the coarsened mesh size is physically conditioned by the extinction coefficient β if it is not zero. Although we use the DOM to solve the RTE, along with a finite volume solver, the conclusions still apply if similar methods are used.

REFERENCES

[1] G. Colomer, M. Costa, R. Cònsul, J. Cadafalch. *Numerical simulation of convection in radiative participating media*, Proceedings of the XIV Congreso Nacional de Ingeniería Mecánica (in spanish), pp. 1691–1696, Leganés (Spain), December 2000.

[2] C. Ollivier-Gooch. *Coarsening unstructured meshes by edge contraction*, Int. J. Numer. Meth. Engng, Vol. 57, pp 391–414, 2003.

[3] P. J. Coelho, J. M. Gonçalves, M. G. Carvalho, D. N. Trivic. *Modelling of radiative heat transfer in enclosures with obstacles*, Int. J. Heat Mass Transfer, Vol. 41, nos 4–5, pp. 745–756, 1998.

[4] O. Lehmkuhl, C. D. Pérez Segarra, R. Borrell, M. Soria, A. Oliva. *TERMOFLUIDS: A new Parallel unstructured CFD code for the simulation of turbulent industrial problems on low cost PC Cluster*, Proceedings of the Parallel CFD Congress, Antalya (Turkey), May 2007.