USE OF CFD AS A REACTIVE TOOL TO ADDRESS ENGINE COMPARTMENT OVERHEATING PROBLEMS IN A MILITARY COMBAT/ TACTICAL VEHICLE

K. Vijaya Sankaran **, G. Sriram**#, J. Catherin Rose*, T.K. Seshachari*, A.P Haran* and J.J Isaac*

Abstract:

The vital engine power package of a military combat / tactical vehicle is designed to operate in a compact, nearly fully enclosed engine compartment that is ventilated only through ballistic grilles so as to protect it from projectiles, bullet splash and fragments. On account of the tight packing of key components in the limited compartment space, the engine cooling airflow path is highly constrained leading to challenges in heat evacuation and consequent overheating under certain operating conditions.

A powerful State-of-the-art analysis, based on 3D-CFD, has been developed as a reactive tool, to address the heat and fluid flow through the engine compartment of an in-production military combat/tactical vehicle in which severe overheating problems have arisen during actual operation, Virtual prototyping, as against the conventional physical prototyping, has been employed together with a comprehensive sensitivity analysis of key component performances to identify triggering sources for the overheating and thereby evolve remedial measures to be incorporated.

Introduction:

Military combat/tactical vehicles are required to operate in hostile enemy terrain even under adverse ambient conditions. Ventilation of the vital engine power package is provided only through ballistic grilles to protect it from projectiles, bullet splash and fragments. The power package is contained in a compact, nearly fully enclosed engine compartment tightly packed with key components and consequently, the cooling air flow path is constrained leading to difficulties in safe heat evacuation. Moreover, the natural advantage enjoyed by commercial vehicles plying on standard roads is absent as the ram air is not available as all the cooling air has to be sucked in and ejected out to the ambient with the aid of a suction fan.

The traditional method of assessing the thermal management of such a complex cooling system has been to build a physical prototype with all the key components, packaged in the available space and actual test it so as to incorporate design changes or even relocation of components in the restricted engine compartment space, if necessary. With the availability of powerful, versatile computational techniques, it is now possible to carry out virtual prototyping to help make the necessary design changes. Moreover, these computational techniques could also be used as a reactive tool to identify reasons for cooling shortfalls which show up during actual trials of an in-production combat/tactical vehicle.

The Army Design Bureau, through its website featuring a ' Compendium of Problem Statements' had appealed for the development of indigenous solutions for the overheating problems experienced by military combat/tactical vehicles. In response, a powerful, State-of-the-art analysis, based on 3D CFD, has now been developed as a reactive tool, to address the heat and fluid flow through the engine compartment of a typical in-production military combat /tactical vehicle in which serious operating problems have been experienced during actual trails.

Fig.1 shows a schematic of the engine compartment. The source of all heat in the compartment is the diesel engine itself. Ambient air is drawn in, at right angles to the vehicle motion direction, through an inlet ballistic grille by suction created in a centrifugal fan. This air then flows past the radiator carrying away the reject heat both from the engine cooling water and the oil cooler. This heat laden air, as well as the cooling air coming from the auxiliary compressor, is sucked out by the fan and ejected to the ambient through the exit ballistic grille.

In general, there are four triggering conditions that could cause excess temperature distress in the engine compartment cooling system; (a) extreme ambient temperatures which could rise up to even 50C, (b) excess engine

Keywords: Underhood simulation, Conjugate Heat Transfer, Combat vehicle cooling, Reactive tool, CFD

* PARK College of Engineering and Technology, Coimbatore, Tamil Nadu.
** VESRAD Technologies Pvt. Ltd, Kalpakkam, Tamil Nadu.
# Corresponding author email: < gsriram@vesrad.in >.

reject heat to be transferred from the radiator to the carrier cooling airstream, (c) degradation of the swallowing capacity of the suction fan and (d) cooling air flow path infirmities and even maldistribution, in that adequate cooling air may not be washing critical areas which need attention.

Solution approach:

It is to be stressed that a successful cooling system design is not determined by the selection of individual components. Rather, it is the result of the vehicle operational requirements, peculiar system installation restrictions and the integration of the cooling system into the complete vehicle. Moreover, mere changing of one component or a group of components of the system will not lead to a successful cooling solution if the initial heat transfer difficulty is elsewhere. Adoption of a holistic approach is essential for examination of the internal cooling airflow path and its heat evacuation capabilities.

The cooling air, on passing through the intake ballistic grille, the radiator, exhaust ballistic grille and while negotiating the complex airflow path would undergo pressure losses and for steady state conditions, the fan would necessarily have to provide sufficient head at this air flow rate while maintaining a reasonable fan static pressure efficiency ($\eta_{fan} \ngtr 70\%$) (Ref 1). Fig.2 shows the importance of matching the system pressure loss creating resistances and the fan characteristics. If the system resistance is too high, the fan volumetric flow will be reduced and this would lead to a cascading effect as it would lead to a reduction of heat evacuation from the radiator. The maximum allowable temperature in either the pressurised water heat transfer loop or the lubricating oil loop should not exceed 110C (Ref 2). Hence, if the cooling air flow rate reduces, it could no longer evacuate all the reject heat from the radiator and there will be a consequent unacceptable increase in the radiator coolant temperature. In addition, the decrease in volumetric flow could also result in an increase in the air temperature. To prevent violation of the Laws of Thermodynamics and to maintain an adequate logarithmic temperature difference, the maximum air temperature is generally restricted to 105C. The air volumetric flow rate increase would also affect the swallowing capacity of the fan. This domino effect can only be overcome by considering increasing the capacity of the fan, which may not be available in an in-production vehicle.

The diesel engine is the primary source of reject heat into the engine compartment. In general, $1/3^{rd}$ of the energy produced by the fuel is converted to useful mechanical shaft power. Roughly around $1/3^{rd}$ is transferred to the atmosphere in the form of thermal energy contained in the high temperature exhaust gases. The remainder reject heat must be removed from the system by forced convection away from the radiator into the atmosphere. Therefore, in general the ratio (α_{RP}) of the radiator reject heat to the mechanical shaft power is $\alpha_{RP} \sim 1$. However, under off design conditions, $\alpha_{RP} > 1$ and this could trigger overheating stress (Fig.3).

Computational analysis:

The general purpose CFD code ANSYS FLUENT 14 had been employed on an Intel Xeon workstation to solve the mass, momentum and energy conservation equations for studying engine compartment cooling air flow in a typical military combat/tactical vehicle. A 3D CFD model had been built in order to accomplish the target involved. All the relevant aspects of the air flow had been taken into account and fully incompressible, viscous simulations with turbulence modelling for steady state conditions had been performed. The $k - \omega$ SST turbulence model had been employed to model the turbulence. The body fitted procedure for grid generation ensured that the critical areas were aligned to the grid.

The resulting CFD model (Figs 4,5) had a cell count of 2.6 million tetrahedral cells and around 0.57 million nodes. The 3D model included the important constituents of the vehicle engine compartment (diesel engine, centrifugal fan, intake and exhaust ballistic grilles, radiator, auxiliary compressor cooling inlet). Mesh refinement had been used around relevant components such as the fan, grilles and the radiator. The external domain had been constructed in a way that realistic and stable boundary conditions could be applied to the region internal and external to the vehicle engine compartment.

The modelling of the key components such as the intake and exhaust ballistic grilles, radiator, fan, engine and auxiliary compressor cooling inlet had employed the actuator disk concept which is a simulation device that adds/subtracts momentum and / or energy uniformly over the entire disk area. Since exact performance details were not available, the conventionally used actuator disk characteristics were used and these are shown in Table.1. A pressure-based coupled solver was used for the simulation. The solution was run till the residuals dropped and the air mass flow rate was stabilized.

Three cases of radiator heat release were considered α_{RP} = 1, 1.64, and 1.91, the first corresponding to a standard heat release and the second and third to an off-design cases where, due to a fall in the overall engine efficiency, the excess heat is released in the radiator itself (Ref.3). It was also assumed that there was no direct heat release from the engine itself as it was surrounded by a water-cooled jacket.

The motion of the cooling air through the fan was simulated using the actuator disk concept. A body force was added to the cells through a source term in the momentum equation. The analysis was executed for a chosen pressure rise (Table.1) until the airflow rate through the fan had stabilized. Typical system static pressure drop range 2.1 - 4.2 kPa was considered for the computation Table 1, (Ref.4). for the passage of the induced cooling air from the exterior of the intake ballistic grille to the exterior of the exhaust ballistic grille. The fluid power corresponding to this pressure rise and airflow rate was computed and the fan efficiency was checked against a given fan power/output power ratio, α_{FP} = 0.049 and 0.078. The former being standard and the latter an augmented fan power. Only if the fan efficiency < 70% was the result accepted (Ref.1). In addition, for a fan computation to be acceptable, the weighted inlet velocity vector angle with the fan axis should have been close to zero (Figs 6 and 7). The low cooling air velocities in the vicinity of the radiator is a matter of concern as it would lead to low heat evacuation from the radiator.

The two radiators (coolant pressure water, lubricating oil) and the intake ballistic grille were considered to be equivalent to a single actuator disk.

Numerical Results:

The heat and fluid flow characteristics related to the cooling air flow path along with its associated components have been computed to evaluate the effects of vehicle operation at ambient temperature (normal (27C) and extreme (50C)) as well as the effects of excess engine radiator reject heat being attempted to transfer to the carrier cooling air stream. In the absence of actual component performance, standard performance characteristics have been assumed and incorporated into the cooling airflow path analysis through the concept of actuator disks (Table 1).

In all cases, it is seen that if the stringent criterion that the radiator coolant temperature should not exceed 110C (383K), is to be met, then higher cooling air flow rates need to be induced by the fan suction. This will entail a corresponding increase in the static pressure losses as the air flows through the intake ballistic grille, radiator, exhaust ballistic grille and constricted engine compartment space.

Figs 8-10 show the streamlines, static temperature contours for the operation at elevated temperature (50C) and reject heat (α_{RP} = 1, 1.64, 1.91) dissipation through the radiator. It is seen that the flow between the engine and the fan is fairly disorganized leading to additional static pressure loss. In order to meet the radiator coolant temperature restriction of 110C, the air flow rate induced by the suction fan has to be increased (Figs 11,12), keeping the overall static pressure within limits. The corresponding fan fluid power is very close to the available fan power (α_{FP} = 0.076) leading to unrealistic fan static pressure efficiencies ($\eta_{fan} \not> 70\%$) If the ambient temperature is close to 50C, whatever be the engine reject heat (normal α_{RP} = 1 or even excessive α_{RP} = 1.91), the cooling system was found to be inadequate as the required fan fluid power exceeded the available fan power sometimes to unrealistic values. In all these cases the radiator coolant temperature criteria were preserved and the second law of thermodynamics was satisfied.

If, however the coolant temperature restriction of 110C, was relaxed, then the airflow rates that need to be induced by the suction fan and the corresponding static pressure losses were relieved and realistic fan static pressure efficiencies were achievable. But now the engine compartment temperatures would have exceeded 403K (130C) as shown in Figs 8-10, which is unacceptable.

Discussions and Concluding Remarks:

A 3D CFD model has been successfully employed to address the cooling system effectiveness in the engine compartment of a typical military combat/tactical vehicle. The complex cooling air path included the intake ballistic grille, radiator (pressurized water, lubricating oil), engine, auxiliary compressor cooling airstream, suction fan, exhaust ballistic grille and the poorly organized airflow distribution in the space between the engine and the fan. Vehicle operation under normal and extreme conditions related to ambient temperature and excess radiator reject heat from the engine have been considered.

In the absence of actual component performance data, standard performance characteristics had been assumed for all the above-mentioned components. In all cases, if the stringent radiator coolant temperature limitation of 110C (383K), was imposed, the cooling system had been found to be inadequate. If this temperature restriction is not adhered to, then overheating will result, for the present cooling system arrangement would be on overload.

Specifically, four triggering conditions had been identified that could cause excess temperature distress to the engine compartment cooling system; (a) extreme ambient temperature, (b) excess engine reject heat required to be transferred from the radiator to the carrier cooling air stream, (c) degeneration of the swallowing capacity of the suction fan and (d) cooling air flow path infirmities and maldistribution. Remedial measures have also been revealed through the CFD analysis.

However, these observations needed to be experimentally validated. The performance characteristics of each of the above components need to be experimentally determined in separate rig tests. If these performance characteristics are now fed into the versatile 3D CFD model specially developed here, the final observations of the adequacy of the cooling system could be considered validated. Limited actual field trials incorporating these recommendations need to be then carried out to complete the solution of the cooling problem.

Acknowledgements:

The valuable discussions with Senior Officers of the Heavy Vehicles Factory, Avadi are gratefully acknowledged.

References:

1. Baumeister T. (Ed), Marks Standard Handbook for Mechanical Engineers, McGraw Hill,1975
2. Engineering Design Handbook: Military Vehicle Power Plant Cooling, US Army Material Command, AMC Pamphlet AMCP 706-361, June 1975.
3. Pulkrabek, W.W., Engineering Fundamentals of the Internal Combustion Engine, Prentice Hall, 1997.
4. Chiou, J.P., Engine Cooling System of Military Combat/Tactical Vehicles, ASME Paper 750030.

TABLE 1

NAME	MODEL NAME	CONDITION
Auxiliary compressor cooling inlet	Actuator Disk 1	Inlet (Ambient pressure and Temperature)
Radiator inlet grille and Radiator	Actuator Disk 2	Pressure drop = k * 0.5*ρ*V^2 k – loss coefficient = 7.0 – 0.2V Heat transfer coefficient h = 1469.1+126.11V+1.73V^2 (Source – ANSYS Fluent guide)
Exhaust grille	Actuator Disk 3	Pressure drop = 2'' of water (496 Pa) (Source – Military Vehicle Power Plant Cooling Handbook) (Ref.2)
Centrifugal Fan	Actuator Disk 4	Pressure rise ΔP = 4.23 kPa (variable)

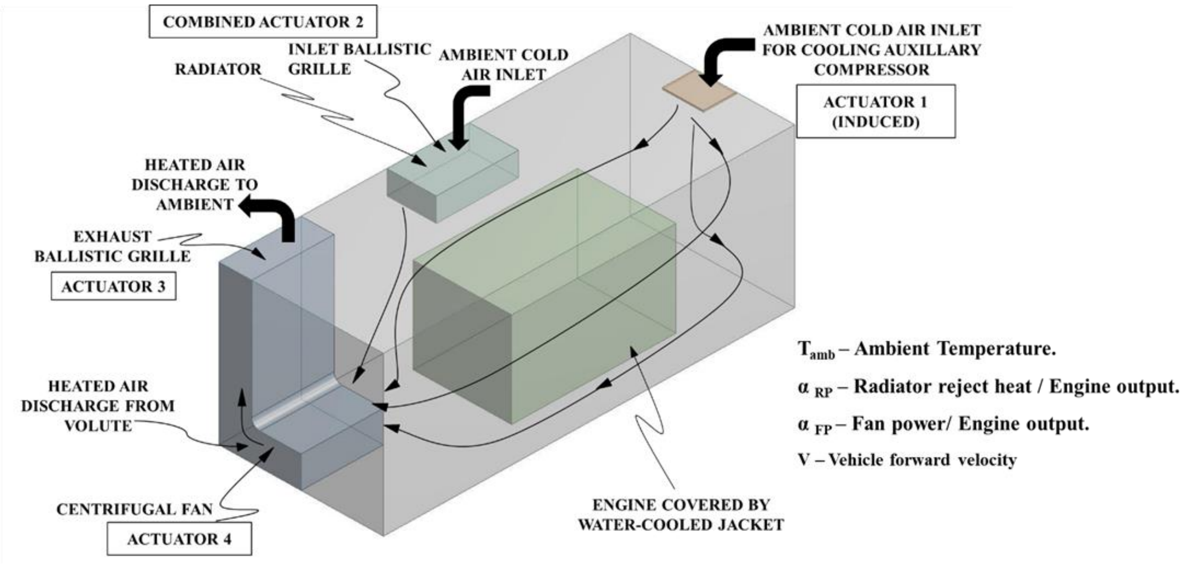

Fig.1 Schematic of the engine compartment – heat and fluid flow paths; location of actuators.

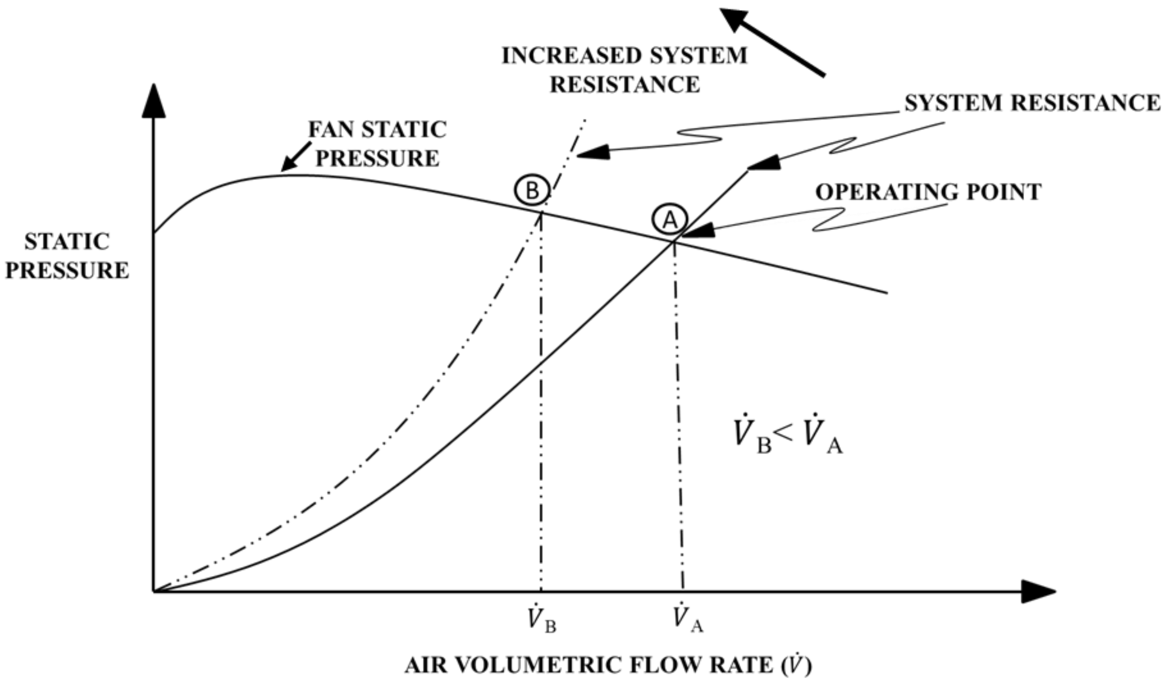

Fig.2 Fan-system resistance matching.

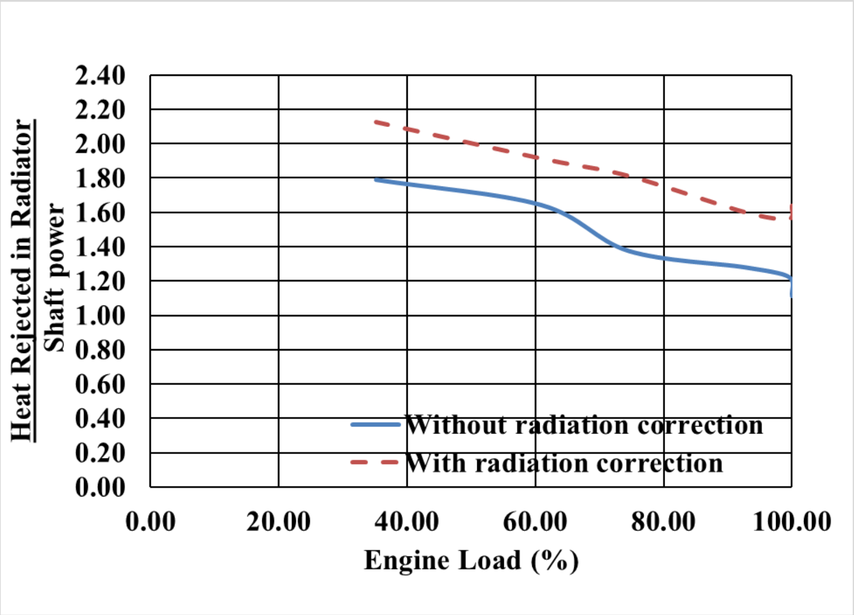

Fig.3 Variation of ratio of radiator reject heat to shaft power vs engine load.

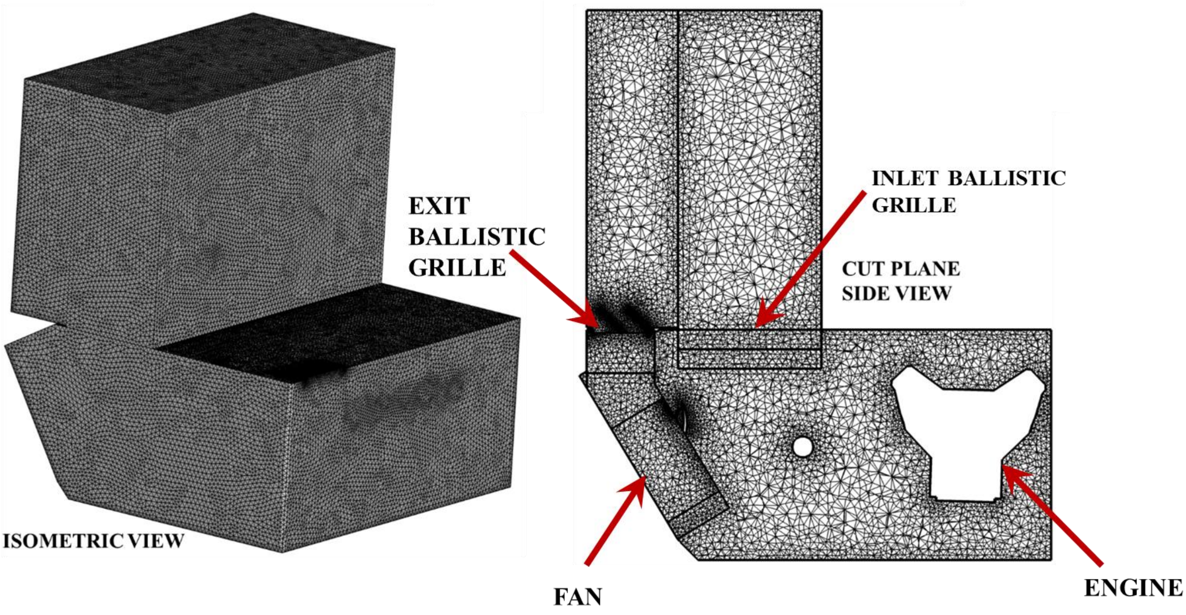

Fig.4 Mesh of the computational domain.

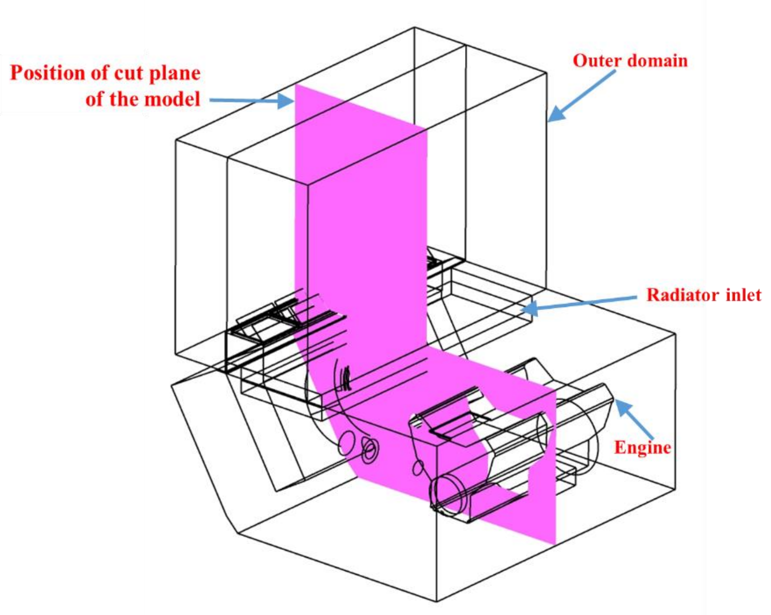

Fig.5 Vertical cut-plane on which are to be shown contours of velocity, static temperature and streamlines.

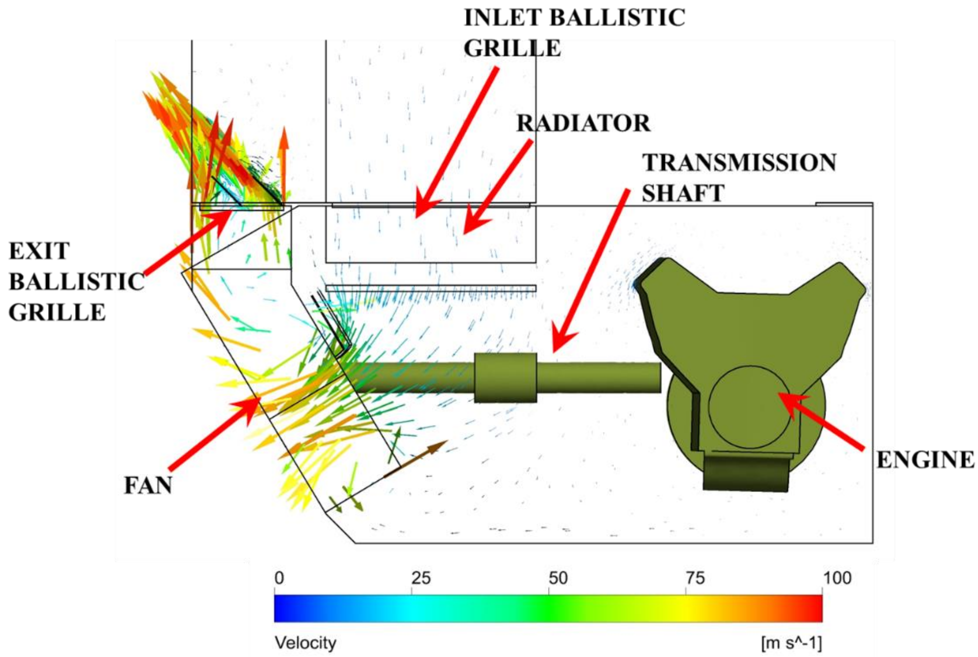

Fig.6 Vectors at the cut plane section, α_{RP} = 1.91(60 percent load); α_{FP} = 0.351 (fan power adjusted);
Ambient temperature $T_{AMBIENT}$ = 323K (27C); V = 16.67 m/s (60 kmph).

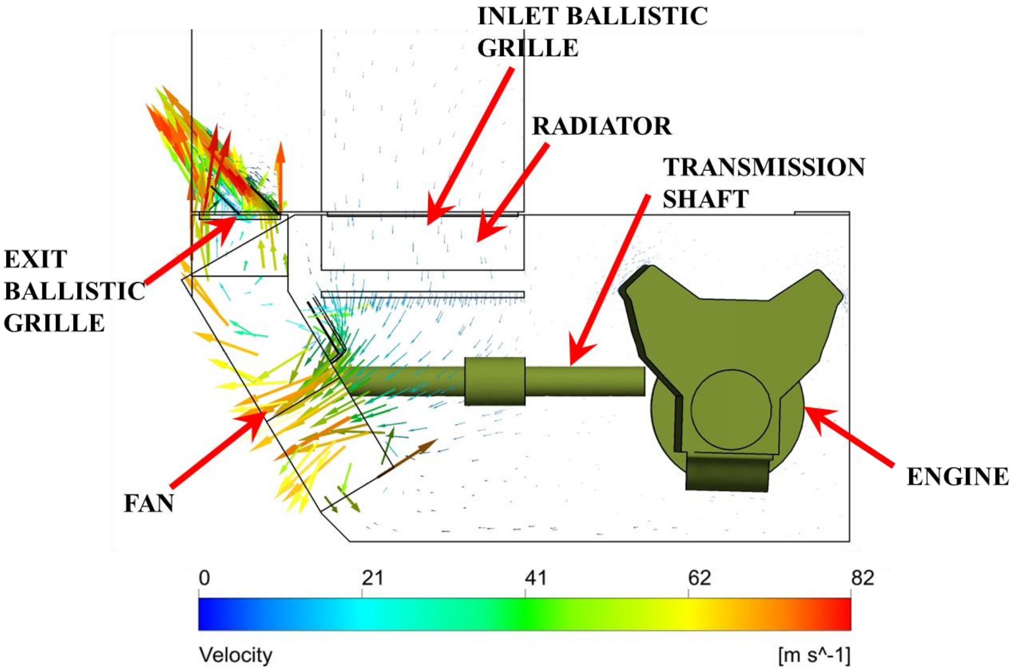

Fig.7 Vectors at the cut plane section, α_{RP} = 1.64 (100 percent load); α_{FP} = 0.078 (fan power adjusted); Ambient temperature $T_{AMBIENT}$ = 300K (27C); V = 16.67 m/s (60 kmph).

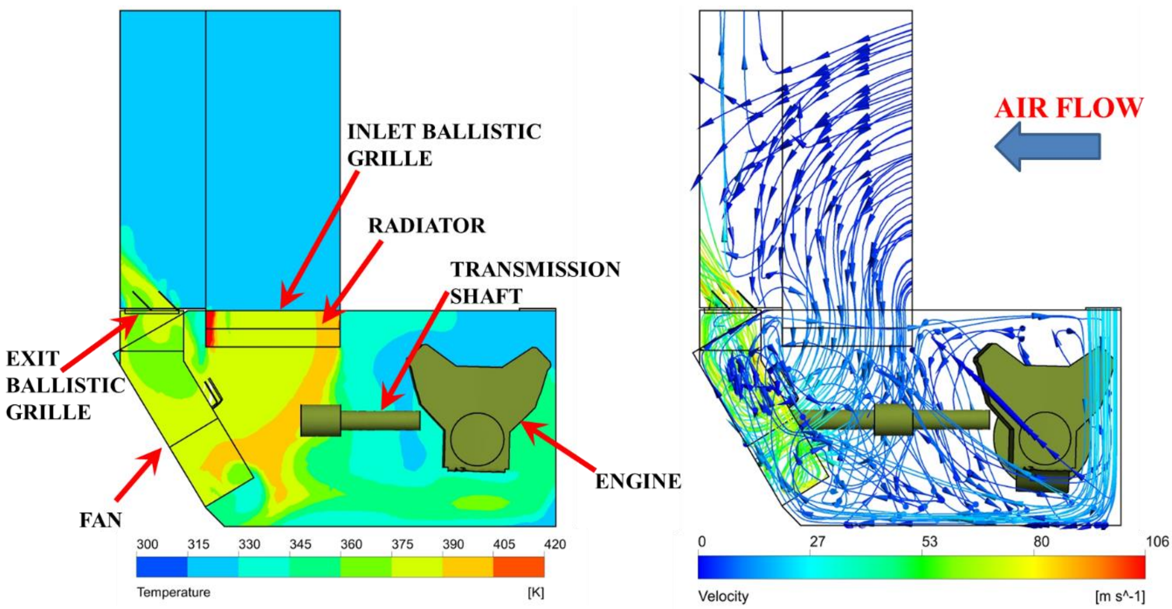

Fig.8 Temperature contours and streamlines showing the internal flow pattern, α_{RP} = 1(100 percent load); α_{FP} = 0.049; Ambient temperature $T_{AMBIENT}$ = 323K (50C); V = 16.67 m/s (60 kmph).

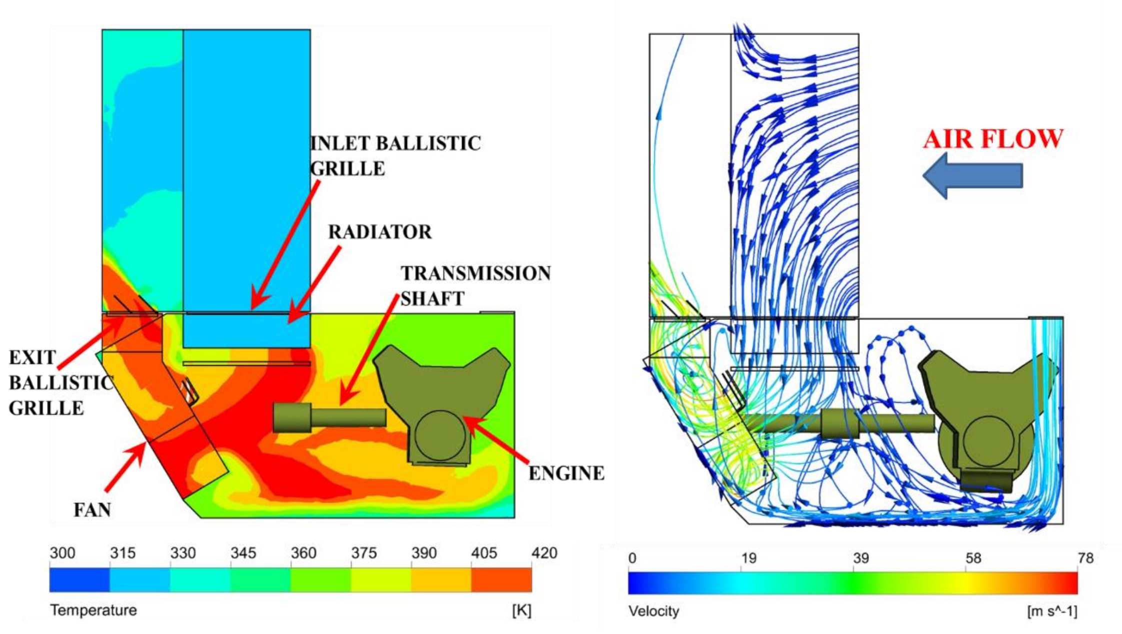

Fig.9 Temperature contours and streamlines showing the internal flow pattern, α_{RP} = 1.91 (60 percent load); α_{FP} = 0.165 (fan power not adjusted); Ambient temperature $T_{AMBIENT}$ = 323K (50C); V = 16.67 m/s (60 kmph).

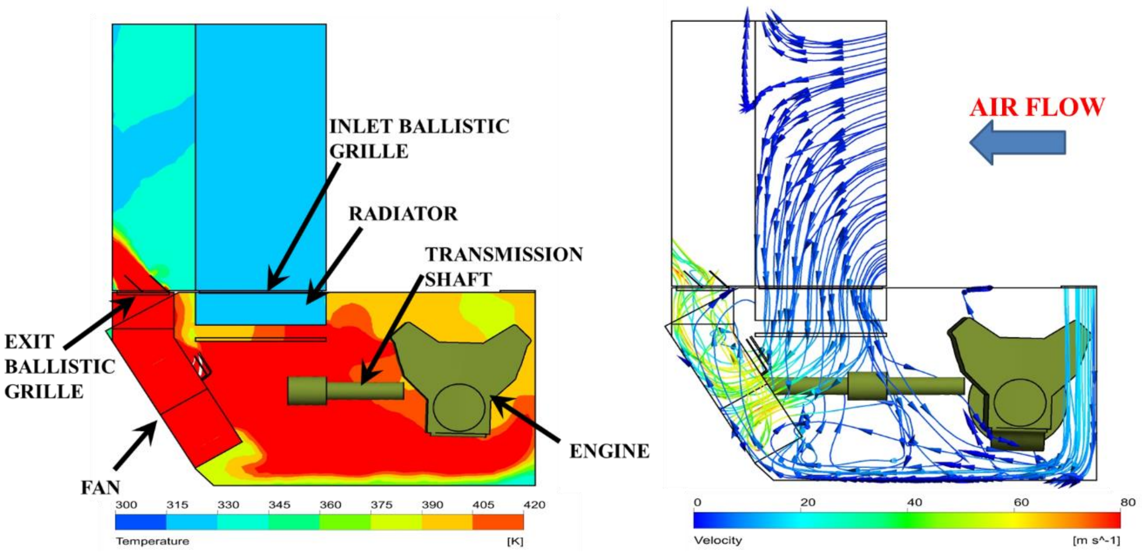

Fig.10 Temperature contours and streamlines showing the internal flow pattern, α_{RP} = 1.64 (100 percent load); α_{FP} = 0.11 (fan power not adjusted); Ambient temperature $T_{AMBIENT}$ = 323K (50C); V = 16.67 m/s (60 kmph).

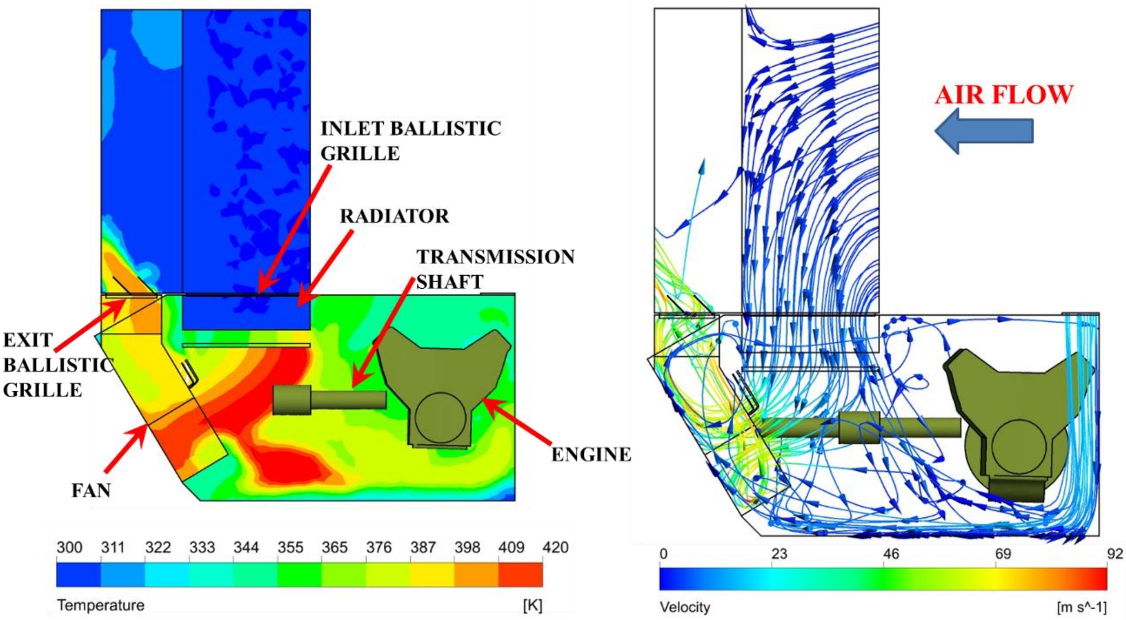

Fig.11 Temperature contours and streamlines showing the internal flow pattern; α_{RP} = 1.64 (100 percent load); α_{FP} = 0.078 (fan power adjusted); Ambient temperature $T_{AMBIENT}$ = 300K (27C); V = 16.67 m/s (60 kmph)

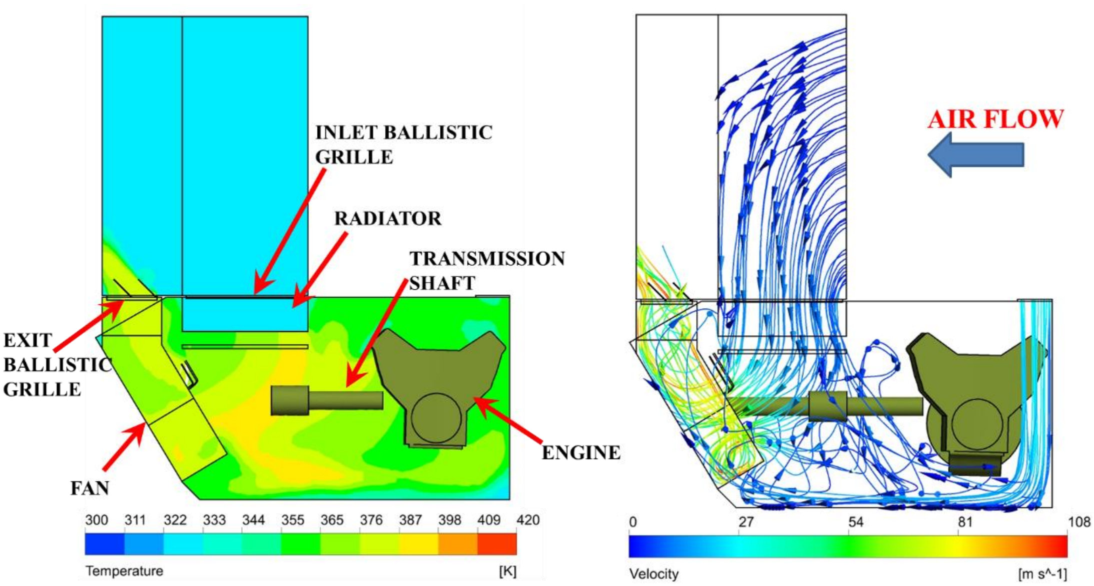

Fig.12 Temperature contours and streamlines showing the internal flow pattern; α_{RP} = 1.91(60 percent load); α_{FP} = 0.351 (fan power adjusted); Ambient temperature $T_{AMBIENT}$ = 323K (50C); V = 16.67 m/s (60 kmph)