Heat transfer in porous media applied to liquid rocket engines

Y. Khamis*, D. Suslov*, B. Desmet**

*Institute of Space Propulsion, German Aerospace Center (DLR),
74239 Hardthausen, Germany
**Université de Lille Nord de France, UVHC, TEMPO EA4542, Campus Mont-Houy,
59313 Valenciennes Cedex 9, France

Abstract

Advances in space propulsion require the use of new materials. The Institute of Space Propulsion at DLR Lampoldshausen (Germany) has been investigating for the application of porous materials in liquid rocket engines. The development of transpiration cooling for combustion chamber is based on such materials in a configuration where hydrogen flows through the wall. Another application is injector heads whish offer better homogenization of the propellants. The efficiency and reliability of such components depend on the permeability of the flow through the porous structure and the heat exchange between the fluid and the porous material. A description of those physical processes is necessary.

A new experimental set up is in development with requirements to reproduce and to control the flow and the heat transfer in a porous material at high temperature, high pressure for cryogenic propellants. It gives the possibility to study the influence of the properties and the structure of the porous material, the properties of the fluid, the parameters of the flow and the heat loads. Appropriate measurements in the fluid and on the material allow determining the pressure loss and the thermal exchanges in the porous sample. The main objective is to develop appropriate models of permeability and heat transfer for real rocket components.

This work presents the investigation made to develop such an experimental device. The design and the methodology of the measurements chosen are explained. The first experimental results and the numerical analysis based on the thermal equilibrium model are also presented.

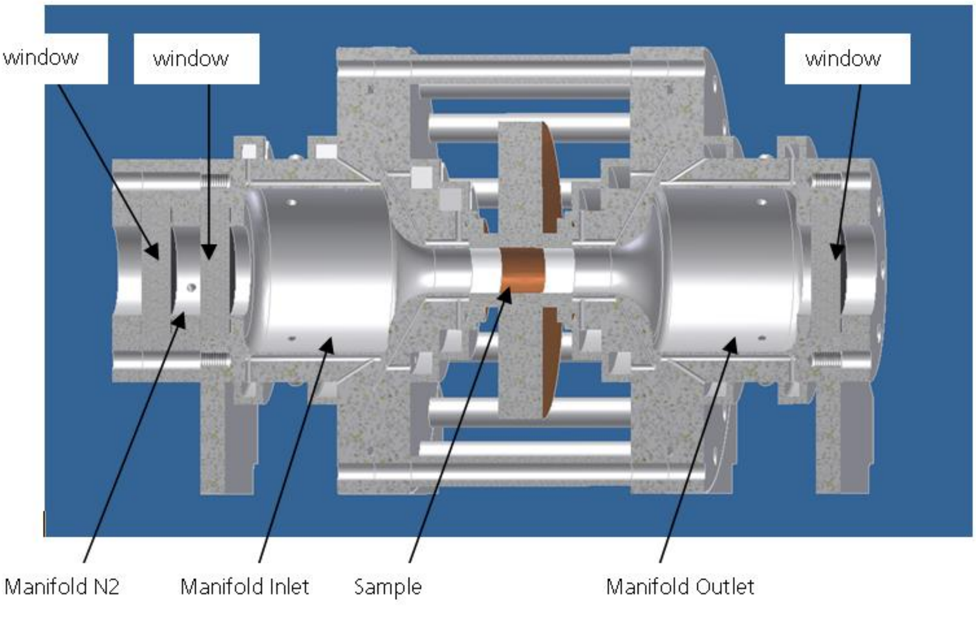
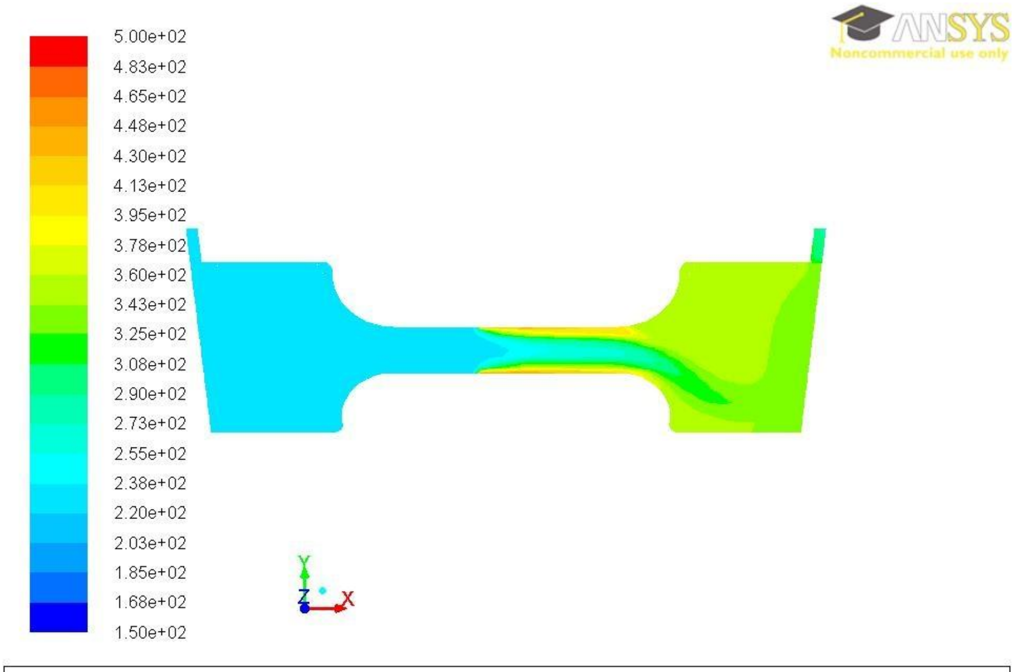


Figure 1: Structure of the experimental set-up



Contours of Static Temperature (k)

Figure 2: CFD analysis