

Condensation Modeling during Automotive Lighting Product Development Using CFD Simulation

J. Christopher Watson, Gennady Dumnov, Alexander Muslaev,
Andrey Ivanov, and Svetlana Shtilkind
Mentor Graphics Corp.

ABSTRACT

Condensation occurrence in automotive headlights can be detrimental to consumer acceptance of a product. This paper describes a technique for transient numerical simulation of liquid film formation on surfaces during lighting thermal analysis performed using Computational Fluid Dynamics (CFD), including how the film's properties influence the thermal solution. The numerical technique presented accounts for the change in the film thermal state and thickness due to heat exchange with external fluid flow and solid bodies, surface evaporation/condensation, melting/crystallization within the film volume, and its motion due to gravity and friction forces from the surrounding airflow. Additionally, accurate modeling of radiation effects is critical for lighting applications, including the attendant influence on the thermal distribution of the solids that may have surfaces subject to condensation. Headlights feature large numbers of reflective surfaces and refractive bodies focusing light on local regions and many of the semitransparent materials have absorption coefficients with distinct spectral dependencies. To simulate the thermal loads adequately, the optical behavior of these elements within the system should be modeled accurately as it potentially impacts the formation of the liquid film on the headlight's components. Radiative heat transfer is often calculated with a "band" Monte-Carlo radiation model where the entire spectral range is split into several spectral bands with material properties and boundary conditions averaged within each band. The CFD simulation study presented utilizes an enhanced Monte Carlo Radiation Model which does not require splitting into bands but allows a continuous, accurate representation of spectral material properties, radiation sources and boundary conditions.

CITATION: Watson, J., Dumnov, G., Muslaev, A., Ivanov, A. et al., "Condensation Modeling during Automotive Lighting Product Development Using CFD Simulation," *SAE Int. J. Passeng. Cars - Mech. Syst.* 9(2):2016, doi:10.4271/2016-01-1409.

INTRODUCTION

In observing evaporation or vapor condensation on the surface of solid bodies the formation and subsequent removal of a liquid film is implied. Heat release or absorption is also observed as a phase change takes place. Furthermore, the heat exchange between the solid body and the external gas flow is affected with the liquid film acting like a thermal resistance.

It is desirable for applications such as automotive lighting design to perform computational simulations of this phenomena. Numerical methods have used approaches modeling a liquid film formation by using a dense computational mesh, where the liquid film along its thickness is sufficiently resolved by the cells. However, this requires large computational resources because of the relatively small scales of the film compared to the solid object considered.

This paper details an engineering approach using mathematical models, with stated assumptions, to adequately capture all the essential physical processes while requiring relatively low computational resources for accurate prediction. This is so that analytical design performance studies with computational fluid dynamics (CFD) software are more

comprehensive. The modeling approach detailed in this paper is implemented in FloEFD software (Mentor Graphics Corporation). Through 2 validation cases, experimental study results are compared to simulation results for agreement.

This paper then describes a report on the results from a numerical thermal CFD simulation study of a vehicle headlight with complex housing geometry. The aim of the study being to predict the formation of liquid film on the internal surface of a headlight's protective lens with the subsequent removal during operation.

SURFACE CONDENSATION AND FILM FORMATION MATHEMATICAL MODELING

This section details derivation of equations for the simulation of film formation on a surface of a solid body. The thermal state of liquid film and its thickness are influenced by several of the following factors: heat exchange with the outer fluid flow and the solid body; film surface condensation/evaporation; melting/crystallization within the film volume; the film motion caused by gravitation and also due to friction forces from the outer flow.

Below are a list of 9 assumptions necessary of the mathematical modeling approach described:

1. The outer flow fluid is considered a gas mixture with nonzero humidity;
2. Mass transport is diffusive between film and vapor in the outer flow;
3. Equations describing outer fluid flow exclude film thickness, although finite;
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. 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 is laminar, if present;
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 is linear along the film thickness.

Equations describing the transport of mass per unit surface and enthalpy can be derived:

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

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

Variable Definitions:

∇_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;

A vector equation for film velocity averaged over its thickness is derived in the following form given assumptions 4-6:

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

Where:

ρ_l, μ_l - density and dynamic viscosity of liquid film material

$\vec{g}$ - gravity force vector projected on the surface

δ - film thickness

$\vec{\tau}_f$ - stress vector on the film's surface.

The equations (1)...(3) are 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 to form a closed system of equations that is fully implemented in the CFD software.

MONTE-CARLO RADIATION MODELING ACCOUNTING FOR SPECTRAL PROPERTIES - A MODIFIED APPROACH

Accurate radiation heat transfer modeling within a luminaire such as an automotive headlight is a complex analytical challenge. The following points below further highlight radiation modeling complexity for an automotive headlight application with halogen bulb:

- A Tungsten spiral of a halogen bulb may have a temperature in the region of 3000K as the radiation source
- Spectral radiation produced by the spiral interacts with semitransparent and opaque headlight elements.
- The bulb's envelope and the gas within it are heated.
- A heated bulb radiates in the IR spectrum.
- A certain fraction of radiative energy produced by the spiral is absorbed by the envelope material, thus the resulting radiation spectrum and polar pattern.

Given the above physical behavior, it is advantageous for heat transfer simulation accuracy overall to account for the absorbed and radiated heat in the IR spectrum comprehensively.

Simulating spectral effects via classic approaches depends on discretization of the entire spectral range into several bands that each have material properties (absorption coefficient, emissivity) and boundary conditions averaged within each band. There are some complexities that arise at the stage when specifying the band ranges as it is important to ensure that the all the materials properties considered within a certain band vary moderately with the change of wavelength. When many materials have to be specified, such as during industrial design of a product, this type of task becomes very detailed as the number of spectral bands becomes large.

The modified Monte-Carlo radiation model presented, and implemented in the CFD software (FloEFD), does not require splitting into bands. In contrast to "classic" Monte-Carlo model where its direct simulation of a "photon" involves having a certain energy assigned at the "photon" inception time, this modified modeling approach considers "photons" having both the energy and the wavelength assigned at the time of inception. The interaction of a "photon" with the surrounding medium is simulated such that the medium properties and the spectral boundary conditions at certain wavelengths affect only the "photons" of matching wavelength. This means that no averaging of medium properties within a spectral band is performed and instead the "real" properties of the radiating objects are used.

To implement this method there are challenges in obtaining accurate spectral distribution of "photons". This is because distribution depends on the temperature of the radiating object as well as its emissivity. The latter is dependent on temperature and wavelength so complex spectral functions have to be integrated.

VALIDATION OF LIQUID FILM MODELING APPROACH VS PUBLISHED EXPERIMENTAL STUDIES

This section describes 2 separate comparisons of experimental data to the numerical simulation of surface condensation and liquid film formation performed in the analytical CFD software.

Surface Evaporation in a Rectangular Duct

Film condensation or evaporation of water in a horizontal rectangular duct was analyzed. The experimental study and results are detailed in Iskra and Simpson (2007) [1].

For the numerical analysis 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 considered duct is part of a transient moisture transfer (TFT) facility, shown in Figure 1.

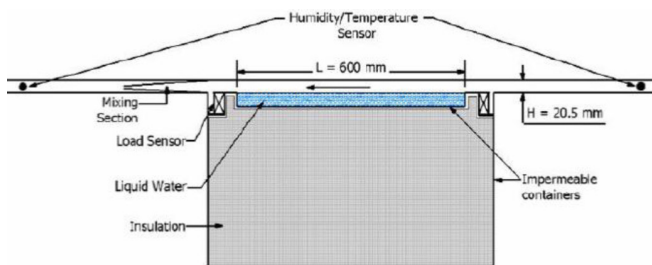


Figure 1. Schematics of the test facility showing a side view of test section (according to Iskra and Simpson (2007)).

Conditions and flow regime assumptions for simulation:

- Water temperature = 12°C.
- Air upstream of the test section is 23°C.
- Laminar ($Re_D=1500$) and turbulent ($Re_D=6000$) flow regimes are considered.
- Air relative humidity is 16.2 % for the laminar flow case and 22.9 % for the turbulent flow case 2D and 3D computational domains were considered.
- For the water surface simulation, a condensed water film initial film thickness of 50 μm was set at the bottom of the pan.

Figure 2 shows the transient development of the predicted evaporation rate compared to the experimental data [1]:

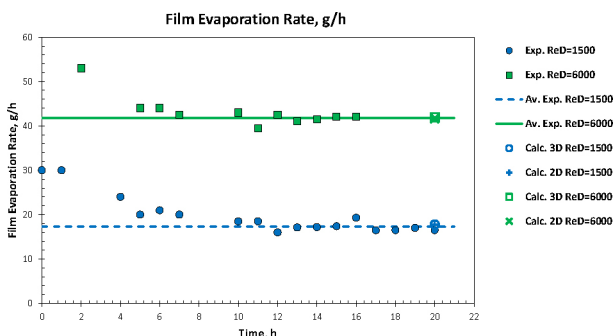


Figure 2. Evaporation rate of water predicted by FloEFD and measured experimentally.

A comparison of the evaporation rates shows there is good agreement between the described CFD numerical method and test results for accuracy. For 2D and 3D calculations, the relative calculation errors do not exceed 1.0% for $Re_D=6000$ and 3.5% for $Re_D=1500$, respectively.

Horizontal Plate - Film-Wise Condensation Simulation

The film condensation process on a cooled surface resulting from the condensation of water vapor from humid air is considered for a horizontal flat plate (area 25 cm²). The results of the experimental study are given in Tiwari (2011) [2] are compared to numerical calculations using CFD software. The compared test results are from condensation progression under controlled air flow conditions.

Experimental Study summary (based on Tiwari (2011)) :

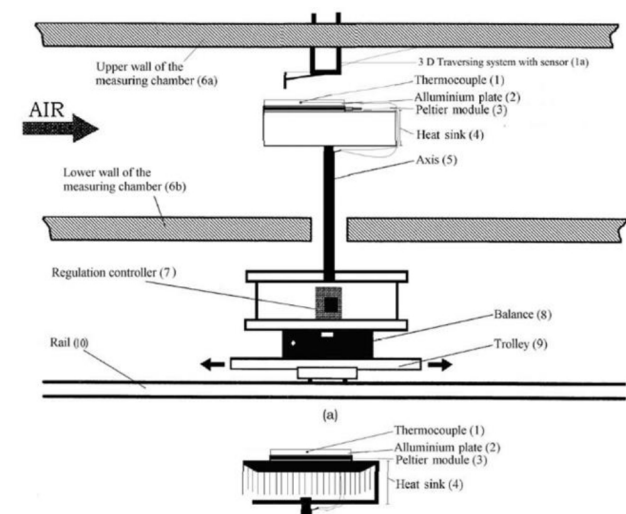


Figure 3. Schematics of the condensation unit (according to Tiwari (2011)): (a) front view, (b) side view of the upper part, which faces the airflow.

Configuration (shown in Figure 3): A square shaped Peltier module sandwiched between a square shaped aluminum flat plate, used as an active surface for the condensation, of dimension 5x5 cm² with thickness of 3 mm 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 airflow hydrodynamics, temperature and hygrometry were regulated.

Measurement: The plate temperature is constant. The condensate formed is monitored by weight of the whole system and measuring changes in weight every 30 minutes.

To complete comparison with experimental and theoretical results, the CEI-1 experiment is considered in this study [2]. This experiment was performed for a mean entrance velocity of 1.0 m/s and the mean relative humidity of 57.2 %, with 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 constant at 11.4°C.

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