SIMULATION OF LIQUID, TRANSCRITICAL AND GASEOUS COOLING FILMS IN ROCKET COMBUSTION CHAMBERS

Christoph Höglauer, Björn Kniesner*, Oliver Knab*, Gregor Schlieben**and Oskar Haidn***
**ASTRIUM GmbH Space Transportation, 81663 Munich, Germany*
Christoph.Hoeglauer@astrium.eads.net, Bjoern.Kniesner@astrium.eads.net, Oliver.Knab@astrium.eads.net
***Institute for Flight Propulsion, Technische Universität München, 85748 Garching, Germany*
schlieben@lfa.mw.tum.de, haidn@lfa.mw.tum.de

Abstract
The focus of the paper is set on the modeling and simulation of liquid, gaseous and transcritical cooling films in rocket combustion chambers. Besides different film fluids such as MMH, NTO, kerosene or methane a major issue is the modeling of films in transcritical state. For the modeling Astrium's in-house code Rocflam-II is applied. The main goal of the used modeling in Rocflam-II is to provide a tool package for the simulation of a wide range of rocket combustion devices, validated against experimental data. This includes the modeling of injection, droplet spray, combustion, wall heat transfer, film cooling as well as additional regenerative cooling simulated by 3D conjugate heat transfer. First successful steps toward this goal are presented in this paper.

Nomenclature

Latin Symbols		
Re	[-]	Reynolds number
Nu	[-]	Nusselt number
k	[m^2/s^2]	Turbulent kinetic energy
p	[bar]	Pressure
T	[K]	Temperature
O/F	[-]	Mixture ratio
l	[m]	Length
u	[m/s]	Velocity
$\dot{q}$	[MW/m^2]	Heat flux

Greek Symbols		
ϵ	[m^2/s^3]	Dissipation rate
η_{c*}	[%]	Combustion efficiency
α	[-]	Blending factor for underrelaxation

Abbreviations	
MMH	Monomethylhydrazine
NTO	Dinitrogentetroxide
LFA	Institute for Flight Propulsion
PPDF	Presumed propability density function
CFD	Computational Fluid Dynamics
LOX	Liquid oxygen
GOX	Gaseous oxygen
RANS	Reynolds Averaged Navier Stokes
CHT	Conjugate heat transfer
Rocflam-II	**Roc**ket **Fl**ow **A**nalysis **M**odule - **II**. Generation
RCFS-II	**R**egenerative **C**oolant **F**low **S**imulation - **II**. Generation

1. Introduction

As engine dimensions get smaller, it becomes more and more difficult to cool combustion chamber only by regenerative cooling via cooling channels; also, the pressure budget often does not allow the additional pressure drop required by regenerative cooling circuits. Therefore, propellant films in operation dependent liquid, transcritical or gaseous state are applied. The analytical description of film cooling in rocket combustion devices still shows significant deficiencies in capturing all the involved physical processes. For that reason the development of analytical or numerical tools is still an important point at Astrium Space Transportation.

Test data from several combustion chambers like a subscale rocket combustion chamber at the Institute for Flight Propulsion (LFA) were used for extending the film competence with the numerical design tool Rocflam-II of Astrium Space Transportation. The main goal of these investigations is to extend the Rocflam-II film cooling model to various propellant combinations, with different film fluids, at different combustion chamber designs, with different film deposition possibilities, different film fluid conditions and different boundary conditions underneath the film. This includes also the possibility to calculate liquid / transcritical (supercritical pressure, subcritical temperature) / gaseous films with both combustion modeling approaches (ppdf-equilibrium based and global chemistry scheme) available in Rocflam-II [3].
Therefore many new features have been implemented into the Rocflam-II Lagrange module for droplet tracking as well as the film model. These new features and modeling approaches are shortly described and finally backed up with simulation results. The focus during the modeling development was set on individual importance of the different physical aspects of cooling film in a rocket combustion chamber.
Another focal point is the convergence of film simulations. The interactions between chamber wall, film, disperse droplet phase and hot gas with heat and mass transfer, hypergolic reactions, film waviness etc. are considered in Rocflam-II by the coupling of three phases (hot gas, droplets and film) and a coupling for consideration of the conjugate heat transfer between hot gas side and regenerative coolant flow in the chamber wall.

2. Physics of cooling films

As already stated in the introduction an often applied method for wall cooling in a rocket engine combustion chamber is film cooling. Figure 1 shows some physical aspects of this cooling method.

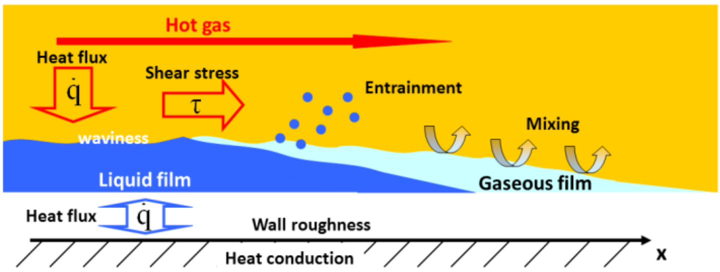

Figure 1: Physical aspects of liquid film cooling

Generally, the hot gas flow is the main driving factor for the development of the film. Nevertheless, the wall side has also an influence on the film. Furthermore the film and the hot gas flow interact very strongly. In the following the main influences on a liquid, transcritical and gaseous film in a hot gas flow environment are described:

- *Shear stress* between the film – hot gas interface influences the flow regime (turbulent or laminar) of the film, the velocity of the film and the mixing.

- *Temperature difference* between the film and the hot gas controls the film heat up and if so evaporation of the film.

- On the wall side of the film, *heat conduction*, *wall roughness* and the *temperature difference* between film and wall control the heat flux between film and wall. In a transient case or film cooling combined with regenerative cooling the film is often cooled by the wall, which results in a longer film.

- *Hypergolic reaction* between the NTO (MMH) film and a penetrating MMH (NTO) droplet. This reaction releases energy on the surface of the film.

- Exothermic or endothermic *dissociation* of film species during heat up or evaporation due to exceeding of the critical dissociation temperature.

- *Hot gas composition* close to the film surface influences the heat flux into the film and diffusion of the film. This has a strong influence on how the released film fluid reacts with the hot gas (for example oxidizer film and fuel rich environment). The species mass fraction controls the diffusion of the film species into the hot gas.

- Depending on the Reynolds number the film shows a laminar or turbulent *flow regime*. This has also an important influence on the heat transfer through and into the film.

- *Hot gas radiation* can heat up the film directly or heat up the chamber wall which on its part heats up the film.

- The released *gaseous film* still protects the wall from the hot gas. The cooling effect results from a mixture ratio trimming and the, compared to the hot gas, cooler fluid. But with increasing length the gaseous film mixes and reacts with the hot gas until it vanishes.

The following influences only apply to liquid film:

- *Liquid film waviness* is a result of the shear stress and influences the heat flux into the liquid film. The liquid film waviness is treated in the heat flux calculation in the same way as the surface roughness of the chamber wall.

- *Droplet entrainment* (droplets ripped out of the liquid film surface) depends on film thickness, waviness, surface tension, film velocity and shear stress. This results in a liquid film mass reduction and a film shortening.

In case of film cooling with a fluid in transcritical state another important aspect appears:

- In the transcritical state, a fluid has no *heat of evaporation* and no or minor *surface tension*. This means that after heat up to saturation temperature (in this case the critical temperature) the film switches instantaneous to gaseous state.

All of these physical aspects have to be considered for a realistic film simulation. But due to the very strong interactions between hot gas, cooling film and wall it is a challenge for the numerical modeling to get a well converged solution. This means that all parts of the simulation for example the Lagrange module, combustion or gas solver have to converge to get a realistic liquid film determination.

3. CFD Tool Rocflam-II

EADS Astrium's in-house code Rocflam-II is a structured, finite volume, compressible and axisymmetric/2-D, multiphase, multi-species Navier-Stokes solver. A standard 2-equation k-ε turbulence model is incorporated with a 2-layer approach at the wall.
Most important and outstanding feature is the Lagrange module for particle injection and tracking. Bi-propellant and multi-class droplets can be injected in a gaseous environment that way. The droplets are injected as a disperse spray with different droplet sizes computed by a log-normal or Rosin-Rammler distribution based on a mass mean diameter and a distribution width. They are tracked, heated up and finally vaporized before they enter the RANS equations as right-sided source terms. Not only evaporation, also diffusion and droplet dispersion due to a turbulent flow in the chamber are considered. This approach is also applied for the injection of trans- or supercritical fluids using a special treatment within the Lagrange module. That means a propellant of transcritical state is modeled like a droplet, but with the consideration of no heat of evaporation and no surface tension.
The Lagrange spray approach also enables the code to handle the boundary conditions at the face plate, where the propellants are injected, as closed wall rather than as opening.
The droplet tracking is also responsible for the liquid film buildup and the hypergolic reactions in the film. If a droplet reaches the wall it will be reflected, or with activated film model it deposits on the wall and builds up a liquid film. In case of a hypergolic propellant combination (i.e. MMH/NTO) a contrary droplet, hence fuel droplet in case of oxidizer film and vice versa, which penetrates the liquid film results in a hypergolic reaction and therefore reduces the liquid film mass.
Due to strong coupling of film, hot gas and droplet tracking a good convergence of the Lagrange module is essential for a simulation. Figure 2 depicts the coupling and interactions of the 3 phases (gas / droplets / liquid film) in a schematical way.

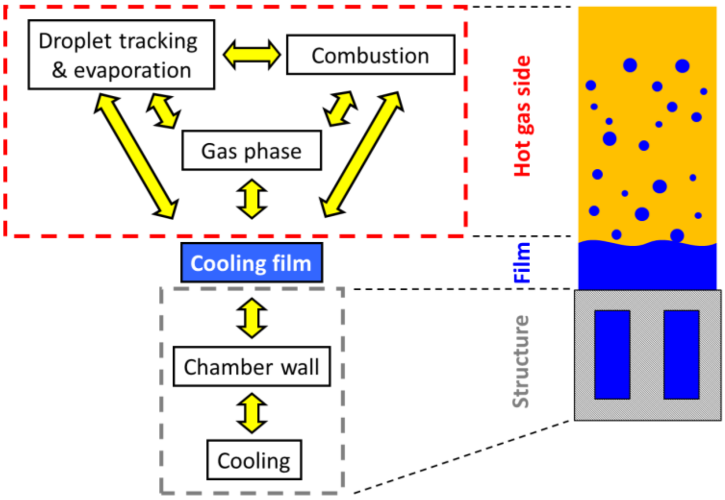

Figure 2: Coupling of the liquid film model with the Lagrange phase, the Euler phase and the structure in a schematical way

Turbulent combustion can be modeled by either a presumed PDF approach based on equilibrium tables or global reaction schemes using a combined eddy dissipation concept or Arrhenius approach. Both modeling approaches are available for diverse fuel/oxidizer combinations. The fluid properties are extracted from a fluid database stored in a table with pressure and temperature dependency or in polynomial equations with temperature dependency. Therefore no separate equation of state is needed in the code. Real gas effects are implicitly included in the experimental fluid data of the used database. For more information about Rocflam-II, see [3] and [4].

4. Film modeling

To calculate cooling films in Rocflam-II different modeling approaches are possible. Crucial point of the film modeling is the thermodynamical state of the different rocket propellants. Here one has to distinguish between liquid, gaseous and transcritical state depending on the combustors operational conditions. Table 1 shows the pressure and temperature conditions of these states.

State	*Pressure*	*Temperature*	*Simulated film fluids in this paper*
Liquid	$p < p_{crit}$	$T < T_{crit}$	MMH, NTO, kerosene and water
Transcritical	$p > p_{crit}$	$T < T_{crit}$	Kerosene and methane
Gaseous / supercritical	$p > p_{crit}$	$T > T_{crit}$	Hydrogen

Table 1: Different thermodynamical states of a film fluid

The transcritical state is situated between the liquid and the gaseous state and therefore has special properties like a density similar to a liquid, no heat of evaporation or no surface tension. These properties are very important for a modeling of transcritical fluids.
All following modeling approaches have in common the treatment of the generated (in case of liquid / transcritical fluid) or injected gaseous film fluid. The gaseous film mass, momentum, enthalpy and species are always transferred into the Navier-Stokes solver via source terms and are thereby considered in the flow and combustion calculation.

4.1 Liquid film modeling

For the liquid film calculation Rocflam-II uses a specially adapted one dimensional liquid film model based on the publications of Nahstoll [1] and Fisher & Pearce [2]. The model was originally implemented for simulating small satellite rocket combustion chambers with MMH liquid film and radiation cooling. Therefore in the original version the liquid film model had the restrictions to use only MMH as liquid film species combined with a radiation cooling on the wall. The current film model does not have these restrictions any longer. The film model is validated against most Astrium satellite thrusters.

In basic the film model calculates after the deposition of the film fluid the heat up and evaporation until the dry out point. The evaporated film mass, enthalpy and momentum are transferred via source terms into the Navier-Stokes equations of the RANS solver and are considered in the chemical reaction. Furthermore, the film temperature and velocity, in case of existing film, is the boundary condition for the gas solver.
The Rocflam-II film model can simulate different film fluids depending on the propellant combination. It can be chosen between fuel, oxidizer or for a H_2/O_2 combustion, water film in case of a ppdf-equilibrium based chemistry, or every in the combustion considered specie in case of the global chemistry scheme. This includes the possibility of the film model to interact with the two different combustion modeling approaches in Rocflam-II: A global chemistry scheme and a tabulated ppdf-equilibrium chemistry [6].
The liquid film model is one-dimensional and derived for shear stress driven films. It solves the mass, enthalpy and momentum conservation equations for the film. As example for the conservation equations the considered effects and the calculation scheme of the momentum in the film is depicted in Figure 3.

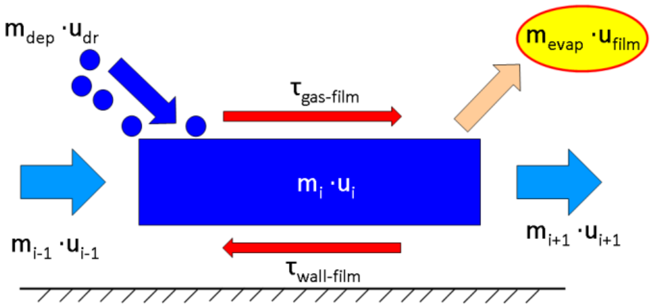

Figure 3: Scheme of the film momentum calculation

Due to the fact that the shear stress between wall and film and the momentum of the evaporated film uses the current film velocity, the momentum equation has to be solved iteratively with a prediction-correction method.

The film build-up occurs through spray droplets which impinge on the chamber wall or injection through a film device. So the film model is, in case of droplets deposition, directly coupled with the Lagrange droplet tracking module. In case of hypergolic bipropellants, a special treatment of hypergolic reactions is considered in the film, as soon as the counterpart droplet hits the film fluid. Furthermore decomposition effects of the film fluids are modeled. The reaction products including decomposition effects are then transferred into the gas phase.

The film thickness is a result of the mass conservation and the calculated mean film velocity. Due to the one dimensional model the thickness is not considered in the gas phase. This means that the gas phase boundary is not displaced due to the film thickness. In the presented applications the film thickness lies in the range of 10 to 100µm which justifies this simplification. Nevertheless the thickness influences the heat transfer coefficient of the film and thus the resulting heat flux into the wall.
The heat flux from the hot gas into the liquid film is calculated in the same way as the usual wall heat flux. For that reason effects like film waviness, hot gas temperature distribution or hot gas composition are considered in the heat flux calculation into the liquid film. To distinguish between the film and the wall a separate treatment for the consideration of wall roughness and film waviness is implemented.
To determine the heat flux between film and wall an one-dimensional Nußelt-correlation is implemented. This correlation considers three different flow regimes of the film: Laminar, laminar-wavy and turbulent film. All three flow regimes are possible in film cooling of rocket combustion chambers. Especially in combination of film cooling and regenerative cooling or predefined wall temperature this heat transfer is very important.
Furthermore axial and radial heat conduction in the chamber wall are considered in the heat transfer calculation between film and chamber wall. The heat is either conducted into the regenerative cooling circuit or radiated to the ambient.

4.2 Gaseous film modeling

Different to the modeling approach for liquid cooling fluids, the gaseous film is directly simulated in the Navier-Stokes-Solver of Rocflam-II. To initialize a gaseous film the following three approaches (two are basically related see bullet 1.) can be used:

1. Set gaseous source terms

In this approach a source term with mass, momentum, enthalpy and specie is set into the Navier-Stokes solver of Rocflam-II. This can be done via the Lagrange module which models the gaseous fluid in the same way as a supercritical spray droplets injected through an injection element (instantaneous “evaporation” without heat up and resistance caused by latent enthalpy) or the source term is directly set into the Navier-Stokes solver without Lagrange module.

2. Inlet boundary condition

The third possibility to inject a gaseous film is with an inlet boundary condition in the wall or the faceplate. Here the film mass flow is injected through an opening in the chamber wall. Because of the opening boundary type as inlet of the gaseous film fluid it is directly considered in the Navier-Stokes solver of Rocflam-II. The proceeding via source terms is in this case not necessary. The inputs are the mass flux, temperature, species distribution or mixture ratio and inlet turbulence.

Compared to method 1 the inlet boundary condition shows the best results, therefore this modeling is preferred for the simulation of gaseous cooling films.

4.3 **Transcritical film modeling**

Currently Rocflam-II offers three different approaches to simulate a film in transcritical state, which are shortly described in the following.

1. Dense-gas approach

This approach is identical to the both approaches explained in the gaseous film modeling. Hence, in this approach the transcritical fluid is treated like a gas in the Navier-Stokes solver.

2. Extended liquid film model

Another possibility to simulate a transcritical film is with the extended liquid film model. The validation of this approach is currently in progress. The basic idea is to simulate the transcritical fluid as a continuous liquid phase but without the evaporation enthalpy and surface tension. This is a little bit tricky for the convergence of a numerical method, especially for a one-dimensional film model without temperature profile in the film normal to the wall. Due to the physical aspect that a transcritical fluid has no resistance against evaporation at saturation temperature, the whole film mass “evaporates” instantaneously in one grid cell when it reaches saturation temperature. The result is a peak source term in this grid cell, which is hard to solve for the CFD code.

The currently implemented approach is to split the absorbed heat flux, depending on the film temperature, into a heat up part and an evaporating part. For example, if the film temperature has reached 60% of the saturation temperature, 80% of the incoming heat flux is used for further film heat up and 20% are used to “evaporate” a part of the film mass. The "evaporated" part of the film mass depends on the difference between film temperature and saturation temperature. This assumption should consider the facts, that the film temperature in reality is not uniformly distributed across the film thickness and a transcritical fluid has no latent enthalpy. Figure 4 shows this separation in a schematical way.

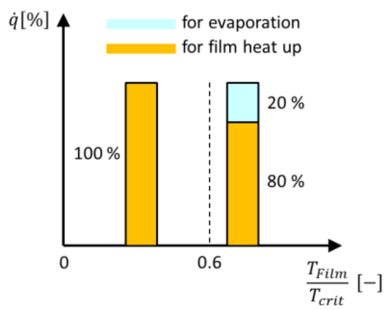

Figure 4: Heat up and evaporation of a transcritical film

The numerical effect of this approach is smoother film "evaporation". The big advantage of the extended liquid film model is that the transcritical film can be modeled as a continuous film flowing alongside the wall. This physical effect has been observed in different tests and out of these three approaches only the extended liquid film can consider this.

Validation simulations for transcritical kerosene as cooling fluid show a very good agreement of the extended liquid film compared to test data. Therefore this approach is the favored method for future simulations.

3. Droplet injection close to the wall (Lagrange)
The last possible way to simulate a transcritical film is to initialize disperse droplets close to the wall through the Lagrange module of Rocflam-II as done for the main propellant injection. This approach can be seen as a compromise between the extended film model and the dense-gas approach. The droplets are tracked along the wall, heated up and become gaseous but do not build-up a closed film at the wall. However the simulation catches the mixture ratio trimming and the low enthalpy flow insertion along the wall.

All three approaches for modeling a transcritical cooling film are schematically illustrates in Figure 5.

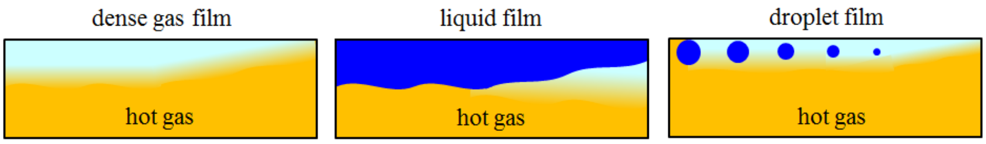

Figure 5: Illustration of different variants to simulate transcritical films

Due to the complex combination of coupled liquid/transcritical film model, Lagrangian approach for the disperse spray, combustion modeling, the hot gas flow and if necessary the conjugate heat transfer it is very challenging to get a well converged CFD solution. To get a good convergence the commonly used numerical utility of under relaxation is implemented. Under relaxation is a numerical method for blending the solution of the former iteration with the solution of the present iteration, so the change from former solution to the present solution depends on the under relaxation factor α. In detail the under relaxation is implemented for the impinged film mass, enthalpy and momentum and the evaporated mass, momentum and enthalpy source terms and refers to two subsequent film solutions.
The results of the film model can be monitored by different outputs. One of the most important is the film length which results in the film effectiveness. Every physical aspect which is considered in the model has a direct or indirect influence on the film length. So to compare a film simulation with test results not only the wall heat flux but also the film length is of major interest.

5. Simulation results

To fulfill the main goal of maximum flexibility and very good agreement to test data, the film modeling in Rocflam-II is validated and tested against several different test cases. They are divided on the basis of the thermodynamical state of the film fluid. So liquid, transcritical and gaseous film fluids are simulated. Additionally to the diverse film fluids, the cases have different propellant combinations, different cooling techniques in addition to film cooling, different types of film build up, different load points and different chamber geometries.
The most important aspect in film calculation is a correct simulation of the hot gas side with combustion and the heat transfer into the wall / film. Hence the first and crucial step is always an anchoring of the spray combustion computation on tests without film cooling for validation by the experimental data.
If regenerative cooling is used, the conjugate heat transfer is calculated via a coupling of Rocflam-II for the hot gas side and the one dimensional Nusselt-correlation based coolant flow solver RCFS-II [5] or the commercial 3D CFD tool Ansys CFX [7] for the solid side including regenerative cooling. In case of a radiative cooled chamber, the cooling is calculated in Rocflam-II. Herein also effects like axial heat conduction in the structure are considered.

5.1 Liquid film cooling fluid

The following simulations were conducted with a liquid fluid for film cooling. Three different test cases were simulated and compared to test data.

5.1.1 Liquid film fluid: MMH

The first case is a MMH/NTO thruster with a liquid MMH cooling film combined with radiative cooling on the outer side of the combustion chamber. The film build up occurs via deposition of droplets injected by several holes in the faceplate close to the chamber wall. The film fluid enters the chamber with about 290 K and the chamber mixture ratio is about 1.65 (fuel rich combustion). The combustion modeling in Rocflam-II for the propellant combination MMH/NTO is the global chemistry scheme [6].
Two challenging aspects for the simulation of this chamber should be explicitly mentioned here. One is the extreme exothermic dissociation enthalpy of the gaseous MMH. That means, the liquid MMH film changes his phase condition to a gaseous MMH film, which further heats up and dissociates. This dissociation enthalpy is released close to the liquid film surface and strongly increases the heat flux into the liquid film which, on the other side, accelerates the film evaporation. Another important aspect of this chamber is caused by the radiative cooling. This cooling mechanism is not very effective, which means that the chamber wall without film reaches very high temperatures. Due to the axial heat conduction in the chamber wall the liquid film receives an additional heat flux peak from the wall side around the dry out point.
Because of the straight axial propellant injection at the faceplate, only a few NTO droplets hit the MMH film. Therefore the considered hypergolic reaction in the film is of minor priority in this injection case. Due to the radiative cooling, the measured outer wall temperature is in this case used for comparison with the simulation results. Figure 6 shows on the right side the calculated wall temperature compared to measured data. Here a good agreement especially in the most critical part of the chamber, the throat, can be observed. The film dry out point is a little bit overestimated and can be identified by the sharp increase of the wall temperature. Note that film efficiency determines the maximum throat temperature and hence thruster life, i. e. its simulation capability is crucial for thruster layout.

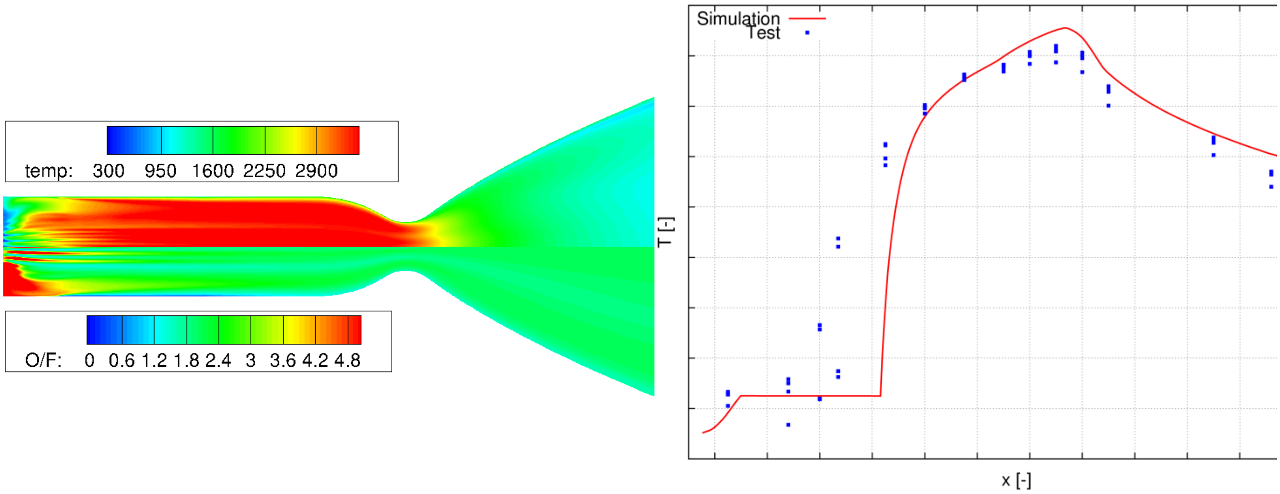

Figure 6: Temperature and mixture ratio distribution (left) and comparison of wall temperature in test and simulation (right)

The left side of Figure 6 shows the corresponding temperature (upper half) and mixture ratio (lower half) distribution in this chamber. Clearly visible is here the cold and fuel rich wall area due to the evaporated liquid MMH film.

5.1.2 Liquid film fluid: NTO

The second test case is also a MMH/NTO engine. Here the liquid film is an oxidizer film, respective a NTO film. The film build up occurs also via droplet deposition on the chamber wall, but these droplets are injected by a single pintle injector in the center of the chamber. That means, the droplets have to cross the whole chamber before reaching the wall. Hence a well validated Lagrange module is essential to simulate the correct droplet deposition. The initial film temperature and the initial film mass therefore strongly depend on the droplet tracking and heat up / vaporization within Rocflam-II.
The NTO enters the chamber with about 290K and the chamber mixture ratio is about 1.65 (fuel rich combustion). The combustion modeling is, as in the first test case, realized with a global chemistry scheme.

A special challenge in this test case is modeling the propellant injection for the pintle injector. The NTO which enters the chamber through the faceplate annulus meets the MMH on the pintle tip. The MMH enters the chamber perpendicular to the chamber axis through small discrete windows. Due to the discrete windows a part of the NTO meets the MMH on the pintle tip and both are redirected, the other part of the NTO still flows straight into the chamber.
Therefore hydraulic CFX simulations were performed to calculate the injection angle after interaction of the NTO and MMH. After interaction at the pintle tip, both propellants are heading parallel with a common angle into the chamber. Due to hypergolic reactions between the NTO and MMH jets and the subsequent expansion on the contact surface of both propellants they get separated. This effect resembles to the known “blow apart” effect. Figure 7 depicts schematically on the left side the finally resulting spray injection and on the right side the MMH (red) which enters the chamber through discrete windows as well as the NTO (blue) through the annulus.

Figure 7: Modelling of the pintle injection (left). Annulus injection of NTO (blue) and MMH (red) injection through windows (right)

The separated MMH droplets later reach the NTO film and they are reacting hyergolically, which reduces the film mass thus resulting in a shorter liquid film. So in this test case the spray modeling has an exceptional importance, not only for the hot gas side, but also for the liquid film.
The unusual case of oxidizer film in a fuel rich chamber results in another special effect. As one can see in Figure 8 (left) two areas with stoichiometric mixture ratio of 2.5 exist where the combustion temperature is very high. The evaporated NTO film fluid reacts with the fuel rich main flow and results in a very hot area close to the chamber wall. The second hot area is caused by the oxidizer rich center flow and the fuel rich core flow, which is a result of the pintle injection and the relatively complex flow field. On the right side of Figure 8 the wall temperature of this combustion chamber is plotted. Clearly visible is the heat up of the film fluid at the beginning, the nearly constant temperature (saturation temperature of NTO) during evaporation and the dry out point of the liquid film.

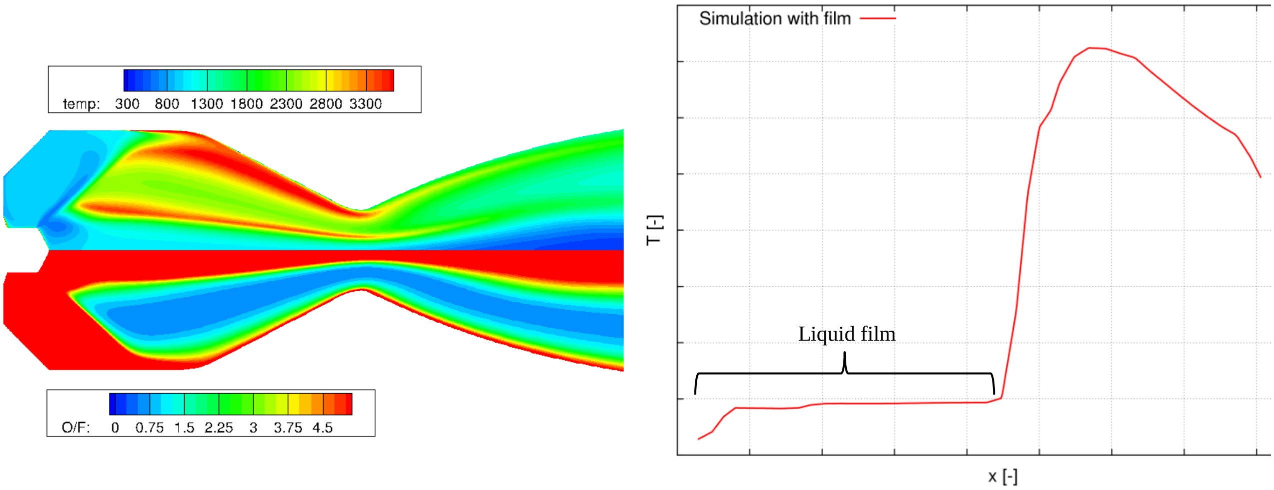

Figure 8: Temperature and mixture ratio distribution (left) and wall temperature (right)

The differences between simulation results compared to the experimental test data are listed in Table 2 showing a very good agreement with the tests.

$\Delta\ p_c$ [%]	$\Delta\ \eta_{c*}$ [%]	$\Delta\ l_{Film}$ [%]
0.2	+2.1	-0.8

Table 2: Comparison between test and simulation

5.1.3 Liquid film fluid: Kerosene

The third example with liquid film is a subscale GOX/kerosene rocket combustion chamber with kerosene film operated by the Institute for Flight Propulsion (LFA) at the Technical University Munich. The film is deposited by a film device located downstream of the second of four chamber segments. The propellant injection occurs via a single double swirl element. The chosen load point has a chamber pressure of 20bar and a mixture ratio of 2.9 in the throat. The GOX/kerosene combustion is modeled in Rocflam-II with an equilibrium based presumed PDF approach [9]. The LFA combustion chamber uses water cooling as primary cooling technique of the chamber wall which has to be considered in the CFD simulations to get a realistic wall temperature boundary condition for the wall heat flux calculation. Therefore all simulations were coupled with the three-dimensional CFD tool Ansys CFX for conjugate heat transfer calculations. Figure 9 shows the coolant flow scheme of the simulations.

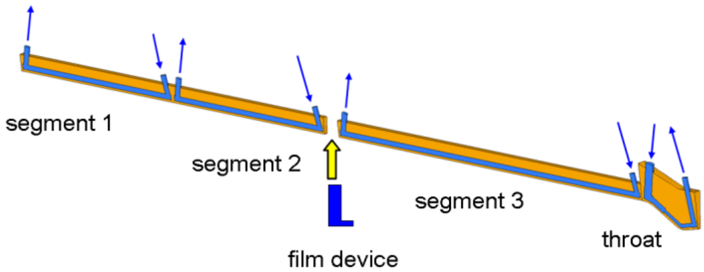

Figure 9: Cooling flow scheme for CHT calculation

In Figure 10 the calculated wall heat fluxes with and without film are compared against measured values from the hot firing test. The red solid curves are the simulated integrate heat load values in each segment, the red dashed curves are the calculated heat flux profiles from Rocflam-II. The blue curves are test data for each segment calculated via the heat up of the regenerative cooling fluid. The spray setting anchoring of Rocflam-II was done on the “no film” test case to validate the hot gas side. The heat flux steps between the first/second and second/third segment are caused by the wall roughness change of the different segments. The roughness of used and new chamber segments was measured and considered in the simulation [4]. Almost perfect agreement of test and simulation is visible, demonstrating that GOX/kerosene combustion and the heat transfer is modeled very well.

On the right side of Figure 10, the simulation result with 7% liquid film, that means 7% of the whole injected kerosene is film, is shown. Due to the more ox-rich injection at the faceplate (constant O/F in the throat), the measured heat flux in the film case is higher in the first segment compared to the no film case. The beginning of the liquid kerosene film downstream of the second segment results in a sharp decrease of the wall heat flux which is calculated between the liquid film and the regenerative cooled chamber wall (a). After injection of kerosene film the fluid heats up and accelerates because of the shear stress of the hot gas flow (b). This increases the heat flux into the wall. At the beginning of the evaporation at saturation temperature, two effects start and result in a decrease of the heat flux (c): First the film stay on constant temperature, but the regenerative coolant and the chamber wall below still heat up, this leads to a decrease of the temperature difference and to a decrease of the heat flux. The second effect reduces the film velocity and therefore the wall heat flux, which is calculated via Nusselt correlation. The evaporated film fluid creates a huge mass, enthalpy and momentum source term compared to the existing gas in the first computational cells. This results in a relatively strong flow perpendicular to the chamber wall, which reduces the shear stress between hot gas flow and film surface and thus results in a reduction of the film velocity.

The dry out point is characterized by a sharp increase of the wall heat flux (d). Shortly after the dry out the heat flux increases to the same level as without film cooling. In this area the vaporized kerosene mixes with the hot gas until this gaseous film vanishes. The reason for the quite low cooling efficiency of the vaporized kerosene and the sharp increase of the heat flux is of the very high critical temperature of kerosene. In a regeneratively cooled chamber without film, the temperature close to the chamber wall is quite similar to the temperature of the evaporated kerosene. Hence the heat flux increase downstream of the dry out point is pretty sharp.

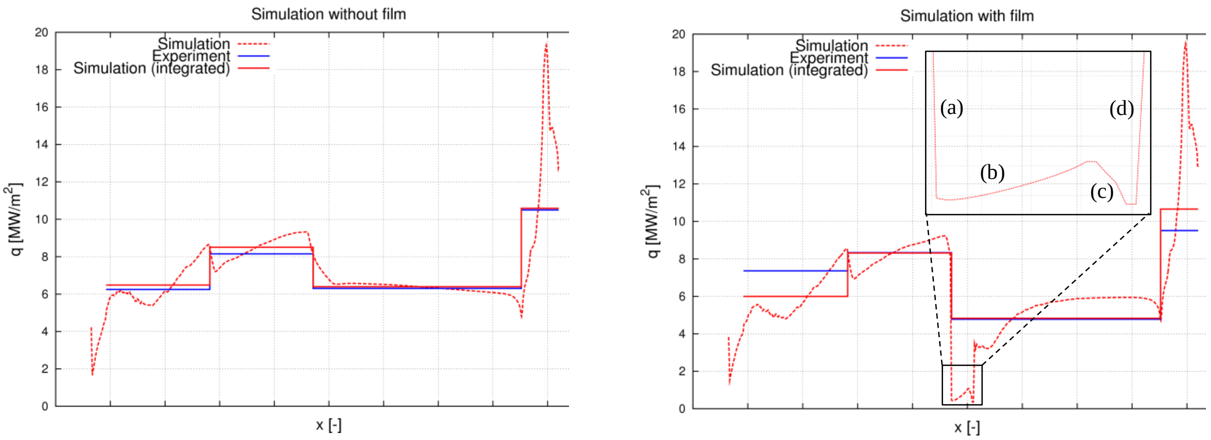

Figure 10: Comparison of simulated and measured wall heat flux without film (left) and with film (right)

Compared to the test data, the calculated heat flux shows a very good agreement in the second and third segment. Taking into account the standard deviation on the measured values in the throat section of up to 17% around the plo tted value, the calculated heat flux value is also within the measured range. The underestimation in the first segment is currently not clarified and will be further investigated.

Figure 11 shows the temperature and mixture ratio distribution in the LFA combustion chamber. On the left side one can see the whole chamber and on the right side a zoom on the liquid film cooled area. Clearly visible is the influence of the evaporating film on the hot gas close to the film. There is a significant difference in temperature and more fuel rich layer due to the cooler evaporated film fluid.

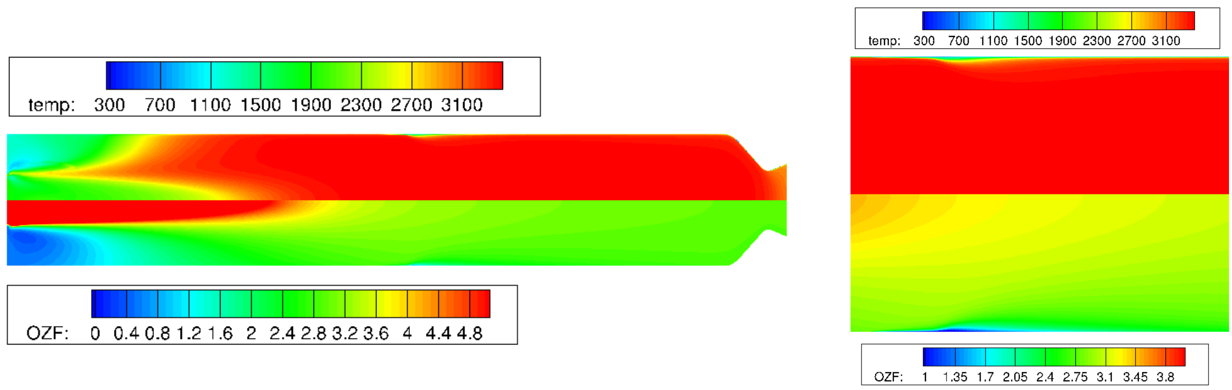

Figure 11: Temperature and mixture ratio distribution (left) and a zoom on liquid film (right)

5.2 Transcritical film cooling fluid

The simulations described in this chapter are performed with transcritical fluids as cooling film. This means that the chamber pressure is above the critical pressure of the fluid and the temperature is below the critical one. All simulations are conducted with the extended liquid film model already described in chapter 4.

5.2.1 Transcritical film fluid: Kerosene

The first test case is the known LFA GOX/kerosene combustion chamber which is operated also at supercritical pressure. At the chosen load point with p_c=60bar and an O/F=2.9 at the throat, the kerosene film is in a transcritical state (p_{crit}=23.44bar and T_{crit}=684K). In this case 17% of the total injected kerosene mass is film. The conjugate heat transfer is again simulated via coupling of Rocflam-II for the hot gas side and Ansys CFX for the structure including the regenerative cooling. The considered wall roughness and the anchored spray combustion settings are identical to the liquid LFA test case with 20bar described in chapter 5.1.3.

Figure 12 shows a comparison of the wall heat fluxes measured in tests and calculated via Rocflam-II. The wall heat flux below the film increases due to acceleration of the film and heat up of film fluid. The difference to the liquid kerosene film is that the evaporation of the fluid starts already before it reaches the critical temperature and at the critical temperature the remaining film mass evaporates instantly. In both cases, the heat flux in the cylindrical part of the chamber agrees very well with integrally measured test data. Consider the deviation on the test data in the throat segment the calculated heat flux is also within the measured range.

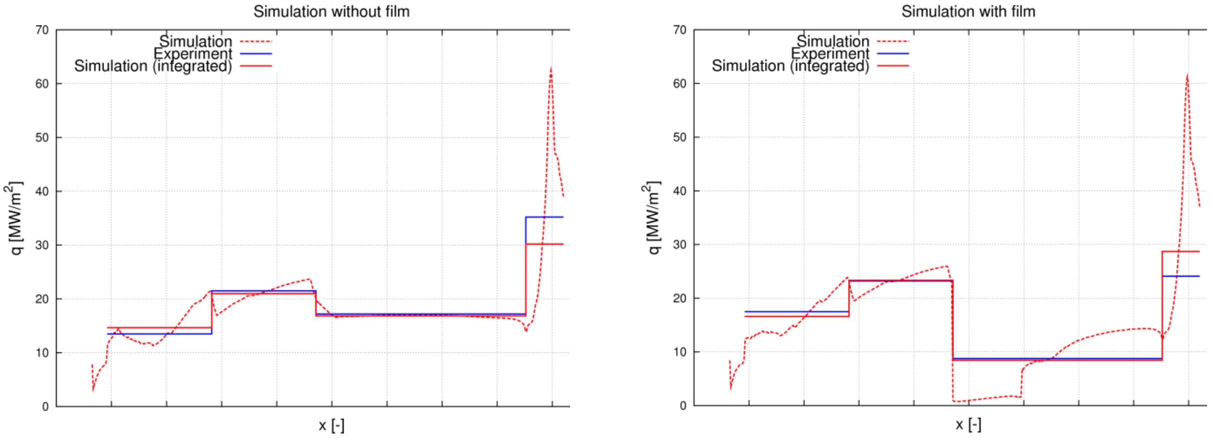

Figure 12: Comparison of simulated and measured wall heat flux without film (left) and with film (right)

Figure 13 depicts the temperature and mixture ratio distribution in case of film cooling. Comparable to other film cooled cases, a clear influence of the film cooling can be seen by a decrease of the temperature and a more fuel-rich mixture ratio. The cooling effect is noticeably higher compared to the liquid kerosene test case because of the higher film mass flow of 17%.

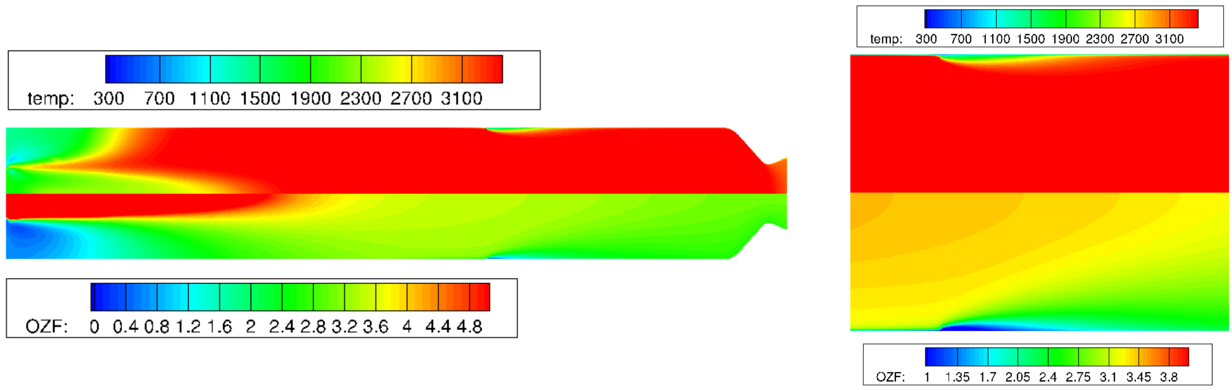

Figure 13: Temperature and mixture ratio distribution (left) and a zoom on transcritical film (right)

5.2.2 Transcritical film fluid: Methane

The second transcritical film fluid test case is conducted with methane as cooling fluid. These simulations are performed by a special film test case for detailed investigation of the film module. The test case is a cylindrical pipe with a hot gas ($T_{Hot\ Gas}$=3000K, $u_{Hot\ Gas}$=100m/s) inflow with a mixture ratio of 2.45 and a chamber pressure of 60bar and an initial film temperature of 170K. This simple test case is reduced to a minimum of complexity to focus strictly on the film calculation. Therefore no spray is injected, the film enters the chamber through a film device, the chemistry influence is pretty low and a constant predefined wall temperature of 700K is used. In Figure 14 the blue curve represents the calculated wall heat flux without film cooling and the red curve shows the case with film cooling. The definition of the heat flux in Rocflam-II is egocentrical, this means that a heat flux which is directed into the chamber is negative. Downstream of the film injection the film accelerates very strongly caused by the fast hot gas flow and results in an increased heat flux into the film up to the dry out point. Downstream of the dry out point a noticeable heat flux reduction can be observed caused by the gaseous methane film.

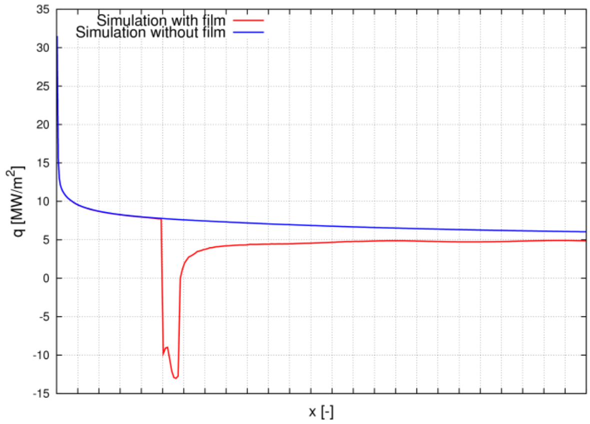

Figure 14: Comparison of simulation with (red curve) and without (blue curve) methane film cooling

Figure 15 shows the temperature and mixture ratio distribution in this film test case with film on the right side and without film on the left side. The evaporated film fluid causes a very strong temperature and mixture ratio stratification close to the wall and provides a good cooling effect.

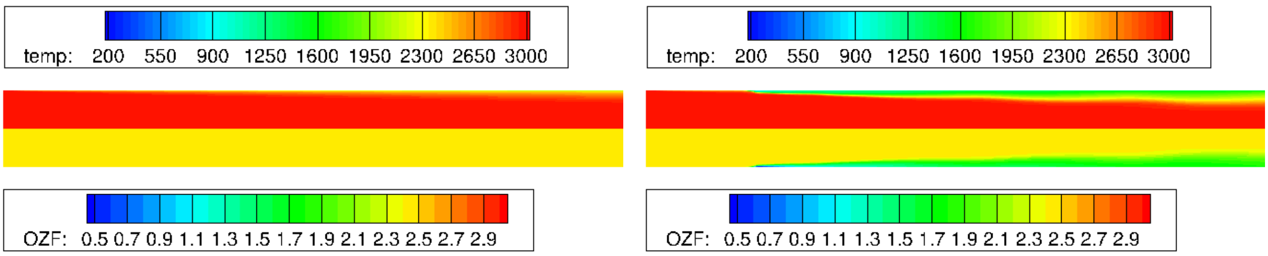

Figure 15: Temperature and mixture ratio distribution without (left) and with (right) transcritical film cooling

5.3 **Gaseous film cooling fluid**

As example for gaseous film cooling simulations within a H_2/O_2 combustion chamber are performed. The chamber pressure is p_c=100bar with a mixture ratio of O/F=6 and a gaseous H_2 film fluid. The film enters the chamber through discrete slots in the faceplate with an initial temperature of 100K. The measured wall heat flux is calculated via the heat up of the regenerative cooling fluid in each segment of the chamber. These are first preliminary simulations, therefore they are conducted with a prescribed wall temperature and a currently not adapted wall roughness. Figure 16 depicts the calculated wall heat fluxes compared to measured heat fluxes for the test case with and without film cooling.

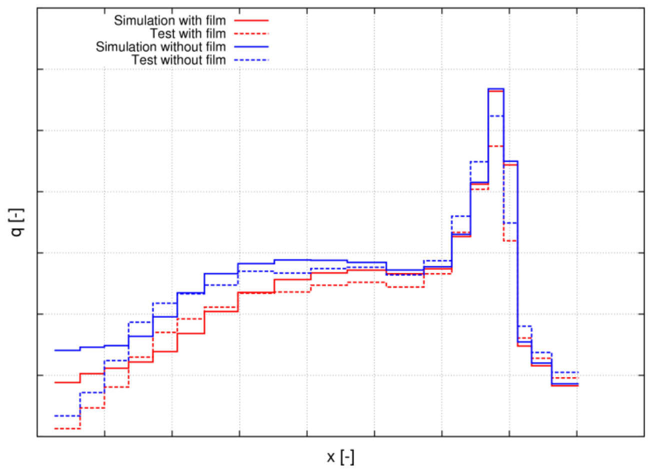

Figure 16: Wall heat flux comparison between test and simulation, with and without gaseous film cooling

In the cylindrical part of the chamber the relative reduction of the heat flux due to film cooling is reproduced quite well. In the throat section the reduction is much smaller than measured in experiments, also the absolute value is over predicted in the simulations. To improve the simulation the next step is to consider the measured wall roughness and the regenerative cooling of the chamber. Here the conjugate heat transfer will be calculated via coupling of Rocflam-II and RCFS-II or Ansys CFX.
In Figure 17 one can see the temperature and the mixture ratio distribution in the combustion chamber with (right) and without (left) gaseous film cooling. As expected for the film cooling case a fuel rich and cooler layer close to the wall exists, which protects the wall against the hot gas and reduces the heat load into the wall.

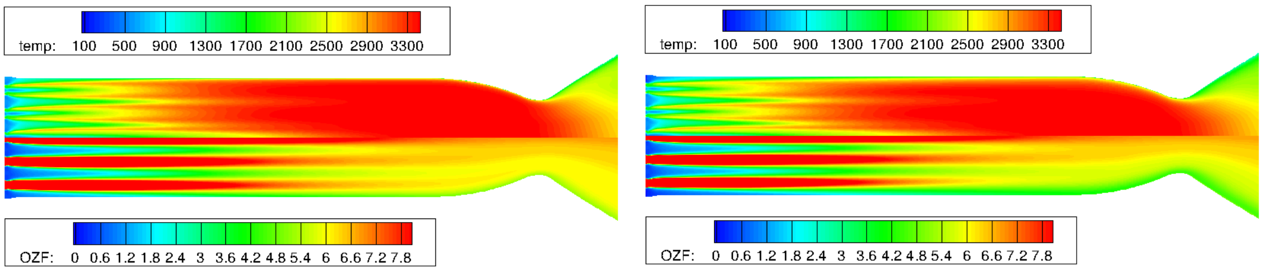

Figure 17: Temperature and mixture ratio distribution without (left) and with (right) gaseous film cooling

5.4 **Inert cooling fluid**

The primary application of the film model is the simulation of reactive film cooling fluids in subsonic flow conditions. However the model has also the ability to calculate inert cooling films in hypersonic flow, for example a liquid water film entering a radiative cooled nozzle extension. The introduced test case is a H_2/O_2 combustion chamber with a subcritical pressure for the film fluid and a partly radiative cooled nozzle. The liquid water film enters the nozzle extension with an initial temperature of 350K. As in the other H_2/O_2 test case in chapter 5.3 the combustion is calculated using the ppdf equilibrium based chemistry.
Figure 18 depicts the wall temperature with (red curve) and without (blue curve) H_2O film. The sharp increase of the temperature without film is caused by the interface between the regenerative and the radiatively cooled chamber part. With film cooling, the wall temperature is clearly lower than without film and the sharp increase of the temperature indicates the dry out point of the liquid film.
The difference between measured film length and calculated film length is in the range of few percent and shows a very good agreement for this extraordinary usage of the film model.

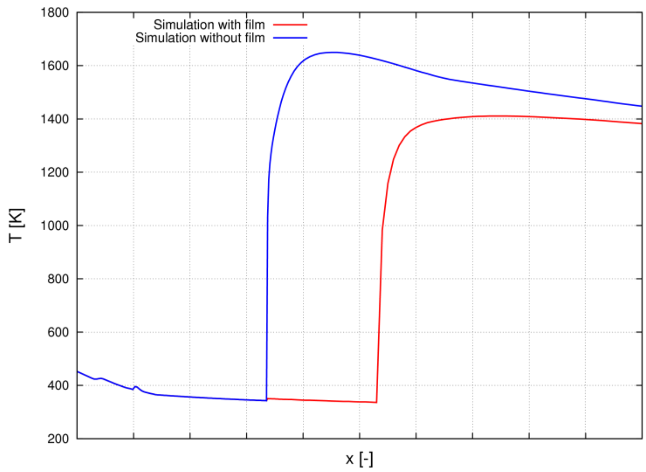

Figure 18: Wall temperature with (red curve) and without (blue curve) liquid water film

6. Conclusion

First steps of the ambitious goal to simulate many different film cooling applications in rocket combustion chambers with the CFD tool Rocflam-II are presented in this paper. Different film cooled combustion chambers are simulated and partly validated against test data. In addition to the different thermodynamical states of the cooling fluids, different film fluids, different film build up methods, different combustion models with several propellant combinations and different cooling combinations with considered conjugate heat transfer are successfully simulated and tested.

In summary the film cooling simulations for the different cases show a good agreement to experimental data and prove the ability of the Rocflam-II tool package to simulate a broad range of cooling films at rocket combustion chamber conditions. Such a simulation capability is crucial for the European leader of rocket thrust chambers to design its products properly.

Next steps will be the simulation and final validation of liquid and transcritical methane under rocket combustion chamber conditions. Also the simulation of the gaseous film test case will be improved by the consideration of the conjugate heat transfer via coupling with Ansys CFX or RCFS-II.

7. Acknowledgement

This work was performed within the national technology program TEKAN 2010 II. This program is sponsored by the German Space Agency, DLR Bonn, under contract No. 50RL0710.

References

[1] J. Nahstoll: "Untersuchung von Flüssigkeits-Filmströmungen bei hohen Verdampfungsraten im Hinblick auf Filmkühlung."*DFVLR-FB 88-08, 1988*

[2] S. A. Fisher, D. L. Pearce: "An annular flow model for predicting liquid carryover into austenitic superheaters." *Int. J. Multiphase Flow Vol. 19, No. 2, pp. 295-307. 1993*

[3] M. Frey, B. Kniesner, O. Knab."Consideration of Real Gas Effects and Condensation in a Spray Combustion rocket Thrust Chamber Design Tool." EUCASS2009-85. *3th European conference for Aerospace Sciences, Versilles, France. 2009*

[4] B. Kniesner, M. Frey, O. Knab."Vorhersage des Wandtemperaturverlaufs in Raketenbrennkammern unter Berücksichtigung zeitlich und örtlicher Wandrauhigkeitsentwicklungen." *DGLR2010-161237. 59. DeutscherLuft- und Raumfahrtkongress. Hamburg, Deutschland. 2010.*

[5] C. Mäding, D. Wiedmann, K. Quering, O. Knab."Improved Heat Transfer Prediction Engineering Capabilities for Rocket Thrust Chamber Layout." *EUCASS2009-90. 3th European conference for Aerospace Sciences, Versilles, France. 2009*

[6] T. Aichner, H. Riedmann, B. Kniesner, O. Knab."Development of an advanced global MMH-NTO combustion model for CFD application." *SPC2356608. Space Propulsion Conference, Bordeaux, France. 2012*

[7] Homepage: www.Ansys.com

[8] G. Schlieben, C. Kirchberger, H.-P. Kau, C. Höglauer, O. Knab."Experiments on film cooling in a GOX-kerosene rocket combustion chamber". *SPC2394111. Space Propulsion Conference, Bordeaux, France. 2012*

[9] C. Höglauer, B. Kniesner, O. Knab, G. Schlieben. „Simulation of liquid and transcritical cooling films in rocket combustion chambers". *SPC2357891. Space Propulsion Conference. Bordeaux, France 2012.*