NUMERICAL INVESTIGATION ON THE POTENTIAL OF STEAM COOLING FOR THE SKYLON SPACEPLANE IN HYPERSONIC FLOW

Th. Eggers*
***German Aerospace Center (DLR) – Institute of Aerodynamics and Flow Technology, Lilienthalplatz 7, 38108 Braunschweig, Germany**
Thino.Eggers@dlr.de

Keywords: *hypersonic flow, reacting flow, effusion cooling, active cooling, Skylon*

Abstract

A preliminary study to assess the existence of shock-shock interactions which might jeopardize the SKYLON structure indicated that the junction of nacelle and wing is the most critical region of the vehicle. Therefore, within the present paper the potential of effusion cooling with steam to control the heat flux in these regions of SKYLON is investigated. The simulations with the three-dimensional finite volume Euler / Navier-Stokes solver TAU investigate the most critical non-equilibrium hypersonic flow conditions along the SKYLON re-entry trajectory. They underline the feasibility of effusion cooling to protect critical structure regions of spacecraft.

1 Introduction

SKYLON is a single stage to orbit (SSTO) winged spaceplane [1] designed to give routine low cost access to space. At a gross take-off weight of 275 tonnes of which 220 tonnes is propellant the vehicle is capable of placing 12 tonnes into an equatorial low Earth orbit. The vehicle configuration consists of a slender fuselage containing the propellant tankage and payload bay with delta wings located midway along the fuselage carrying the SABRE engines in axisymmetric nacelles on the wingtips. The vehicle takes off and lands horizontally on its own undercarriage. The fuselage is constructed as a multilayer structure consisting of aeroshell, insulation, structure and tankage. SKYLON (**Fig. 1**) employs extant or near term materials technology in order to minimize development cost and risk. The SABRE engines have a dual mode capability. In rocket mode the engine operates as a closed cycle liquid oxygen/liquid hydrogen high specific impulse rocket engine. In airbreathing mode (from takeoff to Mach 5) the liquid oxygen flow is replaced by atmospheric air, increasing the installed specific impulse 3-6 fold.

Fig. 1: The SKYLON spaceplane

A preliminary aerodynamic study to assess the existence of shock-shock interactions which might jeopardize the SKYLON structure indicated that the junction of nacelle and wing is the most critical region [2], **Fig. 2**.

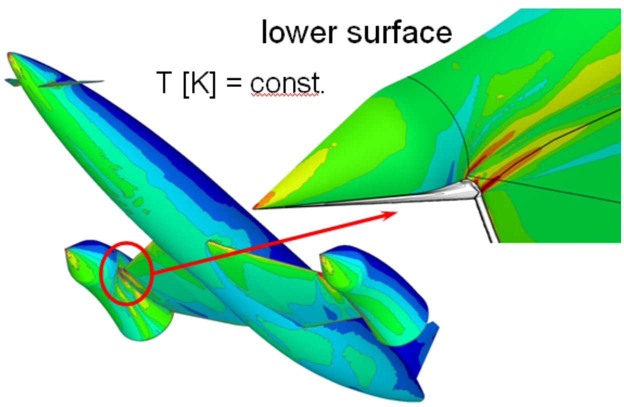

Fig. 2: View of the thermally critical region

Therefore, the present paper focuses on this area. The study is divided into three steps. First of all, in order to outline the principle effects of the effusion cooling, a simple test case of a ramp is investigated for different cooling gases (air, nitrogen, and steam) and mass flows. Thereafter, in the second step the region with complex shock-shock interactions in the junction of nacelle and wing region is discussed based on the assessment of a simplified configuration and finally, the experiences of these parameter studies are applied to the full SKYLON vehicle to prove the feasibility of effusion cooling for regions with excessive heating rates.

2 The DLR TAU Code

The DLR TAU code [3] is a three-dimensional finite volume Euler / Navier-Stokes flow solver based on hybrid structured / unstructured grids. The advantage of a hybrid code is to combine the superior quality of structured grids to resolve wall heat transfer or wall shear stress with the ability of unstructured grids to resolve efficiently flow regions of particular interest.

The TAU code is composed of three independent modules: a pre-processing module, the solver and a grid adaptation module. The pre-processing is decoupled from the solver in order to allow performing grid partitioning and calculating the metrics on a different platform than used by the solver. This provides the possibility to run the solver independently, which in terms of CPU-time requirements is the most critical part of the system. Thus, it is possible to run large-scale calculations also on distributed memory machines with limited memory on each node. The third module is for grid adaptation. It detects regions with insufficient grid resolution and performs local grid refinement. The initial solution is interpolated to the adapted grid.

The TAU code uses a second order finite-volume discretisation. Different central and upwind schemes, like AUSMDV are implemented for sub- and transonic respectively super- and hypersonic flow. Second-order accuracy for upwind schemes is obtained by MUSCL extrapolation, in order to allow the capturing of strong shocks and contact discontinuities. A three-stage explicit Runge-Kutta scheme as well as a point implicit LUSGS scheme are options to advance the solutions in time for steady flow fields. For convergence acceleration local time stepping, implicit residual smoothing and multigrid are implemented. Fast and accurate transient flow simulations are computed by a Jameson tape dual time stepping scheme, as an implicit algorithm, which is not restricted in the choice of the smallest time step in the flow field. To overcome this limit the time derivative in the Navier-Stokes equations is discretized by a second order backward difference, resulting in a non-linear equation system, which converges towards the subsequent time step by using an inner pseudo time. Within this inner loop, all mentioned acceleration techniques are applicable. TAU is extended for hypersonic flow field applications and models to perform chemical and thermal equilibrium and chemical and thermal non-equilibrium flow computations. In addition to a number of one- and two-equation turbulence models, Detached Eddy Simulation is implemented in the code.

The DLR Tau code includes extensions for chemical and thermal non-equilibrium flows in high enthalpy aerothermodynamics. The flow is considered to be a reacting mixture of thermally perfect gases. A dedicated transport equation is solved for each individual species. The chemical source term in this set of transport equations is computed from the law of mass action by summation over all participating reactions. The forward reaction rate is computed from the modified Arrhenius law and the backward rate is obtained from the equilibrium constant which is computed directly from the partition functions of the participating species.

The thermodynamic properties (energy, entropy, specific heat) are calculated from the partition functions or external lookup tables for each individual species in the reacting gas mixture. The advantage of this approach is its high flexibility. Extensions such as multi temperature models to handle thermal non-equilibrium effects are easily possible. Having determined the mixture composition and the thermodynamic

state of the individual species, the properties of the reacting gas mixture are then computed using suitable mixture rules such as proposed by Wilke for the viscosity and by Herning and Zipperer for the heat conductivity.
For fully catalytic wall boundaries, a Dirichlet condition for the species mass fractions is set according to the local equilibrium composition. Non-catalytic walls are modeled using a von Neumann boundary condition imposing vanishing wall-normal gradients of the species mass fractions.
The species diffusion fluxes are modeled using Fick's law applying an averaged diffusion coefficient for all species. This approximate diffusion coefficient is computed using the mixture viscosity and constant laminar Schmidt numbers, *Sc*. Turbulent diffusion is modeled in an analog way by computing a turbulent diffusion coefficient, D_T, from the eddy viscosity, μ_T, and the turbulent Schmidt number, Sc_T. The eddy viscosity is derived from the applied turbulence model (e.g. computed from the turbulent kinetic energy and the length scale when applying a *k*-*ω* model).
Thermal non-equilibrium flows are computed by solving an additional transport equation for the vibrational energy of each molecule in non-equilibrium. The relaxation of vibrational energy is modeled according to the Landau-Teller approach and the vibrational relaxation times are obtained from the correlation of Millikan and White.

2 Introduction into the procedure to assess of effusion cooling

The implementation of the cooling interface in TAU is realized by modification of the boundary condition for a viscous wall. The given cooling mass flux is included in the mass balance and is assumed to enter the flow field normal to the wall, so the tangential components of the velocity are still zero at the wall. The momentum of the gas is set according to the mass flux, in other words the wall normal velocity is set equal to the given mass flow divided by the iterated local density. In equation (1) this correlation is expressed.

$$m_c = \rho \cdot \mathrm{v} \cdot A_c \tag{1}$$

Consequently, for the numerical computation the following values are flexible to define:

- cooling area A_c, results from the geometry
- cooling mass flow m_c
- cooling temperature T_c

Results of the numerical calculations are:

- density ρ, depends on the flow condition (e.g. Alt, α, M)
- velocity v, calculated from $v=m_c / (\rho A_c)$
- heat flux of the cooling panel
- temperature of the cooling panel

In order to outline the principle effects of the effusion cooling, in the following a simple test case is investigated. The test case is illustrated in **Fig. 3** and consists of a simple ramp flow where an oblique shock is developing. For this case different cooling gases (air, nitrogen, and steam) and mass flows are investigated. The free stream conditions are selected as M=25, ρ=1.3e-4 kg/m³ so that a high wall temperature is generated.

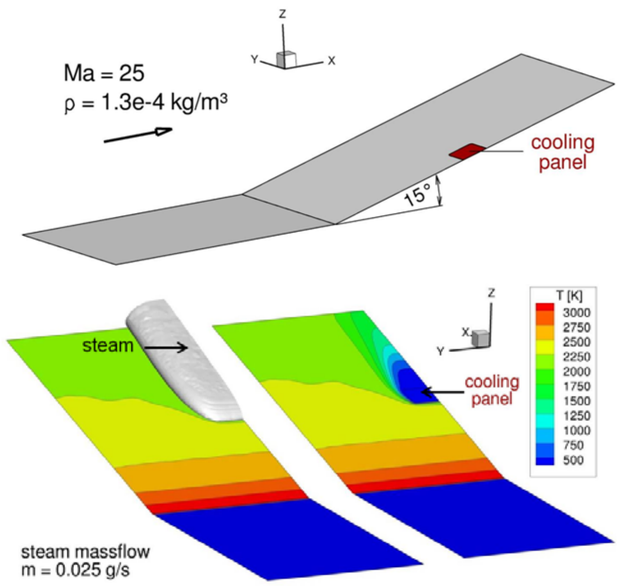

Fig. 3: Wedge geometry with cooling panel

For the injection of the cooling gas a squared panel is installed into the ramp. Along this panel and behind it a cooling film is generated and the wall temperature is decreased (lower right side of **Fig. 3**). However, the cooling effectiveness depends on the applied gas and the chosen mass flow. The effect of different cooling gases is shown in **Fig. 4**. Here, the surface temperatures along the symmetry line (A) without cooling and with injection of air, nitrogen, and steam are compared.

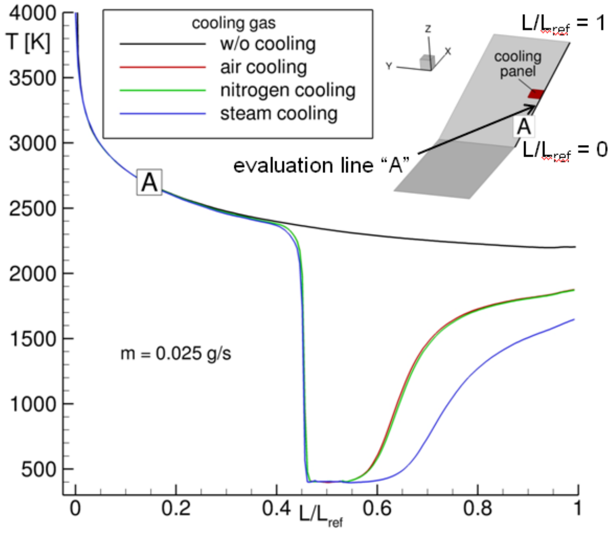

Fig. 4: Influence of the cooling gas on T_{wall}

As air contains nearly 80% of nitrogen and the thermal properties between both gases are very similar, the cooling effectiveness is for the same mass flow almost identical. The second main conclusion is that the cooling effectiveness using steam is much better as in case of air or nitrogen because the molecular weight is lower.
In **Fig. 5** the cooling effectiveness of steam depending on different mass flows (0g/s, 0.005g/s, 0.025g/s, and 0.1g/s) is compared along the symmetry line (A) of the ramp. Based on these three mass flows it can be concluded that an optimal cooling mass flow should be chosen. In case of m=0.005g/s the cooling film is too thin and the wall temperature after the injection panel is similar to the case without cooling. In opposite to that, for m=0.1g/s the mass flow is too large and causes a separation and recirculation flow at the border of the cooling panel. This significantly reduces the cooling effect. The most promising cooling mass flow for this test case and the applied free stream condition is between 0.005g/s and 0.1g/s, for example 0.025g/s.

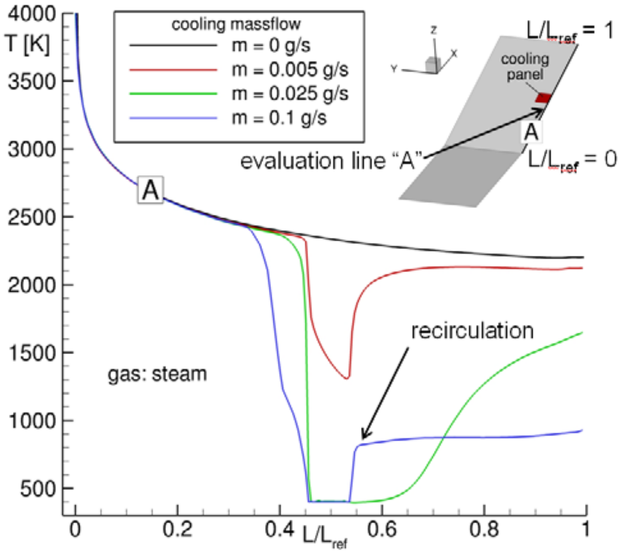

Fig. 5: Influence of the mass flow on T_{wall}

2 Simplified configuration of the wing – nacelle region

The flow topology in the junction between nacelle and wing is very complex due to the appearing shock-shock interactions. Therefore, in a first attempt the complex Skylon geometry is transformed in a simplied test case (double wedge) which enables a parameter study to analyze the cooling possibilities with significantly reduced computational effort. **Fig. 6** indicates that the Skylon wing/nacelle configuration resembles a combination of two wedges.

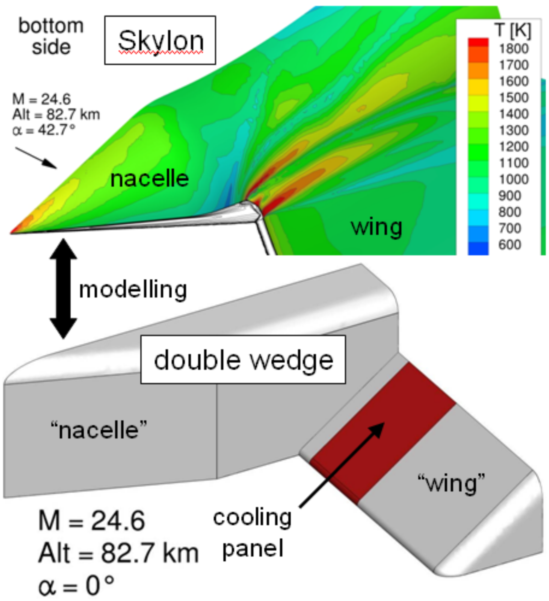

Fig. 6: Introduction of the double wedge test case

The deflection angle of the second wedge ("wing") corresponds approx. to the Skylon wing in combination with α=42.7° as the computations of the test case model are performed with α=0°. The first wedge produces a ("nacelle")-shock which interacts with the ("wing")-shock of the second wedge. The resulting shock-shock interactions are similar to those of the real Skylon spaceplane (compare **Fig. 7**).

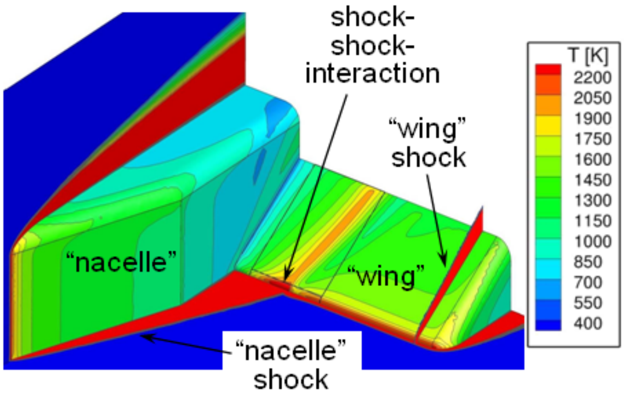

Fig. 7: Visualization of T_{wall} on the double wedge test case

The cooling of the high thermal loads at the leading edge is the objective of the following investigations. In the lower part of **Fig. 6** the two optional cooling panels are red marked. The first one is along the leading edge (A_N=0.0048m²) and the second one is a part of the wedge (A_P=0.0893m²). In the first step it is analyzed whether a cooling of the complete critical region can be established, if the cooling gas is only injected at the leading edge (A_N). For this simulation a steam mass flow of m_c=0.4g/s is selected, which corresponds to an area weighted mass flow of m_c=83.3g/sm² (A_N=0.0048m²). The resulting temperatures and steam outflow velocities along the interpolation line A are illustrated in **Fig. 8** and **Fig. 9**. In addition, the values without cooling (red curve) are shown in order to express the cooling effectiveness. Based on these results it can be concluded that a limited steam injection at the leading edge does not enable to cool the wedge panel too. The temperature reduction is limited to the leading edge itself where it can be reduced from T_N=2500K to T_N=1950K (**Fig. 8**).

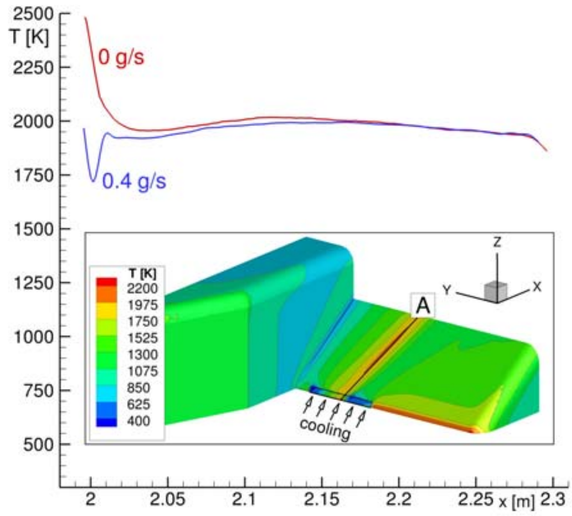

Fig. 8: T_{wall} along line A, m=0.4g/s, steam injected only on the leading edge panel

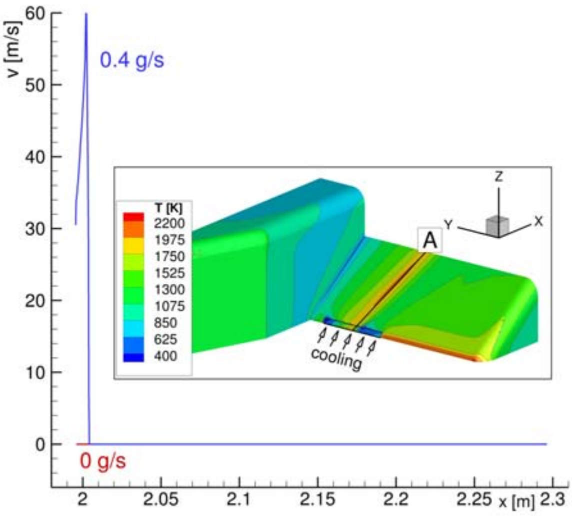

Fig. 9: Steam velocity along line A, m=0.4g/s, steam injected only on the leading edge panel

Fig. 9 shows the outflow velocity of the steam required to generate the target mass flow of m_c=0.4g/s. It varies from 30m/s to 60m/s.

steam mass flow	
[g/s]	[g/sm²]
0.1	1.1
0.5	5.3
1	10.6
2	21.3
4	42.4
8	85.0
16	170.0

Tab. 1: Steam mass flows for leading edge and wedge panels (A_c=A_N+A_P=0.0941m²)

Since, the cooling effect is restricted to the leading edge panel the steam has to be injected through the wedge panel too. This is investigated for different steam mass flows, summarized in **Tab. 1**.

Fig. 10 exemplarily shows the temperature distribution for cooled leading and wedge panels for cooling steam mass flows of m_c=1g/s and 16g/s. For the high mass an extensive temperature reduction is obtained.

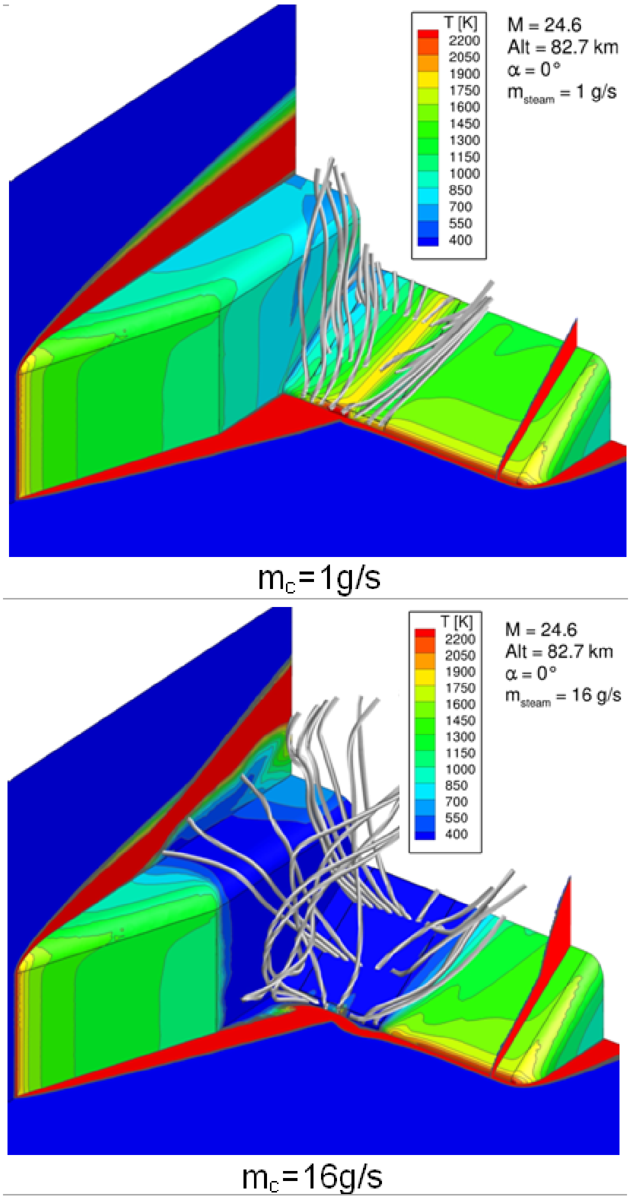

Fig. 10: Surface temperatures for coolant mass flows of m_c=1g/s and m_c=16g/s

The cooling effectiveness for all mass flows is summarized in **Fig. 11**. Here, the worst case trajectory condition for Skylon at Alt=82.7km is assessed. In case of no cooling, the temperature of the leading edge reaches value in the order of T_N≈2500K. The temperature on the wedge cooling panel is heating up to T_P≈2000K.

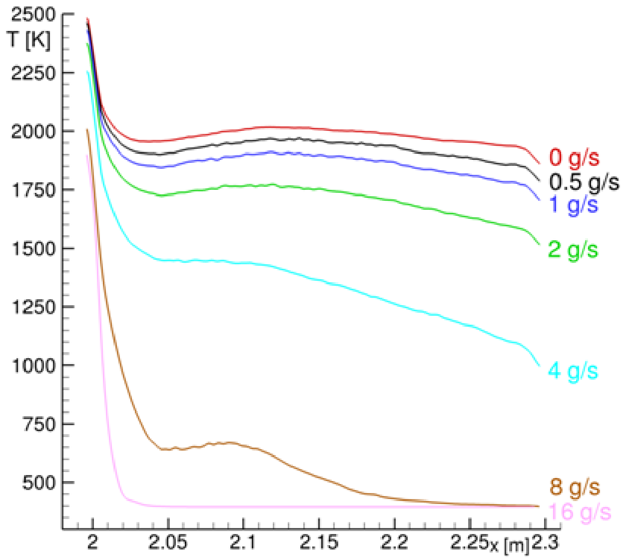

Fig. 11: Temperatures along line A for different mass flows, Alt=82.7km

If both surface panels are taken into account for the active cooling the wall temperature can be reduced depending on the applied steam mass flow. However, in order to produce a clearly sufficient cooling effect a steam mass flow of at least m_c=4g/s which corresponds to a coolant consumption of 42.4g/sm² is required. Nevertheless, even for this value, the leading edge temperature exceeds T_N=2200K. Therefore, at least in the leading edge region, the coolant mass flow has to be increased to obtain realistic surface temperatures.

3 Active cooling of the Skylon spacecraft

For the flow topology around the investigated Skylon configuration in the critical flight point (Alt=82.7km, M=24.6, and α=42.7°) very high gas temperatures within the flow field are present (**Fig. 12**). These are illustrated along two lines (A and B) below the wing in the vicinity of the shocks of nacelle and wing.

The gas temperatures are up to T=5000K and therefore, the gas is influenced by vibrational excitation and dissociation effects. Due to the low flow density (ρ_∞=1.34E-05kg/m³) and the high flow velocity (v_∞=6570m/s) a non-equilibrium flow is likely. In addition, to predict the feasibility of steam cooling for the Skylon spacecraft the non-equilibrium solver of the TAU-Code anyhow has to be applied, because two different gases (air as a combination of N_2, O_2, NO, N, O and steam H_2O) have to be modelled.

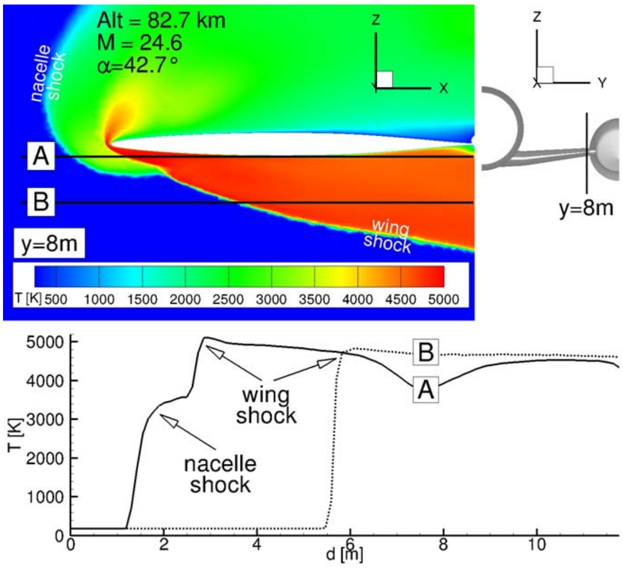

Fig. 12: Gas temperatures in the vicinity of the shock-shock interactions

Consequently, in the following an equilibrium simulation (**Fig. 13**) and a non-equilibrium flow calculation (**Fig. 14**) are compared for the critical flight point.

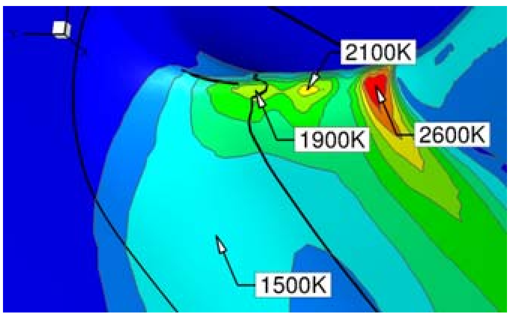

Fig. 13: Surface temperature, equilibrium flow

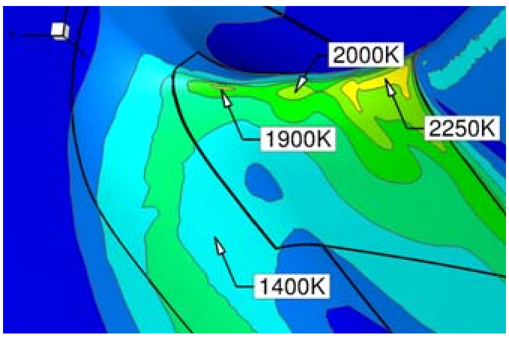

Fig. 14: Surface temperature, non-equilibrium flow

To ensure the worst case with respect to the surface temperatures in case of a non-equilibrium flow, the wall is assumed to be fully catalytic. As a result of the non-equilibrium gas effects, the wall temperature at the leading edge of the wing is significantly decreased from T_{eq}=2600K to T_{noneq}=2250K. Outside the leading edge region the flow is not affected by non-equilibrium effects.
The mass flow rates applied for the assessment or the steam cooling are listed in **Tab. 2**.

steam mass flow	
[g/s]	[g/sm²]
6	10
15	25
31	50
62	100
93	150

Tab. 2: Investigated steam mass flows, A_c=0.6161m²

Within the following discussion the cooling effectiveness is compared along lines A – C which are illustrated in the corresponding pictures. The cooling panel is introduced into the leading edge and its main part is at the lower surface where, due to the high angle of attack of (→α=42.7°) the major heat loads are present. The complete area of the cooling panel is A_c=0.616m². The temperature of the cooling gas (steam) is assumed to T_c=373K.
In **Fig. 15** the effect of the steam mass flow on the surface temperature is illustrated. As expected and previously described for the simplified double wedge test case the cooling effectiveness increases with increasing mass flow rates. Especially for the case m_c=62g/s which corresponds to 100g/sm², nearly the complete panel and the area in its vicinity can be cooled to less than T=500K. Only close to the hot spot of the shock-shock interaction the temperature remains at approx. T=1500K.

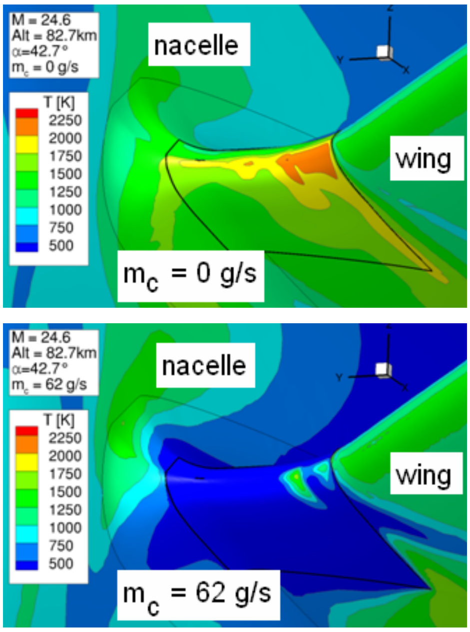

Fig. 15: Influence of the coolant mass flow on the surface temperature

In order to interpret the cooling effect in detail the numerical results for different coolant mass flows are compared along three lines which are going to be introduced in the following pictures. Line A, runs along the leading edge of the wing and catches all temperature maximums. The start point of this line is at the wing and the end point is located at the nacelle. The remaining lines, B and C, are located on the lower side of the wing. For these lines the start points are at the nacelle and the end points are at the wing. Line B and C run parallel to the y-axis and cross the cooling panel at different positions (line B: x=44.42m and line C: x=44.62m).

The results of the surface temperature and the required steam velocity along the interpolation line are illustrated in **Fig. 16** and **Fig. 17** (→ line A), **Fig. 18** and **Fig. 19** (→ line B), and **Fig. 20** and **Fig. 21** (→ line C). Additionally, all diagrams include the case without cooling (m_c=0g/s) in order to enable an evaluation of the cooling effectiveness.

- **Line A (leading edge of the wing/cooling panel)**

Without active cooling the maximum temperature along the leading edge reaches T=2250K (**Fig.16**). In case of an established cooling mass flow rate, it can be reduced. However, the numerical calculations pointed out that at least a steam mass flow of m_c=31g/s corresponding to 50g/sm² is required in order to reduce the maximum temperature of the leading edge below T≈1900K. If a steam mass flow of m_c=62g/s, (100g/sm²) is applied, the maximum temperature of the leading edge is T≈1600K. A reduction of the maximum temperature to T≈1000K is even possible, if the cooling mass flow is increased to m_c=93g/s (150g/sm²). The outflow velocities of the steam, which follow from the selected cooling mass flow ($v=m_c/(\rho A_c)$), vary from approximately 5m/s to 85m/s (**Fig.17**).

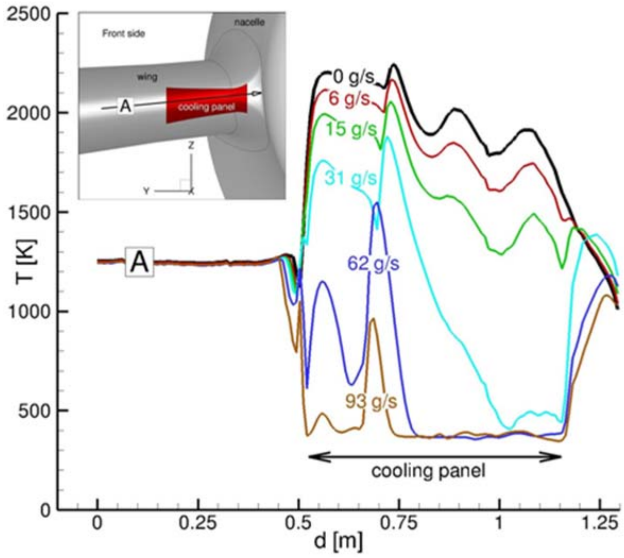

Fig. 16: Influence of the coolant mass flow on the wall temperature, line A

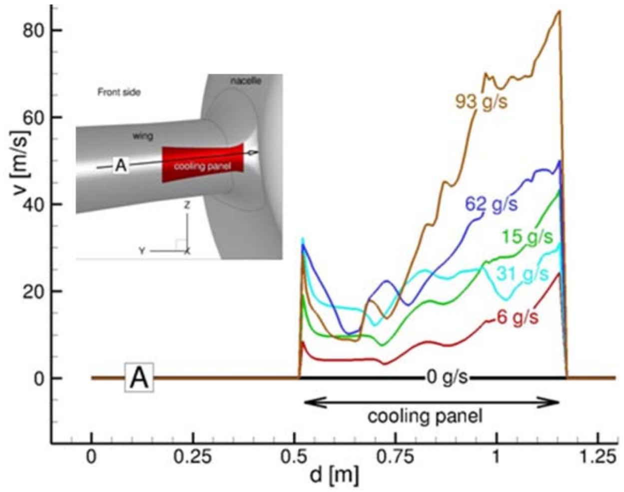

Fig. 17: Influence of the coolant mass flow on the steam velocity, line A

- **Line B (lower side of the wing/cooling panel, x=44.42m)**

The results along line B are shown in **Fig. 18**. This line is in the vicinity of the leading edge.

Therefore, the temperature levels are in the same order of magnitude as for line A (along the leading edge). However, the maximum temperature value without cooling is only T=1950K. For a cooling mass flow rate of m_c=31g/s corresponding to 50g/sm² the maximum temperature can be reduced to T≈1300K. If the mass flow rate is increased to m_c=62g/s (100g/sm²) or more, the temperature along line B is the steam temperature itself (T_c=373K). With respect to the required outflow velocities of the steam the values are from approximately 3m/s to 63m/s (**Fig. 19**).

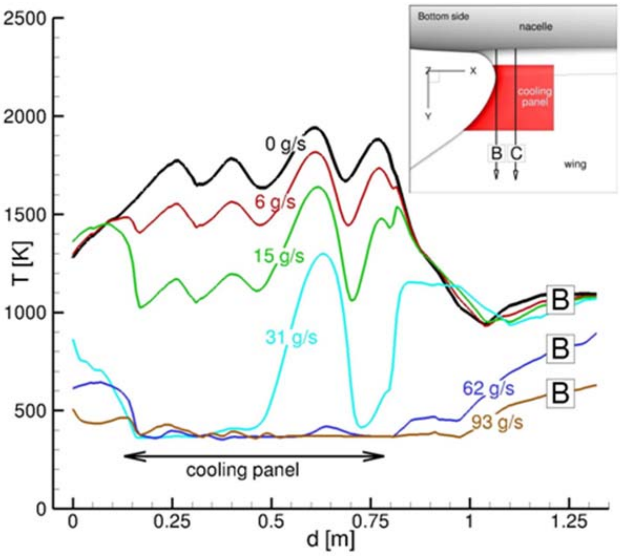

Fig. 18: Influence of the coolant mass flow on the wall temperature, line B

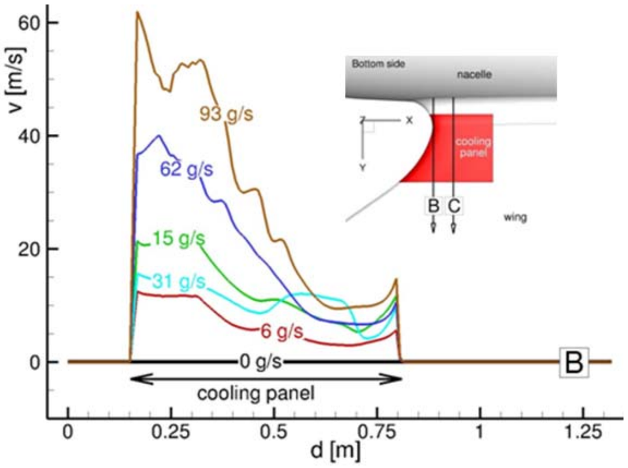

Fig. 19: Influence of the coolant mass flow on the steam velocity, line B

- **Line C (lower side of the wing/cooling panel, x=44.62m)**

The results of the last line C, presented in **Fig. 20**, are extracted in the middle of the lower side of the wing/cooling panel in spanwise direction. Here, the thermal loads are lower than the values of line A and B. The maximum temperature value of line C is T=1850K. If a cooling film is established, only m_c=31g/s (50g/sm²) are required to reduce the temperature of the complete cooling panel to below T=400K. For this mass flow rate the outflow velocities are from approximately 5m/s to 15m/s (**Fig. 21**).

Based on these numerical calculations it can be summarised that the shock induced high thermal loads can be reduced by an active cooling using steam. Depending on the targeted surface temperature and the position on the cooling panel a cooling mass flow rate from 25g/sm² to 150g/sm² is required.

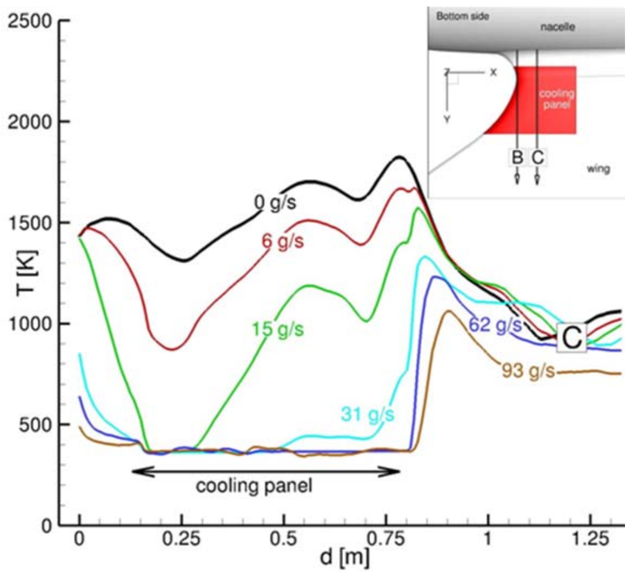

Fig. 20: Influence of the coolant mass flow on the wall temperature, line C

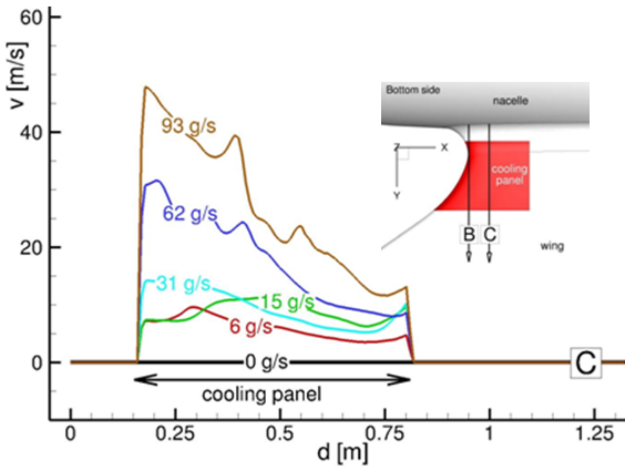

Fig. 21: Influence of the coolant mass flow on the steam velocity, line C

3 Conclusions

The present paper summarizes the numerical analysis of the Skylon spaceplane with the DLR Navier-Stokes code TAU. Target of the simulation is to prove the feasibility of steam cooling for the most critical zone of the vehicle at the junction between nacelle and wing. Here, the interaction of the nacelle shock and the wing shock leads to extensive heatloads which might jeopardize the structure. The simulations for M=24 and α=42.7° at Alt=82.7km clearly point out that the active cooling of the vehicle surface in the region of shock-shock interactions is possible if a effusion cooling with water steam is applied. Even in the stagnation point region the temperature level can be reduced to temperatures below T=1000K. Nevertheless, despite these very promising results until now only the most critical flow condition along the Skylon trajectory is considered and it has to be taken into account that the position of the most critical shock-shock interactions depends on angle of attack and Mach number. Therefore, a proper engineering approach to implement steam cooling in the practical application has to be developed.

References

[1] Varvill, V.; Bond, A.: The SKYLON Spaceplane - Progress to Realisation, *Journal of the British Interplanetary Society (JBIS)*, Volume 61, pp. 412-418, 2008.

[2] Eggers, Th.; Dittrich, R.; Varvill, R.: Numerical Analysis of the SKYLON Spaceplane in Hypersonic Flow, *AIAA Paper 2011-2298*, 2011.

[3] Schwamborn D., Gerhold, Th., Heinrich, R., The DLR TAU-Code: Recent Applications in Research and Industry, *Proceedings of the European Conference on Computational Fluid Dynamics*, ECCOMAS CFD 2006.

Copyright Statement