Influence of Curvature in Regenerative Cooling System of Rocket Engine

Y. Torres, L. Stefanini** and D. Suslov**
** German Institute of Space Propulsion (DLR),*
Langer Grund, D – 74239 Hardthausen, Germany
*** Université de Valenciennes et du Hainault Cambrésis,*
Le Mont Houy, Valenciennes, F- 59300, France

Abstract

Thermo-mechanical loads in rocket engines can be drastically reduced by a reliable cooling system. The regenerative cooling system uses propellants as coolant, which flows through milled cooling channels in the chamber walls. Due to centrifugal forces, dynamic secondary instabilities appear in the cooling channel curvatures, which strongly modify heat transfer characteristics. 3D numerical calculations have been performed in order to compare this heat enhancement with empirical correlations and later with experiments. A panel of turbulence and wall treatment models have been tested to develop a complete numerical data base about asymmetrical (concave side) heat transfer in curved cooling channels.

1. Introduction

For an optimal design of the cooling systems, with minimal hydrodynamic losses, a precise knowledge of the heat transfer processes in the rocket combustion chamber and in the cooling channels is necessary. Nowadays, 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 existing simulations, experiments and empirical correlations. A CFD (Computational Fluid Dynamic) simulation has to be extended over the entire operating domain to test the robustness of the model: the influence of several parameters has to be characterised and evaluated, like boundary conditions, turbulence models, wall treatment, grid precision, fluid models, roughness, etc.

This study focuses on a particular test case that concerns interaction between dynamic instabilities from centrifugal acceleration and asymmetrical heat transfer in a high aspect ratio cooling channel. An actual experimental realisation at DLR delivers the boundary conditions and the geometry of the asymmetrical heated cooling channel.

In reality, the combustion chamber imposes some curvature to the cooling channels, which are milled in the chamber walls. The concave side is heated at the nozzle throat section of the chamber, where the maximal heat flux is reached (up to 80 MW.m-2 for the Ariane 5 radial heat flux main engine). The curvatures impose dynamic instabilities inside the fluid structure and modify the heat transfer conditions and a change of the wall temperature. The centrifugal acceleration induces a pressure gradient from the concave wall to the convex one. By neglecting the centrifugal forces in the near wall region (no velocity region), the variation of the centrifugal forces to pressure gradient ratio in a channel section imply the formation of counter vortices in the entire cross section of the channel: the Dean vortices[1].

These dynamic and centrifugal instabilities can be characterised by a dimensionless number, the Dean number:

$$De = \mathrm{Re} \cdot \sqrt{\kappa} \qquad \kappa = \frac{D_h}{2 \cdot R_c} \qquad \text{(1) (Dean, 1927)}$$

Moreover, highly asymmetrical heating imposes a thermal stratification along the channel, clearly demonstrated by experimental and numerical investigations[15, 13]. The presence of the Dean secondary vortices in concave heated channel perturbs the thermal stratification and enhances locally the mixing ratio. The global heat transfer is then

locally perturbed and the local Nusselt number at the concave side increases (around +25%)[5, 11] although the Nusselt number at the convex side can vary as well (from -10% to +15%)[5], depending on the way of heating (asymmetrical concave heating or constant heating around the perimeter).
Some experimental data has been collected concerning the heat transfer through curved cooling channels, and classic empirical correlations have then been elaborated to characterise the enhancement of the heat flux due to the curvature. Experimental investigations have been provided on rocket engine[3, 6]: Kumakawa and Niino have adapted the initial empirical correlation of Taylor[12]:

$$\frac{Nu_{curved}}{Nu_{straight}} = \left[\mathrm{Re}\cdot\kappa^{2}\right]^{\pm 0.05} \qquad \text{(2) (Taylor, 1965)}$$

$$\frac{Nu_{curved}}{Nu_{straight}} = \left[\mathrm{Re}.\kappa^{2}\right]^{\pm 0.02} \times \left[1+\frac{1}{3}.\sin\left(\pi\cdot\sqrt{\frac{x_c}{L_c+15\cdot D_h}}\right)\right] \qquad \text{(3) (Niino, 1982)}$$

$$\frac{Nu_{curved}}{Nu_{straight}} = \left[\mathrm{Re}.\kappa^{2}\right]^{\pm 0.05\cdot\sin\left(\pi\cdot\sqrt{\frac{x_c}{L_c+37\cdot D_h}}\right)} \qquad \text{(4) (Kumakawa, 1986)}$$

Nevertheless, it can be noted that the extreme thermal, fluid and mechanical real conditions (high heat flux density, cryogenic propellants, complex mechanical construction, etc) bring some difficulties in obtaining similar and reliable experimental results. Experimental realisations have been set up on single cooling channel, electrically heating it to advantage sensors implementation, accessible experimental conditions and reliability[8, 5]. The resulting empirical correlations are more reliable in terms of experimental achievement (large implementation of sensors, reliable results and good repeatability). However, the domain of experimental realisation is far away from real rocket conditions (high heat transfer, cryogenic hydrogen, high aspect ratio of micro-channel). The following correlation comes from experimental investigation on electrical heated channel[11]:

$$\frac{Nu_{curved}}{Nu_{straight}} = \mathrm{Re}^{0.046}\cdot\kappa^{0.1} \qquad \text{(5) (Sturgis, 1999)}$$

Numerical simulations have been performed, through different methods of calculation and for different test configuration (real cooling channels of rocket engine[2, 13], straight channels[10], referent test case[9]). These numerical references have constituted the basis to develop numerical simulations of an experiment at DLR: the EH3C campaign (Electrical Heated Curved Cooling Channels). An interesting comparison can be done between empirical correlations and numerical simulations, which have been performed at DLR of Lampoldshausen. All these numerical simulations have taken the experimental conditions of a campaign that will be published soon. Numerical simulations of the EH3C project have been carried out using three-dimensional CFD calculation, performed with CFX 10.0.
Several parameters have been investigated including grid convergence, fluid and solid model, roughness, some boundary condition parameters and finally turbulence models and near-wall treatments.

2. Numerical simulations

To characterize the turbulent variations, the thermal and dynamical parameters are decomposed into a time averaged component and temporal fluctuation component. Then, the basic equations have been time averaged: the Reynolds-Averaged Navier-Stokes equations (RANS) have been used in this research to model the turbulence. Some closure equations are needed to solve the equations system. The choice of the right turbulence model and the right wall treatment represents one of the most important challenges for engineering numerical applications, to save time costs and to allow representations of dynamics and thermics close to reality. Turbulence has been here modelled with a two-equation (k-ε, k-ω) closure equation system and a seven-equation model (Reynolds Stress Model). The 2-equations closure equation system of turbulence uses the Boussinesq assumption to calculate the Reynolds stress tensor, which is a linear, isotropic and homogenate model. The turbulent component (Eddy viscosity concept) of the viscosity is then directly deduced from the 2 equations closure model:
A large number of scientific and engineering calculations adhering to turbulent flows are established on the k-ε model, complemented by a logarithmic function to patch the core region of the flow in the wall region. The k-ω model[14] seems to be well adapted to conjugate heat and mass transfers. Nevertheless, the k-ω model would present strong sensitivity to free-stream conditions that can be improved by the good compromise of a SST model (Shear Stress Transport). This SST turbulence model uses a blending function to apply the k-ε model in the bulk region and the k-ω inside the boundary layer[7]. Other kinds of closure models do not adopt the Boussinesq assumption and allow a non linear resolution of the Reynolds stress tensor that gives an anisotropic model. This is quite well adapted to

approach recirculation, high shear stress gradient regions and curvatures but needs an important convergent time (7 equations resolution instead of 2) and the convergence of calculations is not so stable as for the 2-equations models. 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.
There are two different approaches for the near-wall treatment. First, there are "wall functions", in which the viscosity-affected inner region is not numerically calculated but replaced by semi-empirical formulas to bridge the viscosity-affected region between the wall and the fully-turbulent region. The other approach is called Low-Reynolds Model, in which the turbulence models are modified to enable the viscosity-affected region to be resolved with a precise mesh near the wall, including the dynamic resolution of the viscous sublayer (where Reynolds number are low). This method is more precise than the logarithmic approximation but requires much more cells near the wall domains.

3. Numerical Model

The curvature effects on asymmetrical heat transfer in rocket cooling channels are investigated numerically and experimentally at the DLR of Lampoldshausen with the project EH3C (Electrical heated curved cooling channels). The experimental set up concerns manufactured test specimens: one single straight channel and one curved concave channel to study the influence of Dean vortices on heat transfer. The experimental conditions are not fully representative of the real engine channels conditions: the pressure level is one magnitude under reality (40 bar instead of 300), as well as heat flux density: 20 $MW.m^{-2}$ instead of 80 $MW.m^{-2}$ at the throat section of the Vulcain 2 main engine of Ariane 5 launcher. However, the medium is supercritical hydrogen at low temperature at the inlet of channel (80K) and the EH3C channels respect real geometry of real engine channels[15] (high aspect ratio channels). The obtained Reynolds number in this test case reaches 3.10^5 ($\sim 2.10^6$ for Vulcain configuration) and the corresponding Dean number is around 2.10^4.
This study summarizes numerical simulations obtained to calculate the EH3C experiments. Figure 1 describes the geometry of the 2 test specimens, which have been numerically simulated.

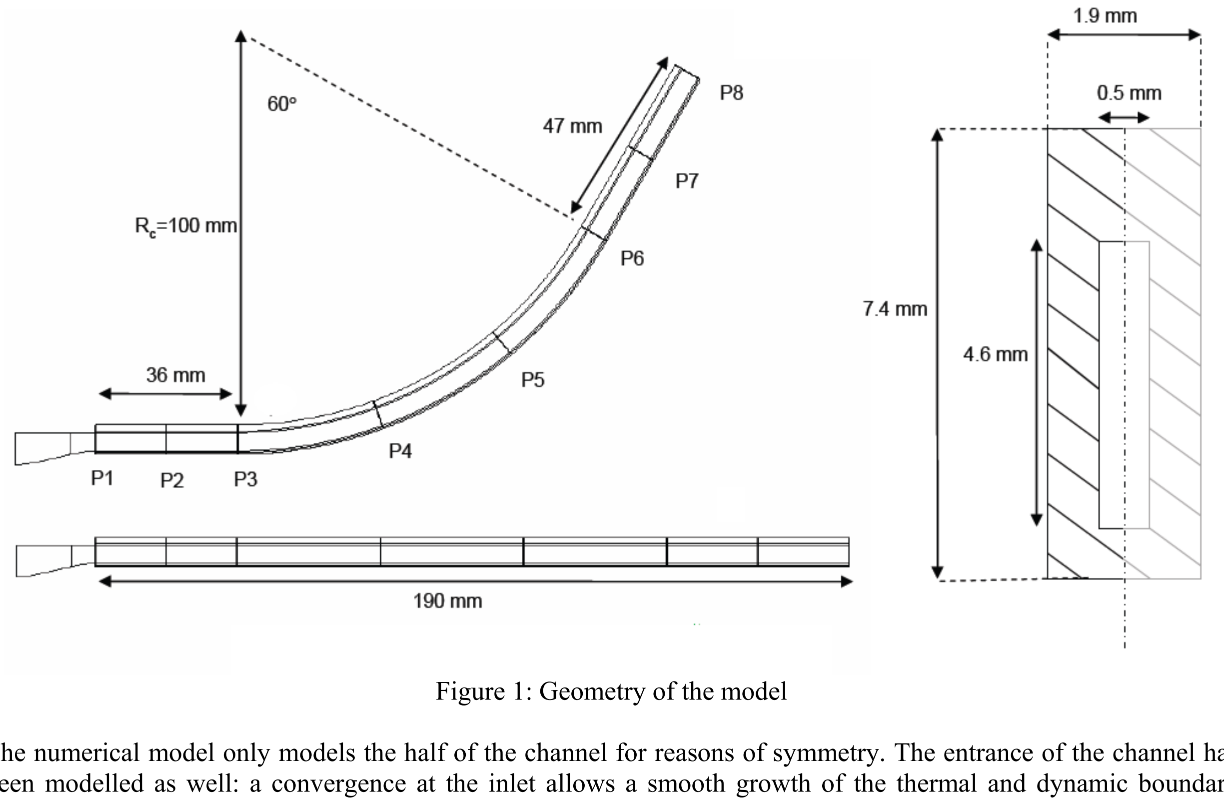


Figure 1: Geometry of the model

The numerical model only models the half of the channel for reasons of symmetry. The entrance of the channel has been modelled as well: a convergence at the inlet allows a smooth growth of the thermal and dynamic boundary layers. Several plane surfaces (P1 to P8) are located along the channel, to illustrate the results.

The boundary conditions used for the numerical model follow the experimental conditions of the EH3C campaign, as it is shown in the figure 2.

- mass flow 4g/s at the inlet ($2g.s^{-1}$ for the modelled half of channel)
- pressure outlet at 15 bar
- symmetry on the median lines
- Heat flux density 20 $MW.m^{-2}$ on the lower wall
- Adiabatic wall on the others extern walls
- Hydrogen as fluid
- Copper alloy as solid
- Inlet pre-mixing turbulence at 1% with an Eddy Length Scale of 0.5 mm
- The roughness has been measured on the manufactured channels and implemented in the numerical model

The inlet turbulence conditions have been changed (from 1% to 10% with varying Eddy length scales) and do not have any influence on the numerical results. The averaged roughness is included in the viscous sublayer, and does not have any influence on the thermal and dynamic behaviour; this has been proven by different numerical simulations with and without roughness.
Several grids with different number of nodes have been designed in order to check the grid convergence. All the calculations used to make this comparison were carried out using a standard k-ε model with a wall function near-wall treatment. The entire grid is structured, both in solid and fluid regions. To ensure better fluid and thermal resolutions in sensitive domains (fluid domains), conformal mesh interfaces have been adopted for the fluid/solid and the fluid/fluid interfaces in the channel. All the solid/solid interfaces are non-conformal to reduce the number of nodes in the solid outer region while keeping the global accuracy of the simulations. Two kinds of grids have been designed, either for using a wall function (first fluid cell in the fully turbulent region of the boundary layer), or for using a low-Reynolds model (around 5 cells inside the viscous sublayer). A first grid, with globally 1 million of nodes, is a 60x15 grid with 530 nodes streamwise in the fluid zone. The solid part (channel wall) has been divided into 2 zones: an inner part in contact with the fluid insuring conform interfaces and an outer part all around. In the inner solid zone, a matching number of nodes was set and the outer solid region uses a coarser grid. Several other grids present alternate distribution of the nodes streamwise with a finer distribution inside the curvature, going from 300 to 1170nodes. This has been done to check the influence of the cell aspect ratio (streamwise length on height of cell) on the results. The change of the streamwise cells distribution does not have a strong influence on the results, and the axial number of nodes has been limited to 530. Another grid has been developed 120x20x530 grid, with a total of 2.5 million nodes and has been designed to observe the influence of the cross section meshing on results. The different heat profiles from the different grids seem to converge with nodes increasing in the cross section, particularly after the curvature. Then, the finest mesh looks appropriate for this numerical model. The Low-Reynolds Model (LRM) requires a finer grid near the walls, so an additional grid has been designed, to have a y^+ between 0.5 and 2 on the entire fluid/solid interface, reaching a grid of 4.5 Millions elements. The details of all investigated grids are presented in the Table 1. The CPU time has been evaluated in hours on a parallel nodes cluster.

	Mesh 1	**Mesh 2**	**Mesh 3**	**Mesh 4**	**Mesh 5**	**Low Re**
Cross Section	60*15*530	60*15*530	80*20*530	80*20*1170	120*20*530	
Nodes [Millions]	1	1	1.7	3.6	2.5	4.5
CPU Time [hours]	5	5	7	17	12	60
y+	12<y+<70	12<y+<70	12<y+<70	12<y+<70	12<y+<70	0.15<y+<2.5

A typical way to analyse results from the numerical simulation is presented here in figure 2. The profile of the heat flux density is drawn around the channel walls. Indeed, for any comparisons, the profile of heat flux density along channel periphery seems to be the most appropriated parameter to compare straight and curved simulations. For this study, only the concave side is heated to represent the hot gas side of rocket engine (see figure 2).

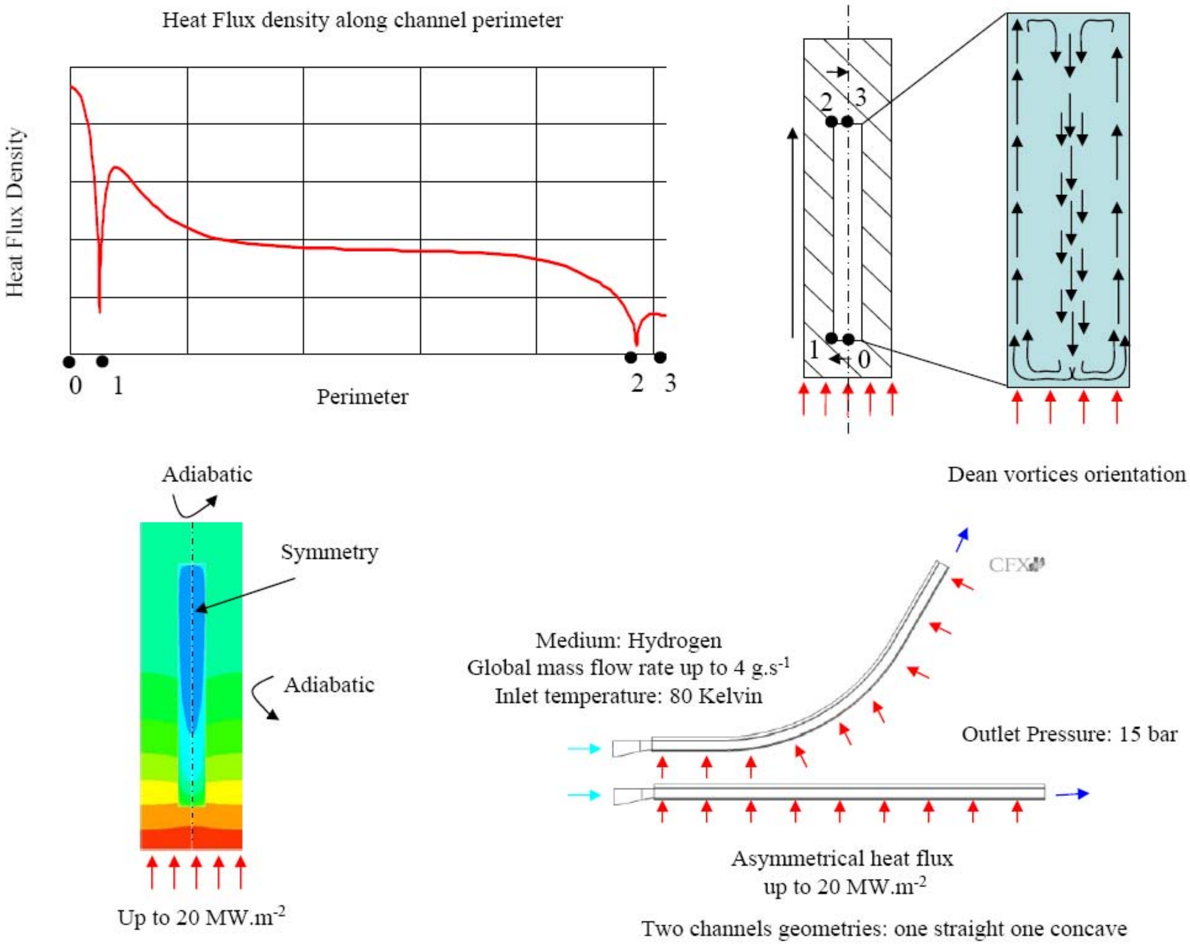


Figure 2: Boundary conditions of the numerical model and typical heat transfer profile along channel perimeter.

4. Effects of Curvature

The effects of a curvature on the heat transfer in a cooling channel are significant. A straight model has been designed exactly with the same geometry, the same grids and the same boundary conditions. The first comparison between the straight and the curved cases were carried out with ideal gas model (hydrogen), constant properties of the solid (copper alloy) and with a k-ε turbulence model.

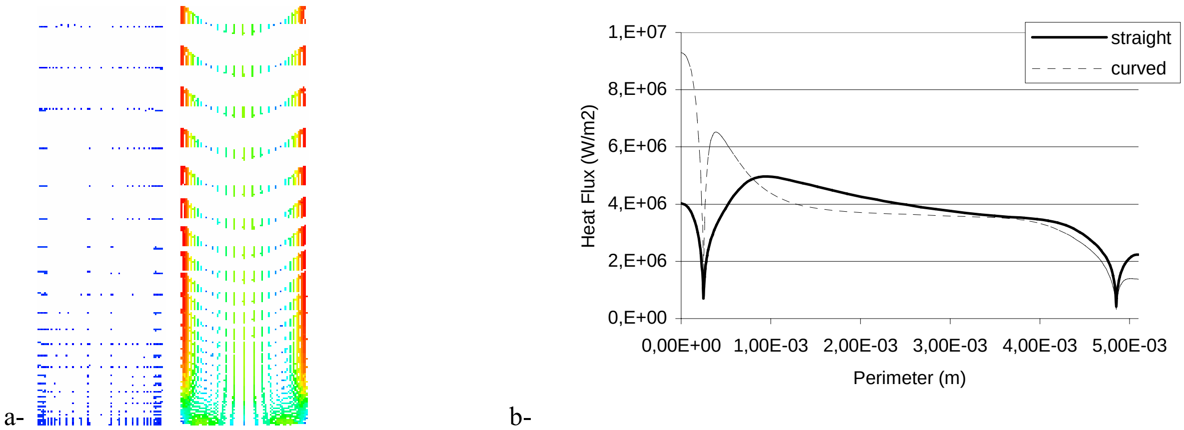
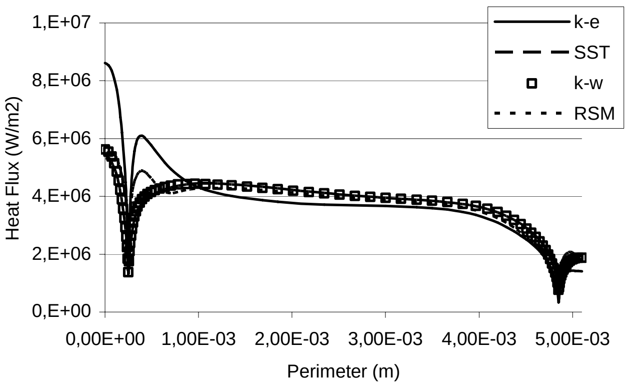


Figure 3: a- Secondary velocities calculated in a straight channel (a- left) and in a concave channel (a- right) (colored by velocity magnitude)
b- Heat flux distribution for straight and curved cases in plane 5

In plane 3, just before the curvature for the curved channel, straight and curved cases provide the same distribution with a difference up to 2 %. The two test cases can be then compared and suppose that the differences are only due to the curvature. Of course, supplementary pressure losses are added for the curved case (pressure losses due to the energy dissipation of the secondary motions) that can explain the small differences. However, the pressure losses for the straight channel are reaching 19.5 bar and for the curved case 19.9 bar, that can be neglected for this case. On the contrary in plane 5, which is located in the curvature, the heat flux distribution is radically different, with a much higher heat flux density on the lower wall (see figure 3-b). Indeed, the heat flux density is enhanced through the concave wall (the heated one in this case) and reduced through the convex wall. The curvature induces Dean vortices: the "cold" fluid coming from the centre of the channel is displaced to the "hot" concave wall, increasing the local heat flux (figure 3 –a). Moreover, the fluid coming from the "hot" wall is moving up along the lateral walls. This explains the decreasing of the local heat flux through the lateral walls near the concave side because of hotter fluid (already heated by the concave side). In this experimental and numerical EH3C case, the global heat flux is supposed constant for any axial sections along the channel and any test configuration (straight or curved). Then, a cooling channel absorbs globally around 7.22 kW, uniformly distributed along its axis. Only the distribution of the heat flux density changes around the perimeter of the channel. The calculations were carried out with a real gas fluid model (Redlich Kwong model). Several turbulence models have been tested in straight and curved channels. Both Near-Wall Treatment models have been investigated, using either SST or RSM turbulence model. The results of the numerical simulation are presented in the figure 4.



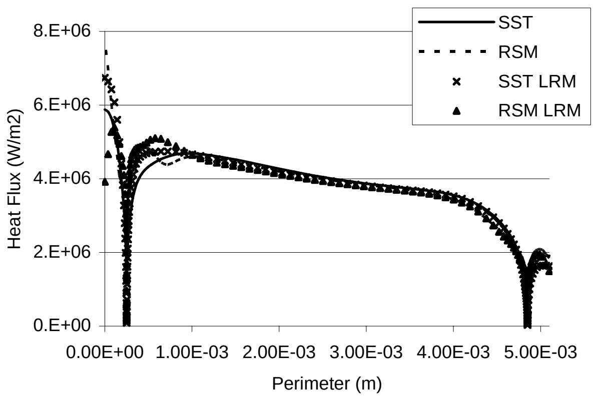


Figure 4: Heat Flux distribution along the perimeter for logarithmic wall treatment (left) and low Reynolds models (right)

All the turbulence models detect the presence of the Dean vortices but indicate several intensities of this secondary motion. Moreover, with the RSM model, some thermal and dynamic oscillations are visible near the corners, which are caused by additional secondary vortices (figure 5). This does not appear for all other 2-equations models. Indeed, in rectangular channels, the orthogonal walls bring shear stress asymmetries. These vortices can be compared to those observed in LES simulations[10] (Large Eddy Simulation) and are in fact very similar in shape and intensity, despite a RANS model is used there. By taking into account the anisotropy of the turbulence, the RSM turbulence models seems to be the most adapted model for the numerical calculations of rectangular channels. In the curvature, the mixing of the Dean vortices and the "corner vortices" are shown in figure 5.

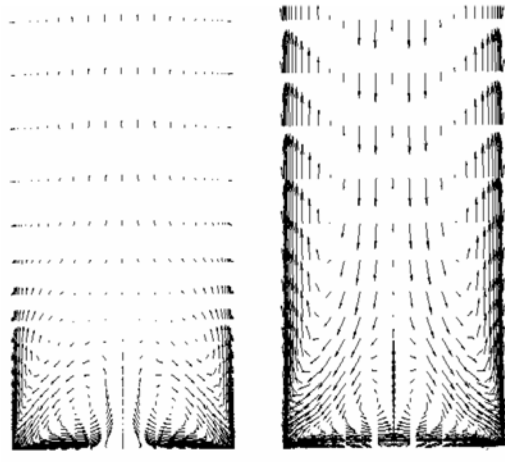


Figure 5: Secondary velocity distribution in plane 5 in straight (left) and curved (right) channel calculated with a RSM model

The Nusselt numbers ratio (curved to straight) has been calculated on the concave side (centre of the channel) by different ways: empirical correlations and numerical simulations (see figure 6). The curvature begins at 36 mm after the entrance and finishes at around 141 mm. The 4 empirical correlations do not converge together and present more than 50% difference. The main defficiency of empirical correlation is the limited expansion and adaptation to different experimental domain (pressure range, fluid, dimensions, heat flux range, temperature ranges, etc). Concerning the numerical simulations, the variation due to turbulence models can reach locally 50%.

The behaviour of the RSM calculation works “in stair” along the channel. It can be explained by the interaction of the “edges vortices” with the Dean vortices. Indeed, it has been noted that the Dean vortices and the upper half vortex from the edge are not distinct anymore (Figure 5) and the “edges vortices” seem to disappear along the curvature (here at the abscissa 80), giving a full access of the Dean cells on the concave side.

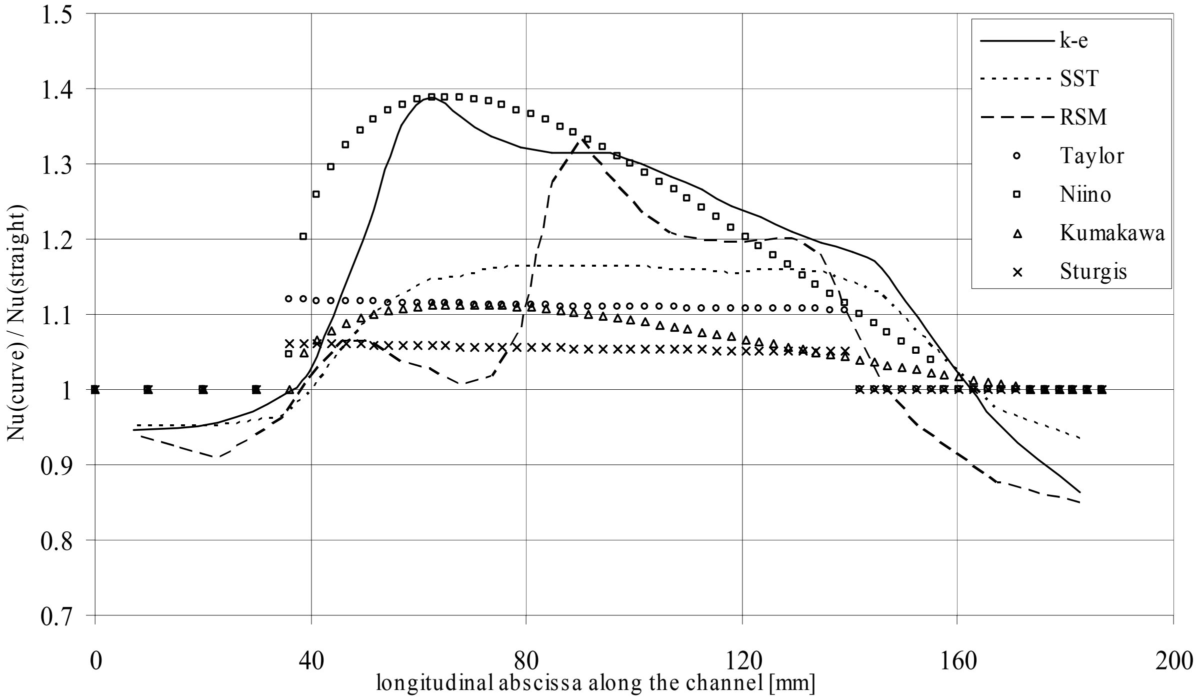


Figure 6: Curve to straight Nusselt ratio along channel length

Further numerical investigations can detect the right model by comparing with other kind of calculations (LES for example). The RSM seems to be the most appropriate model, taking into account “edge recirculation”. Several RSM options have to be tested and improved to check the robustness of these non isotropic models (ε or ω basic equation, BaseLine, Isotropization of Production model, Quasi-Isotropic model…).

Experimental validations would represent the best way to adapt the empirical correlations to this experimental environment and select the right turbulence model for this precise test case: high asymmetrical heat flux in a concave high aspect ratio micro-channel of rocket engine. The experimental realizations (EH3C) are investigated yet at the DLR of Lampoldshausen.

References

[1] Dean, W.R., *Note on the Motion of Fluid in a Curved Pipe*, M.A., Imperial College of Science, Phil. Mag. iv. p. 208, 1927

[2] Fröhlich, A., Immich, H., LeBail, F., Popp, M., Three-Dimensional Flow Analysis in a Rocket Engine Coolant Channel of High Depth/Width Ratio, AIAA 91-2183, 27th Joint Propulsion Conference, June 24-26, 1991, Sacramento, California, USA

[3] Kumakawa, A., Saski, M., Niino, M., Sakamoto, H., and Sekita, T., „Thermal Conduction Characteristics of an Electrically Heated Tube Modeld after the LE-7 Main Burner", presented at the 30th Space Sciences and Technology Conference, October 15, 1986.

[4] Lopes, A., S., Piomelli, U., Palma, J.M.L.M., Large Eddy Simulation of the Flow in an S-Duct, AIAA 2003-0964.

[5] Neuner, F., Preclik, D., Popp, M., Funke, M., Kluttig, H., Experimental and Analytical Investigation of Local Heat Transfer in High Aspect Ratio Cooling Channels, AIAA-98-3439, 34th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, July 13-15, 1998.

[6] Niino, M., Kumakawa, A., Yatsuyanagi, N., Suzuki, A., Heat Transfer Characteristics of Liquid Hydrogen as a Coolant for LO_2/LH_2 Rocket Thrust Chamber with the Channel Wall Construction, National Aerospace Lab of Japan, Ohgawara, Miyagi, Japan. AIAA/SAE/ASME, 18th Joint Propulsion Conference, Cleverland, Ohio, June 21-23, 1982.

[7] Menter, F. R., Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications, *AIAA Journal,* 32(8):1598-1605, August 1994.

[8] Meyer, M., Electrically Heated Tube Investigation of Cooling Channel Geometry Effects, AIAA-95-2500, 31st AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, 1995, San Diego, CA, NASA Lewis Research Center, Cleverland, OH, USA.

[9] Münch, C., Métais, O., Turbulence in Cooling Channels of Rocket Engines, Large Eddies Simulation, Comptes Rendu de Mécanique de l'Académie des Sciences 333 (2005) 574-579.

[10] Salinas Vasquez, M., Métais, O., Large-eddy simulation of the turbulent flow through a heated square duct, J. Fluid Mech. (2002) vol. 453, pp. 201-238.

[11] Sturgis, J.C., Mudawar, I., Single-phase Heat Transfer Enhancement in a Curved, rectangular Channel subjected to Concave Heating, International Journal of Heat and Mass Transfer 42 (1999), pp. 1255-1272, Purdue University, West Lafayette, USA.

[12] Taylor, M.F., A Method of Predicting Heat Transfer Coefficients In The Cooling Passages Of Nerva and Phoebus-2 Rocket Nozzle, NASA Lewis Research Centre, Cleverland, Ohio, AIAA 68-608, 4th Propulsion Joint Specialist Conference, June 10-14, 1968

[13] Wennerberg, J., C., Hogirl, J., Schuff, R., Anderson, W., E., Merkle, C., L., Study of Simulated Fuel Flows in High Aspect Ratio Cooling Channels, AIAA 2006-4708, 42nd Joint Propulsion Conference & Exhibit, 9-12 July 2006, Sacramento, US**.**

[14] Wilcox, D.C, Multiscale model for turbulent flows, *AIAA 24th Aerospace Sciences Meeting, American Institute of Aeronautics and Astronautics,* 1986.

[15] 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.