Numerical simulation of transpiration cooling through porous media

W. Dahmen, V. Gerber, T. Gotzen*, S. Müller

*Institut für Geometrie und Praktische Mathematik
RWTH Aachen
Templergraben 55
52056 Aachen
gotzen@igpm.rwth-aachen.de

ABSTRACT

In combustion chambers of rocket engines the walls are exposed to high temperatures. In order to avoid material damage effective cooling concepts are necessary. Transpiration cooling using ceramic matrix composite (CMC) materials is an innovative concept to reduce the heat load of the wall. However, there is still a severe lack of understanding concerning the fundamentals of both the hot gas and the porous media flow and the thermal interaction.

Transpiration cooling applied to rocket engines is based on the idea of using porous materials for the combustion chamber lining instead of solid materials with boreholes or slits for the cooling gas injection. Therefore, a considerably large amount of heat is absorbed by the coolant already inside the porous wall and carried away by convection. Furthermore, the developing coolant film is very homogenous and sufficient blowing ratios are rather small. Additionally, the porous material might also have advantages concerning weight and structural integrity.

The effectivity of such cooling concepts crucially relies on the comprehension of the mass flow through the porous media and the interaction of the cooling gas with the attached hot gas flow field. A direct numerical simulation (DNS) that adequately resolves all geometric and corresponding flow scales including the porous material is far beyond the capacity of present and near future computer technology.

Therefore in both flow regimes numerical models are used which reduce the complexity of the problem. Concerning the hot gas flow, the compressible Reynolds-averaged Navier Stokes (RANS) equations are solved using the Menter SST model. To obtain results in affordable computing time, the fully adaptive numerical solver QUADFLOW is used, which has already been applied to film cooling applications before [1]. The flow inside the porous material and the temperature distribution in both the solid and the fluid part of the porous material are modeled by combining the continuity equation and the Darcy Forchheimer equation with a two temperature model. The two solvers are then weakly coupled using coupling conditions on the interface between the two integrational domains.

The combined solver is used to simulate the coolant injection through a carbon/carbon material mounted in the sidewall of a subsonic hot gas channel. The coolant (air) is driven through the porous material by a pressure difference between the coolant reservoir and the turbulent hot gas flow, which is characterized by a Mach number of $Ma = 0.5$ and a hot gas temperature of $T_{hg} = 540\,K$. Simulations with different blowing ratios obtained by different pressure differences, see [2], are compared with experiments from Langener et al. [3]. The numerical investigations confirm that the porous media solver adequately reproduces the temperature distribution and the mass flow through the porous material. Nevertheless, there is a strong dependence on parameters like the heat exchange coefficients, which have yet to be determined by experiments.

REFERENCES

[1] Dahmen, W., Gotzen, T., Melian, S., Müller, S., Numerical simulation of cooling gas injection using adaptive multiresolution techniques, *Comp. Fluids* **71**, pp. 65–82 (2013).

[2] Numerical Simulation of Effusion Cooling through Porous Media, Dahmen, W., Gerber, V., Gotzen, T., Müller, S., SFB/TRR40 - Annual Report 2012, pp. 1–15.

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