Numerical Simulation of Automobile Windshield Defogging

Mark Doroudian
Mindware Engineering
Abstract
Visibility in the automotive industry is a major source of concern for the car manufacturers and safety design engineers. Water film (fog) that forms on the windshield during winter times would reduce and disturb the driver’s visibility. This water film is originating from condensing water vapor on the inside surface of the windshield due to low outside temperatures. Primary source of this vapor is the passenger’s breath, which condenses on the windshield.

Warm and dry air, which impinges at certain velocity and angle relative to the windshield, will reduce the thickness of this water film (defogging) and hence improves driver’s visibility. Therefore, a well-designed HVAC system will help reduce this water film thickness to an accepted level for visibility.

CFD technology can be utilized to study the defogging process by tracking water vapor content and predict transient variation of condensed water layer on the glass surface.

An analytical approach to simulate the defogging process including removal of the water vapor from the passenger cabin glass is discussed. The pre-processing including grid generation was performed using ICEMCFD. The transient thermal CFD simulation was performed using Fluent.

Introduction
Visibility in the automotive industry is a major source of concern for the car manufacturers and safety design engineers. Water film (fog) that forms on the windshield during winter times would reduce and disturb the driver’s visibility. This water film is originating from condensing water vapor on the inside surface of the windshield due to low outside temperatures. Primary source of this vapor is the passenger’s breath, which condenses on the windshield.

Warm and dry air, which impinges at certain velocity and angle relative to the windshield, will reduce the thickness of this water film (defogging) and hence improves driver’s visibility. Therefore, a well-designed HVAC system will help reduce this water film thickness to an accepted level for visibility.

This paper presents the results of the application of CFD technology to study the defogging process by tracking water vapor content and predict transient variation of condensed water layer on the glass surface. The pre-processing including grid generation was performed using ICEMCFD. The transient thermal CFD simulation was performed using Fluent.

CFD Grid Characteristics
The passenger cabin of an automobile contains air with a relevant content of water vapors, originating from various sources (breath of passenger(s), outside humidity, etc.). These vapors condense on the inside surfaces of the cabin, especially on the windows (windshield, side windows, rear windows), due to lower temperatures of the outside air and radiation heat transfer effects.

The condensed water film blocks the view from the inside and it has to be removed before the vehicle can be safely operated. In order to remove this condensed water film, the HVAC unit of the car is used in the defogging mode. In this mode, air with lower humidity than ambient supplied at certain velocities and temperatures into the cabin through the demister and defroster registers.

A jet of air through the defroster nozzle and demister are directed at the windshield inner surface. The velocity of this jet of air is usually around 4 to 5 meters per second for a length of scale of .5 to 1.5 meters. The boundary layer thickness near the windshield is approximately 17000 micrometers. The fog layer smaller than 5 micrometer is considered as transparent. The fog layer thicker than 200 micrometer might result in drips and run offs. To study fog layers of that thickness, two-phase flow analysis is required.

The CFD grid for defogging simulation for this study had the following characteristics:

- Ten layers of prism cells adjacent to walls with fog layer.
- The value of the turbulent $y^+ = 1$ in the cells adjacent to walls with fog layer.
- Height of the cells adjacent to walls with fog layer was greater than twice the fog layer thickness.
The mesh was conformal (no hanging node adaptions or non-conformal interfaces) on the walls.

**CFD Grid Generation**

The ICEMCFD Cabin Modeler was used to create the minivan geometry. The Cabin Modeler was used to adjust the size of the default minivan cabin model. The dimensions of the cabin interior were taken off of an actual vehicle with a tape measure.

Most often, the vehicle data exists in a CAD package and the physical model is not available. In that case, the CAD model is imported into ICEMCFD MED using one of the direct cad interfaces. Usually some CAD clean up is necessary before running ICEM grid generation tools. The CAD preparations include checks for single edges (i.e. no gaps, the fluid domain is enclosed within the cabin), proper curves and material points.

For this cabin model, the grid resolution was chosen so that the resulting grid would represent the actual geometry as close as possible and without paying additional penalty of having high number of computational cells.

In this interior cabin flow model, tetrahedral cells occupy most of the interior volume of the vehicle. The triangle faces of the tetrahedral cells, which were on the glass surfaces and some inlet/outlet boundaries, were extruded to represent four domains:

1. The boundary layer near the windows (air side)
2. The glass domain (solid side)
3. The ice domain (solid side)
4. Inlet and outlet regions.

10 layers of prisms were extruded in the boundary region with first layer initial height of 100 micrometers (the one next to the window shells), the height of these prisms was exponentially increased based on a growth rate of 1.1. (the 10th prism layer height, which was attached to the tetras, was 0.235mm). See the figures below:

![Figure 1. The grid on a cut section through the cabin centerline. The red oval shows the defrost nozzle region and is shown in the next figure.](image-url)
Figure 2. The grid on a cut section through the cabin centerline near the defrost nozzle opening into the vehicle cabin

Figure 3. A section cut through the vehicle centerline shows the layout of the prism layers inside the computational domain

The outer shells of the boundary layer prisms were used to extrude the five layer “Glass prisms”. The total height of those equidistant glass layers was 4.6 mm. Then another five equidistant layer was extruded to represent the “Ice prisms” with a total height of 0.4 mm. The inlet and outlet boundary regions were extended by extruding the outer triangular shells. This was done to account for the ducts that supply the defrost and demist registers.

This flow model was constructed to perform both defogging and deicing simulation using this single computational domain. However, in the case of the defogging simulation the ice layer was omitted.
Boundary Conditions and Analysis Set Up

The flow into the cabin was set to 400 kg/hour of air. The air was entering the cabin through the defrost nozzle, the driver side demist, and the passenger side demist. The air was exiting the vehicle through the air extractors in the rear of the vehicle.

Some of the boundary conditions that were applied to the cabin defogging model:

- The walls that needed to be tracked for vapor thickness (inside surfaces of the green house glass) were flagged for the solver.
- The air Temperature at the inlets as a function of time by means of a user defined subroutine (UDF).
- The relative humidity(species mass fraction for water) at the inlet boundaries was assigned as a function of time(UDF).
- Atmospheric conditions were applied at the outlet boundaries
- The glass temperature was set to 256 K.
- The cabin relative humidity was initialized at 50%.
- The thickness of the water condensate film which was assumed to be uniformly distributed on all the surfaces was set to 10 micrometers.

The CFD solver that was used to perform the defogging simulation was Fluent. Fluent is a commercial CFD solver. The following solver functions were used for this defogging calculation:

- Init_water_distribution
- Calc_water_source
- Watervapor_source
- Solid_side_heat_flux
- Fluid_side_temperature
- Inlet_water_vapor
- Watervapor_diffusivity

The strategy for performing the defogging simulation was to decouple the flow solution from the temperature and humidity calculations. This was done by assuming that the flow field in the cabin is driven by the flow coming through the HVAC unit and the buoyancy effects have a small effect on the defogging of the green house surfaces.

Based on this assumption, the flow field in the cabin was solved for steady state conditions. This steady state solution for the flow field is then used as the initial conditions to solve for the temperature and humidity over a simulation period. The time step is usually in the range of 0.01 to 0.1 seconds. For this calculation, the time step was set to 0.06 seconds.

During the simulation, the solver tracks the water vapor content through out the computational domain. The following assumptions were made regarding the water condensation on the walls:

- Water film is a continuous film of condensed water on a wall, contained in the first cell next to the glass.
- Surface tension, gravity effects are neglected on the water film
- Water film is not flowing or moving at any time.
- Evaporation/condensation will happen at specified walls only.
- Water vapour is at saturation conditions at the interface between the fog layer and moist air. The solver delivers temperature, pressure and humidity for the cells next to the walls. The solver dbefogging module will determine the mass transfer process and direction based on the following criteria:
  - Evaporation when $T_{\text{cell}} > T_{\text{saturation}}$, deficit in specific humidity.
  - Condensation when $T_{\text{cell}} < T_{\text{saturation}}$, excess in specific humidity.
  - Equilibrium (no mass transfer) when $T_{\text{cell}} > T_{\text{saturation}}$, and excess in specific humidity or $T_{\text{cell}} < T_{\text{saturation}}$, and deficit in specific humidity.
  - Defogging module will determine the mass transfer rate by a diffusion law for specific humidity.
  - Defogging module will update water film thickness based on the new mass transfer rate determined above.
  - Defogging module will apply appropriate source/sink to the water vapour species in the cells next to glass.
  - Defogging module will compute latent heat of vaporization/condensation and updating thermal boundary conditions to the solid/liquid sides of the glass.

Results
The results of the analysis that are of interest are:
- Species mass fraction of water
- Thickness of the water condensate on the walls
- Source term of water vapor on the pre-selected internal walls. Positive is evaporation, negative is condensation.
- Saturation temperature for water vapor in the cell layers next to the pre-selected walls.
- Gradient of the water vapor mass fraction in the cell layers next to the walls to show the potential for evaporation or condensation.

Some of the results of the analysis are shown below.

Figure 4. Particle traces released from the defrost nozzle and side demist inlets. The traces are colored by velocity magnitude.
Figure 5. Y Plus distribution on the green house surfaces

Figure 6. Temperature (K) distribution on the green house surfaces
Figure 7. Velocity distribution near the green house surfaces

Figure 8. Water vapor source term plot on the green house surfaces
Summary

The ICEMCFD Cabin Modeler was used to create the grid for the interior cabin of a minivan. The ICEMCFD med was used to create to prism layers near the green house surfaces. The Fluent solver was used to perform a flow simulation to track the moisture content in the air and the water film thickness on the vehicle interior surface. These tools could be used to help design the defroster nozzle and the demister register quite sometime before the physical part is ready for testing.

References