Nitrogen Two-phases Flow in Regenerative Cooling System of Rocket Engine

Y. Torres, D. Suslov and O. Haidn*
German Institute of Space Propulsion (DLR),
Langer Grund, D – 74239 Hardthausen, Germany
** Actually in EADS Astrium ST,Lampoldshausen, yohann.torres@astrium.eads.net*

Abstract

Start processes for rocket engines and more particularly for expander cycles are investigated with particular attention. The regenerative cooling system uses propellants as coolant, which flows through milled cooling channels in the chamber walls. During the start sequence, the propellant is still liquid and can evaporate in the cooling channels. Numerical simulations are useful to quantify this phenomenon. Experimental tests with liquid nitrogen were performed in order to validate numerical models, with an asymmetrically heated channel specimen.

1. Introduction

For an optimal design of the cooling systems of rocket engine, with minimal hydrodynamic losses, a precise knowledge of the heat transfer processes in the rocket combustion chamber and in the cooling channels is necessary.

3D-numerical simulation could be considered as the most efficient method to determine mechanical, thermal, and pressure loads on a structure, with relatively minimal expense and requirement of time. The main advantage of this computing method is the universal character and the relative precision of the results, which increases with the recombination of several sources of information such as experiments. To test the robustness of the numerical model, the influence of several parameters has to be characterised and evaluated, like boundary conditions, turbulence models, wall treatment, grid precision, fluid models, roughness, multiphase flow models, etc. Nevertheless, numerical results have to be verified by experimental data.

This study focuses on an experimental test case that concerns evaporation of coolant in asymmetrically heated cooling channel with high aspect ratio. The experimental results (temperature measurements) are compared to different numerical models (turbulence models) of multiphase flows. As conclusion, numerical models deliver different results and they are more or less adapted to characterize the heat flux in cooling channel with phase change (evaporation inside the channel).

A test specimen was built in order to create high heat fluxes in a single cooling channel, allowing implementation of many sensors, which is not possible in real engine configuration. This single cooling channel in mounted on a copper block, which is electrically heated and delivers heat through conduction to the channel. A special profile separating the block and the channel (thermal nozzle) allows reaching high asymmetrical heat flux density.

2. Experimental Set up

2.1 Test specimen

In reference of a DLR-patent [1], a “thermal nozzle” brings asymmetrical heat flux density into a throat section. This principle was adapted from a *3*-dimensional design to a *2*-dimensional one, allowing the machining of a cooling channel at the throat section. A single cooling channel can be mounted on the thermal nozzle at the throat section, in order to provide controlled high and homogeneous heat flux density. The single channel concept allows better sensor implementation in comparison with real engine configurations.

The dimensions of the channel refer to the cooling channels of the combustion chamber HARCC, High Aspect Ratio Cooling Channels [3], with the aspect ratio of *9.2*:

- Height of a channel: $h_c = 4.6mm$
- Channel width: $w_{ch} = 0.5mm$
- Channel cross section: $S = h_c \cdot w_{ch} = 2.3mm^2$
- Wall width (fin width): $w_w = 0.7mm$
- Length of the cooling channel: $l_{tot} = 188mm$

The lower part (manufactured in copper alloy) of the test specimen is heated by cylindrical electrical heaters (cartridges), see figure 1. Afterwards, the “thermal nozzle” concentrates the heat flux through the thinnest section where a cooling channel is milled. Geometrical parameters of the cooling channels can be machined identical to rocket cooling channels, representative fluids can be used as coolant (nitrogen was used for this 2-phases flow investigation), relative high asymmetrical heat flux density can be achieved and quite homogeneous heat flux density can be provided at the throat section. Moreover, a very precise control of the heat flux can be assured by an electrical regulation system.

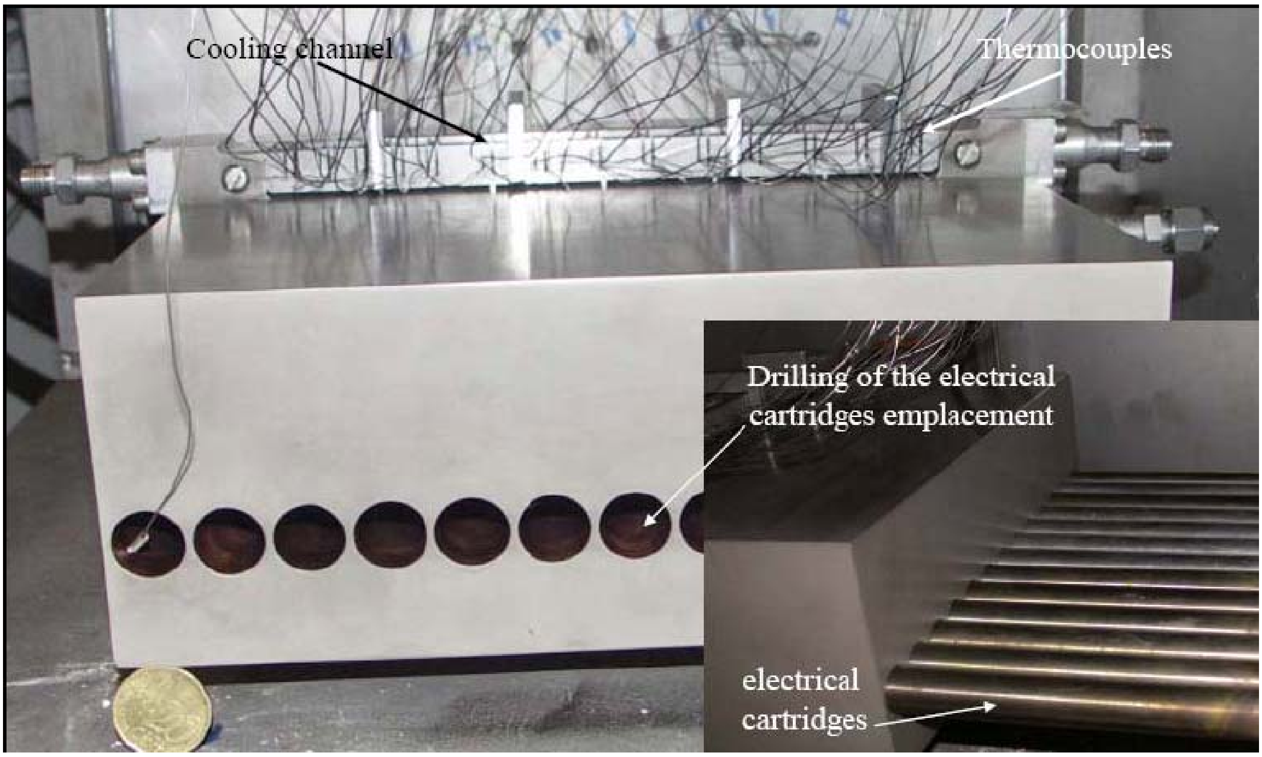


Figure 1: picture of the test specimen, in which electrical heaters are implemented.

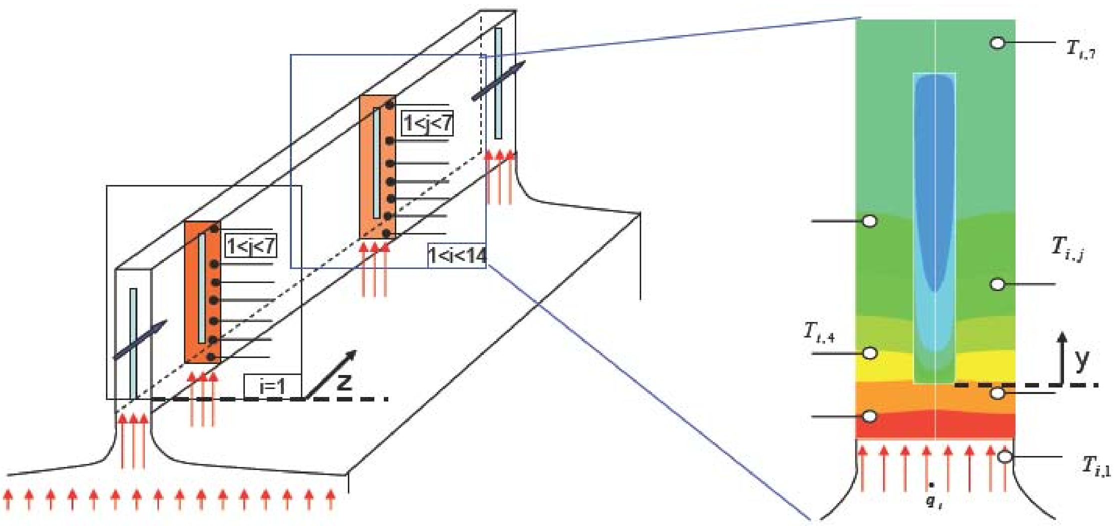
2.2 Instrumentation

The cooling channel is equipped with *98* thermocouples with a diameter of *0.5 mm*. To fix the position of the thermocouples, small holes of *0.1 mm* depth were drilled in the outer wall of the channel. The thermocouples positions are divided into *14* sections along the channel length and *7* heights in the channel wall. Figure 2 shows the positions of the sensors in the *14* axial and the *7* radial positions. The origin of the coordinates is taken at the entrance of the channel and at the bottom of the channel. Along the channel length, *15* D_h or *10* D_h separates two successive axial positions.
Moreover, thermocouples and pressure sensors were implemented at the inlet and the outlet of the channel inside the fluid to control the thermodynamic state of nitrogen before and after heating.
An estimation of the measurement error was done, taking account several sources of error:

- Influence of the reference temperature (cold junction) on the thermocouples accuracy
- Thermocouple positioning in the structure
- Acquisition Error
- Temporal stochastic error

By adding all the errors sources, a global and maximal measurement error was calculated and estimated at 12 K for this test case. The position of the thermocouple brings the main component of this error calculation, but it is quite limited due to the implementation of sensors, which are parallel to the iso-temperature lines.

$$\Delta T_{i,j} = \sum_k \left(\Delta T_{i,j}\right)_k < 12K$$



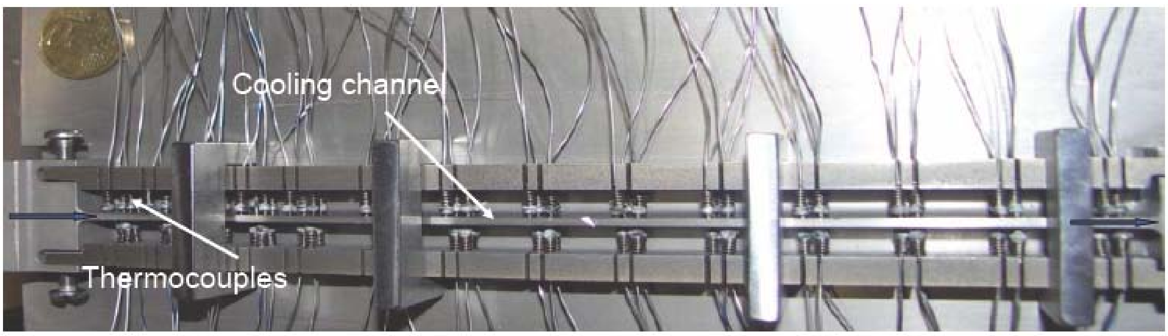


Figure 2: Implementation of the 98 thermocouples in the test specimen walls.

2.3 Test conditions

A test bench was used in order to cool down nitrogen, to control the mass flow and to stabilize the pressure at the inlet of the channel.
At the inlet of the channel, the liquid nitrogen is under 24 bar and has the uniform temperature 80 K. Along the channel, the asymmetrical heat flux is about 4 $MW.m^{-2}$, delivered by electrical heaters delivering a constant temperature in the block (190°C). The inlet Reynolds number of liquid nitrogen reaches $2.2 \cdot 10^4$. The mass flow rate of nitrogen is 8.5 $g.s^{-1}$.
Thermal and dynamic steady states were reached during the tests when the following heat and flow characteristics do not depend on time (less than 1% variation over time): mass flow rate, inlet temperature, outlet temperature, inlet and outlet pressures. A test requires 10 seconds, time-averaged values are provided for the mass flow rate, the fluid temperatures and pressures at the inlet and outlet, and for all temperatures delivered by the wall thermocouples.

3. Numerical simulations

The channel and the block were modelled and meshed with the help of commercial CFD software. The detailed numerical investigation is available in a publication of the same authors [2]. A mesh with 1 million nodes was used for this numerical investigation, where the channel and the block were modelled. The y+ repartition (between 8 and 120) indicates an appropriate mesh for both phases, liquid and gas.
Concerning the two-phases model, an Eulerian multiphase flow model was used and applied to nitrogen (vapour and liquid) with the application of the Redlich Kwong equation of state. This Eulerian model is homogenous, which allows to adapt easily the turbulence models to the multiphase flow problem. The Reynolds-Averaged Navier-Stokes equations (RANS) have been used in this CFD investigation. Turbulence has been here modelled with two-equations (k-ε, k-ω) closure equation systems and seven-equations models (Reynolds Stress Model). Except from the resolution of the Reynolds stresses, the particular BSL RSM “Baseline Reynolds Stress Model” has been used here and is based on a ω- equation, but integrating blending coefficients from the k-equation.

The boundary conditions used for the numerical model refer to the time-averaged experimental conditions:

- mass flow rate 8.51 g/s at the inlet (4.25 $g.s^{-1}$ for the modelled half of channel)
- pressure outlet at 21.4 bar
- symmetry on the median lines
- constant temperature 190°C at the lower wall of the block, where the cartridges are implemented
- adiabatic condition on the others extern walls
- nitrogen as material, real gas nitrogen as gas, liquid nitrogen as fluid
- copper alloy as solid

Three different turbulence models (k-ε, SST, RSM) were investigated and present similarities and differences concerning the evaporation of liquid nitrogen inside the cooling channel. Figure 4 presents the fraction of liquid along the channel length for the three turbulence models.
For all models, more than 70% of the mass was changed into gas at the end of the channel. The evolution of the evaporation zone is different, see figure 3: the k-epsilon and SST models expect a local and sudden phase change directly after the entrance of the channel (30 mm after the inlet). For the RSM model, the phase change really appears at the end of the channel: about 120 mm after the inlet of the channel. For the structure temperature, each model predict different temperature profile, depending mainly where the phase

change appears and how it develops along the cooling channel. The numerical temperatures are read, where the thermocouples are located in reality.

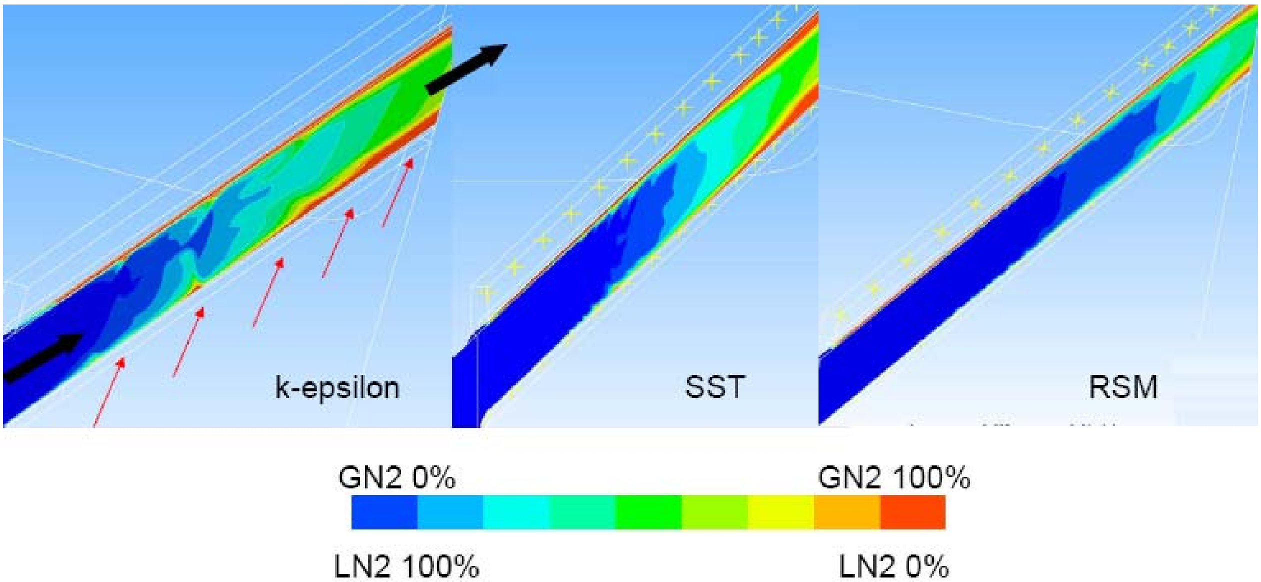


Figure 3: Mass fraction of liquid nitrogen along channel length for different turbulence models.

4. Results

A direct comparison between experiments and numerical simulation can be done. In figure 4, only two thermocouples lines ($T_{i,2}$ and $T_{i,6}$) are depicted in order to improve the clearness of the diagram. The points are the measured temperatures in the channel wall and the lines are the simulated temperature for different turbulence models.

The profile of the measured temperature (increase and then decrease of the wall temperatures) can be partially explained by the latent heat of nitrogen evaporation inside the channel, increasing the local heat transfer. By keeping constant hot temperature in the block, the channel temperatures decrease while heat flux increases. This temperature decrease was predicted by numerical simulations too, but it mainly appears where the main evaporation zone takes place: quite early and sudden for the k-epsilon model, early and progressive for the SST, late and progressive for the RSM, see figure 4.

All simulated temperatures were compared to the experimental temperatures. A least square method was applied over the 98 temperatures to illustrate the difference between experiments and simulations:

$$\Delta T_{num-\exp} = \frac{1}{98}\sqrt{\sum_{i,j}\left(T_{i,j}^{EXP} - T_{i,j}^{NUM}\right)^2}$$

The results are presented in table 1.

	k epsilon	SST	RSM
$\Delta T_{num-exp}$[K]	2.10777043	2.71758091	3.15922544

Table 1: Least square calculation between measured and simulated temperatures.

Clearly, the k-epsilon model presents the less temperature deviation with the experiments. It corroborates the fact that the evaporation zone is local and situated directly at the entrance of the channel, as it is described in figure 4.

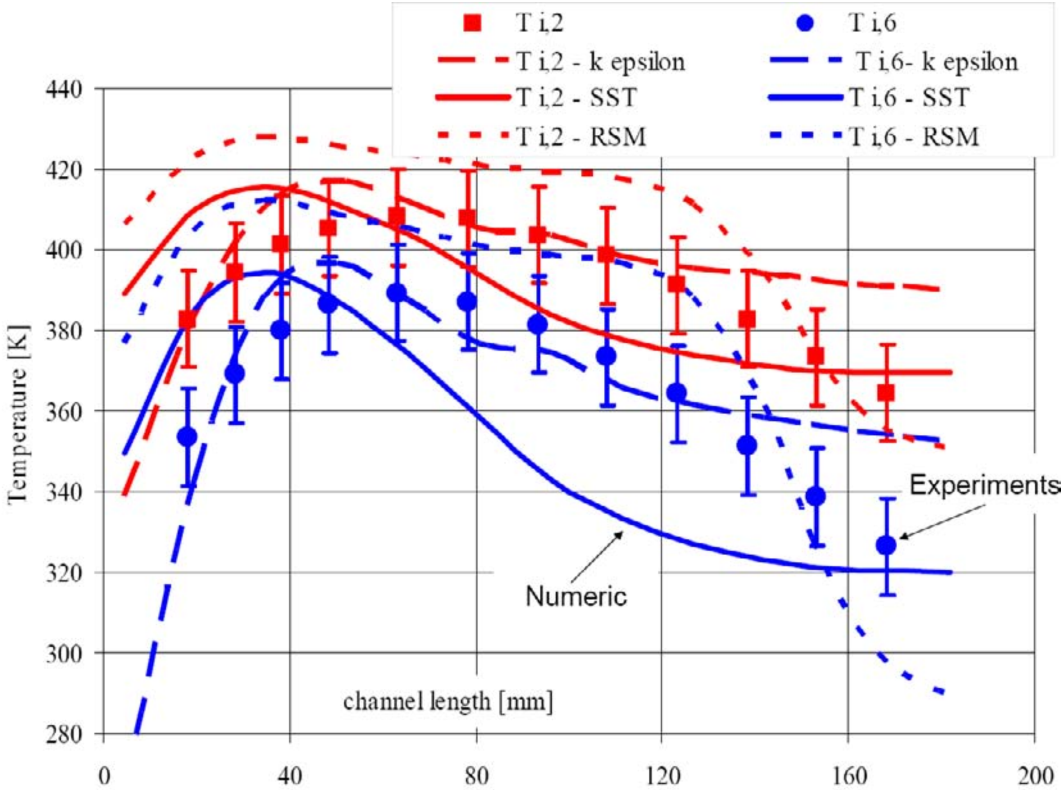


Figure 4: Experimental and numerical temperatures along the channel length at 2 different channel highs.

Concerning the pressure losses, the experiment delivers a pressure drop of 3 bar over the channel. The three turbulence models underestimate this pressure loss, but the RSM model is 50% false, see table 2.

	k epsilon	SST	RSM
P_inlet Simulation [bar]	23.54	24	22.94
P_outlet Simulation and Experiments[bar]	21.4	21.4	21.4
ΔP_{num} [bar] = P_{inlet}-P_{outlet} (Simulation)	2.14	2.6	1.54
ΔP_{Exp} [bar] = P_{inlet}-P_{outlet} (Experiments)	3.0		
ΔP_{exp}-ΔP_{num} [bar]	0.86	0.4	1.46

Table 2: Experimental and numerical pressure drops.

As conclusion, the homogenous Eulerian multiphase numerical model is easy to use and delivers quite reliable results. The k-epsilon model seems to be the most reliable turbulence model to take into account evaporation inside a cooling channel of rocket engine. Of course this investigation can be improved, by using other multiphase numerical models (Lagrange for example), other numerical simulation methods (LES for example), other heat flux processes (developed latent heat transfers), etc.

The experimental results showed that a high asymmetrical heat flux can be obtained with electrical heating, approaching the real heat transfer processes in a rocket engine. This hardware can be used in several configurations, with different coolants in order to investigate precisely different phenomenon impossible to measure in real cooling channels of rocket engines.

References

[1] Suslov, D., Torres, Y., Woschnak, A., Oschwald, M., Vorrichtung und Verfahren zur Erzeugung von Wärmeströmen definierter Wärmestromdichte, 2006-03-02, DLR, European Patent Office, DE102004042901.

[2] Torres, Y., Stefanini, L., Suslov, D., Influence of Curvature in Regenerative Cooling System of Rocket Engine, 2nd European Conference for Aerospace Science, EUCASS, Brussel, July 2007.

[3] Woschnak, A., Suslov, D., Oschwald, M., Experimental and Numerical Investigations of Thermal Stratification Effects, AIAA 2003-4615, 39th AIAA/ASME/SAE/ASEE/JPC Conference and Exhibits.