Numerical investigation of the flow in a high aspect ratio water cooling duct

By T. Kaller, V. Pasquariello, S. Hickel† and N. A. Adams
Institute of Aerodynamics and Fluid Mechanics, Technische Universität München
Boltzmannstr. 15, D-85748 Garching
† Faculty of Aerospace Engineering, Technische Universität Delft
Kluyverweg 1, NL-2629 HS Delft

Current liquid rocket engines are exposed to extremely high combustion temperatures up to 3800 K and heat fluxes up to 160 MW/m². Therefore, efficient cooling mechanisms are required. A major role plays regenerative cooling, where cryogenic propellants are used as coolant flowing through high aspect ratio cooling channels (HARCCs). To study the flow field and the turbulent heat transfer, a combined experimental-numerical test bed for a generic cooling duct has been set up. The duct is operated with subcritical water at $\text{Re} = 111 \cdot 10^3$ and $\text{Nu} = 376$. The main focus is the characterization of the duct flow field and the turbulent heat transfer. The flow is strongly influenced by secondary flow structures, namely corner vortices and Dean vortices. Both play a crucial role for the cooling duct effectiveness. For preliminary numerical investigations a commercial RANS solver was used with different Reynolds stress turbulence models. For a more detailed analysis of the flow field and turbulent heat transfer, our in-house LES solver INCA has been applied to this flow configuration. At first, a detailed grid sensitivity study was performed for a periodic isothermal duct flow. The wall-resolved grid was then used to perform the simulation of the spatially evolving straight and heated cooling duct.

1. Introduction

The main stage of current high performance space launchers uses liquid combustion rocket engines fueled with liquid hydrogen and oxygen. The combustion temperatures of this mixture reach extremely high temperatures up to 3800 K and heat fluxes up to 160 MW/m². To ensure safe operating conditions and structural integrity of the launcher system an efficient cooling of the propulsion system is vital. One of the most important technologies is regenerative cooling. The cryogenic liquid propellants are used as coolant flowing through ducts milled in the combustion chamber and nozzle wall. It has been shown, that high aspect ratio cooling channels (HARCCs) provide a significant increase in cooling effectiveness. Therefore, a deep understanding of the principles and mechanisms of turbulent heat transfer in HARCCs is of special interest.

The flow field in HARCCs is highly affected by secondary flows, anisotropy induced corner vortices and curvature-induced Dean vortices. Even if these flow phenomena are weak compared to the mean flow, they have a great impact on turbulent heat transfer by transporting hot fluid from the heated wall to the cooler core region of the duct.

As duct flows of various cross sections and especially cooling ducts are of considerable engineering interest, several experimental as well as numerical studies have fo-
cussed on this topic. In the sixties and seventies, first measurements concerning the flow through adiabatic quadratic ducts and the characteristic secondary flows have been performed by [1] and [2] using a hot wire system and by [3] using laser Doppler anemometry. Using the same technique the influence of wall heating on the flow structure was investigated by [4]. More recently [5] measured the velocity profile of an adiabatic high aspect ratio duct (AR = 11.7). The velocity profiles were obtained based on the pressure measured by a pitot-static probe.

Whereas experimental data for HARCCs is scarce, several numerical investigations focused on the simulation of cooling ducts over the last decade. Extensive research on this topic has been done by [6] and [7]. Using low-order turbulence modeling in terms of Reynolds-averaged Navier-Stokes equations (RANS), the author is able to simulate HARCCs under realistic conditions found in current launcher systems at an affordable numerical cost. This comprises high pressures and high heat transfer, the consideration of real gas effects, realistic geometries and high Reynolds numbers.

A first Large Eddy Simulation (LES) of cooling ducts with asymmetric heating has been published in [8]. In this case the duct was straight and of quadratic shape. The influence of an asymmetric heat flux applied to the lower wall on the velocity profile and fluctuations has been discussed in detail. [9] extended this approach to an investigation of a heated curved duct, where curvature-induced secondary flows play a significant role. In [10] the spatial development of the turbulent flow in an asymmetric heated straight duct was studied in detail. In order to reduce numerical costs, all the studied turbulent duct flows had a quadratic shape and operated at a low Reynolds number of 6000. In [11] the authors extended their investigation to turbulent heat transfer in rectangular ducts of various aspect ratios ranging from 0.25 to 1.5 at a Reynolds number of 4410.

Recently an extensive DNS study of turbulent flow in high aspect ratio ducts has been performed by [12]. The turbulent flow and the influence of secondary flow structures for straight isothermal ducts with aspect ratios ranging from 1 - 7 have been investigated at a low bulk Reynolds number of 2800. Neither heating effects nor curvature effects have been included in this study.

This report is structured as follows: At first the reference experiment conducted by our colleagues of sub-project D9 from the Technical University of Braunschweig is presented. Then the results of a preliminary RANS simulation of the whole curved cooling duct are shown and the typical secondary flow structures presented. Finally, the LES results are presented, for both an isothermal duct case as well as a heated straight HARCC.

2. Reference experiment

Together with our project partners from the Technical University of Braunschweig we developed a generic cooling duct experiment. A sketch of the experimental setup is shown in Fig. 1 and an overview given in Fig. 2. Figure 3 presents the cooling duct experiment, its instrumentation and measurement equipment in working condition. In the first phase of the project a straight isothermal and heated duct was investigated. In the second phase the setup will be extended by a heated bend of radius $R = 60$ mm. The experiments are conducted using the non-invasive methods Particle Image Velocimetry (2C2D-PIV), Stereo Particle Image Velocimetry (3C2D-PIV) and Volumetric Particle Tracking Velocimetry (3C3D-PTV). Further details can be found in [13].
It is necessary to account for the typical flow conditions and also the rich array of secondary flow features present in cooling ducts of launcher systems. Therefore the Reynolds, Nusselt and Dean-numbers have to be of the same order as found in realistic devices. However, as the simulation costs scale with $Re^3$ for a well-resolved LES, approximations have to be made to keep the numerical costs at an acceptable level. Therefore a trade-off between numerical costs, experimental realizability and the realistic conditions of launcher systems has been done. The main parameters of the generic cooling duct design are given in Tab. 1.

Sub-critical water at a bulk temperature of $T_\infty = 60^\circ\text{C}$ is chosen as a coolant. The flow rate is set to $\dot{V} = 50$ l/min which results in a bulk speed of $u_b = 5.38$ m/s and is controlled via a high precision electromagnetic flowmeter. The rectangular duct has a width of $w = 6.0$ mm and a height of $h = 25.8$ mm, corresponding to a hydraulic diameter $d_h = 9.74$ mm. The aspect ratio of the cooling duct is $AR = h/w = 4.3$.

At the beginning the pre-heated water is pumped into the isothermal feed line of 600 mm to assure a fully developed turbulent duct flow entering the heated test section. After a flow straightener the heated section of again 600 mm and an identical cross section follows. An asymmetric heating is applied to the copper bottom wall with a constant wall temperature of $T_{wall} = 100^\circ\text{C}$, which is low enough to avoid boiling of the water. After the test section the water flows back to the reservoir to close the cycle.

### 3. Preliminary numerical investigations with RANS

As first step the cooling duct experiment was modeled and simulated with RANS using the commercial flow solver ANSYS CFX 15.0. The setup consists of a 300 mm straight isothermal feed line, then a 600 mm heated duct and at the end the $R = 60$ mm bend, see Fig. 4. All the non-heated side walls were defined as smooth, adiabatic walls. The lower heated wall, comprising the 600 mm straight duct and the bend, is set at a fixed temperature of $T_{wall} = 100^\circ\text{C}$. At the inlet the velocity profile and the Reynolds stresses for a fully developed turbulent duct flow are prescribed. This profile was extracted from another RANS simulation of an isothermal straight duct. For the wall-resolved simulation ($y^+ \approx 1$) a structured mesh was generated using ANSYS ICEM CFD. This mesh comprises in total $1.9 \cdot 10^6$ hexahedrons. In the 2D duct cross section 150 cells for the long side and 60 for the narrow heated side are used.

The incompressible steady-state simulation was performed with water as fluid. Viscosity and thermal diffusivity are temperature dependent according to the IAPWS IF97 formulation (International Association for the Properties of Water and Steam). The fluid model was set to thermal energy and buoyancy effects are neglected. Several turbulence models have been tested. In order to capture the corner vortices, an anisotropic Reynolds stress model was used for turbulence modeling. Best results are achieved
FIGURE 1. Sketch of the cooling duct experiment (without bend), [13]

FIGURE 2. Overview of the generic cooling duct experiment, [13]

FIGURE 3. Overview of the experimental setup, [13]

FIGURE 4. Temperature contours in duct midplane (RANS)
using the BSL Reynolds stress model. Details of the used solver and the implemented models can be found in [14].

In each corner of the duct a pair of counter-rotating turbulence induced vortices is formed, which extend after an initial phase over the whole duct cross section. Figure 5 shows one pair of these counter-rotating vortices, which create a weak secondary flow...
(1-2% of the bulk flow velocity) in wall-normal direction. This so called Prandtl’s flow of the second kind transports fluid from the short heated wall side into the core region of the cooling duct. For cooling applications this secondary flow is important for turbulent heat transfer away from the short wall side. Hot fluid is transported from the heated wall in normal direction into the cooler core region of the duct and cold fluid back to the wall. Therefore mixing and the overall cooling efficiency is increased. As the vortex pairs are a consequence of the Reynolds stress tensor anisotropy, most of the simplified turbulence models used for industrial purposes, fail to capture them. These models are mostly based on the Boussinesq approximation of isotropic turbulence. Therefore more complex turbulence models like LES or RANS using a Reynolds stress turbulence model have to be applied for the simulation of HARCCs.

In curved sections of ducts the centrifugal force leads to the formation of Dean vortexes, see Fig. 6. These are relatively strong secondary flows, which in the duct midplane transport fluid from the convex bent side of the channel to the concave bent side. Thus a backflow in the side wall boundary layer is induced. In the duct midplane this secondary flow leads to a transport of hot fluid away from a heated wall to the center of the duct or to the transport of cold fluid to the heated wall, depending on its curvature. So the usage of convex-concave bent sections enhances the mixing in a cooling duct and therefore its cooling efficiency.

Figure 5 shows the mean axial velocity profile and temperature distribution after 500 mm from the beginning of the heated straight duct and Fig. 6 the same quantities at the end of the bent section, see also Fig. 4 for the temperature development at the bottom wall in the duct midplane. The direct comparison of mean velocity profile and temperature distribution between the first two figures shows, that the influence of the Dean vortexes is significantly stronger on both the mean flow profile and the heat transfer from the heated wall into the core region of the duct. Thus the corner vortexes just play a minor role when curvature-induced Dean vortexes are present.

Both secondary flow phenomena, corner vortexes and Dean vortexes, occur simultaneously in curved cooling ducts, interfere with each other and enhance the turbulent heat transfer. Both are also affected by the heating, i.e. the change in fluid viscosity. In the case of cooling channels mostly an asymmetrical heating is applied to one of the walls. As the fluid temperature in the wall near region rises, its viscosity drops and thus turbulence is enhanced. This in return has a significant impact on the generation of secondary flows and as a consequence on turbulent heat transfer.

4. Numerical investigation using LES

4.1. Numerical method (LES)

For the LES simulation we use our in-house solver INCA. The flow is described by the incompressible Boussinesq equations Eq. 4.1, a simplified version of the Navier-Stokes equations. The temperature is treated as a passive scalar using the linearly dependent buoyancy variable Eq. 4.2. Viscosity and diffusivity of the passive scalar, i.e. the thermal diffusivity, depend on temperature and density and are updated each time-step using
Numerical investigation of the flow in a high aspect ratio cooling duct

the IAPWS correlations ([15]).

\[ \nabla \cdot \mathbf{u} = 0 \]
\[ \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = \frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} + \frac{1}{\rho} \mathbf{F} \]  \tag{4.1}
\[ \frac{\partial b}{\partial t} + \mathbf{u} \cdot \nabla b = D_b \nabla^2 b + Q_b \]

with

\[ b = -\rho_0 \alpha \Delta T \]  \tag{4.2}

Where \( \rho_0 \) is the water density at \( T_{\infty} = 60^\circ \text{C} \) of 983.2 kg/m\(^3\) and \( \alpha \) the mean volumetric thermal expansion coefficient of water. The averaging over the interval of possible temperatures, i.e. from 60\(^\circ\) C to 100\(^\circ\) C, leads to a value of \( 6.32 \cdot 10^{-4} \) K\(^{-1}\).

The equation system is discretized by a fractional step method on a block structured staggered Cartesian grid. More complex geometries can be taken into account by utilizing an immersed boundary method. As time advancement method an explicit 3\(^{rd}\) order Runge-Kutta scheme, see [16], is applied, while the time-step is chosen to satisfy \( \text{CFL} = 1.0 \).

For spatial discretization the finite-volume method (FVM) is used. For discretizing the pressure Poisson equation and the diffusive fluxes 2\(^{nd}\) order accurate central difference schemes are implemented. The pressure Poisson equation is solved in every Runge-Kutta substep using a Krylov subspace solver with an algebraic-multigrid preconditioner.

For discretization of the convective fluxes the SALD method is used, which is a computationally more efficient implementation of the Adaptive Local Deconvolution Method (ALDM), [17]. ALDM is a nonlinear finite volume method that provides a physically consistent subgrid-scale turbulence model for implicit LES. Locally one can reconstruct the unfiltered solution by a solution-adaptive combination of Harten-type deconvolution polynomials. Regularization of the deconvolution is achieved by limiting the degree \( k \) of approximation polynomials to \( k \leq K \). All polynomials of degree \( 1 \leq k \leq K \) are used for the reconstruction. The solution-adaptivity is achieved by dynamically weighting these polynomials. The numerical flux function operates on this approximated solution. The ALDM implementation contains several model parameters, for the solution-adaptive stencil-selection scheme and the numerical flux function, which have to be defined.

The required calibration has been performed for canonical inertial-range turbulence test cases [18].

4.2. Numerical LES setup

The LES setup used to simulate the heated straight cooling duct is shown in Fig. 7. The adiabatic periodic duct section serves as time dependent fully developed turbulent inflow data generator and models the isothermal feed line of the experiment. In order to resolve all the relevant turbulent structures, the streamwise length was chosen as 7.5 \( d_h \). Each time-step the outflow profile of the isothermal duct is prescribed as inflow profile for the heated duct, for which the whole length of 600 mm is simulated. Both simulations run simultaneously. All walls are defined adiabatic except the lower wall of the heated duct, where a fixed temperature of \( T_{\text{wall}} = 100^\circ \text{C} \) is prescribed, respectively the equivalent buoyancy value.
The initialization of the straight heated duct consists of several steps. At first the velocity profile for a fully developed laminar duct flow (Eq. 4.3, [19]), superimposed with white noise fluctuations of a turbulence intensity of 5%, is defined as initial solution for a periodic duct piece on a coarse mesh. This simulation is performed until the state of a fully developed turbulent duct flow is reached. This solution is then interpolated to the fine mesh version of the same periodic duct piece and the simulation continued for 8 additional flow through times (FTTs). Finally the isothermal fine mesh solution is taken as initial condition for the full setup of the heated straight duct. To determine the required grid resolution for the wall-resolved LES an extensive grid sensitivity analysis has been conducted. The resulting yz-mesh and the blocking is shown in Fig. 8. The 2D grid remains unchanged in streamwise direction. The objective of a wall-resolved LES leads to a very high grid resolution in the wall boundary layers and subsequently to a
Numerical investigation of the flow in a high aspect ratio cooling duct

great number of cells. In order to reduce numerical costs, the resolution in the duct core region was decreased by using a 2:1 block connection between the boundary layer and the core blocks.

With the chosen grid resolution Fig. 9 shows, that both for the narrow side wall as well as the large side wall the logarithmic boundary layer profile is obtained. Both profiles are taken in the middle of their respective side and averaged with the opposite side profile. At the large side walls the dimensionless wall distance of the first cell is $y^+ = 1.43$ and at the small side walls $y^+ = 1.25$. The definition of $y^+$ is based on the cell edge. A further refinement didn’t show any difference in the obtained results. $\Delta y_{max}/y^+$ as well as $\Delta z_{max}/y^+$ was set to 25 in the boundary layer blocks and consequently to 50 for the duct core flow. $\Delta x/y^+$ is set to 50. In y- and z-direction a hyperbolic meshing law and in streamwise direction a uniform distribution is used. In total the mesh consists of $280 \cdot 10^6$ cells, distributed on 7104 blocks. The simulation runs on 7088 CPUs. For sufficiently converged statistics several FTTs for the whole configuration have to be simulated leading to very high numerical costs.

The grid sensitivity analysis has been done for the adiabatic case, i.e. the used 2D grid is symmetric in y- as well as z-direction. For the heated case the lower wall resolution has been increased in wall normal direction to account for diffusive heat transfer. Therefore the smallest cell size in wall normal direction has been reduced by a factor of $1/\sqrt{Pr_{water}(60\degree C)} = 1/\sqrt{3}$. Thus the grid used for the simulations of the heated duct is just symmetric in the narrow side wall direction. This yz-grid is also used for the isothermal periodic part to avoid interpolation of the inlet profile and to simplify the load balancing.

4.3. Numerical LES simulation results

In this section the results of the heated straight duct simulation using LES will be presented. All the results are based on the statistics obtained after four FTTs based on the heated duct length of 600 mm. The following discussion will be mainly qualitative, the detailed quantitative analysis is in preparation.

Figure 10 compares the u- and v-velocity profiles between experimental and numerical LES data in the duct midplane parallel to the large side wall. The u-velocity denotes the streamwise velocity and the v-velocity the velocity in normal direction to the heated wall. Both profiles are taken from the isothermal case, i.e. for the LES simulation the symmetric grid was used. The LES results are averaged over a thickness of 2.25 mm to account for a finite laser-sheet thickness of the PIV instrumentation. The compari-
son shows for both streamwise and wall normal velocity a good agreement between experimental and numerical data.

In Fig. 11 and Fig. 12 the instantaneous and the mean streamwise velocity in the duct cross section at 600 mm, i.e. at the end of the heated duct, are presented. The first serves as a good illustration for the highly turbulent structure of the duct flow field at this high Reynolds number. The latter shows the averaged streamwise velocity sampled over four FTs. The obtained profile is almost symmetric. This means on the one hand, that the streamwise velocity is nearly converged and on the other hand it shows the weak impact of asymmetric wall heating on the mean flow profile. The same result has already been observed for the RANS simulation.
The secondary flow structures, i.e. the pairs of counter-rotating vortices in each corner of the duct, are shown in Fig. 13. One of these pairs is displayed in the upper left corner using streamlines. To quantify the strength of the vortex pairs the secondary flow velocity is introduced by \( v_w = \sqrt{v^2 + w^2} \). In accordance with available data in the literature the strength of the corner vortices is about 1-2% of the bulk flow velocity. Smaller vortices near the duct center can be observed. These structures are temporary and vanish with increasing sampling time.

The heating of the water leads to a significant drop of the viscosity in the region directly at the heated wall, as can be seen in Fig. 14. The viscosity is directly coupled to the instantaneous temperature field via the IAPWS correlations. The figure also shows that just a relatively small part of the cooling duct is affected by the heat transfer.

The drop of viscosity in the vicinity of the heated wall also affects the turbulent structures in the wall boundary layer. This is displayed in Fig. 15, where the streamwise turbulence intensity in the boundary layer is shown on two cutting planes parallel to the
narrow heated side wall. On the left hand side the adiabatic periodic duct section is shown and on the right the fully resolved heated duct. A comparison of the levels between the adiabatic and the heated section shows a sharp drop of turbulence intensity in the boundary layer. This demonstrates that wall heating has a significant influence on the turbulent structures of the boundary layer and thus the need for a further quantitative investigation.

Figure 16 shows the temperature distribution for three cross sections at 100 mm, 350 mm and 600 mm within the heated duct section. As the upper part of the duct is not affected by the heating applied to the lower wall it is blanked. The comparison of the three positions displays the growth of the heated region at the bottom wall. On the left of each duct streamlines are added to mark the position of the corner vortices as well as their influence on the temperature profile. Its characteristic bent shape is a consequence of these vortices. On the left hand side of the duct hot fluid is transported upwards along the large side wall by the larger clockwise rotating vortex and cold fluid downwards into the left duct corner. In the symmetry plane of the duct hot fluid is transported away from the heated wall by the smaller counterclockwise rotating vortex and cold fluid also transported into the left duct corner.

5. Conclusions

In the framework of the SFB-TRR40, sub-project D4, a generic cooling duct experiment conducted with subcritical water as coolant has been defined together with our partners from sub-project D9. The targets of this generic experiment were on the one hand to mimic the typical flow conditions of current launcher cooling ducts and on the other hand to simplify the configuration to be able to investigate it thoroughly via experimental as well as numerical methods.
A preliminary investigation was conducted with the numerically less expensive RANS method using a commercial solver. Qualitatively good results were obtained using the BSL Reynolds stress model as turbulence model. The simulation could reproduce all the relevant flow features, especially the secondary flow structures, i.e. the counter-rotating corner vortices in each corner of the duct and the curvature-induced Dean vortices. Also the significant effect of these secondary flow structures on turbulent heat transfer was clearly visible.

The main part of the numerical investigation comprises the LES simulation of the cooling duct experiment. For this simulation our in-house code INCA has been adapted and successfully applied. As governing equations the incompressible Boussinesq equations are used. The chosen setup consists of a short periodic isothermal duct as feed line and the fully resolved heated straight duct. The bend following the straight duct has not been tested yet. After an extensive grid sensitivity analysis for the isothermal duct, numerically expensive simulations for the heated configuration have been done. A quantitative comparison between LES and experimental results showed good agreement. Qualitatively all physically relevant flow features of the configuration can be reproduced by the LES, i.e. the counter-rotating vortex pairs in each duct corner. The relatively weak strength of this secondary flow agrees well to available data in the literature. Also their significant influence on the turbulent heat transfer in the absence of curvature-induced Dean vortices is clearly visible.

The next step will be a thorough quantitative investigation of the interaction between secondary flow structures and heat transfer as well as the influence of the heat transfer on turbulent structures in the boundary layer. Furthermore, a detailed comparison with experimental data for the heated duct is planned.

Acknowledgments

Financial support has been provided by the German Research Foundation (Deutsche Forschungsgemeinschaft – DFG) within the framework of the Sonderforschungsbereich Transregio 40, SFB-TRR40 (Technological foundations for the design of thermally and mechanically highly loaded components of future space transportation systems). Computational resources have been provided by the Leibniz Supercomputing Centre Munich (LRZ).

The authors acknowledge helpful discussions with the people from the Institute of Fluid Mechanics, Technische Universität Braunschweig, who conduct the experimental investigation of this case; namely Henrik Rochlitz and Peter Scholz.

References


