Transpiration Cooling Using Liquid Water

Arnold van Foreest[1], Ali Gülhan[2], Burkard Esser[2], Martin Sippel[1],
B.A.C. Ambrosius[3], K. Sudmeijer[3]
[1]Space Launcher System Analysis (SART), DLR, 51147 Cologne, Germany
[2]Windtunnel Department, DLR, 51170 Cologne, Germany
[3] Faculty of Aerospace Engineering, TU Delft, 2629 HS Delft, the Netherlands

At the Space Launcher System Analysis (SART) department of DLR-Cologne, a hypersonic spaceplane for passenger transportation is being investigated. A major challenge is the aerodynamic heating of the vehicle. A possible solution for handling the extreme heatloads will be presented. The solution involves an innovative new way of transpiration cooling, using liquid water. The concept has been tested at the arc heated wind tunnel section of DLR-Cologne. The test campaign will be described and the results will be compared with transpiration cooling using a gas as a coolant.

1. Introduction

1.1 The SpaceLiner

At the Space Launcher System Analysis (SART) department of DLR-Cologne, a hypersonic space plane concept for passenger transportation is being investigated. The space plane is able to fly ultra long distances, e.g. from Sydney to Western Europe, in 90 minutes. This vehicle is called the SpaceLiner [1][2][3][7][8][10]. The SpaceLiner uses rocket engines to accelerate.
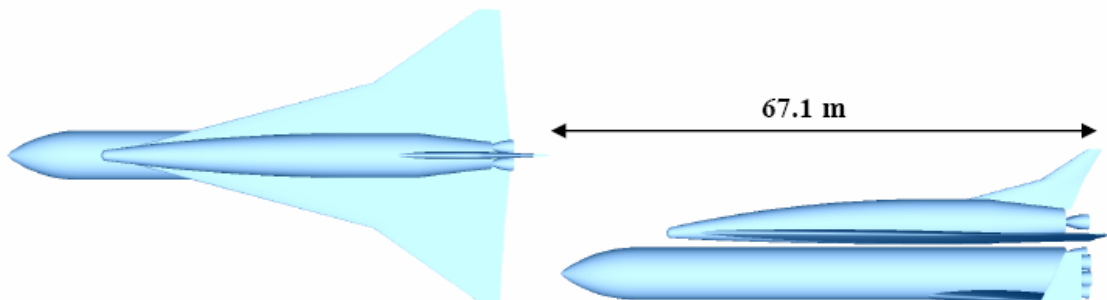

Figure 1. SpaceLiner

After engine burnout the remaining part of the flight is a powerless skip trajectory. Such a skip trajectory introduces extremely high heat loads on the vehicle. To illustrate this, the trajectory flown by the SpaceLiner is compared to the re-entry trajectory of the Space Shuttle in Figure 2. As can be seen the SpaceLiner flies in approximately the same speed regime but at lower altitudes. This means that the heat loads on the SpaceLiner will be more severe as compared to the Shuttle. Analysis has showed that temperatures at leading edge and nose will be approximately 1000K higher [1]. By making the leading edges thicker and choosing a larger nose radius temperatures can be limited. Also, a different (ballistic) trajectory can be flown to reduce temperatures. However, a skip trajectory greatly improves the maximum range of the vehicle. In addition, good aerodynamic performance (which can be obtained by thin wings and small nose radius) is also of major importance for this kind of vehicle. Therefore, it is interesting to look at active cooling options.

One way of active cooling is transpiration cooling using fluids as a coolant. For transpiration cooling a certain coolant mass is required to cool down the vehicle during its flight. Transpiration cooling using gas has already been studied for different concepts [4]. To reduce overall system mass it is important to use a coolant with a high cooling capacity per unit mass. Such a coolant could be liquid water. Together with the wind tunnel department at DLR Cologne, a test campaign in the arc heated wind tunnel L2K has been set up to investigate the feasibility of liquid water as a coolant. In order to verify the advantage of water compared to the gas, additional tests were carried out using nitrogen gas as coolant.
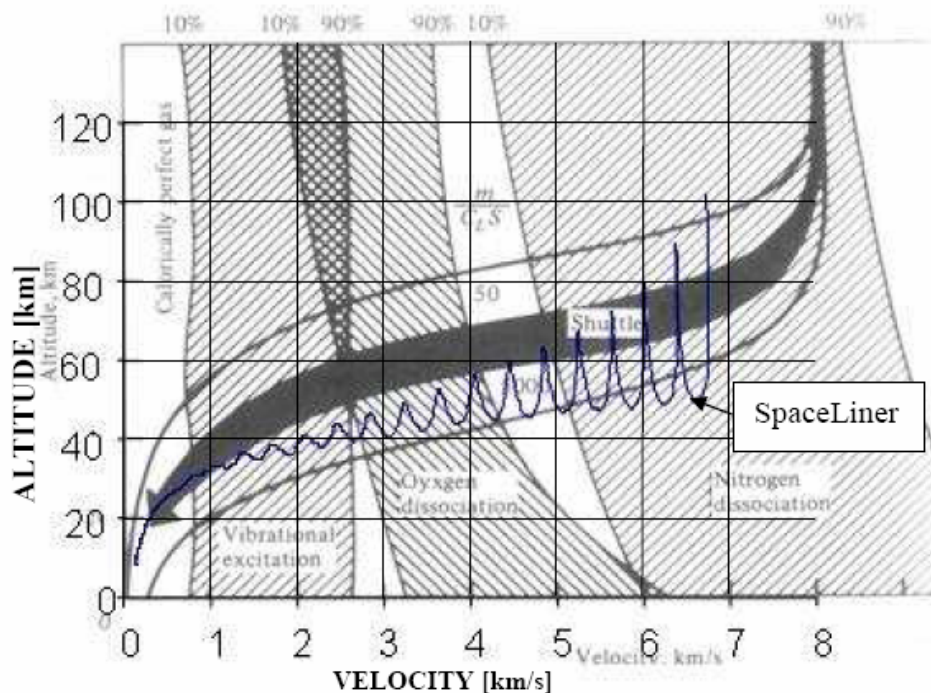
American Institute of Aeronautics and Astronautics Paper AIAA-2007-4034

Figure 2. SpaceLiner Skip Trajectory vs. Shuttle Re-entry [1]

1.2 Transpiration Cooling

By making the heated surface out of a porous material, a cooling fluid can run through this material. The cool fluid absorbs heat by convection and thus cools the material down. Usually, a gas is used as a coolant. However, using a liquid creates the possibility of using the so called 'heat of vaporization'. Because water has the highest heat of vaporization of all liquids, it seems to be the most suitable liquid for this purpose.

Liquids will not become hotter than their boiling temperature. In case of water this boiling temperature is 100°C at 1 bar and increases proportional to the pressure. If water remains in its liquid state during the transportation through the porous material, the convective cooling will be very efficient due to the large temperature difference of liquid water and the uncooled material. When a material with a very high porosity is used, it will be cooled down to approximately the boiling temperature of the water. To prevent water from evaporating within the porous material, new water has to be supplied at a sufficiently high mass flow rate. The higher the heat required for vaporization, the lower the coolant mass flow can be.

The amount of heat which is necessary to evaporate one kg of water depends on the initial temperature of the water, the surrounding pressure and the 'heat of vaporization'. The heat of vaporization is the additional heat needed for the phase change from liquid to gas.

To vaporize an amount of water, it must first be heated up to the boiling temperature. This also requires some energy. This is defined by the specific heat of water, $C_{water} = 4186$ J/kg.K. Assuming the water will be supplied at a temperature of 293K and that the boiling temperature is 373K (at 1 bar), the temperature difference $\Delta T = 80$K. To heat 1 kg of water up to the boiling temperature the energy supplied must be:

$C_{water} * \Delta T = 334.9$kJ/kg. Then, the phase change occurs. This requires an additional 2260 kJ/kg (at 1 bar). As can be seen this 'heat of vaporization' is much more than the energy required to heat up to the boiling temperature. Water has the highest heat of vaporization of all liquids. Therefore it is also the most suitable coolant in this respect.

Using a liquid as a coolant introduces a capillary pressure in the porous material. This pressure will cause water to flow into regions where no water is present. This capillary action will therefore automatically distribute the liquid over the porous material. A simplified model of capillary action in a porous material can be made by assuming a porous material is made up of a bundle of tubes with a certain radius [5]. As soon as a capillary tube has completely filled itself with water, there will be no capillary action anymore. In case of the cooling method using liquid water, this means that when water evaporates at the surface of the material, the liquid water level in

the material will drop. Capillary tubes are not completely filled with water anymore and this then causes capillary action. New water is automatically supplied to the surface at exactly the required mass flow rate.

The evaporation of the water has an additional cooling effect. The vapor enters the boundary layer, creating a protective layer which blocks the incoming heat flux. This effect is called "blocking"[1]. A schematic representation of this cooling principle is given in Figure 3.
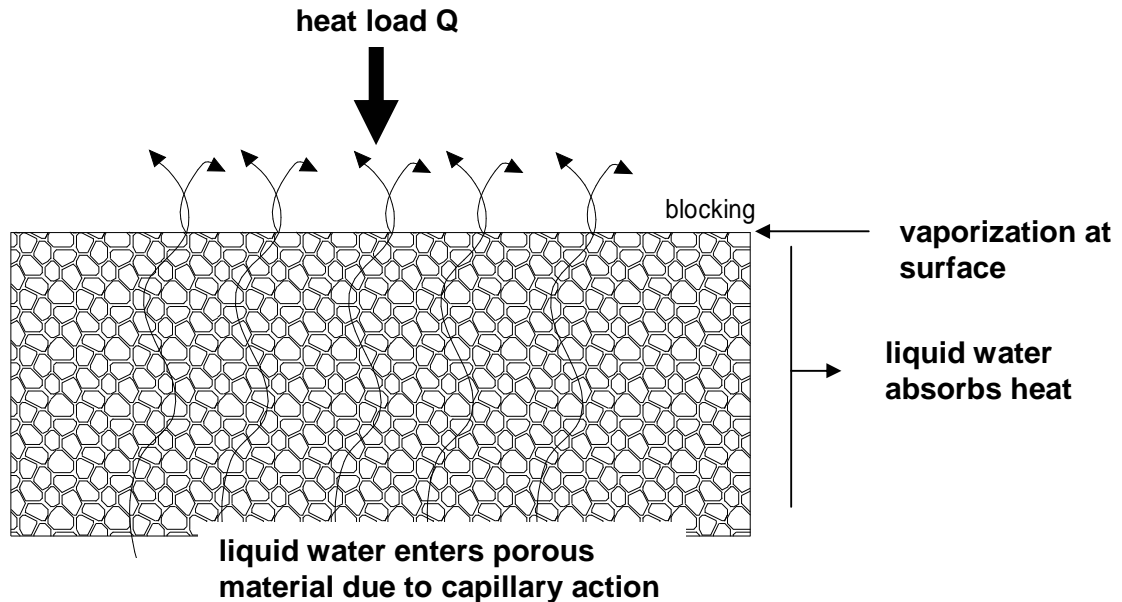
Figure 3. Cooling Principle

2. Wind Tunnel Test Setup

2.1 The L2K and L3K arc heated wind tunnels

The cooling concept described above was tested at the arc heated wind tunnel section of DLR-Cologne. The L2K and L3K arc heated wind tunnels at DLR-Cologne are especially designed for high enthalpy flows. An arc heater is used to give the flow its high enthalpy. The wind tunnels have a long history in qualifying thermal protection systems (TPS). They have been for example used in the Hermes, ASTRA, X-38 and MSTP programs. The gas species can be varied. Thus it is possible not only to simulate earth re-entry, but also for example a mars entry. A schematic view of the wind tunnels is given in Figure 4.

The L3K has a maximal electrical power supply of 6 MW. This allows enthalpies up to 25 MJ/kg at reservoir pressures between 0.15 MPa and 1.8 MPa. Throat diameter can be varied between 200 mm, 300 mm, and 400 mm provide Mach numbers between 5 and 10 at Reynolds numbers up to 10^5/m. Models with a size of 280 mm (W) x 350 mm (L) x 70 mm (H) can be tested in the homogeneous hypersonic flow field of this facility. In the stagnation point configuration, pressures up to 350 hPa can be achieved.

The test facility L2K, with a maximum electrical power of 1.4 MW, can achieve stagnation pressures up to 150 hPa. The different combinations of throat diameters of 14 mm, 20mm, 25 mm and 29 mm with exit diameters of 100 mm, 200 mm and 300 mm provide Mach numbers between 4 and 8 at Reynolds numbers up to 10^4/m. Models with a size of 150 mm (W) x 250 mm (L) x 70 mm (H) can be tested in the homogeneous hypersonic flow field of this facility.

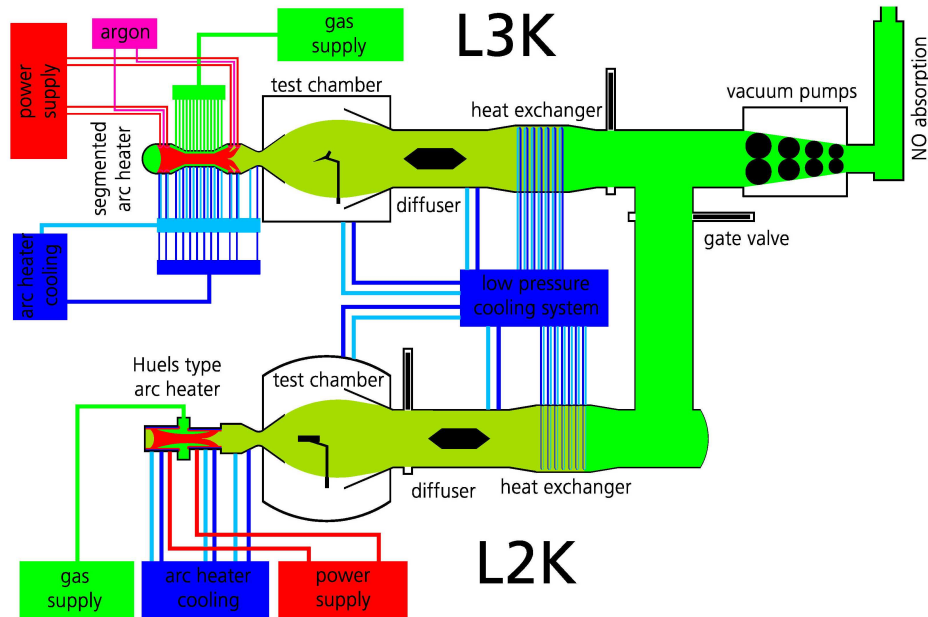The wind tunnel performance parameters are given in Table 1.

American Institute of Aeronautics and Astronautics Paper AIAA-2007-4034

Figure 4. Schematic View of Wind Tunnel Facility

	L2K	L3K
Nozzle exit diameter [mm]	100, 200, 300	100, 200, 300, 400
Mach number [-]	4-8	5-10
Reynolds number [/m]	$<10^4$	$<10^5$
Pitot pressure [hPa]	5-150	15-350
Total enthalpy [MJ/kg]	3-25	6-25
Cold wall heat flux [kW/m^2]	<2000	<4000
Test duration [s]	<7200	<1800

Table 1. Performance of L2K and L3K Wind Tunnels

2.2 Model Construction

To test the lquid water cooling principle, three different nose cone models were made out of a porous material called Procelit 170 (P170). This material consists of 91% Al_2O_3 and 9% SiO_2. This material was chosen because of its high porosity, its ability to withstand temperatures of up to 2000 K and easy manufacturing. The models have a varying nose radius, the smallest radius being 1 cm, the middle radius being 1.75 cm and the largest radius being 2.5 cm. The nose radius was varied to be able to investigate the influence of model geometry on the cooling efficiency. Inside the models, a reservoir has been drilled out. The models were connected to a stagnation probe holder. A copper tube enters the reservoir for water supply, water mass flow could be adjusted using a valve. The models and a cross sectional view are shown in Figure 5.
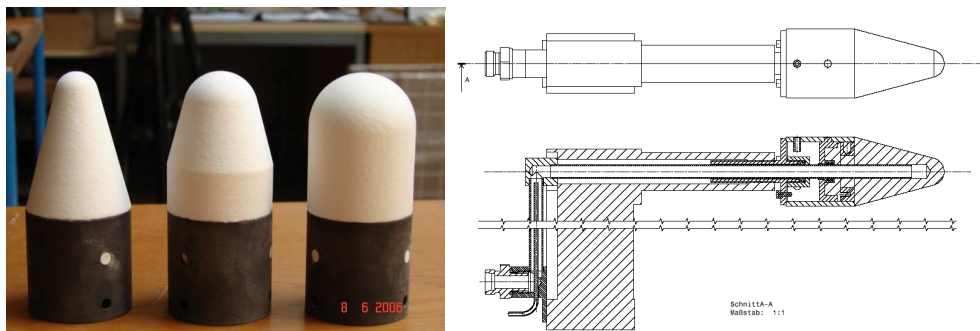

Figure 5. Windtunnel Models [1]

Tests were done using all the models. First, liquid water was used as a coolant. Temperature drops were observed for a certain water mass flow. After these tests had been completed, Nitrogen gas was used as a coolant. The same wind tunnel flow conditions were used in both cases. The surface temperature was measured using an infrared camera. The test procedure was to first insert the models in the flow, without transpiration cooling switched on. Following this procedure, radiation adiabatic temperatures could be measured. Next, cooling was switched on and the temperature drop could be observed.

2.3 Preliminary Testing

For all the tests, the L2K wind tunnel was used. Although its performance is less than the L3K, operation is a lot more straightforward. Since there was no experience with testing this new cooling method, operating simplicity was chosen above performance.

Before any reliable tests could be performed, certain test settings had to be investigated and tried out. Things that had to be tried out were for example the best way of installing measurement equipment, and the water mass flow needed for a certain wind tunnel flow enthalpy. First test were carried out under relatively low wind tunnel enthalpy settings. This was done because the radiation adiabatic temperature of the models needed to be known for analysis of the cooling effectiveness. The maximum allowable temperature for P170 is about 2000K and therefore wind tunnel enthalpy had to be set such that this temperature was not exceeded. The low enthalpy wind tunnel performance is given in Table 2.

First tests were performed with a water mass flow of about 0.5 g/s. These tests showed that the cooling principle worked very well. On infrared images it could be seen how the water was soaked up in the material and was distributed over the whole model. Due to gravity the bottom part of the model was filled with water first. After the bottom part was filled, water started to flow to the upper regions. The last part to be filled with water was the stagnation point. Here, the pressure of the flow on the model is the highest and therefore the water undergoes the highest resistance when flowing into the stagnation point. This causes the relatively long time it takes to wet the stagnation point region. The above described sequence can be seen in Figure 6. Note the temperature scale changes for each picture.

Also visible in this figure is the extreme cooling from temperatures around 2000K to below 488K. Below this temperature, the infrared camera cannot measure the temperature anymore. In these regions, the temperature is probably about equal to the boiling temperature of water at 17 mbar (about 290K).

During these tests, an interesting phenomenon was observed; a huge ice beard was formed on the model. This was not expected to happen, because temperatures in the flow surrounding the model can reach a few thousand degrees. An explanation is found in taking a look at a phase diagram of water, Figure 8. At a pressure of 6 mbar, the so called triple point is reached. Below this pressure, water can only exist in the solid or gaseous phase. The pressure in

Figure 6. Infrared Image Sequence of Water Filling [1]

the reservoir of the model will be equal to the stagnation pressure at the model (17 mbar). Here, the water is in the liquid phase. When the water reaches the surface at points of low pressure (<6 mbar), it undergoes a phase change. Due to the high surrounding temperature, it is expected that vaporization takes place. However, vaporization requires a lot of energy. Thus, when too much water reaches the surface some of the water cannot
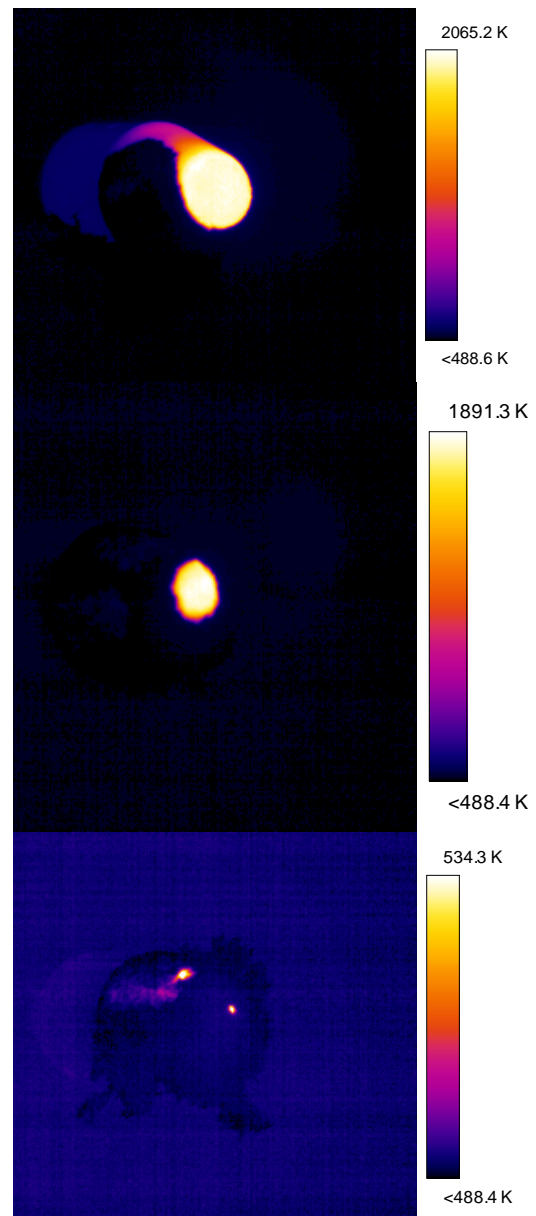
be vaporized. The water which does vaporize extracts energy from its surroundings, causing the temperature of the liquid water to drop to the freezing temperature, and thus causing an ice beard to form (Figure 7).

To prevent such extreme ice building, the water mass flow was reduced to 0.2 g/s. At this flow rate, ice building was still present, but much less extreme. In addition, the building of the ice took a lot more time. A lower mass flow rate than 0.2 g/s was not possible to achieve with the equipment available.

It is noted that the icing only occurs because of the extremely low pressure in the wind tunnel. If the cooling system would be applied to a hypersonic vehicle such as the SpaceLiner, pressure during the flight would be sufficiently high to prevent ice building. Flow separation due to icing is therefore not considered a danger.

p_0 [mbar]	T_0 [K]	h_0 [MJ/kg]	m_{air} [g/s]	p_∞ [mbar]	T_∞ [K]	ρ_∞ [kg/m^3]	V_∞ [m/s]	$pt_{0,2}$ [mbar]	M_∞	Re_∞ (L=1m)	γ_∞
560	3028	4.3	25	0.44	498	2.99E-4	2481	17.3	5.45	2.67E4	1.397

Table 2. Low Enthalpy Wind Tunnel Performance
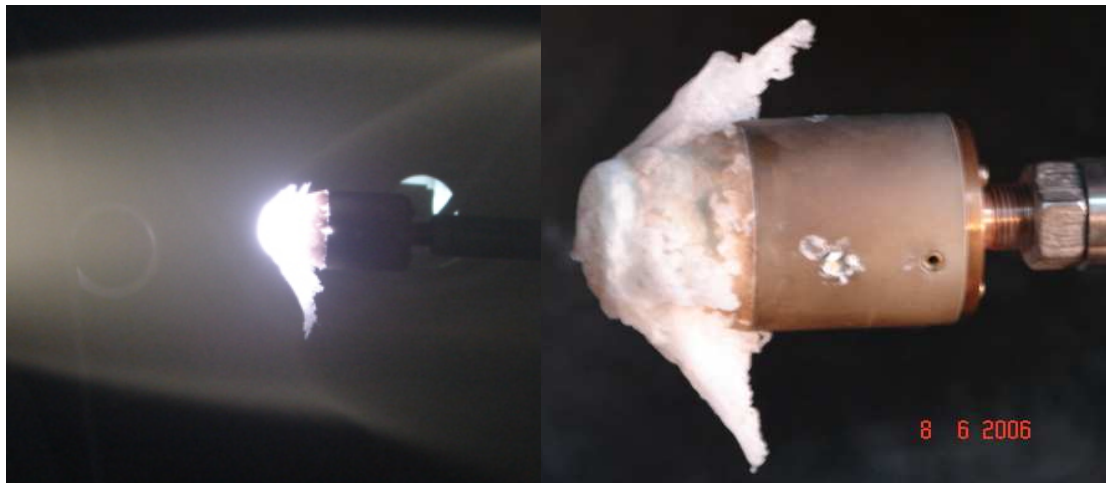
Figure 7. Ice Beard in Hypersonic, High Temperature Flow [1]
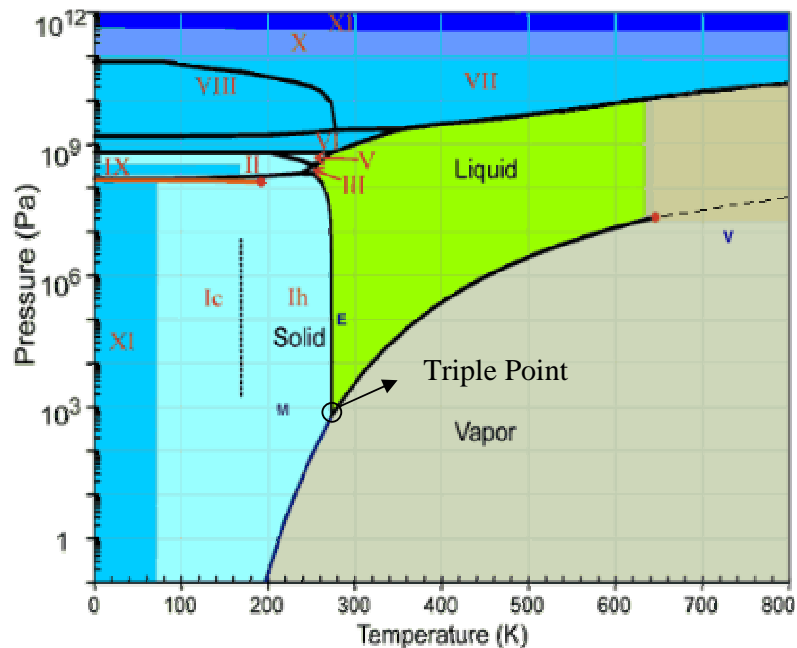
Figure 8. Phase Diagram of Water [1]

American Institute of Aeronautics and Astronautics Paper AIAA-2007-4034

3. Test Campaign

3.1 Introduction

After the preliminary testing to find suitable wind tunnel settings, the test campaign was started. Water mass flow was kept at the value of 0.2 g/s and the wind tunnel performance was kept at the values given in Table 2. All three models were tested but reliable results were only obtained using the models with the largest nose radii (1.75 and 2.5cm). Because of the small radius (1 cm) of the sharpest model, radiation adiabatic temperature became too high and the model showed signs of melting in the stagnation point region. This causes a loss of porosity and therefore water flow is blocked and cooling does not function any longer.

A picture of a test can be seen in Figure 9. Clearly, the shockwave can be seen. Also, the model seems to light up, it is almost as if a light bulb is present in the flow. What happens is that the bright light of the air flow is scattered in all directions. This indicates a high reflectivity of Procelit 170 (and thus low emissivity) in the visible part of the electromagnetic spectrum.

After all these tests had been performed, tests were done using gas as coolant. The gas used was nitrogen (N_2). Different coolant gas mass flows were tried out. These tests were performed to be able to compare the efficiency of using water as a coolant with the 'traditional' transpiration cooling using a gas.
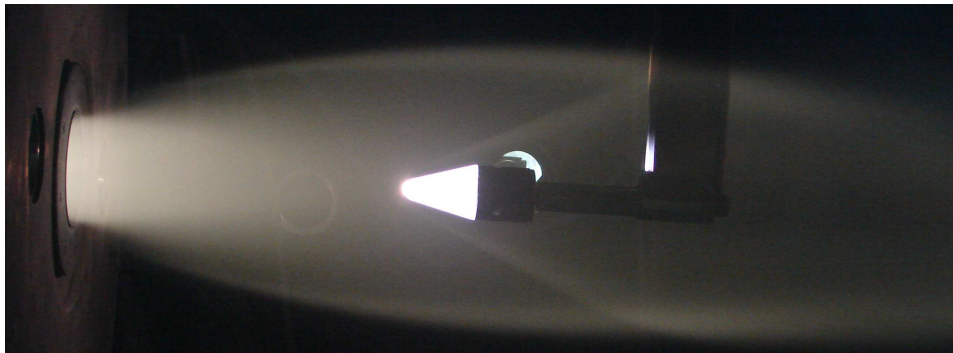

Figure 9. Test in L2K Arc Heated Windtunnel [1]

3.2 Test Results

Test results of cooling using the model with nose radius of 2.5 cm are presented here. Windtunnel conditions for these tests were as specified in Table 2. Figure 10 shows an infrared image of the temperatures in the radiation adiabatic case. As can be seen temperatures in the stagnation point reach over 2040 K. The right part of the image represents the behavior of the temperature on certain spots on the model with water cooling over time. The water mass flow rate was 0.2 g/s. Time is presented in minutes. What can be seen is that the whole model is eventually cooled to temperatures below 500 K. The infrared camera is not able to measure temperatures lower than this value, but as explained before it is expected the temperature will be equal to the boiling temperature of the water (which is about 290 K at wind tunnel conditions).
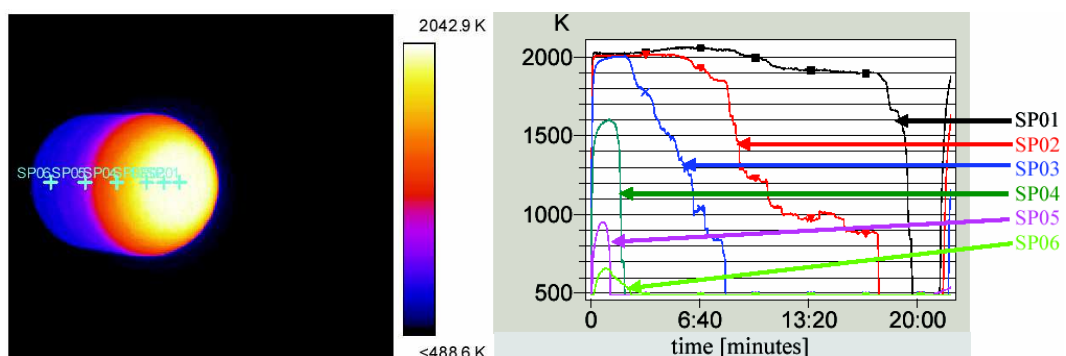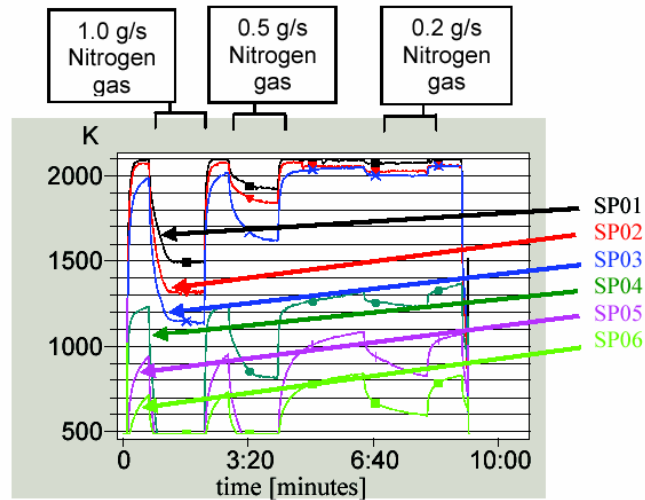

Figure 10. Test Results Using 0.2 g/s Liquid Water [1]

The surface temperature development of the same spots using 1 g/s of Nitrogen can be seen in Figure 11. In this case the stagnation point cooles down to about 1500 K. So even for 5 times higher gas mass flow as water, the temperature drop is still much smaller. In the right part of the figure it can be seen that for the same mass flow

7

rate of the gas as the water (0.2 g/s), temperature drops are extremely small, especially in stagnation point regions. An overview of the test results is presented in Table 3. It can be seen clearly that using liquid water as a coolant is extremely effective as compared to cooling using nitrogen gas. Water cooling can therefore save coolant mass compared to using Nitrogen gas as a coolant.

Figure 11. Test Results Using Nitrogen Gas [1]

	Temperature drop using 0.2 g/s water	Temperature drop using 0.2 g/s nitrogen gas	Temperature drop using 0.5 g/s nitrogen gas	Temperature drop using 1 g/s nitrogen gas
SP01	>1500K	0K	200K	600K
SP02	>1500K	50K	250K	800K
SP03	>1500K	100K	400K	850K
SP04	>1100K	100K	400K	>700K
SP05	>450K	300K	>450K	>400K
SP06	>160K	250K	>200K	>200K

Table 3. Comparison between Gas and Liquid Water as Coolants [1]

Something which was observed during all the water cooling tests is that when the model is completely soaked with water, the stagnation point region is cooled to the lowest temperatures. At other regions, the temperature is higher. The explanation probably is that in the stagnation point, where pressure is the highest, liquid water can exists at the surface of the model. Here, the material cools down to the boiling temperature of the water. At regions of low pressure (<6 mbar) no liquid water can exist. What probably happens is that the water flows from a region with relatively high pressure (the reservoir, 17 mbar) to a region of low pressure (surface pressure, <6 mbar). Somewhere in the material, the surrounding pressure will reach the triple point (Figure 8). Here, ice is formed and some of this ice turns into water vapour. As said, this does not happen at the surface of the material, but somewhere inside the porous material. The water vapour heats up and therefore, the model is not cooled efficiently and the surface temperature will rise. The above described phenomenon is clearly visible in Figure 12.
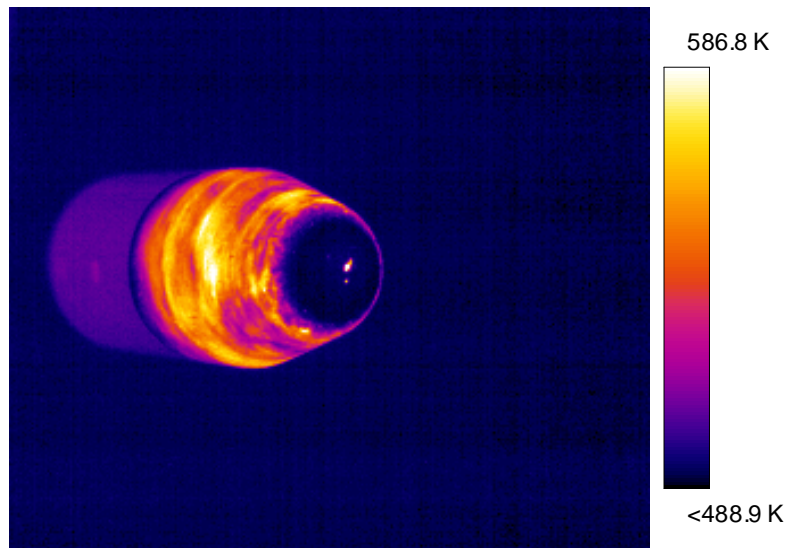
American Institute of Aeronautics and Astronautics Paper AIAA-2007-4034

Figure 12. Cooled Model Temperature Distribution [1]

4. Numerical Analysis

4.1 Determining the Heat Flux

Transpiration cooling using liquid water has been proven to be much more efficient compared to gas cooling. To be able to make predictions of the required water mass flow for cooling, the results have to be quantified. The first step is to determine the heat flux into the model. Assuming all the heat will be absorbed by the water, the heat flux directly determines the evaporation rate of the water and therefore the required water mass flow. Because heat flux was not measured during the tests, it has to be determined numerically. The DLR program HOTSOSE [11] is used for this. HOTSOSE uses the equilibrium gas model to account for real gas effects. In reality, the gas is not in equilibrium. The flow is strongly frozen and Procelit 170 has a strongly catalytic wall. A frozen flow is exactly the opposite of an equilibrium flow. However, a catalytic surface means that the properties of the model material are such that at the surface reaction rate of the air molecules is increased such that at the

surface, equilibrium conditions will be achieved. Figure 13 shows the heat transfer rate $\dfrac{Nu}{\sqrt{\mathrm{Re}}}$ into a wall as a

function of the recombination rate parameter C_1. A large C_1 corresponds to an equilibrium flow and a small C_1 to a frozen flow. As can be seen, for a catalytic wall, the heat transfer is independent on the recombination parameter. For equilibrium flow (the right part of Figure 13), heat will be transferred by conduction into the wall. For both equilibrium flow and frozen flow the heat transfer rate will be the same. The equilibrium gas model therefore seems to be a good approximation for calculating the heat flux into the model wall.

Numerical calculations for heat fluxes at wind tunnel conditions are made. For simplicity these calculations do not include the blocking effect. Results are presented in Figure 14. Here the x axis represents the distance along the centerline of the model and the vertical axis represents the heat flux in W/m^2 at the surface of the model. Note that in case of radiation adiabatic conditions (cooling switched off), heat flux into the model is much smaller than in case of a cooled wall. Cooling decreases the temperature but increases the heat flux into the model. Heat flux into the model depends largely on the difference between the enthalpy of the gas at the boundary layer edge and the enthalpy directly at the wall, $(h_e - h_w)$. In case of a cooled wall the enthalpy directly at the wall will become smaller and heat flux increases.

During the tests the model is cooled down to about 300 K. So this line is representative for the test conditions. By integrating the heat flux over the surface of the model, the total heat flow into the model can be obtained. For the cooled wall, this results in 578 W. At 300K the heat lost due to radiation is minimal. Virtually all this heat will be absorbed by the water.

During testing, the total pressure in the windtunnel is low (17 mbar). At this pressure, water boils at about 17°C, which is only slightly above the initial temperature of the water when it enters the model. In this case energy required to heat the water up to the boiling temperature can be neglected. Only the heat of vaporization is of importance. By assuming all the heat is absorbed by the water, water usage can then be calculated as follows:

American Institute of Aeronautics and Astronautics Paper AIAA-2007-4034

$$\dot{m} = \frac{\dot{Q}_{in}}{H_{vap}}$$

(1)

where $\dot{Q}_{in}$ is the heat flow [W] into the material , $\dot{m}$ the water mass flow in kg/s and H_{vap} the heat of vaporization of water (2460 kJ/kg at wind tunnel conditions).

A required water mass flow of 0.235 g/s is calculated. This is close to the 0.2 g/s of water flow rate, which was measured during the test (a difference of 14.9%). It has already been mentioned in section 2.3 that the 0.2 g/s mass flow was already to much, but that it was not possible to reduce this mass flow further with the exisiting experimental hardware. The difference could therefore also be more than 14.9%, however how much more is unclear. This difference of 14.9% or more,could well be caused by not considering the blocking effect in the calculations. To verify this hypothesis, further tests were executed to investigated the blocking effect in more detail.
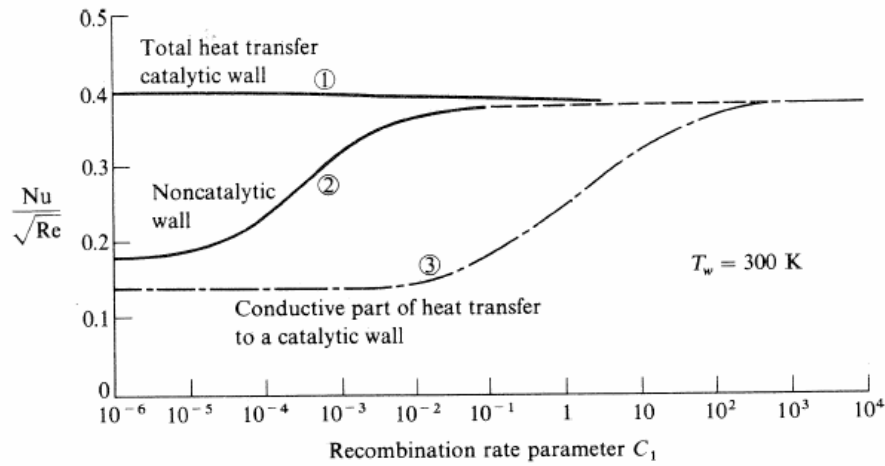
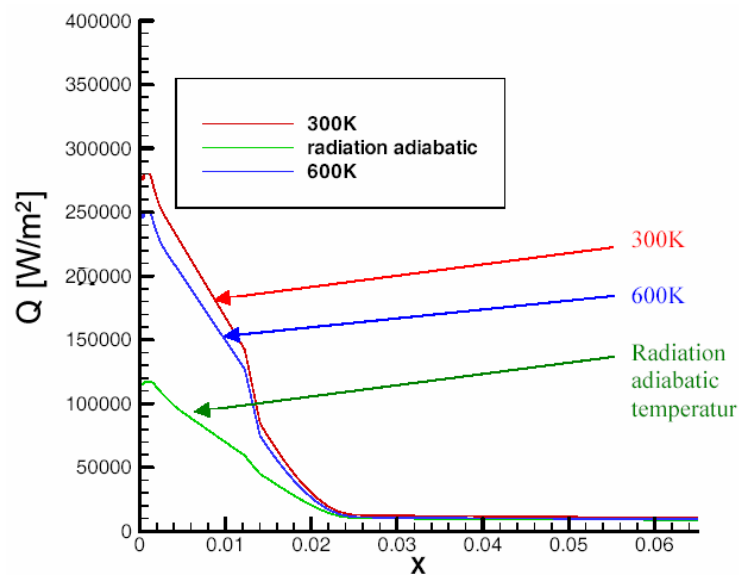Figure 13. Heat Transfer into Wall for different Wall Catalysis [9]

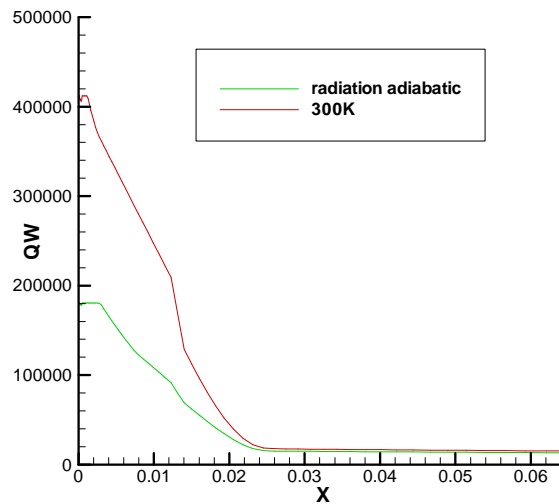Figure 14. Heat Flux Along the Surface of the Model [1]

4.2 Investigation of Blocking Effect

Assuming that the numerical calculations have a sufficiently small error (not investigated), an analysis can be made for the blocking effect. The previous results can not be used to conclude anything about the effectiveness of the blocking, because the water mass flow was too high. Additional experiments had to be carried out.

For the additional experiments, water mass flow was set to 0.24 g/s. The wind tunnel flow enthalpy was increased stepwise. This way, the maximum flow enthalpy at which the 0.24 g/s could cool the complete model down could be determined. The result for the 0.025 m radius model is presented in Table 4. The heat flux along the model is calculated numerically and is shown in Figure 15.

Test	p0 [mbar]	T0 [K]	H0 [MJ/kg]	m_{air} [g/s]	p_∞ [mbar]	T_∞ [K]	ρ_∞ [kg/m³]	V_∞ [m/s]	M_∞	Re_∞
8	800	3906	7.54	30	0.5561	554	$3.038*10^{-4}$	2925	5.719	$3.153*10^4$

Table 4. Maximum Wind Tunnel Performance for 0.24 g/s of Water [1]

Figure 15. Heat Flux Along the Surface of the Model at Maximum Windtunnel Performance for 0.24 g/s Water Mass Flow [1]

Integrating the heat flux over the complete model surface results in 850 W. With equation (1) the resulting water usage is 0.346 g/s. The blocking effect thus saves 0.106 g/s, or 31%. Similar tests with the models with radius 0.0175 m and 0.01 m showed that water mass flow was reduced by 30% and 22%, respectively [1].

Returning to the experiment discussed in the previous section, the difference between the calculated mass flow of 0.235 g/s and the applied mass flow of 0.2 g/s could well be caused by the blocking effect. Assuming 30% blocking, the applied water mass flow only needs to be 0.165 g/s. This is less than the 0.2 g/s and therefore is consistent with the observation that 0.2 g/s is more than required.

Note that the values calculated for the effectiveness of the blocking are only valid under the wind tunnel conditions present during the tests. For other flow conditions, the effectiveness may vary.

5. Future Improvements

5.1 Reducing the Water Usage

A number of potential methods exist to reduce the water usage for cooling. The first method is to make the nose and leading edge radii smaller. This reduces the surface area that needs to be cooled. However, a smaller nose radius also means an increase in heat flux. It can be shown mathematically that the smaller area outweighs the increased heating in terms of water usage.

Looking at the Fay Riddell equation for stagnation point heating [9] it can be seen that:

American Institute of Aeronautics and Astronautics Paper AIAA-2007-4034

$$\dot{q}_{sp} \triangleq \frac{1}{\sqrt{R}} \qquad (2)$$

where $\dot{q}_{sp}$ is the heat flux into the stagnation point in W/m^2 and R is the nose radius in m.

According to [12] the total heat flow into an (almost) half sphere is given by:

$$\dot{Q}_{total} = -\frac{4}{5}\pi R^2 \dot{q}_{sp} \cos^{\frac{5}{2}}\theta \Big|_0^\theta \ , \ 0 \le \theta \le 70° \qquad (3)$$

where $\dot{Q}_{total}$ is the total heat flow into the half sphere in W and θ is defined in Figure 16.
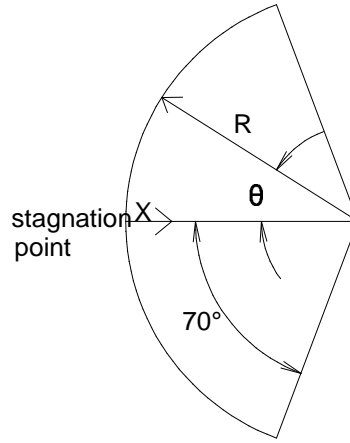
Figure 16. Definition of θ

Inserting (2) in (3) yields:

$$\dot{Q}_{total} \triangleq -\frac{4}{5}\pi R^{1.5} \cos^{\frac{5}{2}}\theta \Big|_0^\theta \ , \ 0 \le \theta \le 70° \qquad (4)$$

Now one can clearly see that the smaller the radius, the smaller the total heat flow into the nose and the smaller the water usage. In case of a nose the total heat flow and thus water usage is shown to be proportional too $R^{1.5}$. For a leading edge the result is [1]:

$$\dot{Q}_{total} \triangleq 2L\sqrt{R} \sin\theta \Big|_0^{70} \qquad (5)$$

In case of a leading edge the total heat flow and thus water usage is shown to be proportional to $\sqrt{R}$.

A second method for water reduction is to reduce the cold wall heat flux. This can be done by covering the porous material (such as Procelit 170) with a perforated skin. This skin would then heat up, and heat is transferred into the porous material via conduction in this skin. By choosing a material with a suitable conductivity, the skin can be allowed to heat up to a certain temperature, thus decreasing the cold wall heat flux. Perforating the skin allows water evaporated in the porous material to escape through the holes in the skin into the boundary layer, thus still being able to use the blocking effect. Figure 17 shows a schematic drawing of this concept. Alternatively, a material with less porosity could be used. The material would thus heat up to higher temperatures, reducing the cold wall heat flux.

American Institute of Aeronautics and Astronautics Paper AIAA-2007-4034
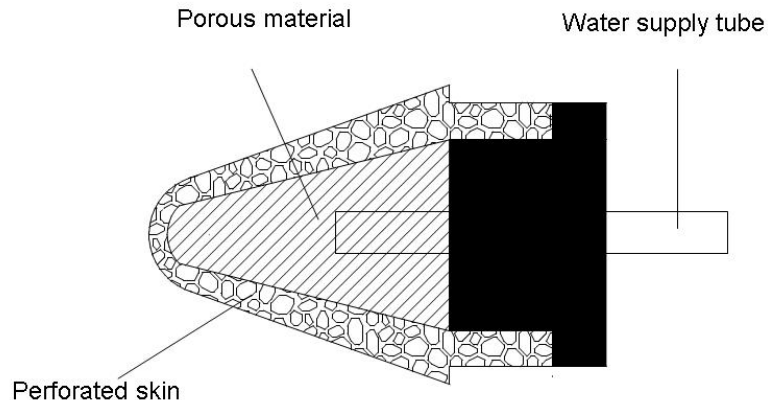
Figure 17. Cooled Nose with Perforated Skin

5.2 Potential Materials to be Used

The Procelit 170 material used during the tests can not be used to make a nose or a leading edge for the SpaceLiner. The reason is that it is far too brittle. Under high stress or in case that something would hit the material (such as a bird) the material would suffer serious damage. A protective layer would have to be placed over the Procelit safeguarding it from damage. Such a layer can have the additional advantage of reducing the cold wall heat flux, as mentioned in the previous section.

Another option is to use a porous material which is stronger and less brittle. CMC (Ceramic Matrix Composites), such as C/C and C/C-SiC are interesting. These materials are very strong. They also have the property that during manufacturing the porosity can be varied, making it possible to control the amount of temperature decrease during cooling. Less temperature decrease means less cold wall heat flux and a reduced water mass flow.

Temperature resistance of C/C is not very high in oxidizing atmospheres (450˚C). C/C-SiC can withstand temperatures of up to 1750˚C [3].

6. Conclusions

The water cooling is proven to be extremely effective. The models are cooled down from temperatures over 2000K in the stagnation point to temperatures lower than 300K using only little water. Compared to transpiration cooling using a gas (in this case nitrogen), a water mass flow of only 0.2 g/s cools the models down to much lower temperatures than is achieved for gas cooling using a coolant mass flow five times as high.

By comparing numerical predictions of water usage to actually needed water usage, an analyses of the blocking effect is made. These suggest that under wind tunnel flow conditions, blocking can save 30% of the water mass flow.

Numerical analyses of the tests show that cold wall heat fluxes are much higher than radiation adiabatic heat fluxes. Water mass flow could therefore be reduced if it was possible to let the surface of the material heat up. This could be done by covering the porous material by a perforated skin. The skin can be made of a material with certain conduction, such that this skin heats up. This would then decrease the heat flux into the porous material. Another advantage is that additional strength is provided to the model. The Procelit 170 material used during the tests is very brittle and can not be applied to 'real' flight vehicles without a protective layer.

Alternatively, water mass flow could be reduced by using a material with less porosity. In this case, the material would be cooled down less and therefore cold wall heat flux would be reduced. Potential materials are C/C or C/C-SiC which can withstand high temperatures and are very strong. Porosity of these materials can be varied during manufacturing.

Research into the cooling method will continue at DLR. New tests will include models made of C/C-SiC, leading edge models and nose cones. Test will also be executed in the L3K arc heated wind tunnel, allowing higher flow enthalpies.

References

[1] Van Foreest, A.: Investigation on Transpiration Cooling Methods for the SpaceLiner, DLR-IB 647-2006/05, SART TN-004/2006, 2006

[2] Van Foreest, A.: Trajectory Analysis and Preliminary Design of a Future Spacecraft for Intercontinental Rocket Powered Passenger Travel, DLR, SART TN-008/2005, 2005.

[3] Sippel M., Klevanski J., van Foreest A., Guelhan A., Esser B., Kuhn M.: The SpaceLiner Concept and its Aerothermodynamic Challenges, Arcachon Conference, 2006

[4] Kuhn, M.; Hald, H.; Gülhan, A.; Esser, B.: Experimental Investigations of Transpiration Cooled CMC's in Supersonic Plasma Flows, 5th European Workshop on Thermal Protection Systems and Hot Structure, 17. – 19. May 2006, ESA/ESTEC, Noordwijk

[5] Richardson M., Theory and Practice in Capillary Force Vaporizer Devices, Vapor Inc. 510-235- 4911, May 20 2004

[6] Gülhan, A.; Esser, B.; Koch, U.: Experimental Investigation on Local Aerothermodynamic Problems of Re -entry Vehicles in the Arc Heated Facilities LBK. AIAA Journal of Spacecraft & Rockets, Volume 38, Number 2, Pages, 199-206, March-April 2001

[7] Sippel, M., Klevanski, J., Steelant, J.: Comparative Study on Options for High-Speed Intercontinental Passenger Transports: Air-Breathing-vs. Rocket Propelled, IAC-05-D2.4.09, October 2005

[8] Sippel, M.: Introducing the SpaceLiner Vision, 7th International Symposium on Launcher Technologies, April 2007

[9] Anderson, J.D., Jr.: Hypersonic and High Temperature Gas Dynamics, McGraw-Hill Book Company, 1989

[10] Van Foreest, A., Sippel, M., Klevanski, J., Gülhan, A., Esser, B.: Technical Background and Challenges of the SpaceLiner Concept, 7th International Symposium on Launcher Technologies, April 2007, Barcelona, Spain

[11] Reisch, U.,Anseaume, Y.: Validation of the Approximate Calculation Procedure HOTSOSE for Aerodynamic and Thermal Loads in Hypersonic Flow with Existing Experimental and Numerical Results, DLR, Institut für Entwurfsaerodynamic Braunschweig, Forschungsbericht 98-23,1998

[12] Buursink, J: On the Development of a Water-Cooled Metallic Thermal Protection System, TU Delft, The Netherlands, 2005