EVALUATING WATER FILM AND RADIATION MODELING TECHNOLOGIES IN CFD FOR AUTOMOTIVE LIGHTING

G.Dumnov, A.Ivanov, A.Muslaev, M.Popov
(Mentor Graphics Corp., Russia)

J.C. Watson
(Mentor Graphics Corp., United States)

Abstract

There is a growing need in certain industrial applications to simulate the effects condensation can have on a surface. One example of this area is automotive lighting, where understanding the complex physical processes occurring on the surface of a component including phenomena such as surface condensation, liquid film motion, film freezing, or evaporation has become essential.

This paper describes the technique used to perform transient numerical simulation of the liquid film formation process on the surface of a solid part, where the film's influence on the part's thermal loads is accounted for, which has been implemented in Mentor Graphics' computational fluid dynamics software FloEFD. This will include the capability to account for the change of the film's thermal state and thickness due to heat exchange with the external fluid flow and solid body, surface evaporation and condensation, melting and crystallization within the film volume, and motion caused by gravitational and frictional forces from the outer flow. Two examples are presented to validate the numerical models for water film condensation and evaporation processes.

In the first example, the film condensation of water vapor on a cooled surface from humid air represented as a horizontal flat plate with controlled air flow conditions (flow regime, hygrometry, temperature) is investigated. The second example deals with the calculation of the convective mass transfer coefficients for water film evaporation in a hydro-dynamically developed airflow in a short rectangular duct for Re number between 570 and 8100. For both examples calculation results are compared with experimental data with good agreement found for calculated condensation rate and weight of condensate as well as for convective mass transfer coefficients.

When designing automotive lighting devices vapor condensation on the surface of headlights' optical system is a major challenge. However to

perform adequate simulation of thermal loads on a particular headlight component it becomes essential to consider the whole optical system in the calculation, including the components where film is not present. A certain degree of precision is also required to calculate convective and radiative fluxes.

Among the design features of headlights is the use of semitransparent materials (glass or plastic), each having a distinct spectral dependence of the absorption coefficient and the presence of a large number of reflective and refractive surfaces that focus light in local regions.

To perform adequate simulations of radiative heat transfer in such complex systems, a "band" Monte-Carlo radiation model is usually implemented. This model assumes that the whole spectral range is split into several spectral bands with material properties and boundary conditions averaged within each band. Calculations are performed independently for each band. Certain complexity arises at the stage of specification of the band ranges, when one must ensure that these are specified correctly. Also since the material properties are averaged within each band, subtle spectral effects may not be captured during the simulation. This paper also describes an enhanced Monte-Carlo radiation model which does not require splitting into bands. As a result, spectral material properties, radiation sources, and boundary conditions are accounted for precisely without averaging over the spectral bands. The effectiveness of the proposed technique to simulate radiative heat exchange is validated for the fogging evolution processes inside a headlight, with the example of the Mini Farol Auxiliar Retangular ForteLuz (PAR) floodlight simulation.

1. Introduction

Phenomena such as evaporation or vapor condensation on the surface of solid bodies are typical examples of complex physical processes that can be observed both in industry and everyday life. In general, these processes imply the formation and subsequent removal of liquid film. Additionally, since a phase change takes place either heat release or absorption is observed. Correspondingly, the heat exchange between the solid body and the external gas flow is affected with the liquid film acting like a thermal resistance.

To perform the numerical simulation of the above mentioned phenomena two options can be considered. The first option suggests the use of a dense computational mesh, where the liquid film along its thickness is sufficiently resolved by the cells. Obviously this would require large computational resources because of the relatively small scales of the film compared to the solid object considered.

The other option deals with the, so-called, engineering approach described in detail in the present paper. Although several assumptions were taken to develop this approach, the resulting mathematical model adequately captures all the essential physical processes and requires relatively low computational resources. The model is implemented in FloEFD (Mentor Graphics) and its features include the capability to account for the change of a film's thermal state and thickness due to the heat exchange with the external fluid flow and the solid body, surface evaporation/condensation, melting/crystallization within the film volume, motion caused by gravitation and friction forces from outer flow. Special attention was paid to the problems related to automotive lighting, where vapor condensation and evaporation on the headlight's surface is crucial.

To validate the code's capability to simulate the liquid film condensation and evaporation processes several problems were considered, where good agreement with experimental data was obtained. Also this paper reports on the results obtained from the numerical study of a headlight having complex geometry. During the study of this device, such physical processes as heat transfer by convection, conduction and radiation as well as phase change were accounted for.

2. Mathematical model of surface condensation and film formation

This section describes the technique used for deriving equations to simulate the film formation on the surface of a solid body. It is known that the following factors affect thermal state of film and its thickness: heat exchange with the outer fluid flow and the solid body, condensation/evaporation on the film surface, melting/crystallization within the film volume as well as film motion caused by gravitation and friction forces from the outer flow.

To develop the corresponding mathematical model the following assumptions were made:

1. The fluid in the outer flow is a gas mixture with non-zero humidity;

2. Mass transport between film and vapor in the outer flow is diffusive;

3. The film thickness, although finite, is neglected when solving the equations that describe the outer fluid flow;

4. The derivatives of all primary flow parameters along the normal to the body surface are infinitely large comparing to the ones along the surface;

5. The forces due to inertia within the film are negligibly small compared to viscous and mass forces;
6. Surface tension force is neglected;
7. Film motion, if present, is laminar;
8. The film along its thickness is uniform and is either in a liquid or solid state, no sub-layers are considered;
9. The temperature profile along the film thickness is linear.

Once all these assumptions are accounted for the following equations describing the transport of mass per unit surface and enthalpy can be derived:

$$\frac{\partial m}{\partial t} + \nabla_S m\vec{U} + q_Y = 0 \quad (1)$$

$$\frac{\partial m\bar{h}}{\partial t} + \nabla_S m\vec{U}\bar{h} + q_Y h_v + q_T + q_s = 0 \quad (2)$$

where ∇_S - surface divergence operator, m - film mass per unit surface, $\bar{h}$ - specific enthalpy of film averaged over its thickness, $\vec{U}$ - film velocity vector, q_Y - mass flux to the gaseous phase (due evaporation or condensation), h_v - specific enthalpy of vapor in the outer flow calculated at the temperature of film's external surface; q_T - heat flux to the gaseous phase, q_s - solid surface heat flux;

Under the assumptions 4…6, a vector equation for film velocity averaged over its thickness is derived in the following form:

$$\vec{U} = \frac{\rho_l \delta^2}{3\mu_l} \cdot \vec{g} + \frac{\delta}{2\mu_l} \cdot \vec{\tau}_f \quad (3)$$

here ρ_l, μ_l - density and dynamic viscosity of liquid film material, δ - film thickness, $\vec{g}$ - gravity force vector projected on the surface, $\vec{\tau}_f$ - stress vector on the film's surface.

The equations (1)…(3) coupled with the equations describing the transport of mass, momentum and energy in the outer flow as well as the equation describing heat conduction in the solid body form a closed system of equations that is fully implemented in FloEFD.

3. The modified approach to account for the spectral properties in the Monte-Carlo radiation model

One of the most complex problems that require radiation heat transfer to be accounted for is the simulation of lighting devices such as headlights. The typical source of radiation in the headlight is a halogen bulb with its tungsten spiral having a temperature of about 3000 K.

The spectral radiation produced by the spiral interacts with other semi-transparent and opaque headlight elements. Also, both the bulb's envelope and the gas within it are heated. Once heated the bulb itself begins to radiate in the IR spectrum. Only a certain fraction of radiative energy produced by the spiral is absorbed by the envelope material, thus the resulting radiation spectrum and polar pattern of the halogen lamp become very complex.

It is crucial to account for the absorbed and radiated heat in the IR spectrum accurately enough in order to perform an adequate simulation of the radiative heat transfer processes. A lot of effort is put into engineering new solid materials, which have a complex dependency of the absorption coefficient in the IR region.

The classic approach to simulating spectral effects deals with the discretization of the entire spectral range into several bands with material properties (absorption coefficient, emissivity) and boundary conditions averaged within each band. Certain complexities arise at the stage of specifying the band ranges, when one must ensure that the properties of all the materials considered within a certain band vary moderately with the change of wavelength. For an industrial problem, where many materials have to be specified, such a task becomes very complicated as the number of spectral bands becomes large.

To eliminate this disadvantage, the enhanced Monte-Carlo radiation model was proposed and implemented in FloEFD, which does not require splitting into bands. While the "classic" Monte-Carlo model assumes the direct simulation of a "photon" having a certain energy assigned at the time of its inception, this enhanced model deals with

“photons” having both the energy and the wavelength assigned at the time of the inception. The interaction of a “photon” with the surrounding medium is simulated in such a way that the medium properties and the spectral boundary conditions at certain wavelengths affect only the “photons” having the same wavelength. Thus, no averaging of medium properties within a spectral band is performed; instead the “real” properties of the radiating objects are used.

A certain complexity of this method arises when it comes to obtaining accurate spectral distribution of “photons”. This distribution depends on the temperature of the radiating objects as well as their emissivity. The latter depends both on temperature and wavelength. As a result, complex spectral functions have to be integrated.

4. Liquid films – validation examples

This section describes the problems used in the validation of the proposed approach to simulate surface condensation and liquid film formation. The numerical simulation was performed in FloEFD. The results were compared with the experimental data.

1. Surface evaporation in the rectangular duct

Film condensation or evaporation of water in a horizontal rectangular duct is considered. In this case the convective mass transfer coefficient between air flowing in the duct and a pan of water is estimated. A pan of water forms the bottom surface of the duct. The results of the experimental study are given in Iskra and Simpson (2007) [1].

The considered duct is part of a transient moisture transfer (TFT) facility, shown in Figure 1: .Figure 1:

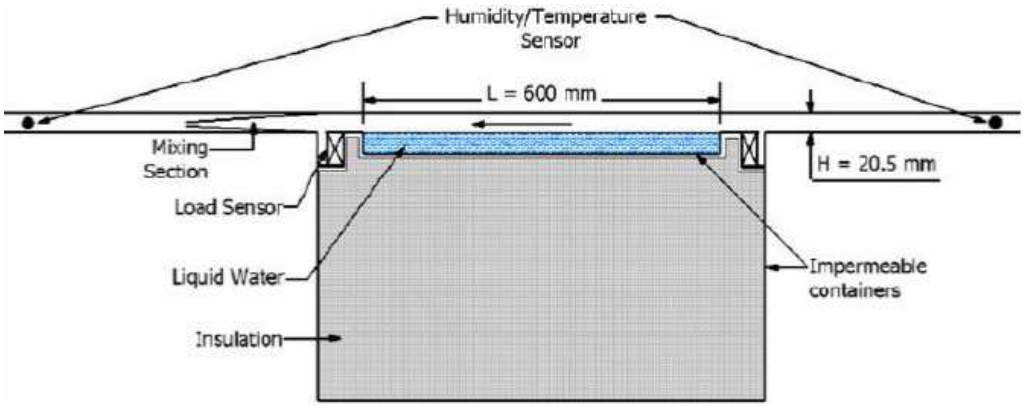

Figure 1: Schematics of the test facility showing a side view of test section (according to Iskra and Simpson (2007).

The temperature of the water is 12°C. The temperature of the air upstream of the test section is 23°C. The laminar (Re_D=1500) and turbulent (Re_D=6000) flow regimes are considered. The air relative humidity is 16.2 % for the laminar flow case and 22.9 % for the turbulent flow case.

For the calculation, 2D and 3D computational domain was considered. To simulate a water surface, the condensed water film with the Initial Film Thickness of 50 μm was specified as the initial condition at the bottom of the pan.

The evolution of the evaporation rate, which was predicted by FloEFD and compared against the experimental data (Iskra and Simpson, 2007), is shown in Figure 2: .

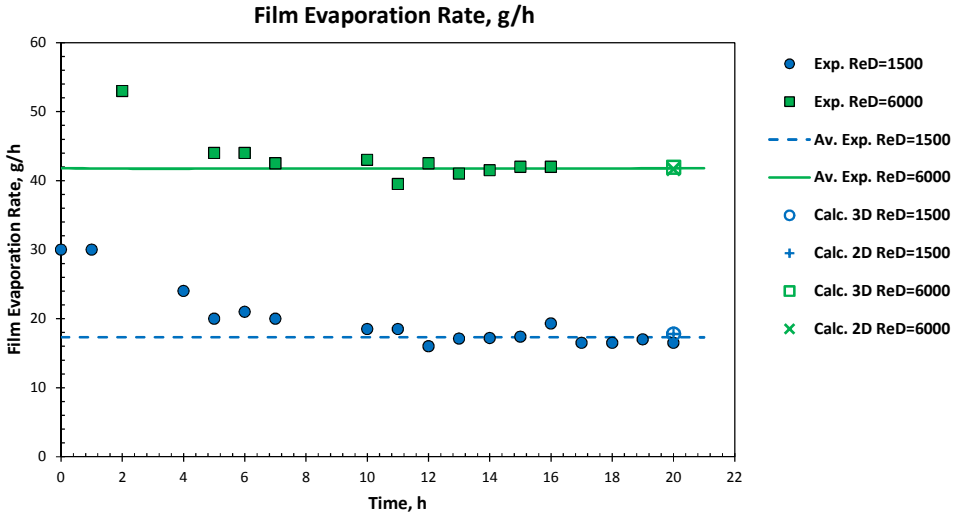

Figure 2: Evaporation rate of water predicted by FloEFD and measured experimentally.

A comparison of the evaporation rates shows that the described method predicts the evaporation effects with good accuracy. The relative calculation errors for 2D and 3D calculations do not exceed 1.0% for Re_D=6000 and 3.5% for Re_D=1500, respectively.

2. Filmwise condensation simulation

The film condensation process on a cooled surface resulting from the condensation of water vapor from humid air on a horizontal flat plate is considered. The plate is of small size (area 25 cm^2). The process progresses under controlled air flow conditions (flow regime,

hygrometry, temperature). The results of the experimental study are given in Tiwari (2011) [2].

The temperature of the plate is kept constant. The resulting condensate is regularly monitored by weighing the whole system. The weight of condensate is recorded every 30 minutes. The system consists of a square shaped Peltier module sandwiched between a square shaped aluminum flat plate of dimension 5x5 cm^2 with thickness of 3 mm, which is used as an active surface for the condensation, and a heat exchanger device and a temperature regulator that controls the power supply of the Peltier module. The overall arrangement was placed in a test section of a wind tunnel in which the hydrodynamics, temperature and hygrometry were regulated, shown in Figure 3: .

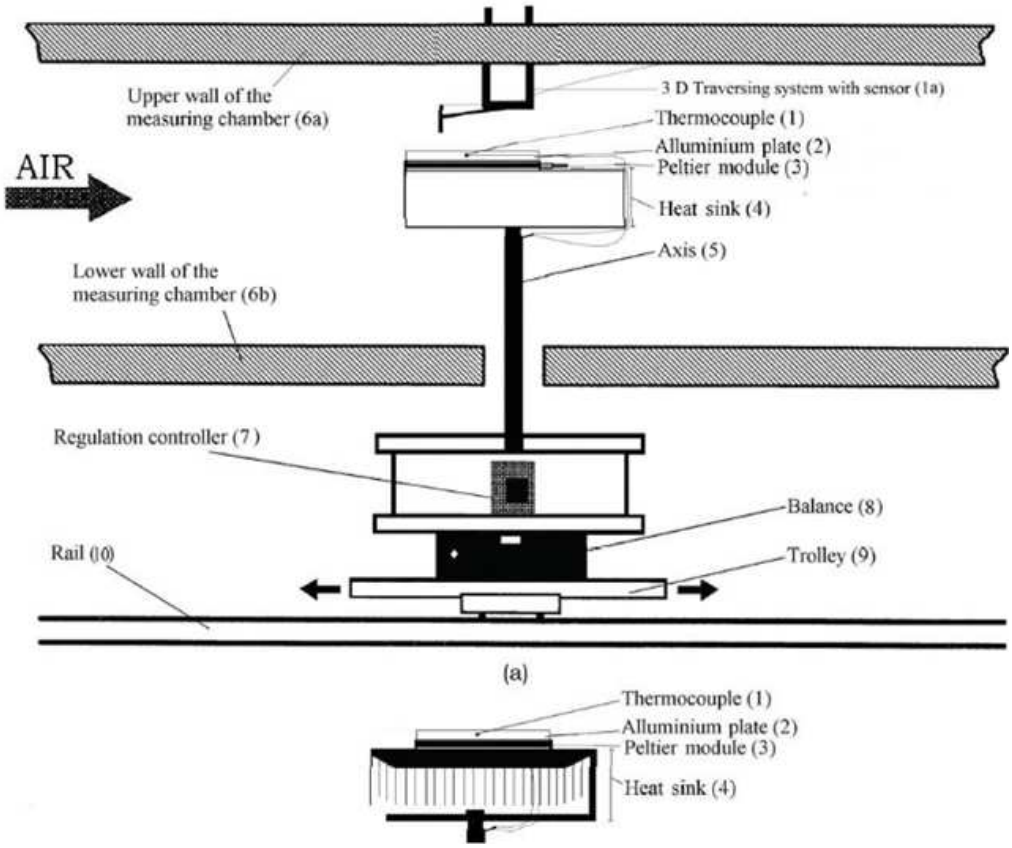

Figure 3: Schematics of the condensation unit (according to Tiwari (2011): (a) front view, (b) side view of the upper part, which faces the airflow.

For the comparison with experimental and theoretical results the CEI-1 experiment is considered in this study (Tiwari, 2011). This experiment was performed for the mean entrance velocity of 1.0 m/s and the mean

relative humidity of 57.2 %, the ambient temperature and pressure far from the condensing plate were 23.2°C and 926 mbar respectively, and the mean surface temperature of the plate was kept at 11.4°C.

The details of the experimental average values according to Tiwari (2011) are given in Table 1: .

Table 1: Input data of the experiment.

Name of exp.	Atmospheric pressure, millibar	Amount of condensate collected, g	Data acquisition time, h:min	Ambient temperature range (mean °C)	Surface temperature (mean °C)	Relative humidity (mean %)
CEI-1	926	2.3	7:50	23.2	11.4	57.2
Name of exp.	(Total w_t/ /Total time), g/h	Mass flux, kg/m^2/s	(m_2-m_1)/ /(t_2-t_1)	Mass flux, kg/m^2/s	Theoretical rate of condensation, g/h	Theoretical mass flux, kg/m^2/s
CEI-1	0.3142	$3.49 \cdot 10^{-5}$	0.2938	$3.26 \cdot 10^{-5}$	0.3175	$3.53 \cdot 10^{-5}$

Here (Total w_t/Total time) - cumulative weight of condensate divided by the total time in collecting that weight; taking the average of all data points for one experiment obtained as (m_2-m_1)/(t_2-t_1) the increase in the weight of condensate in each step (every 30 min or every 0.1g increase) divided by the time taken in that step.

For the calculation a 3D model was considered. The computational mesh with 100,000 cells was sufficient to obtain a reliable mesh converged solution.

The experimental rate of condensation in grams per hour has been calculated in two ways:

$$Mean\ rate\ of\ condensation\ (g/h) = \frac{Total\ weight\ on\ balance\ (g)}{Total\ time\ (h)} \qquad (3)$$

$$Rate\ of\ condensation\ (g/h) = \frac{Increment\ in\ mass\ (m_2 - m_1)\ (g)}{Time\ this\ increment\ in\ mass\ took\ (h)} \qquad (4)$$

The second method seems more accurate because the experimental conditions were not constant during the entire time of the test. The variation of an environmental parameter can affect the rate of

condensation for that particular time, which is better taken care of in the second method, as the first method only produces the global data.

The amount of condensate, which was predicted with the described method and compared against the experimental data (Tiwari, 2011), is shown in Figure 4: .

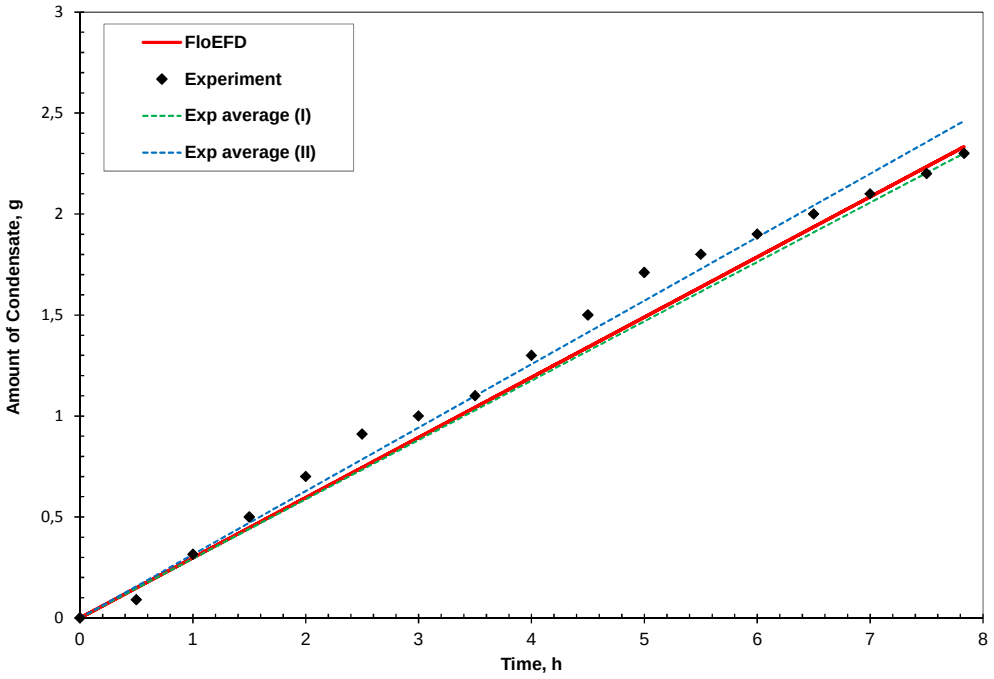

Figure 4: Amount of condensate as a function of time predicted by FloEFD and measured experimentally.

A comparison of the condensation rate shows that FloEFD predicts the condensation effects with good accuracy. The relative calculation error is about 5%.

5. Simulation of condensation/evaporation on the glass of a headlight

This section describes the results obtained in FloEFD to simulate the transient process of condensation on the internal side of a headlight's protective glass with the subsequent evaporation of condensed film caused by switching on the halogen bulb. The geometry model (Figure 5:) used in the simulation was based on the headlight "Mini Farol Auxiliar Rectangular ForteLuz (PAR)",

Figure 5: Simulated headlight.

The headlight was installed on a horizontal rectangular plate emulating a car body. The size of the plate is 450mm x 450 mm. The following parameters of humid air inside the headlight at the initial time moment were specified: T=50° C, P=1 atm, humidity 95%. The headlight's cooling was simulated during the first 30 minutes of physical time. The parameters of the external air were as follows: T=-10°C, humidity 50%. The heat exchange between the headlight and the external air flow included natural convection and radiation. Since the headlight was not fully contained a certain amount of the surrounding cold air was transported inside the headlight.

Such intensive cooling leads to the formation of a condensed vapor film on the internal side of the protective glass with the film mass (and thickness) rising as the cooling progresses.

After 30 minutes of physical time, a 55 W bulb was switched on leading to the rapid heating of the internal elements, including the protective glass. As a result the evaporation of the film was observed.

To simulate the radiative heat transfer the proposed earlier enhanced Monte-Carlo model was used.

The material for the headlight's case was plastic and the reflector was made of aluminum. The protective glass was made of quartz. The spectral dependency of the absorption coefficient was specified for the latter.

The bulb was simulated as a glass envelope filled with krypton at 2 atm. Inside the envelope a tungsten spiral (main source of radiation) was placed.

The computational mesh only contained about 190,000 cells and can partially be seen in Figure 6: .

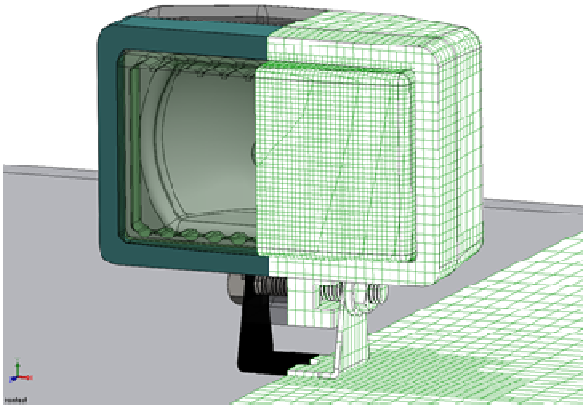

Figure 6: Computational mesh.

The change of the condensed phase mass on the surface with time is shown in Figure 7: The rate of condensation has a direct connection to the temperature of the internal surface of the protective glass.

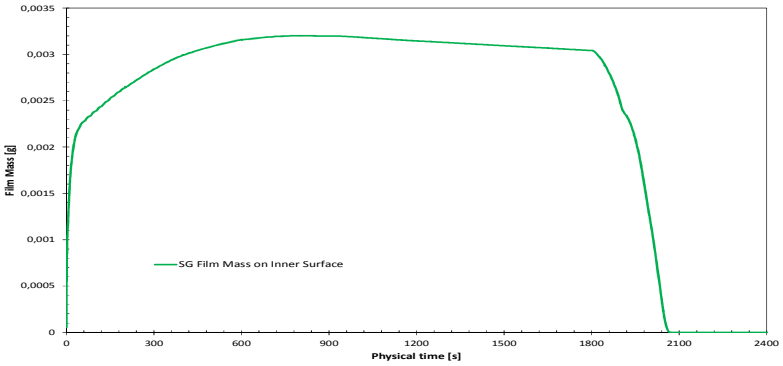

Figure 7: Change of the condensed phase mass with time.

The contour plot of film thickness on the internal side of the protective glass at different time moments is shown in Figure 8: and Figure 9: . One can see the peculiarities of its shape during the process of condensation and evaporation. Temperature isolines are also shown.

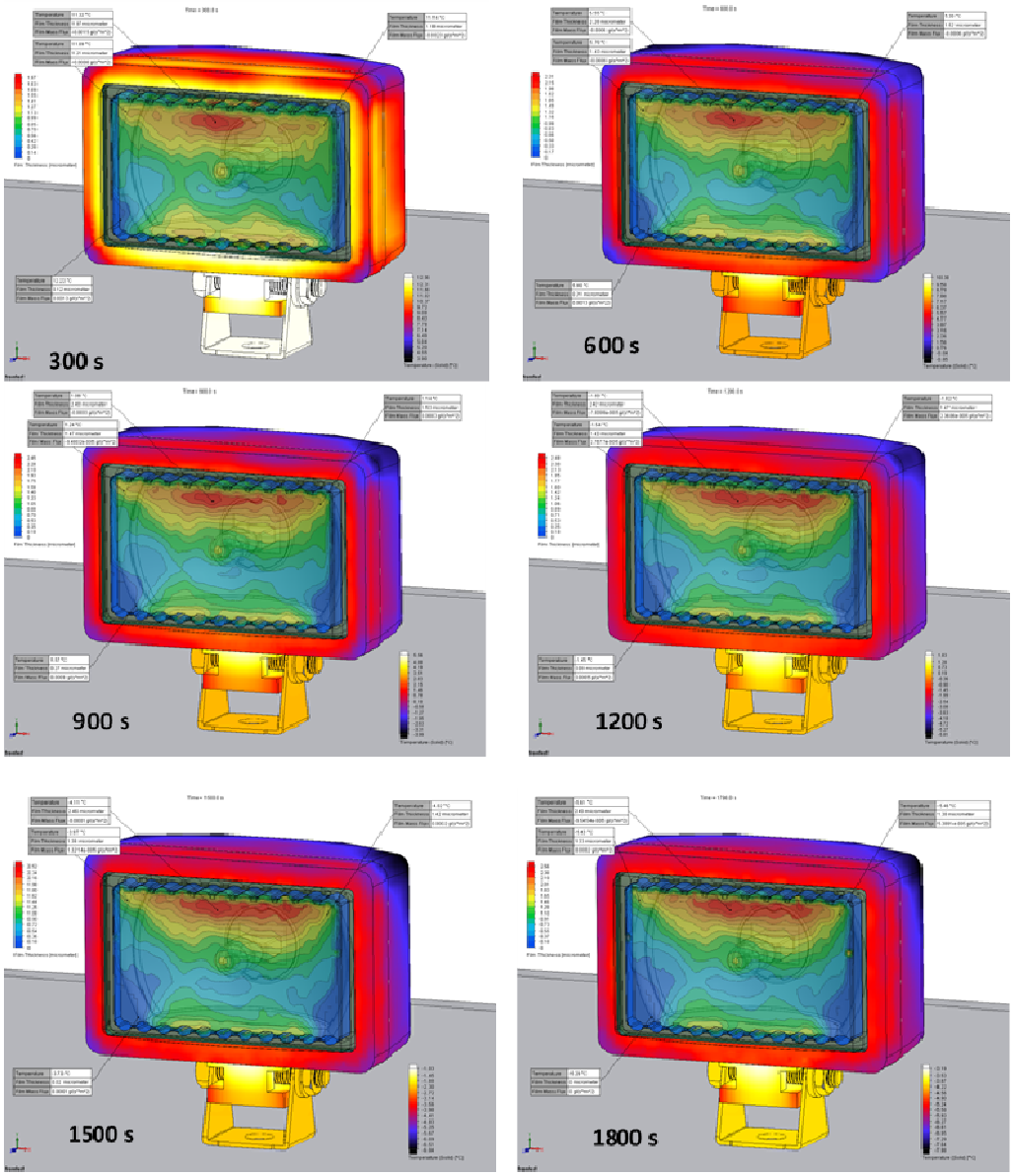

Figure 8: Contour plot of film thickness at different time moments.

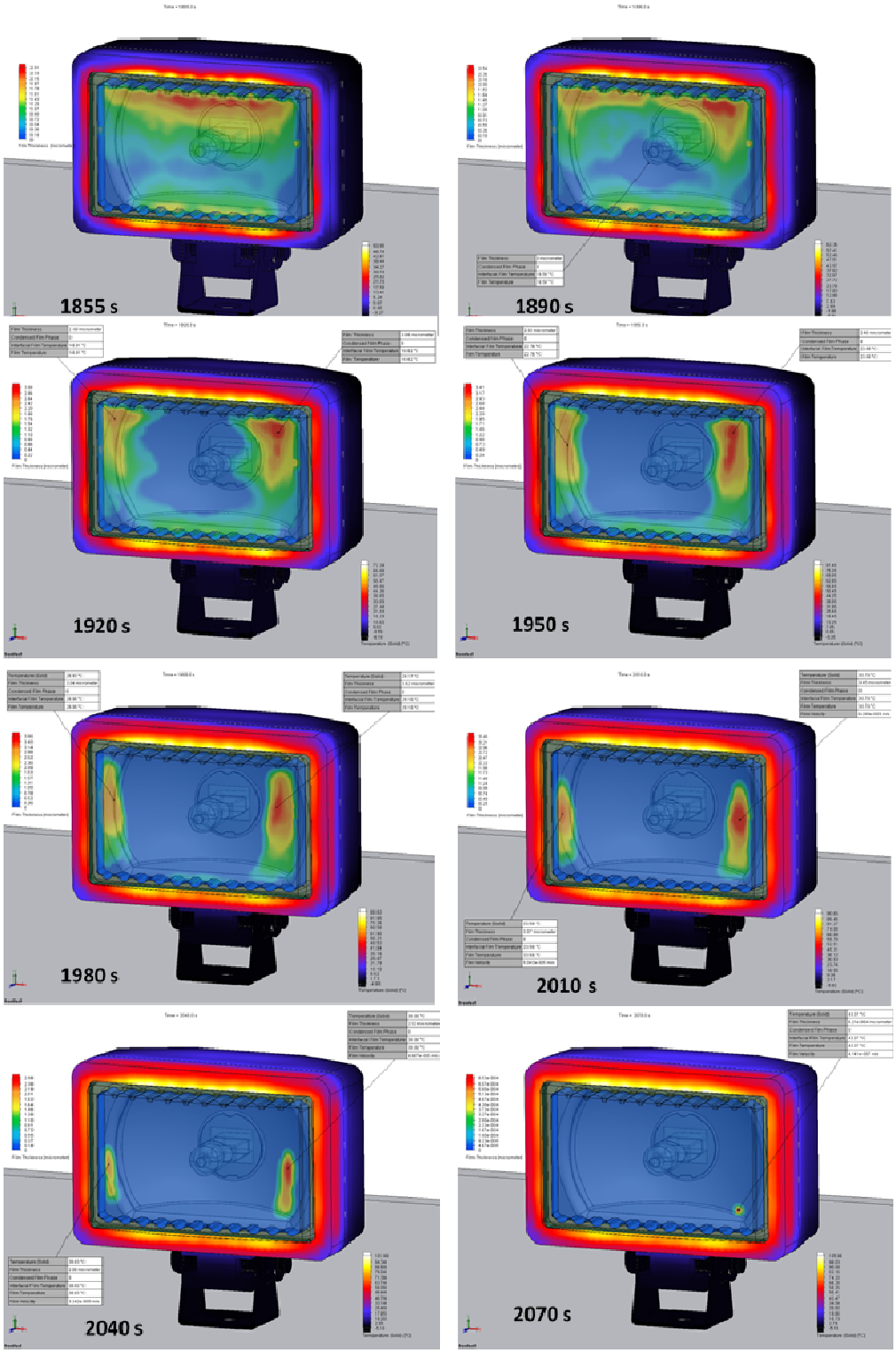

Figure 9: Contour plot of film thickness at different time moments (continued).

According to the images shown the following observations can be made:

1. Local dry regions on the internal glass surface are present, where no film was present.

2. During the cooling of the headlight (about 1000 sec from the initial time moment) the temperature at certain regions of the headlight and the protective glass are below 0°C, thus freezing (crystallization) of the film occurs.

3. Between the start of the crystallization and the switching on of the bulb (1800 sec from the initial time moment) a minor decrease in the liquid film mass (due to evaporation and sublimation) is observed

4. Once the bulb is switched on the removal of the film occurs. The processes of melting and subsequent evaporation last no more than 300 seconds. It takes additional 1800 seconds to obtain steady thermal state of the headlight.

5. Although the film mass and its average thickness decrease after switching on the bulb, there still local region on the glass, where film thickness increases. This points to evidence of the vapor transport within the headlight's internal space. To be exact, the evaporated mass from the hotter regions of the surface is condensed on the relatively colder ones.

It is also interesting to see how the gravitational forces initiate the film motion. Figure 10: shows how the film velocity is distributed along the surface at t=900 sec, just before the crystallization processes occur. The maximum value on the plot is about 1.1 mm/min and it is the highest value observed in the simulation.

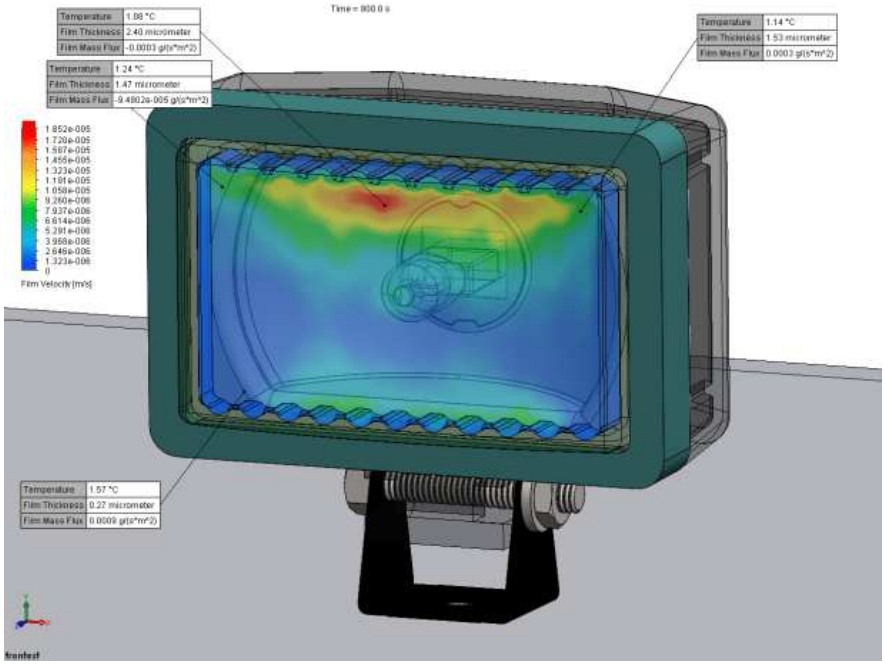

Figure 10: Film velocity distribution at t=900 sec.

6. Conclusion

A special mathematical model of surface condensation and evaporation used to perform transient simulations of film formation and removal was presented. It is currently implemented in FloEFD. Several problems were considered to validate the model. Good agreement with experimental data was obtained.

The application of the proposed mathematical model to simulate the formation of liquid film on the internal surface of a headlight's protective glass with the subsequent removal caused by switching on the bulb is demonstrated.

7. References

1. Iskra, C. R. & Simpson, C. J. (2007) *Convective mass transfer coefficient for a hydrodynamically developed airflow in a short rectangular duct*, *Int. J. of Heat and Mass Transfer* vol. 50, pp. 2376-2393.

2. Tiwari, A. (2011) *Characterisation of mass transfer by condensation of humid air on a horizontal plate*, PhD Thesis, Blaise Pascal University, Clermont-Ferrand, France.