Numerical design and optimization of underhood cooling system based on coupled 1D and 3D flow modeling

Gowtham Chandrasekharan Parameswaran, Sagar Kulkarni, Dig Vijay

Gamma Technologies GmbH
Stuttgart, Germany

gc.parameswaran@gtisoft.com, sagar.kulkarni@gtisoft.com, d.vijay@gtisoft.com

Abstract: Underhood cooling systems are crucial for ensuring efficient thermal management leading to improved vehicle performance in terms of range and thermal derating of key powertrain components, while meeting packaging constraints. This work presents a synergetic 1D-3D solution by modeling external air flow with 3D computational fluid dynamics (CFD) and internal flow (coolant, refrigerant and gas) with 1D CFD in GT-SUITE. The presented methodology helps in achieving higher computational speed owing to 1D-2D spatial discretization of flow inside the heat exchangers and high accuracy due to detailed 3D CFD for external air flow. Moreover, the iterative exchange of boundary data between 1D and 3D domain ensures solution accuracy and stability while reducing computational efforts. Accordingly, a test case with inclined cooling module is presented, where the design optimization focuses on key parameters such as degree of inclination of different heat exchangers (HXs), HX dimensions, flow space geometry, fan characteristics and grill design. Optimization techniques like design of experiments and multi-objective Pareto optimization are deployed to balance cooling efficiency and energy consumption within given system constraints. In conclusion, this coupled 1D-3D modeling approach provides a unique numerical solution that is suitable for performing high number of design iterations desired for pre-design of vehicle cooling systems to make faster and accurate decisions.

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

1 Introduction

The thermal management of electrified vehicles (xEV) presents unique engineering challenges that significantly impact vehicle performance, efficiency, and reliability. Underhood pre-design, a critical phase in xEV development, requires systematic consideration of multiple interacting thermal systems within stringent spatial constraints [1]. This paper examines the fundamental challenges of underhood thermal architecture and their implications for overall xEV performance. The complexity of xEV thermal management stems from the need to simultaneously address several competing requirements. Space constraints necessitate efficient packaging of cooling systems and battery thermal management components while maintaining adequate airflow pathways. Thermal interaction presents another significant challenge, as various components operate at different optimal temperature ranges, batteries typically prefer 20-40°C, while power electronics can function at higher temperatures, requiring careful thermal isolation or strategic integration. Airflow optimization represents a critical design consideration, demanding properly engineered ducting and flow paths to ensure sufficient cooling reaches all components while minimizing aerodynamic drag penalties. Furthermore, the integration of multiple cooling circuits operating at different temperature ranges (battery cooling, motor/electronics cooling, cabin HVAC) requires sophisticated control strategies to maintain system harmony and efficiency. The significance of effective underhood pre-design extends beyond basic thermal control, directly influencing vehicle range through cooling system efficiency, battery longevity through temperature management, fast-charging capabilities, passenger comfort, and overall vehicle reliability [2]. This paper explores these challenges and their implications for next generation xEV design methodologies.

The 3D computational fluid dynamics (CFD) with porous media approach to represent heat exchangers [3] is one of the detailed modeling methodologies for underhood thermal management in battery electric vehicles. This approach replaces detailed heat exchanger geometries with homogenized porous blocks defined by flow resistance and heat transfer characteristics. This enables efficient full-vehicle simulations while capturing key effects such as recirculation, flow maldistribution, and thermal interactions. Implementation involves defining heat exchanger volumes, assigning pressure drop and heat transfer data, and integrating them into the flow domain, with advanced models adding anisotropic resistance and non-equilibrium effects. While less computationally demanding than fully resolved CFD for internal flow inside heat exchangers, the method still requires complex setup, geometry preparation, meshing, solver tuning, and parameter calibration. Design changes often necessitate remeshing and re-simulation, limiting rapid design exploration despite efficiency gains.

In contrast, GT-SUITE, a multiphysics CAE software from Gamma Technologies, offers several methodologies for modeling underhood thermal systems. One of the two approaches is known as Quasi-3D approach. This approach provides a 3D environment to model heat exchangers, fans, shrouds, flow inlets/outlets, and structural blockages. Unlike traditional full 3D CFD approaches, this methodology runs significantly faster by using a 2D discretization of control volumes representing heat exchangers, and fans, while utilizing quasi-3D airflow solution [4]. This makes it efficient in computational speed for analyzing multiple operating conditions and running design of experiments while producing results comparable to 3D CFD for overall system performance metrics. A key advantage of this approach is its ability to provide increased control over discretization (both tangential and axial) and specify spatially varying flow conditions at heat exchanger faces. The predictive capabilities eliminate the need for user-imposed flow distribution assumptions, as it can naturally predict flow redirection around structural elements. Thus, the quasi-3D approach can be naturally used to analyze how heat exchanger stacking affects the heat exchange and cooling system design. Despite its advantages, this approach provides low fidelity spatial resolution than full 3D CFD, making it less suitable for analyzing highly localized temperatures or complex flow patterns that involves flow turning and multiple recirculation regions.

GT-Auto-3DFlow, which is a detailed 3D CFD solution of GT-SUITE, combines the strengths of both the quasi-3D and traditional 3D CFD approaches to overcome their respective limitations while providing enhanced accuracy [5]. It integrates porous media heat exchanger modeling with 1D CFD modelling of heat exchanger, automating geometry handling, meshing, and solver settings. The workflow is designed for system focused engineering. It offers much faster turnaround, which is relatively faster than traditional porous media approach by using an optimized fan discretization scheme and automated parameter exchange between air flow domain and heat exchanger internal flow domain. The detailed 3D approach maintains the ability to capture key 3D flow phenomena such as recirculation, maldistribution, thermal interactions with high accuracy, but with far lower setup complexity and computational overhead than traditional porous media approach.

2 Material and method

This paper utilizes the detailed CFD framework, a high-fidelity computationally efficient 3D computational fluid dynamics (CFD) solution tailored for system focused analysis [6]. Developed through the integration of 3D CFD solver from Simerics Inc. with GT-SUITE, the framework enables accurate prediction of complex fluid flow and thermal phenomena within a system-level modeling context. The approach eliminates the need for extensive 3D CFD expertise, making it particularly well-suited for underhood thermal management, where multiple heat exchangers and intricate flow domains must be evaluated concurrently. In the following sections, a brief description of how different components of an underhood system are modeled is discussed.

2.1 Heat exchanger modeling

Heat exchangers are modeled using two fluid convection models and dedicated geometrical components. The heat exchanger is discretized, and correlations are utilized to create predictive models [4]. When measured performance data is available, it can be calibrated to preselected heat transfer and friction correlations. If measured data is not available, correlations can be chosen or custom correlations can be implemented with scaling multipliers to achieve desired performance. This correlation-based approach enables the heat exchanger model to predict performance both within and outside the entered data range under various conditions such as temperature, pressure, fluid properties, and geometry scaling.

2.2 Fan modeling

Fans are modeled using a non-dimensional theory that converts performance data into dimensionless coefficients for pressure (Ψ), flow (Φ), and power/torque [4]. This method enables predictive fan performance across all operating ranges (windmilling and positive pressure rise), including conditions outside the original test data. During simulation, instantaneous pressure rise governed by the system resistance and shaft speed/torque inputs are converted into non-dimensional coefficients to determine the flow rate and power consumption of the device. The performance of the fan at other temperatures is possible as density variations that significantly affect fan performance is accounted by correcting to reference conditions. The non-dimensional data also enables to scale the fan geometry to appropriately size the fan according to the system cooling performance needs.

2.3 Underhood Air Modeling

2.3.1 Numerical Approach

In this work, the airside is discretized using the finite volumes and the Darcy-Forchheimer formulation is utilized to model momentum source terms for porous media regions like heat exchangers. The equation is:

$$S = -\mu/\alpha \cdot v - \rho \cdot C \cdot |v| \cdot v$$

where:

- S is the momentum source term
- μ is fluid viscosity
- α is permeability
- ρ is density
- C is inertial resistance factor

- v is velocity

This formulation adds resistance terms to the standard momentum equations to represent pressure drop across heat exchangers without modeling their detailed internal geometry.

For the numerical solution, the finite volume scheme is deployed to discretize 3D Navier-Stocks equation with first and second order upwind and central differencing schemes for convective and diffusive terms. The solver applies relaxation factors (ranging from 0 to 0.8) to control solution stability during iterations for velocity, pressure, turbulent kinetic energy, turbulent energy dissipation rate, and energy variables. For turbulence modeling, users can select from standard K-Epsilon, RNG, or Realizable K-Epsilon models, with appropriate transport equations for each, in this paper K-Epsilon model is implemented.

2.3.2 Mesh Generation

Meshing is based on Cartesian (block-structured) base mesh and mesh generation is fully automated and controlled through resolution settings categorized as Coarse, Medium or Fine. Surface mesh export is also resolution-dependent, with configurable edge length parameters for accurate geometry representation in CFD domains.

2.3.3 Co-Simulation Strategy

A co-simulation methodology couples the 3D CFD air domain with the 1D CFD heat exchanger internal flow domain through iterative data exchanges as shown in Figure 1. The process operates as follows:

- The 1D CFD model in GT (referred for GT-SUITE) executes for a defined time interval.
- Boundary data is exchanged with the 3D CFD air domain.
- The 3D solver advances the solution for predefined number of iterations.
- The exchange repeats until user-defined convergence criteria are met or the maximum simulation time is reached.

This iterative scheme enables accurate prediction of key thermal-fluid performance metrics such as pressure drop, temperature distribution, heat transfer rates, and flow distribution, with a balance between accuracy and computational efficiency.

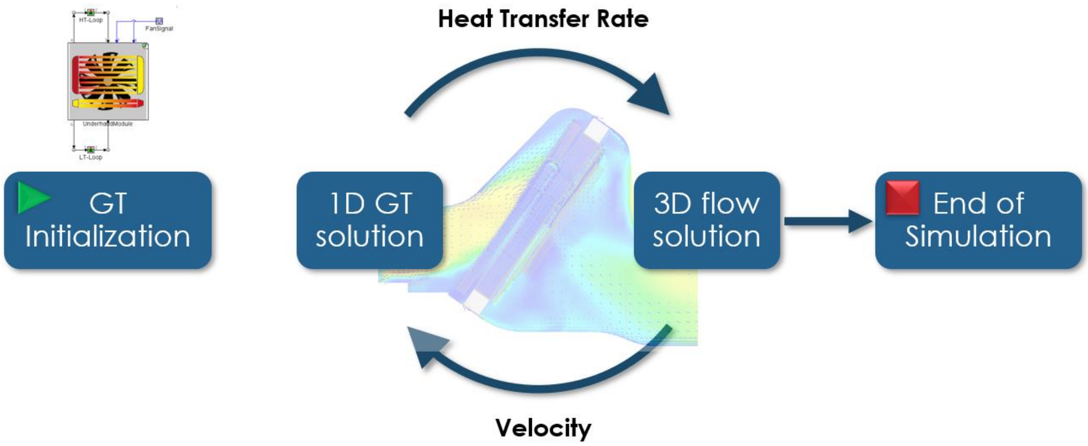

Figure 1: Co-simulation strategy between 1D flow domain (internal) and 3D flow domain (external).

3 Results

This paper employs the detailed 1D-3D approach to investigate the flow and thermal performance of an underhood system representative of a hybrid powertrain which is shown in Figure 2. The underhood system comprises of high and low temperature radiators, a condenser and dual electric fans that aid airflow management. The analysis focuses on the effects of cooling module orientation (0°, 15°, and 30°), grille flap angles (0°, 15°, 25°, 35°, 55°, and 75°), and variable vehicle velocity on overall air mass flow rate and coolant temperatures, which in turn influence the performance of powertrain components such as battery, e-motor, inverter and engine. Finally, it is also demonstrated how the scaling features can be leveraged to optimize the heat exchanger and fan sizes to obtain maximum air flow rate through the underhood to achieve lower coolant outlet temperatures.

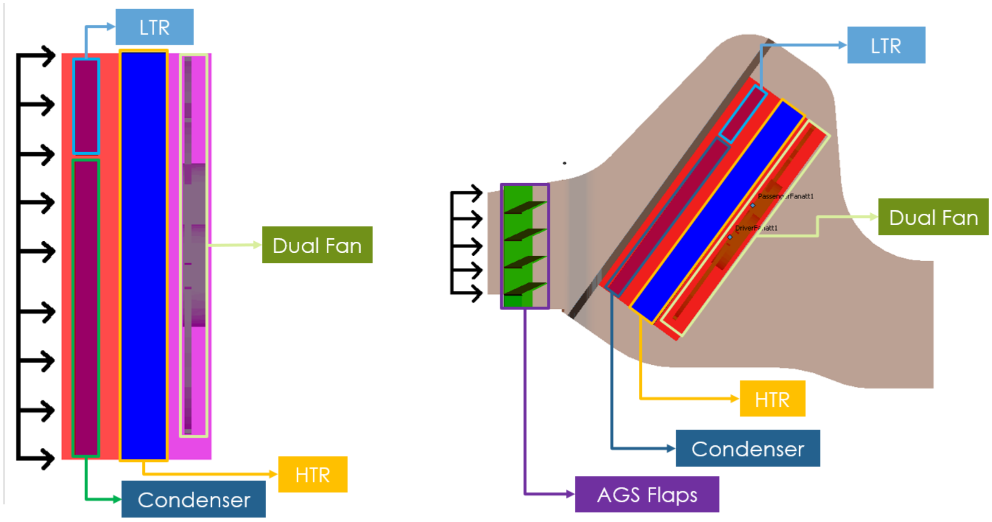

Figure 2: Typical underhood model for vehicle thermal management.

3.1 Impact of flap angle

Flap angle is one of the important parameters that controls the airflow rate and cooling efficiency as this determines how the heat exchangers interact with the incoming air. As a first study the flap angle varied from 5° (open) to 70° (close) for a fixed vehicle velocity to analyze the overall air mass flow rate through the underhood system, see Figure 3. While higher flap angles of 55° and 70° resulted in lower airflow and higher coolant temperatures, flap angle of 5° which represent almost a fully open configuration also resulted in high coolant outlet temperatures >100 °C, which may negatively impact the temperature sensitive components. From Table 1, it can be observed that flap angles of 15°, 25° and 35° facilitate higher airflow and lower coolant outlet temperatures (HTR Coolant Tout and LTR Coolant Tout), air outlet temperatures are listed under HTR Tout and LTR Tout.

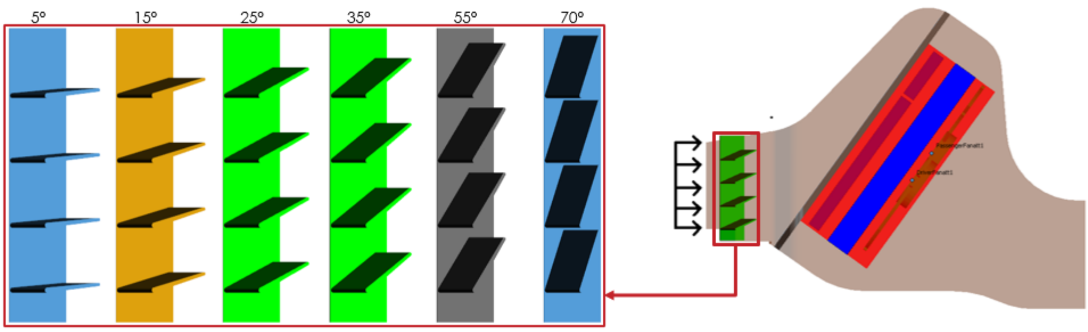

Figure 3: Different flap angles at the inlet of the underhood cooling module.

Table 1: Results of the various flap angles at the inlet

Flap Angle (°)	Vehicle Speed (km/h)	Flow In (kg/s)	HTR Tout (°C)	LTR Tout (°C)	HTR Coolant Tout (°C)	LTR Coolant Tout (°C)
5	50	1.463	76.368	83.49	91.409	108.262
15	50	1.445	75.599	81.328	90.627	105.961
25	50	1.390	77.479	80.38	92.414	104.971
35	50	1.308	80.891	79.265	95.684	103.715
55	50	0.979	98.041	86.707	112.417	111.406
70	50	0.643	131.839	104.856	145.083	127.923

3.2 Impact of vehicle velocity

The vehicle velocity affects the ram air pressure and creates higher pressure at the grille and controls the airflow and heat exchanger cooling performance. Subsequently, for the three best grille angles that resulted in higher airflow and lower coolant outlet temperatures, the effect of low and high vehicle velocity was studied as shown in Table 2, where it can be observed that 15° and 25° flag angle configurations results in overall higher airflow rate and relatively lower coolant outlet temperatures.

Table 2: Results of different vehicle speeds and flap angle

Flap Angle (°)	Vehicle Speed (km/h)	Flow In (kg/s)	HTR Tout (°C)	LTR Tout (°C)	HTR Coolant Tout (°C)	LTR Coolant Tout (°C)
15	0	1.123	85.359	84.907	100.103	109.498
	130	2.593	57.04	66.635	72.754	91.143
25	0	1.097	86.982	87.591	101.539	112.457
	130	2.542	57.403	64.777	73.068	89.133
35	0	1.045	89.876	82.202	104.267	106.548
	130	2.397	58.957	62.159	74.579	86.392

3.3 Impact of cooling module orientation

Though the flag angle and vehicle velocity affect the airflow, the cooling module (radiators, condensers, charged-air-cooler, fans) orientation also dictates the amount of surface area that is available for heat exchange and affects the pressure and flow distribution. During predesign analysis, cooling module orientation in an important aspect that can be examined to determine the optimal layout and packaging constraints and choose appropriate heat exchanger sizes, see Figure 4. In this paper, it is demonstrated how the detailed 1D-3D approach can be leveraged to analyze the effect of different pack angles on the cooling performance.

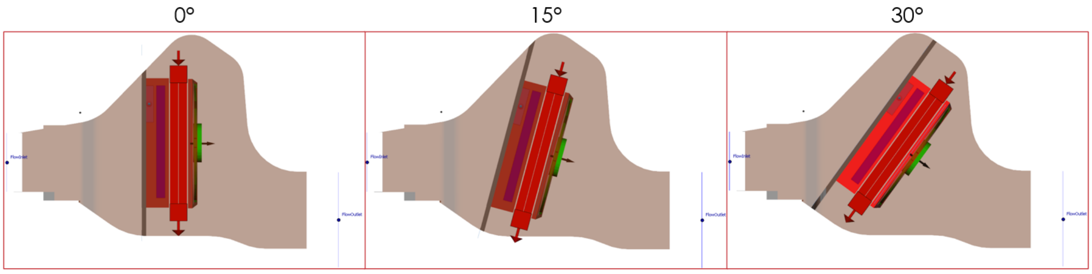

Figure 4: Orientation of cooling modules at different angles.

Table 3 shows the results of airflow rate and coolant outlet temperatures for three different cooling module orientation (0° (planar), 15° and 30°) and different flap angles and vehicle velocities. From the results presented below, the best performance is obtained from 30° cooling module orientation due to enhanced effective surface area that is available for heat exchange.

Table 3: Results of different orientation of the cooling module

Module Orientation (°)	Flap Angle (°)	Vehicle Speed (km/h)	Flow In (kg/s)	HTR Tout (°C)	LTR Tout (°C)	HTR Coolant Tout (°C)	LTR Coolant Tout (°C)
0	15	0	1.136	88.093	87.003	102.995	112.322
		130	2.239	55.966	58.12	71.97	82.56
	25	0	1.115	89.994	84.95	104.703	110.194
		130	2.182	56.735	56.523	72.531	80.807
	35	0	1.07	93.609	83.918	108.143	109.045
		130	2.084	58.09	54.996	73.689	79.017
15	15	0	0.94	100.95	141.911	117.964	167.349
		130	2.498	55.902	59.037	72.068	83.285
	25	0	0.9017	104.145	132.409	121.352	157.53
		130	2.402	57.18	58.161	73.308	82.29
	35	0	0.85	109.413	132.326	127.154	157.499
		130	2.257	59.682	55.907	75.535	79.87
30	15	0	1.123	85.359	84.907	100.103	109.498
		130	2.593	57.04	66.635	72.754	91.143
	25	0	1.097	86.982	87.591	101.539	112.457
		130	2.542	57.403	64.777	73.068	89.133
	35	0	1.045	89.876	82.202	104.267	106.548
		130	2.397	58.957	62.159	74.579	86.392

3.4 Sizing of heat exchanger for performance optimization

The coolant outlet temperatures for vehicle velocity of 0 km/h, representative of idling or charging case are still above the desired value of 100 °C for flap angle of 15°, heat exchanger design is scaled by 25% to achieve the desired performance. The baseline and scaled heat exchangers are shown in Figure 5.

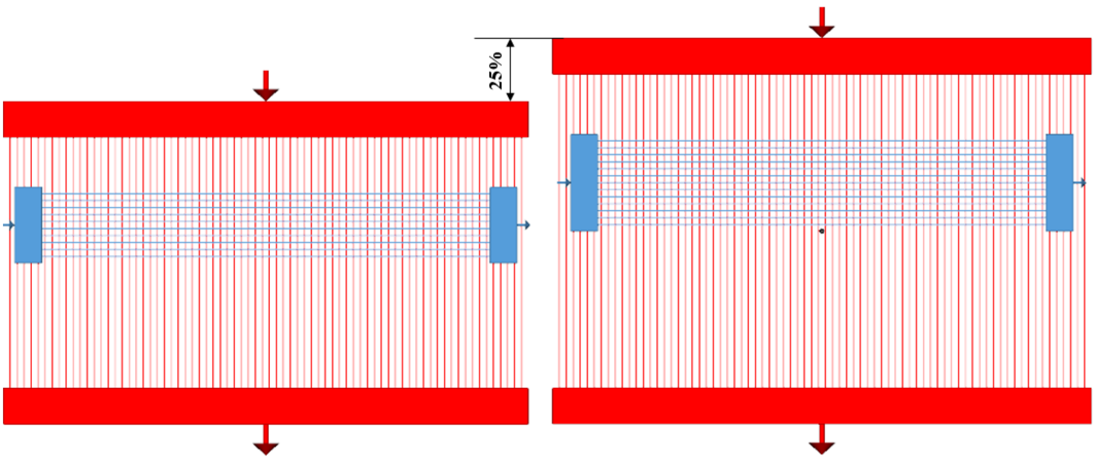

Figure 5: Comparison of baseline size of the heat exchangers with the scaled size.

Increasing the size of the heat exchanger results in lower coolant outlet temperatures as seen in **Error! Not a valid bookmark self-reference.** due to increased heat transfer area for coolant to air heat transfer, however, for some configurations the temperatures are still above 100 °C.

Table 4: Results of different heat exchanger

Size	Module Orientation (°)	Flap Angle (°)	Flow In (kg/s)	HTR Tout (°C)	LTR Tout (°C)	HTR Coolant Tout (°C)	LTR Coolant Tout (°C)
Base	30	15	1.123	85.359	84.907	100.103	109.498
		25	1.097	86.982	87.591	101.539	112.457
		35	1.045	89.876	82.202	104.267	106.548
Scaled	30	15	1.202	81.668	78.322	94.645	99.118
		25	1.162	83.877	77.937	96.930	98.722
		35	1.097	87.437	79.653	100.601	100.602

3.5 Selection of fan for performance optimization

The coolant outlet temperatures are still not in desired range, therefore, as a next step, the fan size is also scaled by 12 % to facilitate higher airflow rates, see Figure 6. The non-dimensional modeling of fans allows the performance to be extrapolated and can be predicted for scaled fan using the baseline performance data. As can be seen in Table 5, the scaled fan result in higher airflow rates and subsequently lower coolant outlet temperatures for both LTR and HTR that are below the desired limit of 100 °C. Velocity and temperature contour plots for this final design are shown in Figure 7 and Figure 8, which shows higher core airflow rates and regions where maximum air temperature is encountered. The detailed 3D methodology also helps in visualizing the streamlines (Figure 9) of the airflow which highlights the recirculation regions. Such information is crucial in airflow management and the cooling performance of the underhood.

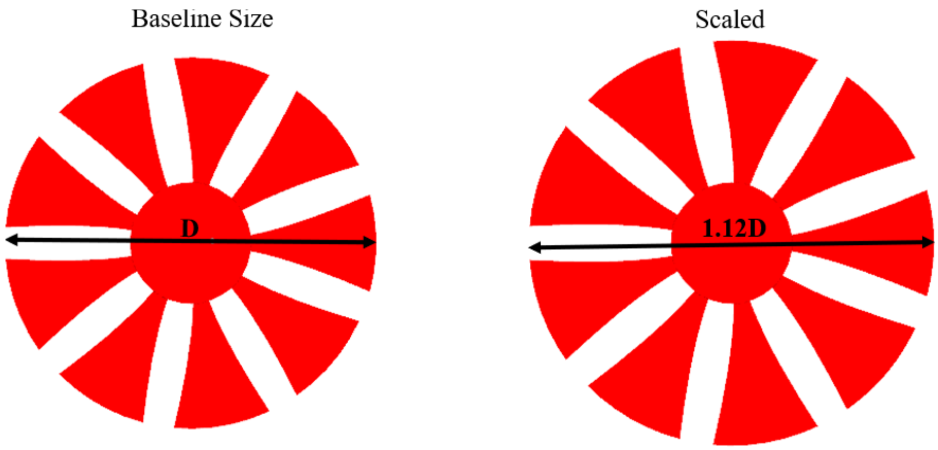

Figure 6: Comparison of baseline size of the fan with the scaled size

Table 5: Results of different fans

Size	Module Orientation (°)	Flap Angle (°)	Flow In (kg/s)	HTR Tout (°C)	LTR Tout (°C)	HTR Coolant Tout (°C)	LTR Coolant Tout (°C)
Base	30	15	1.202	81.668	78.322	94.645	99.118
		25	1.162	83.878	77.936	96.930	98.722
		35	1.097	87.437	79.653	100.604	100.602
Scaled	30	15	1.26596	79.004	78.134	91.256	98.871
		25	1.22083	81.012	74.658	93.272	95.457
		35	1.14412	84.418	74.572	96.620	95.257

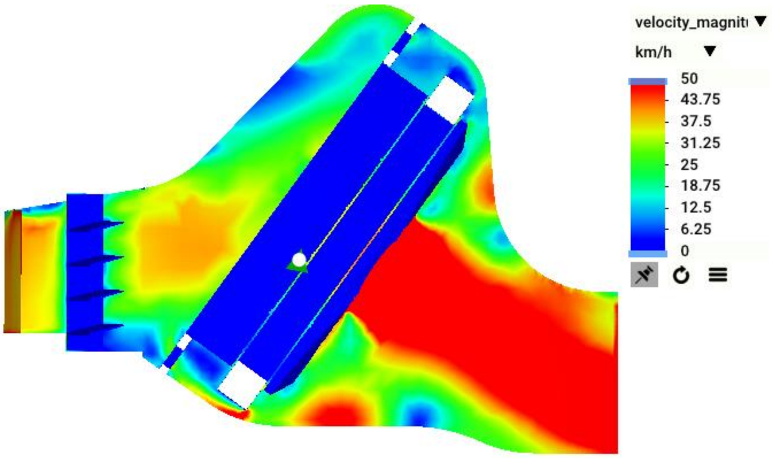

Figure 7: Velocity contour along the cross-section of the underhood flow space.

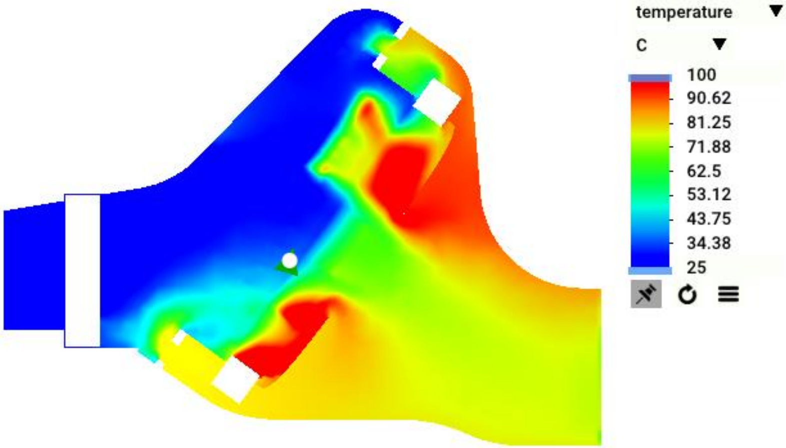

Figure 8: Temperature contour along the cross-section of the underhood flow space.

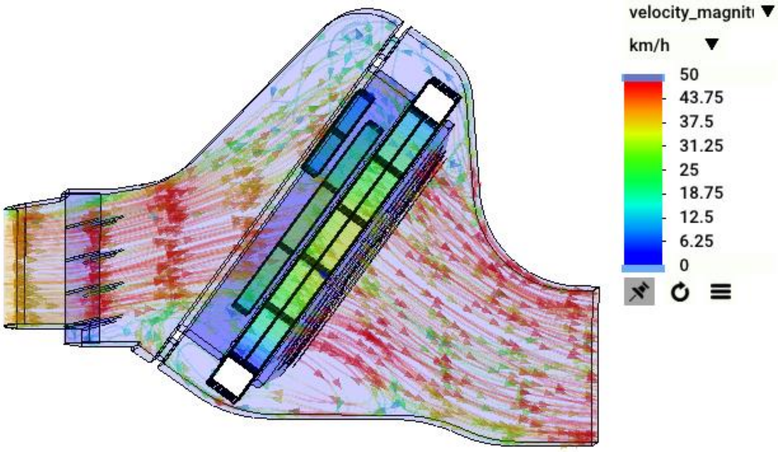

Figure 9: Streamlines along the cross-section of the underhood flow space.

4 Conclusion

This study introduced a coupled 1D–3D CFD methodology for underhood cooling system pre-design, combining detailed airflow modeling with predictive 1D heat exchanger representations. The case study showed that grille flap angle, cooling module orientation, and vehicle speed strongly affect airflow and coolant temperatures, with a 30° module orientation and moderate flap angles (15–25°) giving the best performance. Scaling of heat exchangers and fans further reduced coolant outlet temperatures below 100 °C, ensuring reliable operation even under idle conditions. The approach also provided valuable insight into airflow recirculation and thermal interactions. The results reported in this paper are the outcome of 3D numerical simulations performed on a standard PC with 8 CPU cores. The total number of cells per simulation is around 1.5 million elements, the computation time needed for each simulation was between 15 to 20 minutes. Overall, the 1D–3D co-simulation framework enables rapid design iterations, accurate performance prediction, and efficient optimization, supporting faster and more reliable vehicle thermal management development.

5 Bibliography

[1] A. Cappuccio, „BEV Thermal Management 1D Modeling for high performance system design," Politecnico di Torino, Torino, 2022.

[2] P. Dinakar, „Modelling and Simulation of Cooling Systems for BEV," Chalmers University of Technology, Göteborg, 2016.

[3] J. Nordin, „CFD Study of Optimal Under-hood Flow for Thermal Management of Electric Vehicles," Chalmers University of Technology, Göteborg, 2017.

[4] Gamma Technologies, „GT-SUITE Flow Theory Manual and Template Help," Gamma Technologies LLC, 2025.

[5] Shawn Harnish, Gamma Technologies LLC, „GT-Auto-3DFlow: New Underhood Fluid-Thermal Solution – Fast, Easy and Accurate," Gamma Technologies LLC, 2023.

[6] J. Varghese, Y. Wu, A.J. Juanos, R. Saha, S. Dhar, Simerics Inc., „Numerical Simulation of Class 8 Tractor Trailer Geometries and Comparison with Wind Tunnel Data," *SAE International,* Nr. 2024-01-2533, 2024.

[7] C. Boettcher, N. Framke, C. Altenhofen, T. Vevaud, Gamma Technologies GmbH, „Simulation based Development Methodology for the Thermal Security of Fuel Cell Powertrains," in *SIA Powertrain Conference*, Port-Marly, France, 2023.

[9] Sujan Dhar, Simerics Inc., „Numerical Analysis Using Fast RANS Simulations and Comparison with Experimental Measurements for Closed and Open Grille Realistic Car Models," *SAE International,* Nr. 2019-01-0655, 2019.