Advances in Multiphase HVB Modelling: Coupling of Particle Depositions and Channel Blocking

David Koti, Dr. Enes Aksamija

Magna Vehicle Engineering
MAGNA STEYR Fahrzeugtechnik GmbH & Co KG
Liebenauer Hauptstrasse 317

david.koti@magna.com
enes.aksamija@magna.com

Abstract: Thermal runaway and venting of HV-Batteries are multi-physical fields which include many interconnected phenomena under extreme flow and heat transfer conditions (velocity, pressure, temperature and their respective gradients). Radiation, phase change (boiling, melting, solidification), housing deformation, combustion, arcing and particle/wall interactions play a significant role but only a few of these mayor contributors can be modelled in an effective manner.

So far, only thermal coupling was implemented in previous work which did not fully represent the modified flow behavior due to particle deposition. Mayor breakthrough was achieved in the current study by combining the heat transfer of particulate depositions with the effect of channel blocking (sudden closure of flow passages).

A combined multi-phase approach was used to account for these effects simultaneously. The framework was successfully implemented in the STAR-CCM+ environment and was tested from module to complete pack level.

Keywords: CFD, HVB, thermal runaway, venting, particle deposition, channel blocking

Contribution: 2025 FKFS Conference on Vehicle Aerodynamics and Thermal Management
15 – 16 October 2025 | Leinfelden-Echterdingen

1 Introduction

It has been observed by multiple research groups that large glowing particles, which manage to escape the pack after a venting event are one of the major sources for ignition of the highly flammable gas mixture (see Figure 1). It is a common strategy in the automotive industry to implement particle separation strategies into the venting channel design, by means of baffles, traps, mazes, grids or dedicated venting devices. CFD simulations can only contribute value to channel design if understanding of particle-wall interactions is further deepened and calibrated with measurements. Modelling workflows need to account for all the interconnected physical effects of which there are many: high Mach number flow, multiple reacting species, thermal radiation, particle deposition with phase change, gradual channel clogging, geometry deformation, etc. The present project aims at closing one of the largest and last missing links in the modelling chain, namely the thermal and gas dynamical impact of particle depositions on cell-to-cell propagation and therefore channel design.

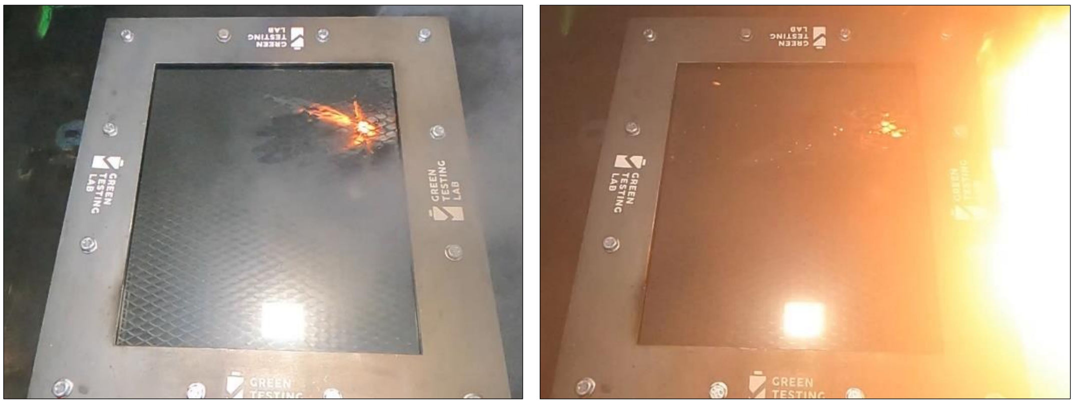

Figure 1: Glowing particles ejected during venting (left), ignition outside caused by the escaped large glowing particles (right), [1].

2 Modelling

Modelling of venting phenomena is a novel and lively field with many different approaches among research institutes and OEMs for tackling the complex multi-physical challenges. It is common practice to increase modelling complexity step-wise and add physical phenomena depending on the problem statement. Due to the increasing geometrical complexity of particle separation strategies in modern HVB venting channels, the particle-wall interactions (incl deposition and clogging) need to be modeled reliably in order to evaluate different design with CFD. As a modelling foundation, the multi-component turbulent gas phase was modeled together with solid walls (CHT) (Figure 2a). The first step-up of complexity is the addition of a lagrangian phase representing molten particles, including thermal radiation and temperature dependent probability for deposition (Figure 2b). As a third complexity step, the deposition is modeled as a separate continuous phase, which can exchange mass and heat with walls and gas while blocking the flow locally for the gaseous phase (Figure 2c).

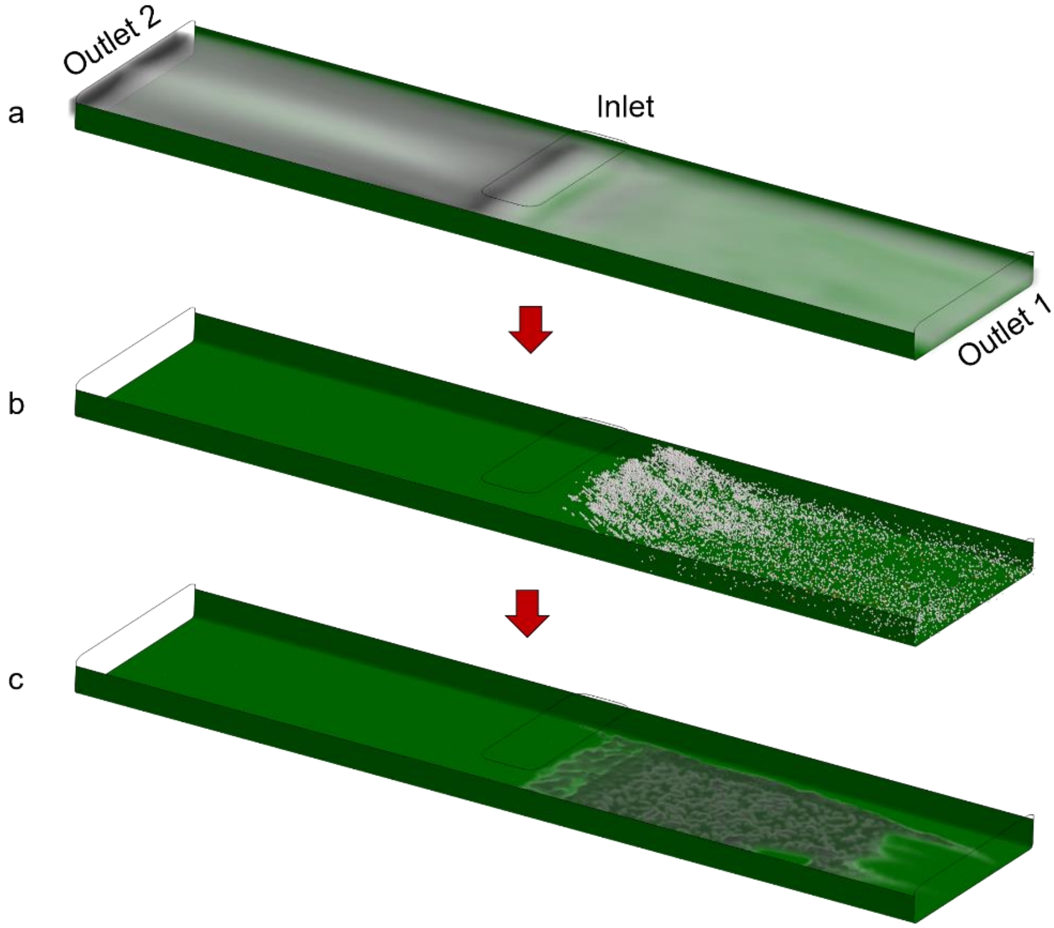

Figure 2: Schematic overview of different modelling level strategies for multi-physical problems.

2.1 Demonstration geometry

A generic T-junction was chosen as a demonstrator geometry as it represents many typical flow scenarios within battery packs and modules. Figure 3 shows an isometric view of the geometry with the respective in/outlets. The venting direction was chosen downwards with realistic distances between cells and solid walls which represent battery/cell housings. The injection angle of the particles was varied between three discrete injection angles relative to the main gas flow in order to demonstrate the increased deposition rate and clogging behaviour on one side.

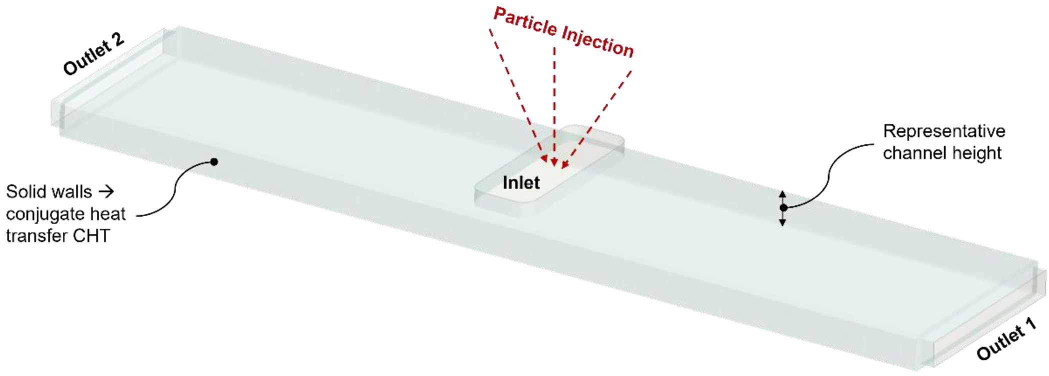

Figure 3: Schematic overview of the simulated geometry

3 Results

Figure 4 shows the velocity distribution on a cut plane in the middle of the channel comparing two modelling approaches. While modelling without particulate phase results in a perfectly symmetrical flow distribution between the two outlets (Figure 4a), inclusion of a lagrangian phase leads to accumulation of a deposition phase in the respective channel (Figure 4b-4d).

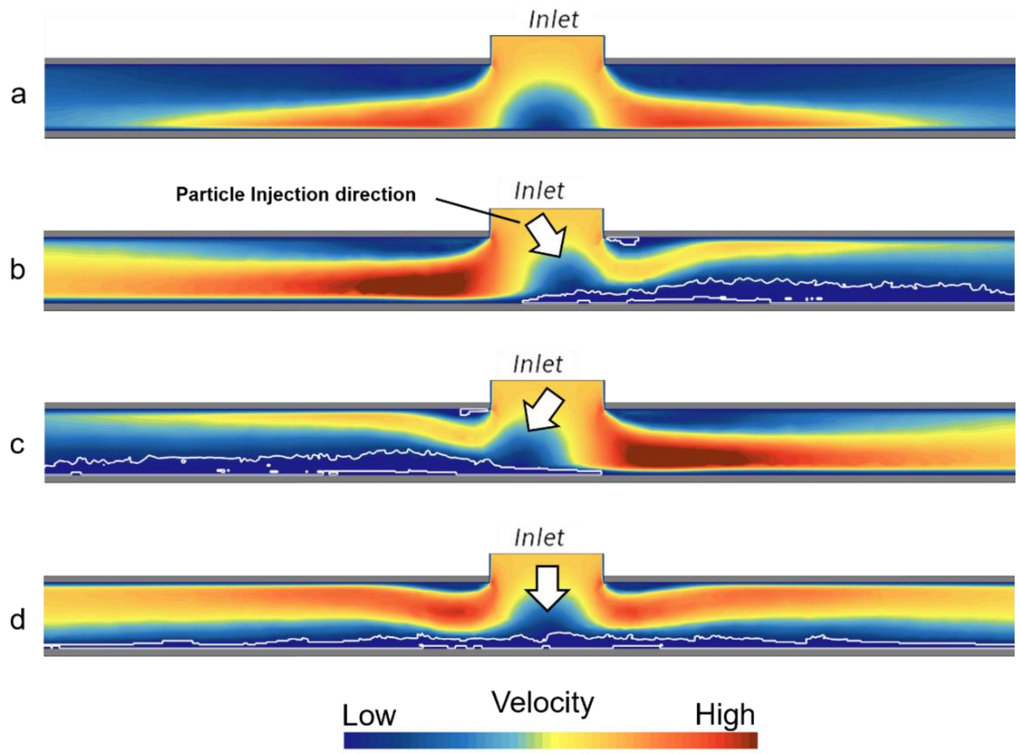

Figure 4: a: gas flow only, b: particle injection to the right, c: particle injection to the left, d: neutral particle injection direction; phase boundary to the deposit phase is indicated by the white line.

Depending on the particle injection direction, increased gas flow rate can be observed at the opposite side of the channel, proving the clogging behaviour of the deposited phase.

At the outlets, gas mass flow rates and average temperatures were recorded (see Figure 5). The simulation with only gas is represented by the black line for both outlet 1 (right) and outlet 2 (left). The case with inclined particle injection towards one outlet is represented by the blue line. Two main findings can be observed: the flow rate is lower at outlet 1 due to deposition towards this direction (Figure 5b) and respectively the flow rate is higher in the opposing channel where the cross section is free (Figure 5a). In contrast, the temperature profiles do not show the same symmetrical behaviour when channel blocking occurs. The shape and level of the gas outlet temperature strongly depends on the particle deposition probability (Figure 5d) as this parameter determines which portion of the enthalpy is removed from the gas stream during a particle-wall interaction.

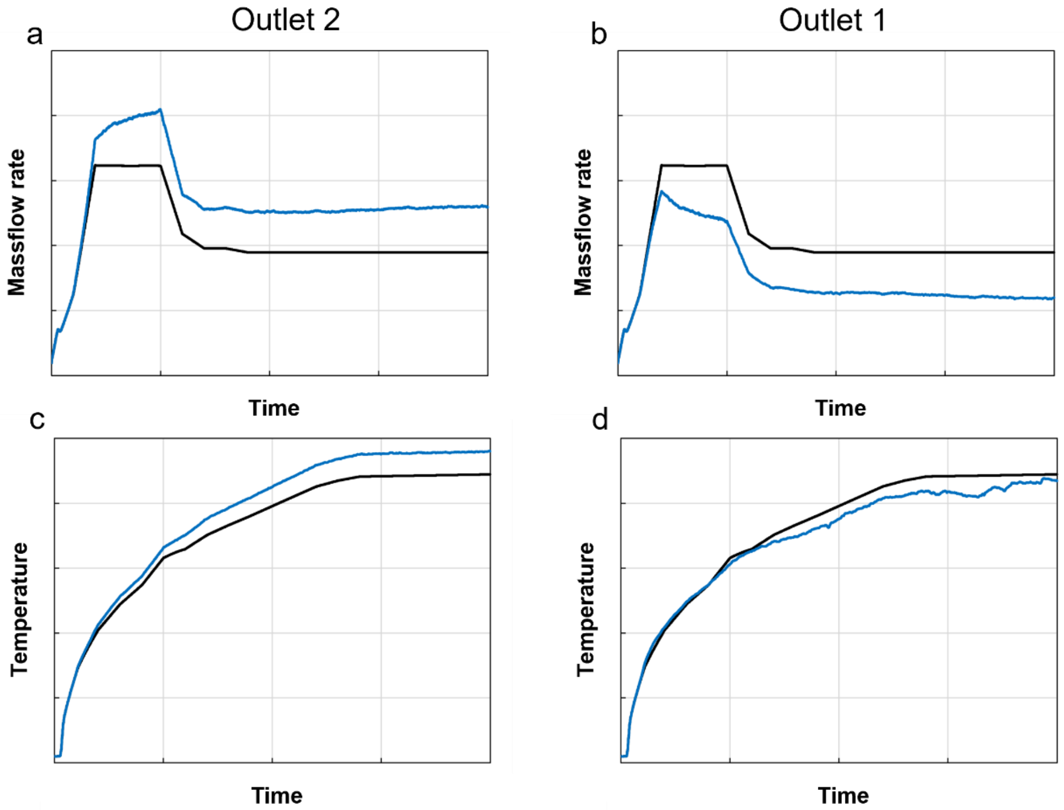

Figure 5: Mass flow- and temperature profiles at the channel outlets for two modelling approaches; gas only (black), deposition modelling with injection towards outlet 1 (blue).

The distinct differences of flow fields when changing modelling depth is also reflected in vastly intensified heat transfer and significantly higher local wall temperatures, as seen in Figure 6. Heat transfer from a pure gas stream is typically homogeneous (see Figure 6, top) and significantly less intense compared to conductive heat transfer from a sticking molten solid. Local wall temperatures at the deposition sites can be higher by several hundred degrees (see Figure 6, bottom) and therefor cause thermal propagation if located on a neighboring cell wall. This example shall demonstrate the importance of the correct modelling depth for the prediction of cell-to-cell propagation.

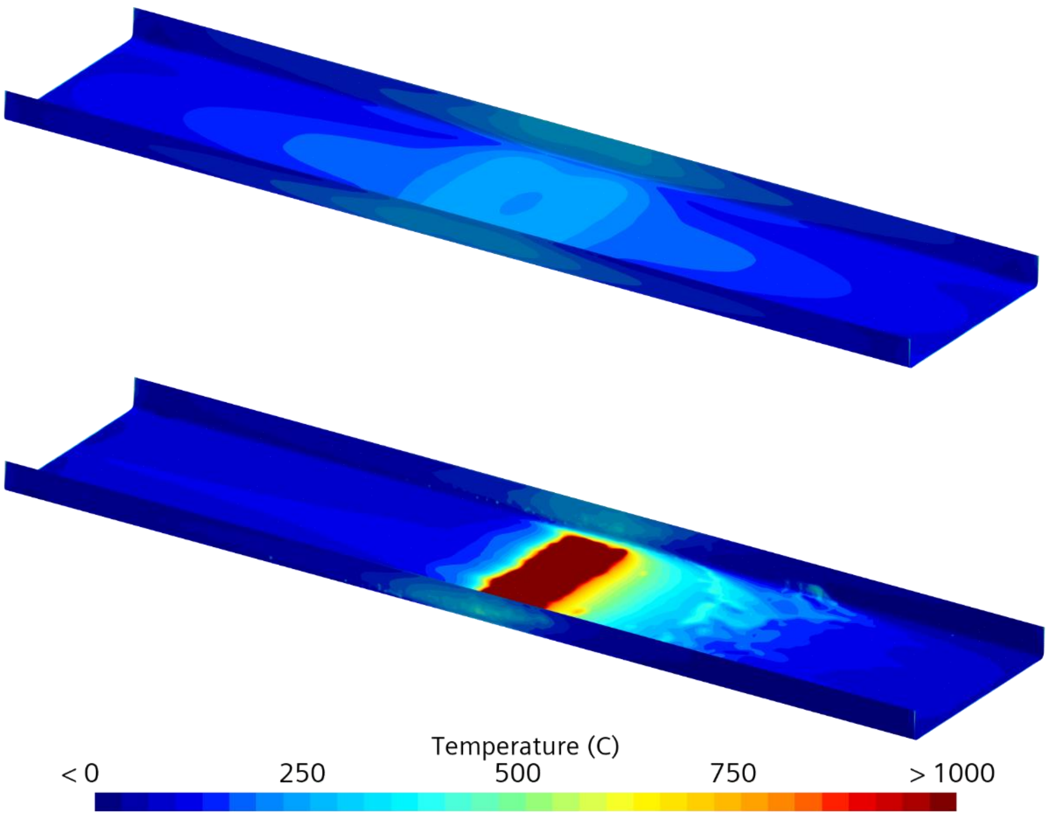

Figure 6: Comparison of wall temperatures depending on the modelling approach; smooth distribution with the gas-only approach (top) and increased temperature levels with heterogenous patches when particle deposition is modelled (bottom).

4 Validation

Validation of venting simulations is extremely challenging due to the multi-physical nature of the problem and the extremely harsh conditions which can damage equipment and falsify results. In general, CFD can be an indispensable tool in venting channel design if the adequate modelling depth is chosen and all relevant phenomena are captured. While generic flow field understanding can be obtained from straight-forward CHT simulations, where only the gas phase is considered, higher modelling depth is required for the evaluation of propagation risk. As shown in Figure 7, the measured outlet temperature of the glass battery matches the confidence band of the simulated values when the heat of the particles is considered (red band) using the above-mentioned approach.

Future focus will be put on validation of deposition probabilities. The model parameters need to be calibrated against recorded video data until matching deposition patterns are obtained in the simulation. As particle sizes and composition vary vastly between cell chemistries, accompanying validation will remain mandatory in the foreseeable future.

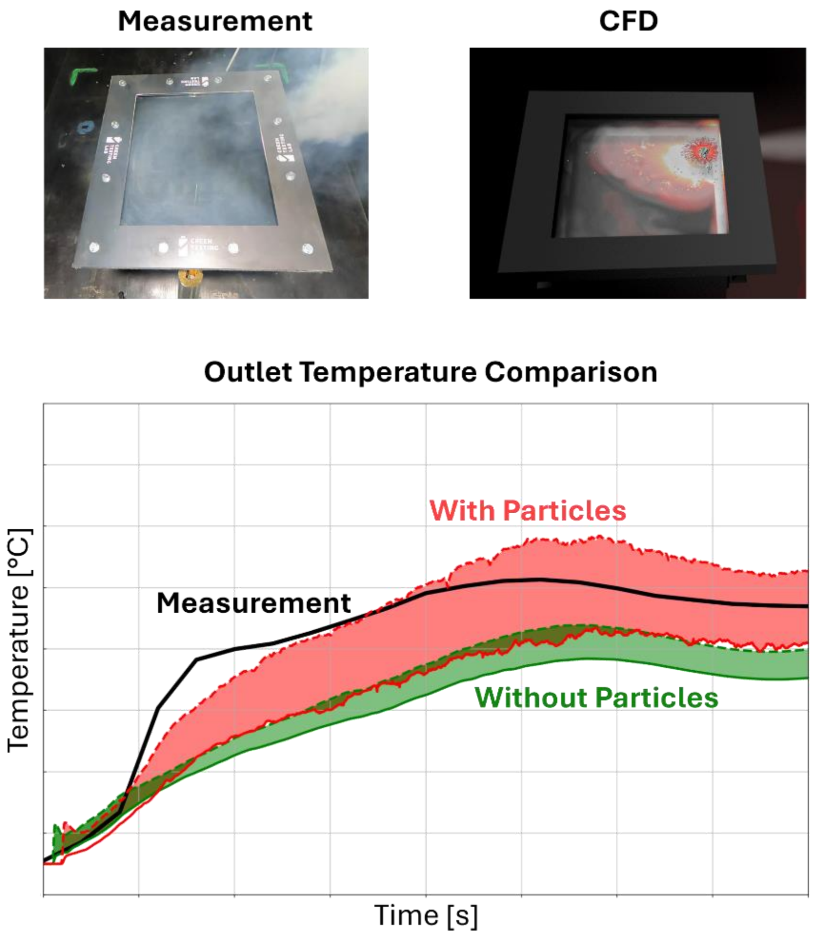

Figure 7: Experimental validation of two modelling approaches for temperature probes placed at the channel outlet.

5 Conclusion & Outlook

Due to the ongoing trends of stricter legislation for HVB pack design (no thermal propagation & no fire within increasing time frames), the requirements for CFD Simulations as a predictive and supportive tool are also increasing. The current work demonstrated the closure of one significant gap in the modelling chain of HVB venting modelling, namely the thermal coupling of local particle depositions as well as the subsequent local clogging of the channel. Only by taking into account the change of the velocity flow field due to the deposited phase, CFD becomes a viable tool for complex channel design with multiple possible gas paths, obstacles and particle traps.

The authors want to thank the partners from the FFG funded project *Prevent+* in which the results from present paper were elaborated: *Virtual Vehicle Research GmbH*, *Green Testing Lab*, *Kreisel Electric GmbH & Co KG*, *Technische Universität Graz* and *Magna Vehicle Engineering*.

In the upcoming follow-up project SafeSustain [2] deeper focus will be put on particle-wall interactions during venting as well as the implementation of ecofriendly materials for thermal protection.

6 Reference list

[1] PREVENT+, FFG funded project, “Understand PaRticlE VENTing and Arcing for Safer Batteries Plus”, 2022-2025.

[2] SafeSustain, FFG funded project, “Safety by Sustainability”, 2025-2028.