11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20–25, 2014, Barcelona, Spain

NUMERICAL SIMULATION OF TRANSPIRATION COOLING WITH A MIXTURE OF THERMALLY PERFECT GASES

Wolfgang Dahmen[1], **Siegfried Müller**[1] **and Michael Rom**[1*]

[1] Institut für Geometrie und Praktische Mathematik, RWTH Aachen University, Templergraben 55, 52056 Aachen, Germany, www.igpm.rwth-aachen.de

Key words: *Transpiration Cooling, Porous Medium Flow, Darcy-Forchheimer Equation, Compressible RANS Equations, Coupled Finite Element-Finite Volume Scheme.*

A coupled finite element-finite volume scheme for the numerical simulation of transpiration cooling of a hot gas flow in a channel is presented. The motivation for this work is the development of more effective cooling strategies for the reduction of heat loads in future space transportation systems such as rocket thrust chambers. Since passive cooling techniques, e.g. radiation cooling, are limited and nonadjustable during flight, transpiration cooling as an active solution might offer a promising alternative. In particular, such a technology would allow for the reduction of the weight of the structural component. The basic idea of transpiration cooling is to inject a coolant through a porous material into the hot gas flow boundary layer such that the temperature boundary layer thickens and a coolant film develops. This leads to a significant reduction of the heat load at the wall. The coolant is driven through the porous material by a pressure difference between the coolant reservoir and the turbulent hot gas flow.

For the configuration of cooling gas injection through a porous material into a hot gas channel flow, experimental data are available from Langener et al. [1] using ceramic matrix composite (CMC) materials, in particular composite carbon/carbon (C/C) materials, and air or argon as cooling gas. Argon as a nonflammable and nontoxic gas has been chosen to investigate the cooling effect of a gas which has a higher molecular weight than air.

The simulations are carried out using separate solvers for the hot gas flow and the flow in the porous medium, respectively. First results for air as coolant, modeled as a thermally and calorically perfect gas, are documented in a technical report by Dahmen et al. [2]. As an advancement, the hot gas in the channel is now modeled as a mixture of thermally perfect gases. The two solvers are directly coupled with each other through alternating data exchange at the interface. Since the goal is to achieve a steady state solution, the coupling is realized in a weak sense, i.e., both solvers are applied alternately and converged to a steady state with respect to the coupling conditions generated from the solution of the other solver at the particular step of the iteration process. This process is continued until the pressure difference at the interface is smaller than a threshold.

Concerning the hot gas flow, the adaptive parallel solver Quadflow [3] is used. It solves the compressible Reynolds-averaged Navier-Stokes equations using the Menter SST turbulence model [4]. The core ingredients are (i) the flow solver concept based on a finite-volume discretization, (ii) the grid adaptation concept based on wavelet techniques and (iii) the grid generator based on B-spline mappings. All three modules are interleaved and frequent data exchange is required during the solution cycle.

The porous medium flow is modeled by the continuity equation, the Darcy-Forchheimer equation and two temperature equations for both fluid and solid material. This model is discretized by a finite element scheme using the deal.II library [5].

Both numerical solvers and the coupling are validated by comparing the simulation results with experimental data. This is done for the injection of different cooling gases into the hot gas channel flow using varying blowing ratios and miscellaneous hot gas flow conditions regarding the temperature and the Mach number.

REFERENCES

[1] T. Langener, J. von Wolfersdorf, M. Selzer and H. Hald. Experimental investigations of transpiration cooling applied to C/C material. *Intl. J. Thermal Sc.*, Vol. **54**, 70–81, 2012.

[2] W. Dahmen, T. Gotzen and S. Müller. Numerical Simulation of Transpiration Cooling through Porous Material. IGPM Report 374, 2013.

[3] F. Bramkamp, Ph. Lamby and S. Müller. An adaptive multiscale finite volume solver for unsteady and steady state flow computations. *J. Comp. Phys.*, Vol. **197**(2), 460–490, 2004.

[4] F.R. Menter. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. *AIAA J.*, Vol. **32**(8), 1598–1605, 1994.

[5] W. Bangerth, R. Hartmann and G. Kanschat. deal.II - a general-purpose object-oriented finite element library. *ACM Trans. Math. Softw.*, Vol. **33**(4), 24/1–24/27, 2007.