Transient thermal simulation process for cabin-electronic components

Gabor Matulik, Rakesh Dontham

SIMULIA
Dassault Systemes Deutschland GmbH
Meitnerstraße 8, 70563 Stuttgart-Vaihingen

gabor.matulik@3ds.com
rakesh.dontham@3ds.com

Abstract: The present work introduces a transient coupled aero-thermal simulation process focusing on the temperature of the electronic components inside the cabin of a vehicle. The simulations are carried out with the PowerFLOW Lattice-Boltzmann solver for the fluid domain coupled with the PowerTHERM finite element solver for the solid domain. The process represents a two-hour transient measurement (one-hour hot soak and one-hour cooldown). The validation is ongoing, the process was running on the Dassault Systemes' Car Bleu.

1 Introduction

The transportation and mobility industry is going under major changes due to the electrification. The topic of cabin-electronic components however can arise in cars with internal combustion engine as well as in electric vehicles. The number of electrical components connected for example to driver assistance, autonomous driving, and infotainment systems in passenger vehicles is growing. These systems' control units consume more and more power as they become more complex and are installed into smaller and smaller spaces inside the cabin. To determine whether adequate cooling is provided in various transient scenarios requires a streamlined simulation process.

With the PowerFLOW/PowerTHERM coupled approach, the current work presents a highly automated transient thermal simulation process for cabin-electronic components. The sample procedure deals with a measurement situation that includes two stages: first, the car is turned off, parked under the sun for an hour, then all the systems are turned on and the HVAC system is cooling down the cabin for an hour. The PowerTHERM/PowerFLOW Time Ratio and the Adaptive Coupling, two recently added elements of the PowerFLOW program, are crucial to reduce the turnaround time of the modeling process. In the case of a quasi-static flow field, the former makes it possible to calculate extended thermal transients; the latter permits the chosen adaptive scheme to automatically modify the time ratio in dependence on the rate of temperature change between thermal solutions. The flow field is in a quasi-static state aside from the first two to three seconds of the second stage, so the three steps of the introduced simulation process are as follows:

- hot-soak under the sun (1 h) - adaptive coupling
- initial flow development after starting HVAC (3 s) – high resolution transient
- transient cooldown (1 h) – adaptive coupling

2 Methodology

The simulation process is highly automatized from case preparation until post-processing. The base of automation is the folder structure (see Figure 1.), and a naming convention of the parts, which enables an organized workflow. The user is required to place the surface mesh files, the boundary condition data for the HVAC, the csv file with measurement probe locations and the PowerTHERM model into the corresponding folders. It is possible to introduce new parts (for example a mobile phone) between Soak and Flow Development phases, hence the different Mesh folders for them. A Shell script governs the three steps of the process, and the communication with the PowerCASE API is done in Python. Reducing the user interaction was a high priority goal during the development of this process. We achieved the automatic importing of mesh files, probe locations, PowerTHERM Model and Boundary conditions. When the naming convention requirements are met, then the template of the PowerCASE sets the parameters automatically. The Initial coupling between PowerFLOW parts and PowerTHERM parts are set on name-to-name basis, user interaction is needed, when special coupling settings are required.

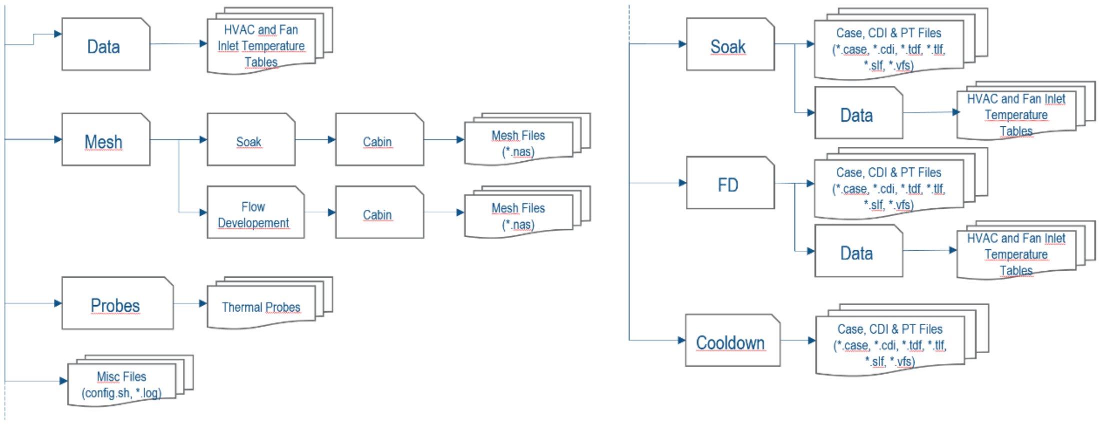

Figure 1: Folder structure of the process

2.1 Hot-Soak

In this simulation step the vehicle is standing under the sun for (at least) one hour. Goal of this step is to calculate the surface temperatures of the parts, and to get the thermal layering effect of the air inside the cabin. We represent the sun with a Solar Lamp in PowerTHERM. For its setup we use pyranometer(s). With this approach we can mimic the solar lamp parameters of a thermal wind tunnel measurement, in case the customer has such data. On the outer surfaces of the cabin we assume natural convection with T_{env}=45°C and h=5W/(m^2K). In this phase natural convection develops inside the cabin, the velocities are very low, the heat transfer coefficient is around 2-5 W/(m^2K). After 1-2 minutes the flow field reaches a quasi-static state, so in order to allow its development the starting PowerTHERM/PowerFLOW time ratio is set to 1. In order to reduce the simulation time, the PowerTHERM/PowerFLOW time ratio will be gradually increased with the help of adaptive coupling, when the temperature gradients are below a certain limit.

2.2 Flow Development - Cooldown

In the second simulation step the HVAC system and the board electronics are starting. This phase aims to calculate the transient flow field until it reaches a quasi-static state. This takes 2-3 seconds depending on the size of the cabin. Since the flow field is highly transient, the PowerTHERM/PowerFLOW time ratio is fixed to 1. The method uses velocity inlet boundary conditions with transient temperature field for the HVAC inlets. Some electronic components have active cooling via fans. In order to save on the computational effort, those fans are represented by Inlet-Outlet boundary condition pairs, where the average temperature of the outlet face is applied onto the inlet face. Heat generation of electronic devices is modelled on chip level in PowerTHERM. Starting from this phase until the end of cooldown we assume a 15 km/h head wind for the car (this comes as a customer request, the parameter is flexible). A standard external aerodynamic simulation provides the heat transfer coefficient data to be mapped on the cabin's outer surfaces.

In the third simulation step the flow field is already quasi-static, the PowerFLOW/PowerTHERM time ratio starts from 40, and with adaptive coupling it can gradually reach maximum of 180. The rest of the settings are same or very similar to the flow development phase.

2.3 Meshing strategy

The PowerFLOW surface mesh is triangle based, we recommend to follow the Best Practices Guideline: using tesselation for flow-critical components (Coarse – Very fine; Surface Tolerance: 0.5 mm – 0.01 mm; Normal Vector Tolerance: 30° - 20°) and using wrap for non-flow-critical components (mesh size between 5 mm – 25 mm).

The volume mesh is built up from voxels (hexahedrons), created automatically by the discretizer. The refinement region's parameters are set in PowerCASE. The soak simulation uses the coarsest mesh in the process, the minimum voxel size is 13 mm. In order to avoid excessive surface mesh count in a voxel, it is recommended to coarsen the very fine mesh parts. That is another reason why the folder structure offers different Mesh folders for Soak and Flow Development. Due to its highly transient nature, the Flow Development phase has the finest mesh, the minimum voxel size is 0.25 mm or 0.5 mm, depending on the required minimum gap size (1.25 mm and 2.5 mm respectively) to be resolved. In order to reach a reasonable runtime for the long transient, the cooldown phase is one level coarser then the Flow Development, i.e: 0.5 mm or 1 mm minimum voxel size.

PowerTHERM allows hybrid meshes, consisting of shell and solid parts. A balanced approach is to use triangle surface and tetrahedron solid elements on the important electronic components, and mixed quad+tria surface elements on the rest of the geometry.

3 Bleu car - cabin simulations

The Dassault Systemes' Bleu car is a car model, on which we can display our processes. Albeit in a simplified manner, but the cabin of the Bleu car contains everything, which usually occurs in a customer case (e.g. monitor on the middle console, electronics with and without cooling fan, HVAC system, etc.). Figure 2. shows the cabin of car Bleu from the outside, and Figure 3. from the inside.

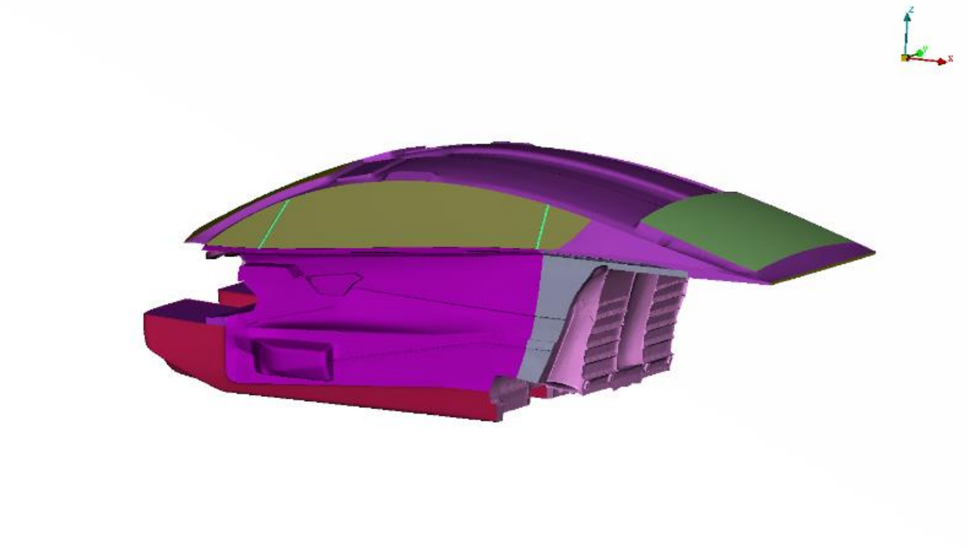

Figure 2: Bleu Car cabin outside

Figure 3: Bleu Car cabin inside

On Figure 4. we can see the surface temperature-, and on Figure 5. the air temperature distribution after one hour of soak. The hottest faces are the ones which are parallel to the solar lamp (below the windshield), hence higher solar irradiation. The thermal layering effect is clearly visible on the air temperatures. Altough one hour seems a very long time, it is not enough time to reach saturation temperature for all the parts. The dashboard probe is almost there, but the probe on the middle console monitor shows a positive temperature gradient by the end of the simulation (Figure 6. and Figure 7. respectively). This is the reason why we need to run a transient simulation for this phase, and cannot have a cheap steady-state solution.

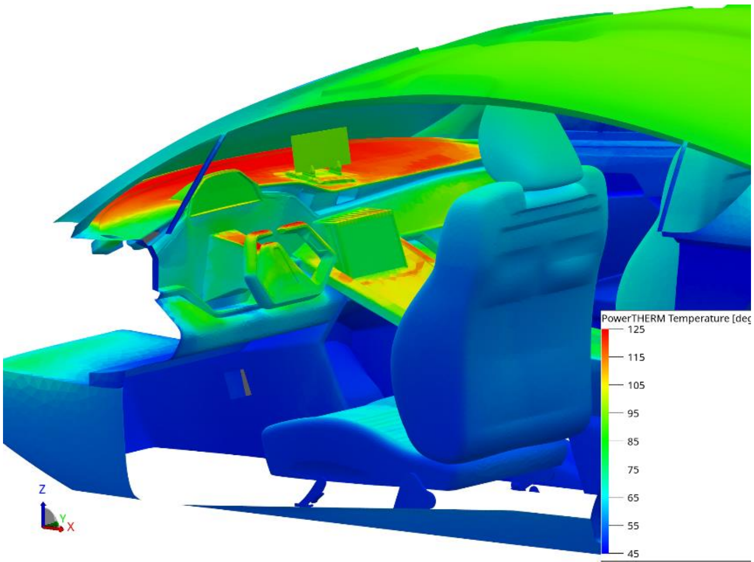

Figure 4: Surface temperature distribution after the soak

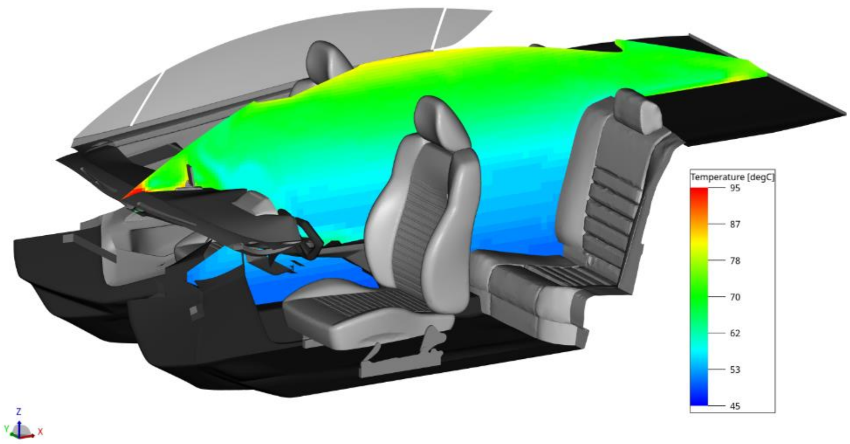

Figure 5: Air temperature distribution after the soak

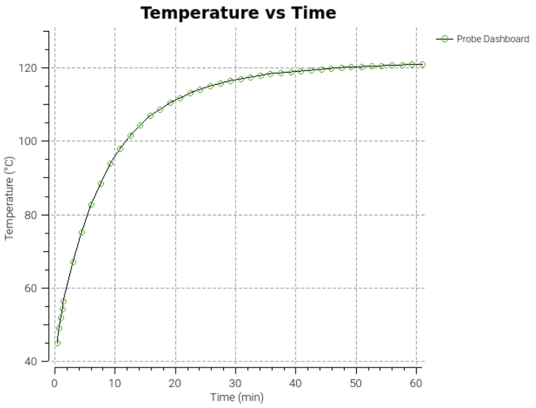

Figure 6: Probe temperature on dashboard

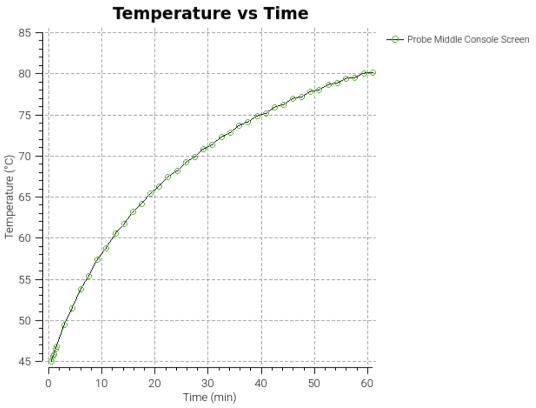

Figure 7: Probe temperature on middle console display

Figure 8. and Figure 9. shows the quasi-static velocity field in the cooldown simulation in a vertical and in a horizontal plane respectively. On Figure 10. we see the surface temperature distribution at the beginning and at the end of the cooldown phase.

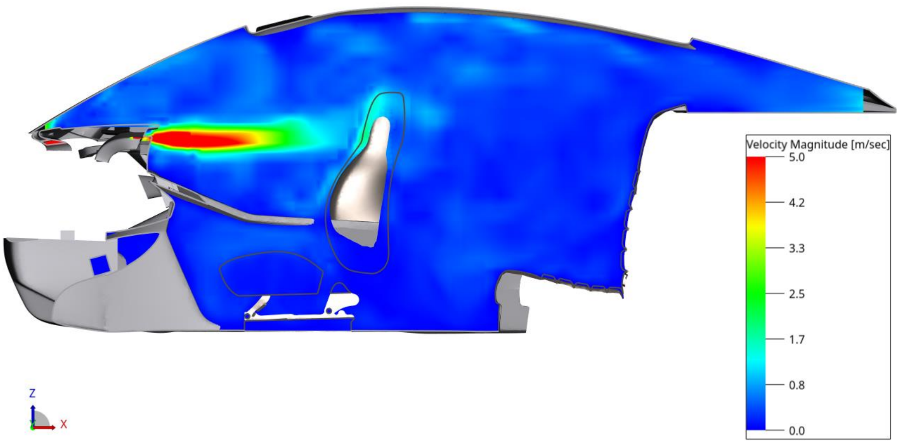

Figure 8: Velocity distribution in a vertical plane (cooldown phase)

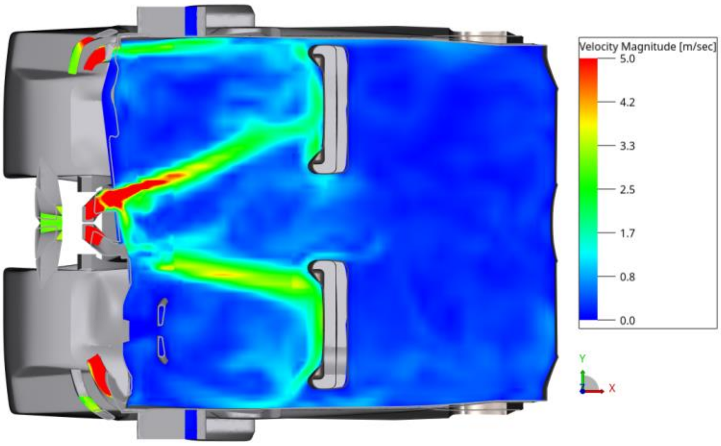

Figure 9: Velocity distribution in a horizontal plane (cooldown phase)

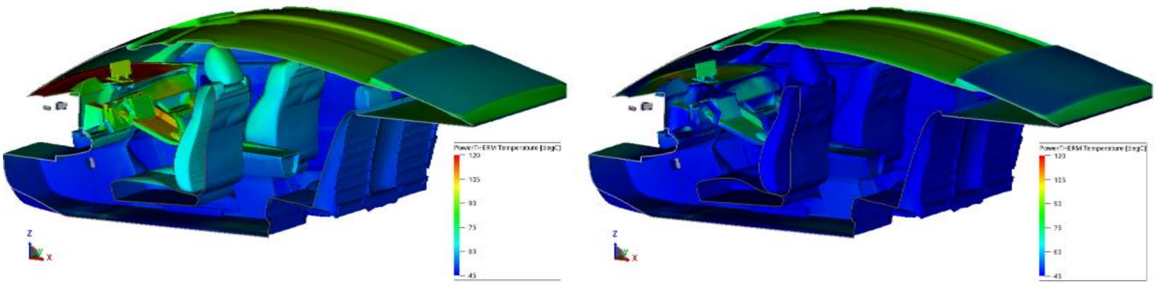

Figure 10: Surface temperature distribution in the beginning (left) and in the end (right) of the cooldown phase

4 Summary and future points

We have introduced an automated transient thermal simulation process, which aims for calculating the temperature of cabin electronic components. The process is not yet validated, this task will be carried out, when we can get access to detailed customer measurement data. Nevertheless, the expected physical phenomena are appearing in the simulation results, and other SIMULIA PowerFLOW/PowerTHERM coupled transient simulation processes are showing good agreement with the measurements.

A possible improvement of the methodology is to replace the inlet-outlet boundary condition pairs in electronic cooling fan models. With a small calculation expense, the use of transient boundary seeding would allow the flow field around those fans to be much closer to the real one.