Underhood Thermal Analysis on a Platoon of Heavy-Duty Vehicles

Venkat sai Dhushetty

Abstract— **This project evaluates the aero-thermal behavior of the heat exchangers in truck platooning. The end goal is to find the optimal following distance between the trucks in platoon formation to benefit from Aerodynamic drag while still maintaining the optimum thermal conditions in trucks forming a platoon. This end goal is achieved by simulating a series of Aerodynamic and thermal analysis tests using computational fluid dynamic analysis software on 3D CAD graphic model imitating the real-world platooning scenario. This paper briefly describes the various methodologies and techniques used to achieve the end goal. This report describes the software required and appropriate modeling approaches to employ for simulations of both aerodynamic and thermodynamic processes. Platooning Simulations are further discussed defining and justifying initial parameters, equations and illustrates the losses occur due to improper cooling and how these losses are important as aerodynamic analysis. This research gives aid for further research.**

Keywords— Platooning, Aerodynamic Analysis, Thermal Analysis, Fuel efficiency, Spacing policy,

I. INTRODUCTION

HDV platooning is described as a group of vehicles following and imitating the lead vehicle while driving closely behind one another. Platooning significantly increases roadway capacity while reducing CO2 emissions and fuel consumption. In active platoon formation HDVs benefit from accelerating and decelerating or braking simultaneously between HDVs, which results in causing low probability to form congestion enabling to extend road capacity. Many research studies have illustrated and demonstrated that platooning can achieve significant fuel savings because of aerodynamic resistance applied on HDVs while platooning.

A. Statement of purpose

The purpose of this project is to find the optimal following distance for a platoon of HDV by conducting aerodynamic and under-hood thermal analysis using computational fluid dynamic evaluation in Flow simulation. This project mainly focuses on the modelling and simulation of the HDV platoon, to understand the thermal behavior of the vehicles under platooning conditions.

B. Scope

The scope of this project is to establish a rough estimate of the ideal following distance for a platoon of HDVs to benefit from the aerodynamic drag benefits and thermal performance drop in both leading and trailing vehicles. Calculating the exact fuel efficiency as a numerical number does not include in the scope of this project.

C. Ideas

Vehicle platooning has the potential of reducing fuel consumption in both leading and trailing vehicles. Recent studies have shown aerodynamic considerations in vehicle platooning. In platooning configurations, lower cold mass air flow rates were observed through the cooling package particularly in trailing vehicles by assuming constant heat rejection rates for fuel condenser, Charge Air Cooler (CAC) and radiator.

II. METHODOLOGY

We understood that platooning means two or more trucks moving one after another in the formation of a platoon going in the same direction. In the following simulated framework, a constant velocity situation (CVS) is simulated to find the aerodynamic drag and thermal benefits of lead and trailing vehicles. Constant velocity situation is selected demonstrating the communication between two trucks is perfect with no communication lag and relative velocity between lead and trailing trucks would be zero. At CVS acceleration of the trucks is ignored since their value would be 0 mph. This is assumed to be ideal and applied for both the trucks and simulation on these vehicles will occur after these HDVs reach specific speed.

$$V_1 = V_2 = V \tag{8}$$

The above equation explains that both HDVs moving at same velocity, therefore acceleration would be zero. The HDVs will be operating on a 0% road grade, this is done to simply the simulation and cut down the simulation time as adding more parameters like acceleration curve and road grade would make the simulation complex and these could be subject to future research on the topic. Therefore, the initial simulations of HDVs will operate at constant velocity and Since, Engine temperature of the HDVs varies with time, the highest temperature on HDV engine is chosen.

After these simulations are complete, variables like road gradient, vehicle mass, and vehicle design can be changed to get a more accurate analysis of the situation. Future research may address these modifications to the initial simulation, as they fall outside the scope of this paper.

A. Equipment Used

The simulation is performed using multiple software involving various steps and processes are listed below.

Software 1: Solidworks 2022 Student Edition for 3D CAD model and Flow simulation

Software 2: Ansys 2022 Student Edition for Flow simulation and Thermal analysis.

Each software has unique features and benefits in terms of aerodynamic and thermal analysis respectively.

B. 3D CAD Model Description

Various parts of the truck including truck, trailer, engine, radiator, cooling package, fuel condenser and heat exchangers are designed using Solidworks 2022 Student edition. The dimensions of the design model are assumed to be.

To create a highly accurate model, the various parts of 3D CAD model need to assemble properly and should have the feature to import the CAD model to another software for further analysis. The best approach to create a highly accurate model is to contact truck manufacturing companies and request a 3D CAD model of the truck, which will give guaranteed accuracy of the results. Although, many companies do not share their designs due to many concerns involved. Alternatively, we could design our own 3D model of truck to perform the analysis by taking dimensions of the real truck model.

C. CAD MODEL REQUIREMENTS

The 3D CAD model needs to be detailed perfectly to make accurate calculations with as low marginal errors as possible. Simultaneously, the CAD model is further simplified as a greater number of complex designs would result in consuming more time to perform Aerodynamic or thermal analysis on the 3D model.

The dimensions of the radiator, engine, and heat exchangers whose involvement may affect the air flow into the grills, curves and body spaces of various parts have been modelled accurately to get best accurate simulations. Thereby, the engine and other heat exchangers are model are simplified as excessive detailing will make the thermal analysis more demanding and reduces overall efficiency. The block representation of 3D CAD model of the engine is shown in the Figure below,

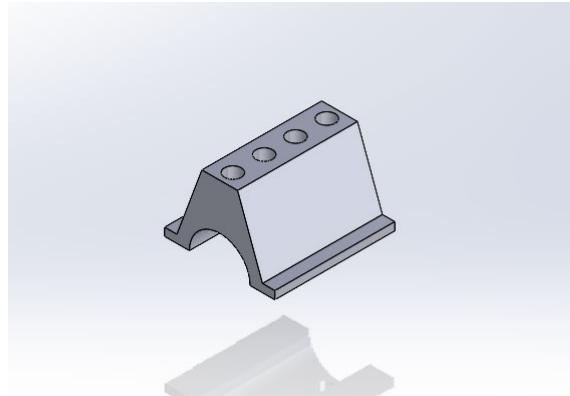


Figure 1: Block representation of Truck engine

The Block representation of Engine is simplified as above to make the calculations faster.

The model of the truck is detailed and simplified as shown below in the Figure below. This will allow us to get accurate results when the air fluid is flowing over the body of the truck. The truck design is kept minimal as stated above complex structures would result in flow air passing through various openings and hence gives less accuracy. In addition, complex structures lead to consuming more time to design and analysis and needs more computing to perform thermal analysis. Thereby, mirrors, interior, doors, and other parts are not included in the below design. The model is further simplified, as the model does not have cutouts for the wheels, this has decreased model complexity as while simulation the air flow will get caught in the arches and decreases the accuracy in the location of interest.

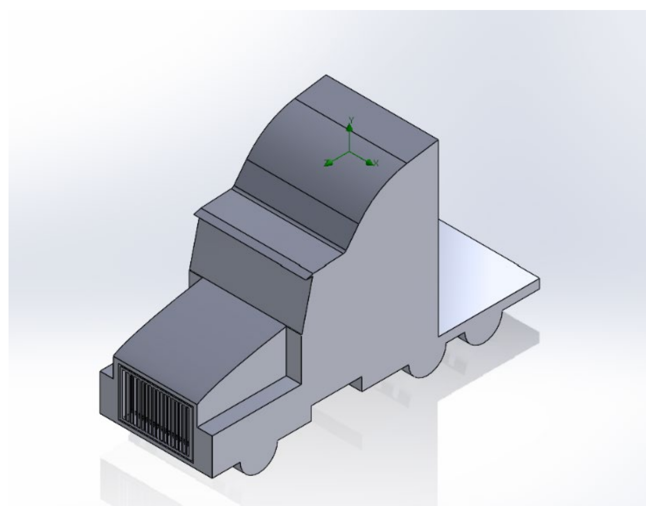


Figure 2: Simplified design of Truck

Once, truck and engine block and other heat exchangers are modelled, they are placed inside the truck. This can be achieved by coinciding the centerline of the engine block in the centerline plane of the truck as shown in the below figure. Proper measurements have been taken to make space for the air to flow from the grills of the truck to the engine block where it is placed.

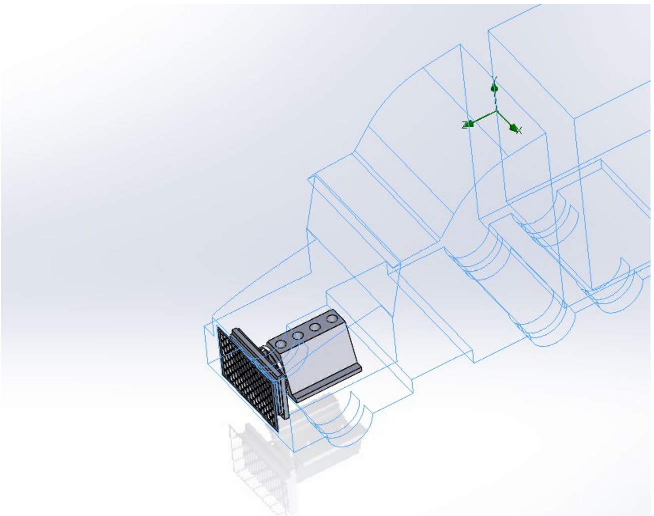


Figure 3: Engine centerlined with truck

The below figure is the simplified 3D CAD model of the trailer and needs to be attached to the backside of the truck as the air flows from the trailer to the radiator of the trailing vehicles. To make sure the air flows from beneath the truck a groove is modelled to cut through both the truck and trailer. This will enable to pass the air under the vehicle and enable it keep it minimal for unneeded complexity while performing thermal analysis.

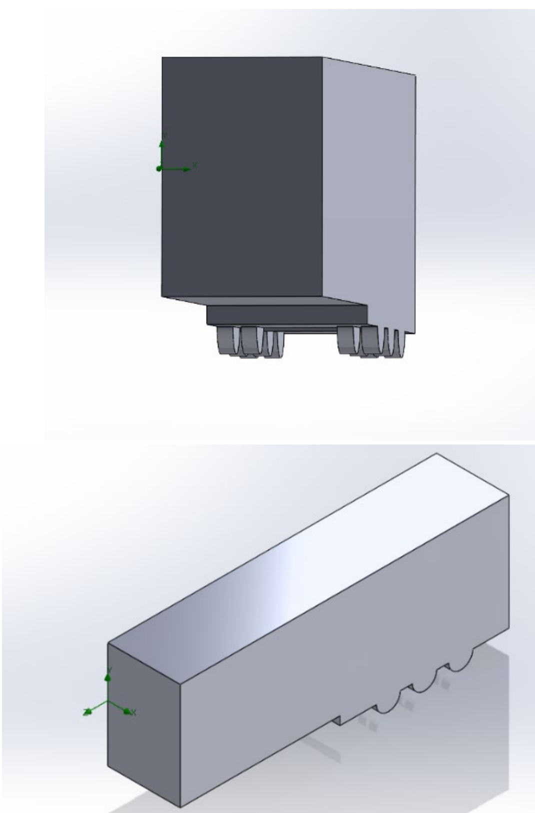


Figure 4: Trailer and air grove designs

D. HDV Assembly

All the parts of the model are put together and linked to each other and will be seen in the figure below.

Once a single HDV is assembled, a platoon of vehicles can be easily assembled by simply copying and pasting behind a HDV with a separation distance. Ensure that the ground level of both HDV should be equal and HDVs must be on the same side and must be parallel to each other. This would look as shown in the figure below.

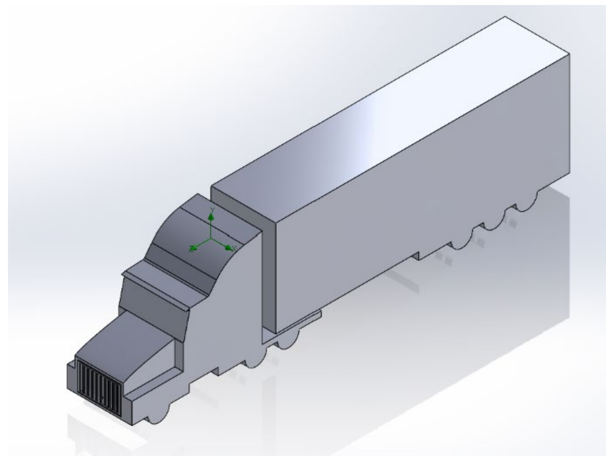


Fig 5: HDV Assembly

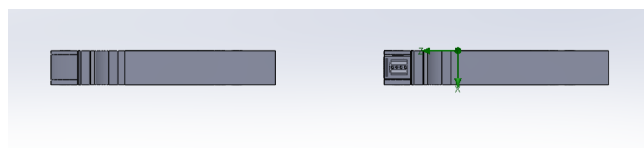


Figure 6: HDV Platoon Formation

E. Mesh Creation

A good mesh must be developed to run the simulations correctly, guaranteeing the stability of the simulation and accurate calculations. Construction of Improper mesh results in obstacle mesh and failed mesh returning errors and the simulation will not run. Even if it does run the results will be inaccurate.

A higher mesh density can be generated on the object of interest such as engine, heat exchangers and other parts of the assembly, while a lower density can be used for objects of lesser relevance. The mesh should have even lines that cover the entire object. A maximum density can be applied to the interest regions after a mesh has been made for each component. This will increase the simulation time with a tradeoff of accurate results.

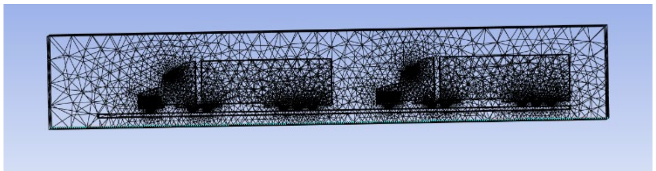


Figure 7: Mesh Creation

III. DATA ANALYSIS AND RESULTS

A. *Aerodynamic drag simulations*

Operating Conditions	Values
Velocity Inlet [mph]	55
Temperature Inlet [K]	300
Outlet Pressure [bar]	1
Computational Domain [m3] [Length x Width x Height]	200x500x100

Table 1: Input parameters for aerodynamic simulations

B. *Computational Domain*

The below figure shows the computaional domain used to run the simulations.

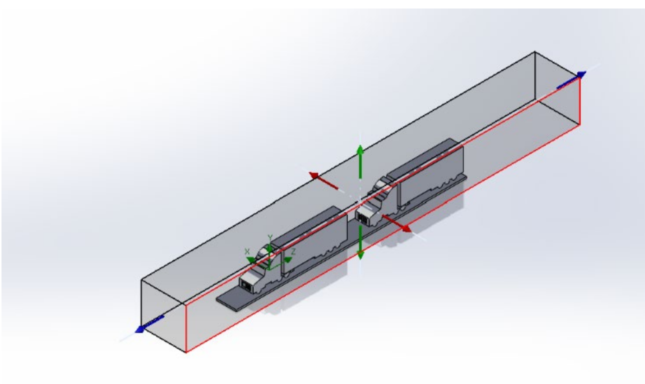


Figure 8: Computational Domain

C. *Velocity characteristics in aerodynamic analysis*

An input velocity of 55mph has given at the inlet of the computaional domain, the highest velocity seen is at the Truck front part which is 55mph flowing over the truck. Air is also flown through the engine compartment.

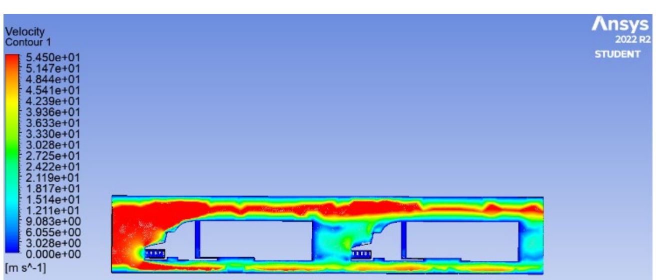


Figure 9: Aerodynamic Analysis on Platoon of two trucks

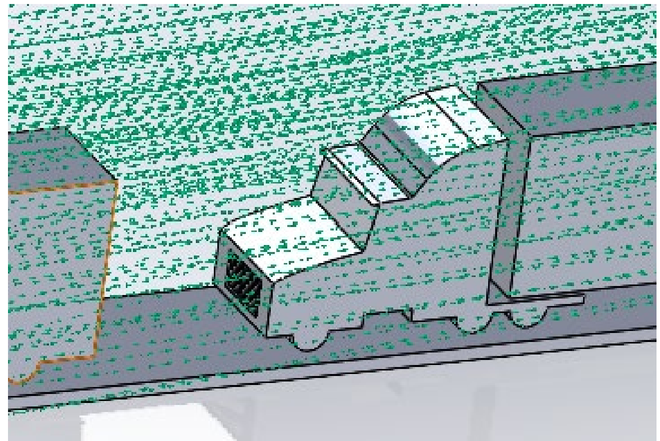


Figure 10: Air flowing through the engine compartment of the trucks.

D. *Pressure characteristics in aerodynamic analysis*

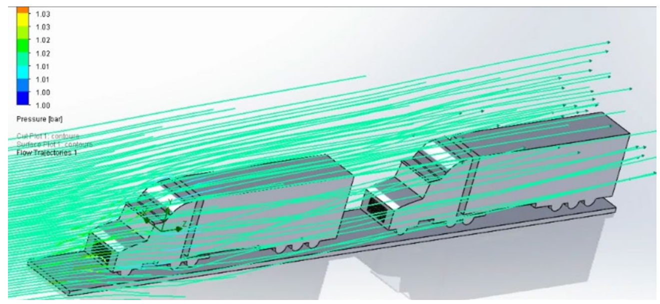


Figure 11: Pressure analysis

Another issue raised in this research is that the pressure drop that would occur (between the grills of the lead and following HDVs) would result in inadequate air pressure to overcome the pressure required to flow through the radiator. This simulation demonstrates that there is a pressure reduction at the tail of the leading HDV's trailer. If the trailing vehicle follows more closely than the results might not be accurate. To comprehend this phenomenon better, more research should be done.

E. *Underhood thermal analysis*

The Boundary conditions of the heat exchangers parameters are refereed and cited from [4]. In this section, underhood thermal analysis boundary conditions are determined by constant Heat Rejection Rate Model with an Influence of Aerodynamic Drag on Fuel Consumption. 3D steady state thermal analysis simulations were carried out using fluid flow fluent and steady state thermal in Ansys Student version.

Heat exchangers were modeled using a porous substance with an inertial and viscous resistance as stated in the below table

The below equation is used to calculate the resistances from experimental pressure drops and velocities.

$$\frac{dp}{L} = \alpha u + \beta u^2 \qquad (9)$$

Here, dp is pressure drop, L is thickness, α is viscous resistance, β is inertial resistance, u is velocity.

Equation 10 was used to determine these resistances using experimental pressure drop and velocity data.

$$\dot{Q} = \dot{Q}_{no-traffic}\left[1 - \frac{\left(1 - \frac{C_{d,i}}{C_{d,no-traffic}^{yaw=0^0}}\right)}{2}\right] \quad 10$$

Here, $\dot{Q}$ denotes heat rejection rate and C_d is the aerodynamic drag coefficient of the vehicle.

F. Modelling of heat exchangers

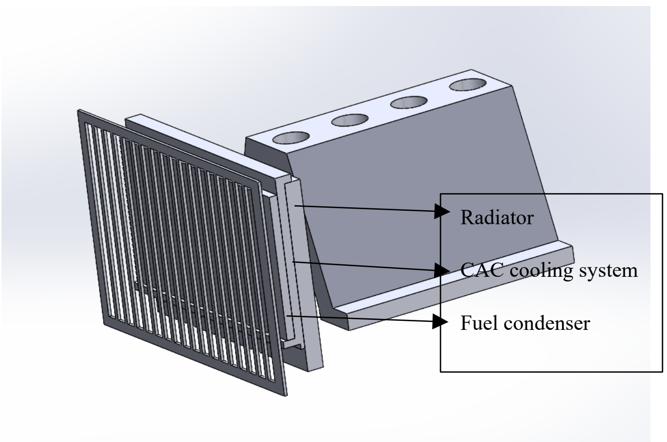


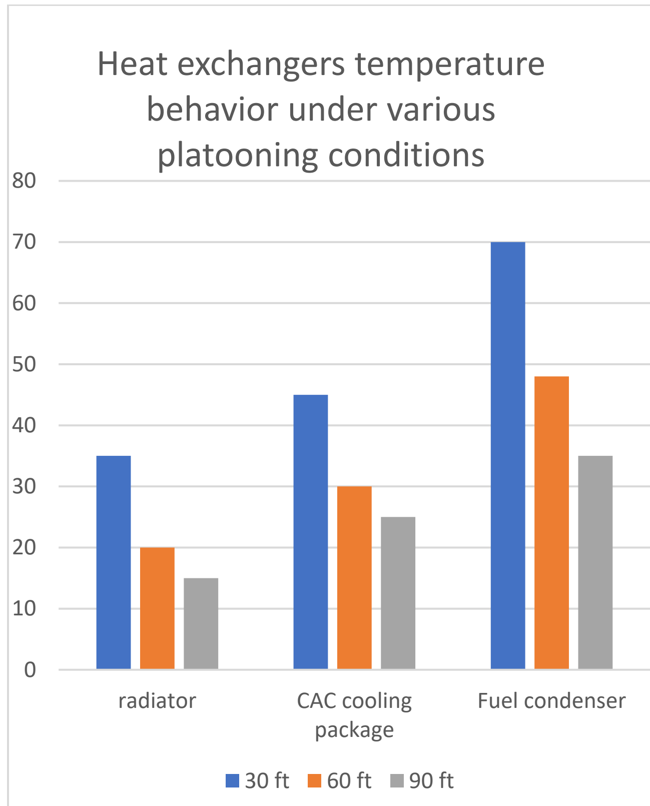
Figure 12 : Modelling of Heat Exchangers

	Fuel condenser	CAC	Radiator
Inertial Forces [kg/m4]	66	67	330
Viscous forces [kg/m3-s]	66	415	335
Constant heat rejection rate [kW]	11	24.25	55

Table 2: Input parameters of the Heat Exchangers

The above table displays the porous resistance values for the fuel condenser, radiator, and CAC. These porous body resistances that were provided were obtained from Vegendla et al. [5]. A constant fan speed is assumed to be 1400 rpm as citied in. Vegendla et al. [5].

After performing the simulations on 3 different separation distances, we can observe that at 30 ft separation distance the trailing vehicle Heat exchangers including cooling package, radiator, fuel condenser, temperature raises when compared to lead HDV. At 60 ft separation distance, the trailing vehicle Heat exchangers temperature drops and less than when compared to the 30ft separation distance, this is because of the low velocity vortex region between both leading and trailing vehicles. At 90 ft HDV separation distance, the trailing vehicle cooling package temperature will decrease further when compared to 60 ft separation distance. This was mainly due to the amount of cold mass air flow rates raises with an increase of the vehicle separation distance. The Graphical representation of the Cooling package temperature rise over vehicle separation distance is shown below



Graph 1: Heat exchangers temperature behavior under various platooning conditions

As shown in the figure, the temperature rise drops in the trailing vehicles was observed with an increase of vehicle separation distance from 30 ft to 90ft in all the heat exchangers.

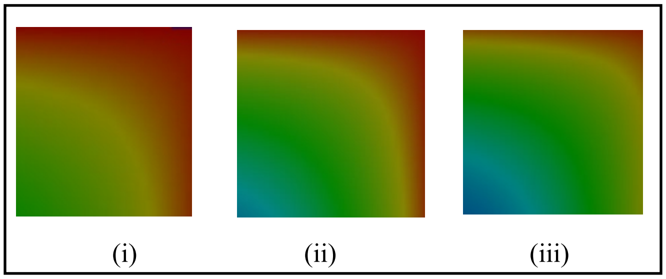


Figure 13 :Radiator at (i)30 ft (ii) 60 ft (iii) 90 ft Vehicle separation distance

The above figure illustrates the simulation results of the Thermal analysis performed on Radiator.

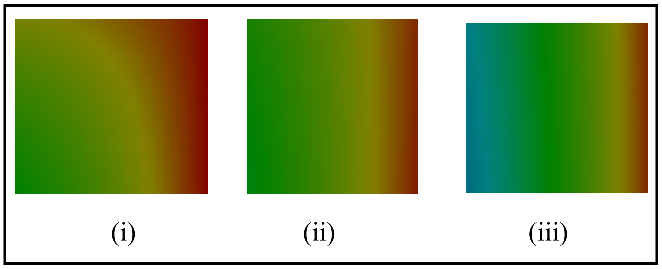


Figure 14 :CAC Temperature at (i)30 ft (ii) 60 ft (iii) 90 ft Vehicle separation distance

The above figure illustrates the simulation results of the Thermal analysis performed on Cooling package.

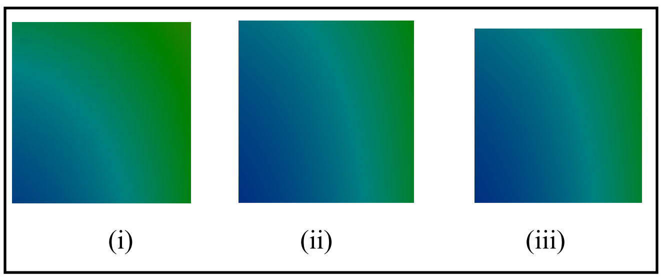


Figure 15 : Fuel condenser at (i)30 ft (ii) 60 ft (iii) 90 ft Vehicle separation distance

The above figure illustrates the simulation results of the Thermal analysis performed on fuel condenser.

IV. CONCLUSIONS

In this paper, proof has shown that thermal degradation is as important as aerodynamic analysis for effective fuel efficiency over time. A proper 3D model and simulation environment of platooning is also presented to understand the aerodynamic and underhood thermal behavior of the HDVs under platooning. Various methods on fuel consumption, spacing policy, Adaptive cruise control and Cooperative adaptive cruise control are shown how HDVs share information of the truck to each other. Results of the simulation of HDV platooning are presented as well to provide proof of the benefits of truck platooning to some extent. Assumed initial parameters based on the real world situations, which provided proof of values which are hard to calculate by hand accurately and gave scope for further research in terms of increasing the accuracy of the model.

Considering the requirements of the vehicle's transmission will result in a fall in thermal efficiency since more energy will be required to provide the same amount of labor output. Unless the following distances or cooling capacity is increased, this should lead to higher engine running temperatures.

V. Lessons Learned

I have researched and learned about a lot of theories related to truck platooning, some of which include spacing policy, cruise control and vehicle dynamics. I have learned how to design a model in Solidworks and how to perform aerodynamic and thermal analysis on the model which I have designed in Solidworks using Ansys software. I have joined various courses to learn how to use the Solidworks and Ansys software which helped me understand this project even further.

VI. References

[1] QICHEN DENG, Heavy-Duty Vehicle Platooning– Modeling and Analysis https://www.diva-portal.org/smash/get/diva2:901974/FULLTEXT02.pdf

[2] Vegendla, P., Sofu, T., Saha, R., Madurai Kumar, M. et al., “Investigation of Aerodynamic Influence on Truck Platooning,” SAE Technical Paper 2015-01-2895 , 2015, doi:10.4271/2015-01-2895.

[3] Ellis, M., Gargoloff, J., and Sengupta, R., “Aerodynamic Drag and Engine Cooling Effects on Class 8 Trucks in Platooning Configurations,” SAE Int. J. Commer. Veh. 8(2):732-739, 2015, doi:10.4271/2015-01-2896.

[4] Investigation on Underhood Thermal Analysis of Truck Platooning https://www.sae.org/publications/technical-papers/content/02-11-01-0001/

[5] Vegendla et al. Underhood thermal analysis of Tractor-Trailer https://www.researchgate.net/publication/310843320_Underhood_thermal_analysis_of_Tractor-Trailer

[6] T. Oguchi, M. Katakura, and M. Taniguchi. Carbon dioxide emission model in actual urban road vehicular traffic conditions. Infrastructure Planning and Management, No. 695/IV-54, pp. 125-136, January 2002. (2002).