Combined Aero and Underhood Thermal Analysis for Heavy Duty Trucks

Final CRADA Report

Nuclear Engineering Division

About Argonne National Laboratory

Argonne is a U.S. Department of Energy laboratory managed by UChicago Argonne, LLC under contract DE-AC02-06CH11357. The Laboratory's main facility is outside Chicago, at 9700 South Cass Avenue, Argonne, Illinois 60439. For information about Argonne and its pioneering science and technology programs, see www.anl.gov.

DOCUMENT AVAILABILITY

Online Access: U.S. Department of Energy (DOE) reports produced after 1991 and a growing number of pre-1991 documents are available free via DOE's SciTech Connect (<http://www.osti.gov/scitech/>)

Reports not in digital format may be purchased by the public from the National Technical Information Service (NTIS):

U.S. Department of Commerce
National Technical Information Service
5301 Shawnee Rd
Alexandria, VA 22312
www.ntis.gov
Phone: (800) 553-NTIS (6847) or (703) 605-6000
Fax: (703) 605-6900
Email: **orders@ntis.gov**

Reports not in digital format are available to DOE and DOE contractors from the Office of Scientific and Technical Information (OSTI):

U.S. Department of Energy
Office of Scientific and Technical Information
P.O. Box 62
Oak Ridge, TN 37831-0062
www.osti.gov
Phone: (865) 576-8401
Fax: (865) 576-5728
Email: **reports@osti.gov**

Disclaimer

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor UChicago Argonne, LLC, nor any of their employees or officers, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of document authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof, Argonne National Laboratory, or UChicago Argonne, LLC.

Combined Aero and Underhood Thermal Analysis for Heavy Duty Trucks

Final CRADA Report

prepared by
Prasad Vegendla and Tanju Sofu
Nuclear Engineering Division, Argonne National Laboratory

Participants:
R. Saha, L.K.Hwang, M.K.Madurai
Cummins Inc. Columbus, Indiana

01-31-2017

This page intentionally left blank

**Non Proprietary
Final CRADA Report**

Date: 01-31-2017

CRADA Number: C1201001

CRADA Title: Combined Aero and Underhood Thermal Analysis for Heavy Duty Trucks

CRADA Start/End Date: Oct 2012 to Jan 2017

Argonne Dollars: \$625K

Participant Dollars: \$1050K

Argonne PI: Tanju Sofu

Participant(s):

Cummins Inc. \$1050K
Name Participant Dollars

1900 Mckinley Avenue, MC: 50040, Columbus, IN, 47201
Complete Address

Name \$ _____
Participant Dollars

Complete Address

Name \$ _____
Participant Dollars

Complete Address

DOE Program Lee Slezak
Manager:

Summary of Major Accomplishments:

The overall averaged aerodynamics drag reduction through the design modifications were shown up to 22.3% through aerodynamic considerations alone, which is equivalent to 11.16% fuel savings. The main identified fuel efficiencies were based on second generation devices, including wheel covers, side skirts, roof fairings, and rounded trailer corners.

The optimum curvature radius of the rounded trailer edges found to be 125 mm, with an arc length of 196.3 mm. The aerodynamic drag reduction increases with dropping clearance of side skirts between wheels and ground. The aerodynamic drag reduction increases with an extension of front bumper

Summary of Major Accomplishments:
towards the ground.

In medium-duty vehicles, the boat-tail reduces the aerodynamic drag upto 12%. The optimized slant angle and plate width were 15 deg and 1.2m, respectively.

Significant fuel savings were observed in a conventional truck (G7C) when compared to cab-over engine (M7) configurations at all yaw angles. The major reduction in fuel consumption was observed mainly from the cabin designs alone (up to 7.5%). Based on coolant temperatures, 12% and 10% thermal efficiency improvements were observed in conventional trucks when compared to cab-over engine for charge air cooler (CAC) and radiator, respectively.

In vehicle underhood, fan-shroud optimization found significant raise (1.4%) in cooling air mass flow rate.

In vehicle platooning, significant Yaw Averaged Aerodynamic Drag (YAD) reduction was observed in both leading and trailing vehicles. In single lane traffic, YAD reduction was up to 8% and 38% in leading and trailing vehicles, respectively.

In both leading and trailing vehicles, YAD reduction effects were dropped with higher separation distances.

The trailing vehicle following the leading vehicle immediate to the next separate lane shown a YAD reduction up to 3% in the leading truck at lower separation distances. Further increasing the separation distance, YAD reductions were nullified in the leading vehicle. On the other hand, the trailing vehicle YAD was reduced up to 15% with increase in the separation distance. In multiple lane side-by-side vehicle traffic, the YAD performance was dropped up to 22% in all vehicles.

In single- and two-lane leading and trailing vehicle platooning, the significant drop in cold mass flow rates (temperature rise) were observed in cooling package of trailing vehicle due to a big vortex (low velocities) zone between both leading and trailing vehicles.

At 0 rpm fan speed; 70% lower air mass flow rates (negative performance) observed in trailing vehicle due to low velocity zone from leading vehicle. In single-lane traffic, air flow rates improved with vehicle separation distance.

At 1400 rpm fan speed; 30% lower air mass flow rates observed in trailing vehicle due to low velocity zone from leading vehicle. In single-lane traffic, air mass flow rates improved with vehicle separation distance.

In two-lane traffic, at 0 and 1400 rpm, lower air mass flow rates, but negligible compared to single-lane traffic and insignificant to the vehicle separation distance.

In variable heat rejection, the temperature rise is lower (<4 deg C) compared to constant heat rejection (<15 deg C) in trailing vehicle [lower amount of heat rejection due to reduced aerodynamic drag and thus leads to lower fuel consumption and lower heat rejection from engine].

In two-lane traffic and constant heat rejection; the temperature raise was independent to the vehicle

Summary of Major Accomplishments:

separation distance and negligible in variable heat rejection case compared to no-traffic vehicle.

Summary of Technology Transfer Benefits to Industry:

Cummins Inc. and his foreign subsidiaries (DFLZ) were implemented the proposed aerodrag devices in heavy-duty vehicle (M7). Also, the identified improvements are suggested to include in their original designs of medium-duty vehicles (9600).

Other Information/Results: (Papers, Inventions, Software, etc.)

Vegendla, P and Sofu, T. Investigation of aerodynamic drag on tractor-trailer in underground road traffic. *Int. Journal of Aerodynamics*, Vol. 5, No. 2, 2016; pp:133-142.

Vegendla, P., Sofu, T., Saha, R., Kumar, M.M., Hwang, LK., Dowding, S. Fan shroud optimization using adjoint solver. *SAE Technical Paper*, 2016-01-8070, doi:10.4271/2016-01-8070.

Vegendla, P., Sofu, T., Saha, R., Hwang, LK. Comparison of Aerodynamic drag and Underhood Thermal Analysis of Two Heavy-duty Vehicles. *Int. Journal of Aerodynamics*, Vol. 5, No. 2, 2016; pp:105-124.

Vegendla, P., Sofu, T., Saha, R., Kumar, M.M., Hwang, LK. Investigation of Aerodynamic Influence on Truck Platooning. *SAE Technical Paper*, 2015-01-2895. DOI:10.4271/2015-01-2895.

Vegendla, P., Sofu, T., Saha, R., Kumar, M.M., Hwang, LK. Investigation on Underhood thermal Analysis of Truck Platooning (under review), *Int. Journal of Commercial Vehicles*

Vegendla, P., Sofu, T., Saha, R., Hwang, LK. Aerodynamic Drag Optimization of Medium-duty Delivery Truck (under Review), *Int. Journal of Aerodynamics*

SUMMARY

The optimum design of the new/existing medium/heavy vehicles is valuable to increase the fuel economy. To meet today's energy demand the improved designs are necessary for achieving energy efficient vehicles in the transport sector. In this work, aerodynamic and underhood thermal analyses of the medium and heavy vehicles were performed to optimize the design.

A reliable analytical model capable of predicting air flow around the complete truck including underhood can have a significant impact toward achieving the most reliable design and can also help make a realistic aerodynamic tradeoff for the best fuel efficiency.

The four base case vehicle configurations provided by the Cummins Inc. from their subsidiaries were studied at Argonne focusing on methods and model development, analysis, and validation efforts.

Aerodynamic analyses were performed for the provided base-case vehicle configurations, namely M3, M7, 9600 and G7C trucks. The 3D CFD modeling and simulations were carried out for several combinations of modified designs with detailed external component analysis, i.e, cabin mirrors, roof deflector, trailer etc.

The aerodynamic drag results were validated with two different commercial CFD codes StarCCM+® and PowerFOW®. Good agreement was observed for aerodrag while comparing these codes.

The overall fuel efficiency in heavy- and medium-duty (M3, M7, G7C and 9600) vehicle models were shown upto 1.2-14% with improved external design of the vehicles. The main identified fuel efficiencies were based on second generation devices, i.e. modified roof deflectors, side extenders, curvy surfaces of mirrors and trailer, flat bed and trailer and tail boat. Also, fan-shroud optimization was performed using an automated Adjoint solver and helps to avoid manual interaction and cooling air flow rate increases 1.3%.

In addition to the optimization of vehicle aerodynamic drag, the aerodynamic characteristics of vehicles were analyzed in five different platooning configurations with two and three vehicles in single and multiple lanes. A significant aerodynamic drag reduction was observed in both leading and trailing vehicles. The aerodynamic drag reduction was up to 8% and 38% in leading and trailing vehicles, respectively. Furthermore, the platooning effects on thermal performance of leading and trailing vehicles were also analyzed.

Table of Contents

Summary.....	i
Table of Contents	ii
List of Figures	v
List of Tables.....	xiii

1. Aerodynamic Drag Optimization of Medium-duty Delivery Truck

1.1. Introduction.....	1
1.2. Vehicle Configurations.....	2
1.3. Modeling Details.....	3
1.4. Results.....	4
1.4.1. Base configuration.....	5
1.4.2. Case-1 (Wheel Covers) Vs. Base Configuration.....	6
1.4.3. Case-2 (Side Skirts) Vs. Base Configuration.....	7
1.4.4. Case-3 (Side Skirts + Lowered Bumper) Vs. Case-2 (Side Skirts).....	9
1.4.5. Case-4 (Side Skirts + Lowered Bumper + Roof Fairings) Vs. Case-3 (Side Skirts + Lowered Bumper).....	9
1.4.6. Case-5 (Side Skirts + Lowered Bumper + Roof Fairings + Rounded Corners) Vs. Case-4 (Side Skirts + Lowered Bumper +Roof Fairings).....	10
1.4.7. Optimized Design Aerodynamics Drag.....	12
1.5. Conclusions.....	13

2. Aerodynamic Drag Optimization of M3 Medium-duty Truck..... 14

2.1. Vehicle Configuration Studied.....	14
2.2. Boundary Conditions.....	15
2.3. CAD models and Mesh.....	15
2.4. Results and Discussion.....	15
2.5. Conclusions.....	16

3. Comparison of Aerodynamic Drag and Underhood Thermal Analysis of Two Heavy-Duty Vehicles..... 17

3.1. Introduction.....	17
3.2. Vehicle Configurations studied.....	18
3.3. Modeling Development.....	19
3.4. Results and discussion.....	20
3.4.1. Aerodynamic analysis.....	20
3.4.2. Cabin only.....	21
3.4.3. Cabin plus flatbed, with or without side skirts.....	22
3.4.4. Base case, with or without side skirts.....	26

3.4.5. Base case, with and without wheel covers.....	28
3.4.6. Cabin-trailer gap sensitivity aerodynamic drag analysis.....	30
3.5. Underhood thermal analysis.....	32
3.6. Conclusions.....	36
4. Boat-Tail Influence on Tractor-Trailer Aerodynamic Drag.....	37
4.1. Introduction.....	37
4.2. Modeling.....	37
4.3. Results.....	38
4.4. Conclusions.....	42
5. Fan Shroud Optimization Using Adjoint Solver.....	43
5.1. Introduction.....	43
5.2. Vehicle Underhood Compartment.....	44
5.3. Modeling.....	46
5.4. Results.....	48
5.5. Conclusions.....	50
6. Investigation of Aerodynamic Influence on Truck Platooning.....	51
6.1. Introduction.....	51
6.2. Vehicle Configuration Studied.....	52
6.3. Modeling.....	54
6.4. Results and Discussion.....	55
6.4.1. Two Vehicle Configurations.....	55
6.4.1.1. Vehicle-to-Vehicle (V2V) simulations in single-lane traffic.	55
6.4.1.2. Vehicle-to-Vehicle (V2V) simulations in multi-lane traffic.	58
6.4.1.3. Side-by-side vehicle simulations in multi-lane traffic.....	61
6.4.2. Three Vehicle Configurations.....	62
6.4.2.1. Multiple side-by-side vehicle simulations in multi-lane traffic	62
6.4.2.2. Two side-by-side leading vehicles and one trailing vehicle simulations in multi-lane traffic.....	64
6.5. Conclusions.....	66
7. Investigation on Underhood Thermal Analysis of Truck Platooning.....	67
7.1. Introduction.....	67
7.2. Vehicle Modeling.....	68
7.3. Results.....	70
7.3.1. Underhood Thermal Analysis by a Constant Heat Rejection Rate Model.	70
7.3.2. Single-lane traffic.....	71
7.3.3. Two-lane traffic.....	73
7.4. Underhood Thermal Analysis by Varying Heat Rejection Rate Model with an Influence of Aerodynamic Drag on Fuel Consumption.....	75
7.4.1. Single-lane traffic.....	75
7.4.2. Two-lane traffic.....	77
7.5. Conclusions.....	79

8. Investigation of Aerodynamic Drag on Tractor-Trailer in Underground Road Traffic.....	80
8.1. Introduction.....	80
8.2. Vehicle and road-tunnel configurations.....	81
8.3. Results and Discussion.....	84
3.1.1. Single/No-traffic Vehicle.....	84
3.1.2. Multiple Vehicles.....	85
8.4. Conclusions.....	91
9. References.....	92
Acknowledgments.....	95
Appendix A.....	96
Appendix B.....	96

LIST OF FIGURES

Figure 1.1. Vehicle configurations: (i) Base configuration, (ii) Case-1, (iii) Case-2, (iv) Case-3, (v) Case-4, and (vi) Case-5.

Figure 1.2: Numerical setup [2].

Figure 1.3. Cumulative normalized aerodynamics drag along the vehicle length.

Figure 1.4. Velocity fields at (i) 0° yaw and (ii) 6° yaw.

Figure 1.5. Static pressure distribution at (i) 0° yaw and (ii) 6° yaw.

Figure 1.6. Cumulative normalized aerodynamics drag along the vehicle length using wheel covers (i) Yaw 0° and (ii) Yaw 6° .

Figure 1.7. Underbody velocity vector (i) base configuration and (ii) wheel covers (Case-2) of medium-duty vehicle in 0° yaw angle

Figure 1.8. Cumulative normalized aerodynamics drag along the vehicle length using side skirts.

Figure 1.9. Velocity streamlines of medium-duty vehicle (i) no side skirts and (ii) side skirts [yaw 6°].

Figure 1.10. Cumulative normalized aerodynamics drag along the vehicle length.

Figure 1.11. Velocity stream lines: (i) base configuration and (ii) lowered bumper.

Figure 1.12. Cumulative normalized aerodynamics drag along the vehicle length: (i) 0° yaw and (ii) 6° yaw.

Figure 1.13. Velocity stream lines: (i) base configuration and (ii) optimized roof.

Figure 1.14. Cumulative normalized aerodynamics drag along the vehicle length using rounded corners [yaw 0°].

Figure 1.15. Velocity streamlines of medium-duty vehicle (i) sharp corners and (ii) rounded corners [yaw 0°].

Figure 1.16. Cumulative normalized aerodynamic drag along the vehicle length: (i) 0° yaw and (ii) 6° yaw.

Figure 1.17. Velocity streamlines of medium-duty vehicle (i) original mirrors and (ii) rounded mirrors [yaw 0°].

Figure 1.18. Comparison of YAD reduction with a base case for each configuration.

Figure 2.1. Base Case truck configurations

Figure 2.2. Drag coefficient along the length of the M3 truck

Figure 3.1. Heavy-duty vehicle configuration of (i) conventional and (ii) cab-over engine

Figure 3.2. Wind tunnel numerical setup

Figure 3.3. Conventional truck performance over cab-over engine configuration (i) YAD and (ii) fuel savings

Figure 3.4. Normalized drag comparison of conventional and cab-over engine cabin over different yaw angles

Figure 3.5. Normalized drag coefficient along the length of the (i) conventional and (ii) cab-over engine cabin

Figure 3.6. Normalized drag comparison of conventional and cab-over engine cabin-flatbed, over different yaw angles

Figure 3.7. Velocity field in central vertical plane of (i) cab-over engine and (ii) conventional truck [yaw: 0°]

Figure 3.8. Cab-over engine pressure coefficient contours (i) yaw: 0° and (ii) yaw: 6° .

Figure 3.9. Conventional truck pressure coefficient contours (i) yaw: 0° and (ii) yaw: 6° .

Figure 3.10. Normalized drag coefficient along the length of the conventional and cab-over engine cabin-flatbed vehicles using side skirts

Figure 3.11. Cab-over engine underbody velocity field in horizontal plane at center of the wheels (i) without side skirts and (ii) with side skirts [yaw: 6°]

Figure 3.12. Conventional underbody velocity field in horizontal plane at center of the wheels (i) without side skirts and (ii) with side skirts [yaw: 6°]

Figure 3.13 Normalized drag comparison of conventional and cab-over engine base case (cabin plus trailer) over different yaw angles

Figure 3.14. Normalized drag coefficient along the length of the conventional and cab-over engine vehicle (i) conventional and (ii) cab-over engine

Figure 3.15. Velocity field in central vertical plane (i) cab-over engine base case and (ii) conventional base case [yaw: 0°]

Figure 3.16. Conventional truck pressure coefficient contours (i) yaw: 0° without side-skirts, (ii) yaw: 6° without side-skirts, and (iii) yaw: 6° with side skirts.

Figure 3.17. Normalized drag coefficient along the length of the conventional vehicle using wheel covers (i) yaw: 0° and (ii) yaw: 6°

Figure 3.18. Conventional truck underbody velocity vector in horizontal plane at center of the rear wheel axles (i) base case [yaw: 0°] and (ii) wheel covers [yaw: 0°]

Figure 3.19. Normalized drag coefficient along the length of the cab-over engine vehicle using wheel covers (i) yaw: 0° and (ii) yaw: 6°

Figure 3.20. Cab-over engine gap sensitivity analysis between cabin and trailer (i) large gap (ii) reduced gap.

Figure 3.21. Cab-over engine and conventional cabin-trailer gap sensitivity on normalized aerodynamic drag over different yaw angles

Figure 3.22. Cabin-trailer gap sensitivity on normalized aerodynamic drag along the length of the cab-over engine vehicle (i) yaw: 0° and (ii) yaw: 6°

Figure 3.23. Thermal performance comparison of conventional and cab-over engine (i) front side of conventional truck-CAC, (ii) front side of cab-over engine-CAC, (iii) rear side of conventional truck-radiator, and (iv) rear side of cab-over engine-radiator

Figure 3.24. Comparison of velocity in conventional and cab-over engine cooling packages (i) front side of conventional truck-CAC, (ii) front side of cab-over engine-CAC, (iii) Rear side of conventional truck-radiator, and (iv) rear side of cab-over engine-radiator

Figure 3.25. Comparison of underbody velocity field in conventional truck and cab-over engine: (i) conventional truck: fan centered horizontal plane, (ii) cab-over engine: fan centered horizontal plane, (iii) conventional truck: fan centered vertical plane, and (iv) cab-over engine: fan centered vertical plane

Figure 4.1. Medium-duty delivery truck (i) no side-skirts and boat-tail (base case) and (ii) side-skirts boat-tail.

Figure 4.2. Boat tail plate width; (i) 0.3 m , (ii) 0.6m and (iii) 1.2m

Figure 4.3. Aerodynamic drag for different boat-tail plate width for wind velocity of 90 kph and slant angle of 15° ; (i) with side-skirts and (ii) without side-skirts.

Figure 4.4. Aerodynamic drag comparisons of with and without boat-tail plate towards ground side for wind velocity of 90kph.

Figure 4.5. Aerodynamics drag coefficient along the vehicle length for boat-tail plate width of 1.2m and wind velocity of 90kph (no side-skirts).

Figure 4.6. Velocity field at trailer rear side (horizontal plane across the center of the vehicle height)

Figure 4.7. % of aerodynamic reduction Vs. yaw angle

Figure 4.8. Comparison of aerodynamic drag force with and without boat-tail vehicle configuration

Figure 4.9. Comparison of fuel savings for two different boat-tail slant angles for vehicle speed of 90kph and with side-skirts.

Figure 5.1. Geometry (i) vehicle underhood compartment and (ii) computational domain

Figure 5.2. Computational Mesh, (i) side view and (ii) top view

Figure 5.3. Control points over fan-shroud

Figure 5.4. Adjoint position sensitivity vector (2nd Iteration)

Figure 5.5. Convergence/residuals plot (un-normalized)

Figure 5.6. Comparison of (i) original and (ii) optimized fan-shroud

Figure 5.7. Position sensitivity magnitudes at 2nd and 6th iteration [surface optimizes @ higher position sensitivities]

Figure 5.8. Pressure distribution, (i) shroud: original and (ii) shroud: optimized

Figure 5.9. Pressure distribution, (i) original and (ii) optimized flow.

Figure 5.10. Velocity contours

Figure 6.1. Medium-duty delivery truck.

Figure 6.2. Configuration of different truck positions for investigation; (i) **Configuration 1:** V2V in single lane, (ii) **Configuration 2:** V2V in multiple lanes, (iii) **Configuration 3:** Two side-by-side vehicles [LV1-LV2] (iv) **Configuration 4:** Three side-by-side vehicles [LV3-LV1-LV2] and (v) **Configuration 5:** Two side-by-side vehicles and one trailing vehicle [LV1-LV2-TV1].

Figure 6.3. Numerical setup.

Figure 6.4. Mesh top view; (i) 150 ft V2V separation distance and (ii) cabin.

Figure 6.5. Configuration 1

Figure 6.6. Yaw Averaged Aerodynamic Drag (YAD) reduction of leading and trailing vehicles [averaged values of three different yaw angles 0°, -6°, and 6°].

Figure 6.7. Static pressure distribution in 0° yaw angle (i) no-traffic, (ii) 30 ft, and (iii) 90 ft

Figure 6.8. Estimated fuel savings in leading and trailing vehicles [averaged values of three different yaw angles 0° , -6° , and 6°].

Figure 6.9. Velocity contours at 3 ft in front of a trailing vehicle in different vehicle separation distances at 6° yaw angle.

Figure 6.10. Configuration 2

Figure 6.11. Yaw Averaged Aerodynamic Drag (YAD) reduction of leading and trailing vehicle in multi-lane traffic [averaged values of three different yaw angles at 0° , -6° , and 6°].

Figure 6.12. Estimated fuel savings in leading and trailing vehicle in multi-lane traffic [averaged values of three different yaw angles at 0° , -6° and 6°].

Figure 6.13. Velocity contours at 3 ft in front of trailing vehicle at 0° yaw angle; (i) 90 ft, (ii) 60 ft and (iii) 30 ft.

Figure 6.14. Velocity magnitude of 30 ft vehicle separation; (i) 0° yaw, (ii) 6° yaw and (iii) -6° yaw.

Figure 6.15. Static pressure distribution in 6° yaw angle (i) 30 ft, (ii) 60 ft and (iii) 90 ft.

Figure 6.16. Configuration 3

Figure 6.17. Yaw Averaged Aerodynamic Drag (YAD) of multiple vehicles in multi-lane traffic [averaged values of three different yaw angles at 0° , -6° and 6°].

Figure 6.18. Static pressure distribution in 6° yaw angle (i) no-traffic and (ii) two vehicles in configuration 3.

Figure 6.19. Configuration 4

Figure 6.20. Yaw Averaged Aerodynamic Drag (YAD) of multiple vehicles in multi-lane traffic [averaged values of three different yaw angles at 0° , -6° , and 6°].

Figure 6.21. Normalized cumulative aerodynamic drag along the vehicle at 0° yaw angle for three vehicles side-by-side in multiple lanes (LV3-LV1-LV2).

Figure 6.22. Configuration 5

Figure 6.23. Yaw Averaged Aerodynamic Drag (YAD) of multiple vehicles in multiple lanes [averaged values of three different yaw angles at 0° , -6° and 6°].

Figure 6.24. Velocity magnitude in 0° yaw for (i) configuration 1: one leading and one trailing vehicle and (ii) configuration 5: two side-by-side leading vehicles and one trailing vehicle (LV1-LV2-TV1).

Figure 6.25. Velocity magnitude in different yaw angles (i) configuration 1: 6° yaw (ii) configuration 5: 6° yaw and (iii) configuration 5: -6° yaw.

Figure 7.1. Medium-duty Delivery Truck.

Figure 7.2. Platooning configurations; (i) Single-lane traffic and (ii) two-lane traffic [6]

Figure 7.3. Computational domain; (i) numerical setup and (ii) mesh top view

Figure 7.4. Vehicle underhood components; (i) vehicle top view and (ii) cooling package

Figure 7.5. Averaged temperature rise in heat exchangers of trailing vehicle at a constant fan speed; (i) cooling package, (ii) fuel condenser, (iii) CAC, and (iv) radiator [averaged values of three different yaw angles 0° , -6° , and 6°].

Figure 7.6. Temperature distribution in 0° Yaw angle at 30ft vehicle separation distance [horizontal plane at fan center position].

Figure 7.7a. Radiator temperature in yaw 0° deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.7b. CAC temperature in yaw 0° deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.7c. Fuel condenser temperature in yaw 0° deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.8. Averaged temperature rise in heat exchangers of trailing vehicle at a constant fan speed; (i) cooling package, (ii) fuel condenser, (iii) CAC, and (iv) radiator [averaged values of three different yaw angles 0° and 6°].

Figure 7.9. Temperature distribution in 0° Yaw angle at 30ft vehicle separation distance [horizontal plane at fan center position].

Figure 7.10a. Radiator temperature in yaw 0° deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.10b. CAC temperature in yaw 0° deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.10c. Fuel condenser temperature in yaw 0° deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.11. Yaw averaged cooling air flow performance; (i) Leading vehicle and (ii) Trailing vehicle

Figure 7.12. Yaw averaged temperature rise over no-traffic vehicle in heat exchangers; (i) fuel condenser, (ii) CAC and (iii) radiator package [averaged values of three different yaw angles 0° , -6° , and 6°].

Figure 7.13a. Radiator temperature in yaw 0° deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.13b. CAC temperature in yaw 0° deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.13c. Fuel condenser temperature in yaw 0° deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.14. Yaw averaged cooling air flow performance; (i) Leading vehicle and (ii) Trailing vehicle

Figure 7.15. Yaw averaged temperature rise over no-traffic vehicle in heat exchangers; (i) fuel condenser, (ii) CAC and (iii) radiator package [averaged values of three different yaw angles 0° , -6° , and 6°].

Figure 7.16a. Radiator temperature in yaw 0° deg ; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.16b. CAC temperature in yaw 0° deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft.

Figure 7.16c. Fuel condenser temperature in yaw 0° deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 8.1. Medium-duty delivery truck

Figure 8.2. Vehicle platooning configuration top view, (i) single lane, (ii) vehicles are in same lane but two-lane traffic and (iii) vehicles are in two separate lanes.

Figure 8.3. Comparison of normalized aerodynamic drag for two different height clearance between vehicle and road tunnel.

Figure 8.4. Comparison of normalized aerodynamic drag along the vehicle length; (i) different road-tunnel height in one-lane and (ii) one- and two-lane for 1.23m height clearance between vehicle and road tunnel.

Figure 8.5. Aerodynamic drag performance of leading and trailing vehicles at different vehicle-to-vehicle (V2V) separation distances; (i) One lane (Figure 8.2i), (ii) Two lane: Leading and Trailing Vehicles are in same lane (Figure 8.2ii) and (iii) Two lane: Leading and Trailing Vehicles are in two separate lanes (Figure 8.2iii).

Figure 8.6. Velocity fields across the center of the truck height (1.27m) at separation distances of 30ft, 90ft and 150ft; (i) One lane, (ii) Two lane: Leading and Trailing Vehicles are in same lane and (iii) Two lane: Leading and Trailing Vehicles are in a separate lane.

Figure 8.7. Static pressure across the center of the truck height (1.27m) at different separation distances of 30ft, 90ft and 150ft; (i) One lane (Figure 8.2i), (ii) Two lane: Leading and Trailing Vehicles are in same lane (Figure 8.2ii) and (iii) Two lane: Leading and Trailing Vehicles are in two separate lanes (Figure 8.2iii).

Figure 8.8. Comparison of normalized aerodynamic drag along the trailing vehicle length for two different vehicle-to-vehicle (V2V) distances of 30 and 150 ft; (i) One lane (Figure 8.2i), (ii) Two lane: Leading and Trailing Vehicles are in same lane (Figure 8.2ii) and (iii) Two lane: Leading and Trailing Vehicles are in two separate lanes (Figure 8.2iii).

LIST OF TABLES

Table 1.1. Tested vehicle configurations

Table 1.2. Vehicle operating conditions

Table 1.3. Normalized aerodynamics drag comparison with respect to 0° yaw in each software

Table 2.1. Base Case Vs. Improved design

Table 3.1. Aerodynamic drag simulation operating conditions

Table 3.2a. Cab-over engine vehicle normalized aerodynamic drag

Table 3.2b. Conventional vehicle normalized aerodynamic drag

Table 3.3. Cab-over engine vehicle drag comparison with and without wheel covers

Table 3.4. Cab-over engine gap sensitivity normalized aerodynamic drag analysis

Table 3.5. Conventional and cab-over engine dimensions

Table 3.6. Conventional and cab-over engine calculated thermal performance data of CAC, radiator and fan

Table 4.1. Vehicle operating conditions

Table 4.2. Comparison of Aerodynamic drag Force at different yaw angles [Boat-tail plate width: 1.2m, Wind speed: 90 kph]

Table 5.1. Operating Conditions

Table 5.2. Heat exchanger porous modeling

Table 5.3. Heat exchanger cooling air mass outflow rate

Table 6.1. Aerodynamic drag simulation operating conditions

Table 6.2. Normalized aerodynamic drag of leading and trailing vehicles in a single lane

Table 6.3. Normalized aerodynamic drag of leading and trailing vehicles in multiple lanes

Table 6.4. Normalized aerodynamic drag of two leading and one trailing vehicles in multiple lanes [30 ft separation]. Drag values were normalized by no-traffic condition at 0° yaw.

Table 7.1. Operating conditions

Table 7.2. Heat exchanger porous modeling

Table 8.1. Road tunnel dimensions [**LV:** Leading Vehicle; **TV:** Trailing Vehicle]

Table 8.2. Vehicle operating conditions

Table 8.3. Blockage Ratio [vehicle cross sectional area: 9.478 m^2]

Table 8.4. Comparison of aerodynamic drag for fine and coarse grid

1. Aerodynamic Drag Optimization of Medium-duty Delivery Truck

Abstract

Aerodynamic analysis of the medium-duty delivery truck was performed to achieve vehicle design optimization. Three dimensional CFD simulations were carried out for several improved designs, with a detailed external component analysis of wheel covers, side skirts, roof fairings, and rounded trailer corners.

The overall averaged aerodynamics drag reduction through the design modifications were shown up to 22.3% through aerodynamic considerations alone, which is equivalent to 11.16% fuel savings. The main identified fuel efficiencies were based on second generation devices, including wheel covers, side skirts, roof fairings, and rounded trailer corners.

The important findings of this work were; (i) the optimum curvature radius of the rounded trailer edges found to be 125 mm, with an arc length of 196.3 mm, (ii) aerodynamic drag reduction increases with dropping clearance of side skirts between wheels and ground, and (iii) aerodynamic drag reduction increases with an extension of front bumper towards the ground.

1.1. Introduction

In automotive industry, aerodynamic efficiency considerations are vital to make fuel economy vehicles [1-3]. The Computational Fluid Dynamics (CFD) analysis offers a very less expensive and faster alternative to costly experiments. The commercial CFD simulation tools, such as Star-CCM+® and PowerFLOW®, are used to analyze and assess the influence of various design parameters on the aerodynamics drag vehicle performance. The optimized aerodynamic vehicle designs helps to reduce the fuel consumption and emissions.

Sirenko et al. [4] had shown higher fuel savings using rear screens and rear fairings for a light-duty General Motors SUV. Song et al. [5] performed aerodynamic optimization of a sedan for drag reduction using artificial neural networks and CFD approach. Similar to Song, Muyl et al. [6] performed aerodynamic external vehicle shape optimization by stochastic genetic algorithms and deterministic Broyden–Fletcher–Goldfarb–Shanno (BFGS) hill-climb approach. Al-Obaidi and Sun [7] performed numerical and experimental analysis of the radiator/cooling channel influence on drag in an open wheel race car. Ahmed and Chacko [8] had shown the aerodynamics drag reduction by modifying external surfaces of cars.

Lee [3,9] had shown the aerodynamics drag reduction devices and their implementation in heavy-duty truck configurations. Liao and Quail [10] performed numerical and experimental studies on a medium-duty tactical truck for fuel vehicle efficiency. Wood and Bauer [11] had shown low-cost aerodynamics drag reduction devices for tractor-trailers. Chowdhury et al.

[12] performed experimental investigation of semi-trailer truck by using aerodynamics drag reduction devices, such as side-skirts, front fairings, and gap fillings.

In this project, an aerodynamics analysis of medium-duty vehicle was performed to optimize the design. The following model design sequences were implemented to meet the aerodynamic targets as discussed in Driant at al. [13] and Vegendla et al. [1]: (i) conducting a detailed study to identify the efficient external vehicle designs, (ii) developing a numerical model for aerodynamic analysis, and (iii) redesigning the external vehicle surfaces that generates a significant drag to improve overall aerodynamics drag performance.

In this project, the base configuration of medium-duty delivery truck was studied with several modification designs to achieve optimized aerodynamics drag. Modeling and simulations were carried out using commercially available CFD analysis software for several combinations of the second generation drag reduction devices, including wheel covers, roof fairings, side skirts, and the avoidance of sharp trailer corners.

1.2. Vehicle Configurations

In Table 1.1 and Figure 1.1, aerodynamics drag analyses of six different medium-duty vehicle configurations, including a base configuration, was investigated. The base configuration has some built-in drawbacks in terms of fuel economy. The major external features with large impact on aerodynamics consist of cabin, trailer, mirrors, side extenders, tires, front bumper, axles, and cabin roof. The truck dimensional length, width, and height were considered to be 12 m, 2.55 m, and 3.72 m, respectively.

Figure 1.1. Vehicle configurations: (i) Base configuration, (ii) Case-1, (iii) Case-2, (iv) Case-3, (v) Case-4, and (vi) Case-5.

Table 1.1. Tested vehicle configurations

Run#	Model Details
Base configuration	Original medium-duty vehicle
Case -1	Base config. + wheel covers
Case -2	Base config. + side skirts
Case -3	Case -2 + no-klaxon + lowered front bumper
Case -4	Case -3 + optimized side extender and roof deflection
Case -5	Case -4 + aerodynamic mirrors + rounded corners

1.3. Modeling Details

As shown in Figure 1.1, six different configurations were analyzed with two separate commercially available CFD software packages.

StarCCM+®: Incompressible turbulent 3D isothermal steady-state simulations were implemented using a segregated-flow solver. The steady state simulations were selected since no significant changes observed in the aerodynamic drag results when comparing to the transient state results (not shown). The gas-phase turbulence is modeled using k- ϵ turbulence with standard parameters. The software uses the mass conserved finite volume discretized technique to solve the Reynolds Averaged Navier-Stokes (RANS) equations numerically. Partial Differential (PD) model equations of continuity, momentum, and turbulence can be found elsewhere [14].

PowerFLOW®: Incompressible turbulent 3D isothermal unsteady-state/transient simulations were implemented in a Lattice-Boltzmann solver [15]. The gas-phase turbulence is modeled using the state-of-the-art renormalization group form of the k- ϵ equations. This software uses the discrete particle approach, which means it mimics micro-physical process in a discrete manner rather than solving the governing differential equations.

Numerical simulations were performed in a computational domain with specified outlet, inlet, floor, and side wall surfaces in addition to the vehicle configuration. To capture air flow recirculation/vortex shredding at the trailer rear end, a very large computational domain was considered as shown in Vegendla et al. [1]. A pressure, velocity and periodic boundary conditions were imposed at outlet, inlet and side walls, respectively. The no-slip wall boundaries were imposed for the roof and floor.

Figure 1.2: Numerical setup [2].

As shown in Table 1.2, the following vehicle operating conditions were used to analyze the aerodynamics drag performances.

Table 1.2. Vehicle operating conditions

Vehicle operating conditions	
Inlet velocity	55 mph
Yaw angle	0 and 6°
Outlet pressure	1 atm
Computational domain [len x wid x heig]	100 m x 50 m x 30 m
Volume mesh	~32 mi

StarCCM+ uses an unstructured, cartesian trimmed hexahedral mesh for all configurations (Table 1.1). The maximum and minimum cell edges in the vehicle configurations were considered to be 40 mm and 3.5mm, respectively. For fine mesh size, volume controls were selected near the vehicle surroundings to limit the cell edge size of 40 mm. A coarse mesh size was allowed towards the computational domain boundaries with a surface growth rate of 1.3. The maximum cell edge was kept at 600 mm. Five prism layers with a thickness of 25 mm were used to provide an accurate estimation of the velocity near the wall in high Reynolds number external flows ($\sim 5.7E6$). Standard wall functions were used by keeping the y-plus value within an acceptable range from 20 to 200 [16].

In contrast, PowerFLOW uses a structured, cartesian cubic mesh (voxels) and a grid refinement scheme to equally and successively refine the grid size in each spatial dimension by a factor of two. The boundary surface was tessellated into triangular planes (surfels), of which each side has a length less than 1/10 the linear dimension of a voxel in the region.

1.4. Results

Aerodynamics drag analysis was performed for several combinations of modified configurations. The presented results were after thorough investigation of independent of grid/mesh as described in Appendix B.

The aim of this project was to find an optimized aerodynamic drag configuration for an existing medium-duty delivery truck. The optimized aerodynamic drag can be obtained by removing Klaxons, aligning the cabin roof with trailer height, modifying the sharp edges in trailer and mirrors by curvy edges and covering the underhood truck parts (e.g. engine) by adjusting front bumper to reduce air flow impact on underhood truck parts. To found an optimum curvature radius and arc length, three different trailer configurations were studied and analyzed to obtain final optimized aerodynamic drag configuration (Case-5). Also, external aerodynamic drag devices like side skirts and cabin side extenders further reduce the significant amount of aerodynamic drag as discussed in Vegendla et al. [2]. The detailed discussion can be found in the following sub sections for each configuration and its impact on aerodynamic drag values.

1.4.1. Base configuration

As shown in Figure 1.3, aerodynamics drag rise was observed with increasing yaw angle due to high vehicle surface pressures. This can be justified in Figure 1.4, where maximum air velocities hit at the trailer rear tires, leading to higher surface pressure and aerodynamics drag values in 6° yaw angle. It should be noted that the aerodynamics drag values were normalized by 0° yaw. In Figures 1.3 and 1.5, the main rise of aerodynamics drag in base configuration was mainly due to four following reasons: (i) high pressure impact on a front cabin, (ii) recirculation zone in a cabin-trailer gap, (iii) high velocity impact on the tires, and (iv) recirculation zone at trailer rear position. The major contribution of aerodynamic drag values are from pressure drag ($\sim 95\%$) and shear drag ($\sim 5\%$).

Figure 1.3. Cumulative normalized aerodynamics drag along the vehicle length.

Figure 1.4. Velocity fields at (i) 0° yaw and (ii) 6° yaw.

Figure 1.5. Static pressure distribution at (i) 0° yaw and (ii) 6° yaw.

Table 1.3 shows that the predicted aerodynamics drag values were found to be comparable in both StarCCM+® and PowerFLOW®. The normalized aerodynamics drag values were based on the 0° yaw in each type of software.

Table 1.3. Normalized aerodynamics drag comparison with respect to 0° yaw in each software

Yaw angle [deg]	StarCCM+- v9.06.011	PowerFLOW- 4.4d
0	1.00	1.00
6	1.11	1.10

1.4.2. Case-1 (Wheel Covers) Vs. Base Configuration

As shown in Figure 1.6, 2% and 2.8% aerodynamics drag improvement (reduction) was predicted with wheel covers when compared to base configuration in 0° and 6° yaw angles, respectively. This is primarily because air flow velocity tends to deflect away from the drive and trailer wheels, as shown in Figure 1.7.

Figure 1.6. Cumulative normalized aerodynamics drag along the vehicle length using wheel covers (i) Yaw 0° and (ii) Yaw 6°.

Figure 1.7. Underbody velocity vector (i) base configuration and (ii) wheel covers (Case-2) of medium-duty vehicle in 0° yaw angle

1.4.3. Case-2 (Side Skirts) Vs. Base Configuration

As shown in Figure 1.8, 3% and 8.6% aerodynamics drag improvement (reduction) was predicted with side skirts, when compared to base configuration in 0° and 6° yaw angles, respectively. Similar observations were found in Ross et al. [17]. They had shown the overall Yaw Averaged Aerodynamics Drag (YAD) reduction was up to 7.3% in Class-8 trucks. It is

noted that YAD is the average of 0° and 6° yaw angles (Appendix A). Also, as shown in Figure 1.9, side skirts help in deflecting high velocity flow away from the vehicle underbody, which will drastically reduce the aerodynamics drag.

Figure 1.8. Cumulative normalized aerodynamics drag along the vehicle length using side skirts.

(i) No side skirts

(ii) Side skirts

Figure 1.9. Velocity streamlines of medium-duty vehicle (i) no side skirts and (ii) side skirts [yaw 6°].

1.4.4. Case-3 (Side Skirts + Lowered Bumper) Vs. Case-2 (Side Skirts)

As shown in Figure 1.10, the higher aerodynamics drag was observed at front cabin due to lowered front bumper, as in Case-3 (Figure 1.1). On the other hand, the observed lower aerodynamics drag was due to removal of klaxon, reduced pressure on front axle, and reduced pressure on the drive tires and trailer tires. Figure 1.11 shows the lowered front bumper reduces the high velocity flows entering the underbody and leads to a lower static pressure on front axles and tires. It is known fact that with

Figure 1.10. Cumulative normalized aerodynamics drag along the vehicle length.

Figure 1.11. Velocity stream lines: (i) base configuration and (ii) lowered bumper.

1.4.5. Case-4 (Side Skirts + Lowered Bumper + Roof Fairings) Vs. Case-3 (Side Skirts + Lowered Bumper)

As shown in Figures 1.12 and 1.13, the aerodynamics drag improvement was observed using cabin roof fairings due to smooth air velocities. As shown in Figure 1.13, cabin roof streamlines were smooth enough to flow off the roof due to absence of klaxon and the

modified roof fairing. The observed flow regions were smaller than in the base configuration. The modified side extenders and roof fairing reduced stagnation pressure on trailers corners, leading to lower aerodynamics drag.

Figure 1.12. Cumulative normalized aerodynamics drag along the vehicle length: (i) 0° yaw and (ii) 6° yaw.

Figure 1.13. Velocity stream lines: (i) base configuration and (ii) optimized roof.

1.4.6. Case-5 (Side Skirts + Lowered Bumper + Roof Fairings + Rounded Corners) Vs. Case-4 (Side Skirts + Lowered Bumper +Roof Fairings)

As shown in Figure 1.14, a 2.1% increase in fuel economy was predicted with rounded trailer rear side corners alone, when compared to base configuration in 0° yaw angle. It was noted that the fuel economy is assumed to be half of the drag reduction. As shown in Figure 1.15, a lower turbulence was observed with rounded trailer rear side corners, which reduce the aerodynamics drag by about 4.2%. The optimum curvature radius of the rounded trailer rear side corners was found to be 125 mm with an arc length of 196.3 cm.

Figure 1.14. Cumulative normalized aerodynamics drag along the vehicle length using rounded corners [yaw 0°].

Figure 1.15. Velocity streamlines of medium-duty vehicle (i) sharp corners and (ii) rounded corners [yaw 0°].

Figure 1.16 shows that the significant amount of drag reduction was observed in a rounded trailer corner configuration (see Figure 1.1 for Case-5) when compared to sharp corners (see Figure 1.1 for Case-4). Lower turbulence predictions seen in rounded trailer corners are the main factor in this drag reduction shown in Figure 1.15. Rounded mirrors also generate lower turbulence, as shown in Figure 1.17.

Figure 1.16. Cumulative normalized aerodynamic drag along the vehicle length: (i) 0° yaw and (ii) 6° yaw.

Figure 1.17. Velocity streamlines of medium-duty vehicle (i) original mirrors and (ii) rounded mirrors [yaw 0°].

1.4.7. Optimized Design Aerodynamics Drag

As shown in Figure 1.18, a total of 22.3% Yaw Averaged Aerodynamics Drag (YAD) reduction was observed in final optimized design (see Figure 1.1 for Case-5). In Table 1.1, the modification of the each run is specified. It is noted that YAD is the average of 0° and 6° yaw angles. In addition, 2.37% YAD reduction was observed in the wheel cover configuration. The optimized design consists of the cumulative effect of side skirts, lowered front cabin bumper, absent klaxon, modified roof fairings, and rounded trailer and mirror corners.

Figure 1.18. Comparison of YAD reduction with a base case for each configuration.

1.5. Conclusions

Aerodynamic drag analysis was performed for several modified medium-duty vehicle (delivery truck) configurations. A significant fuel saving potential was observed with wheel covers, side skirts, modified roof, and rounded trailer edges and mirrors. The overall YAD reduction was up to 23.3%, which is equivalent to 11.16% fuel saving with an optimized design (Case-5).

YAD reduction based on individual contribution of wheel covers, side skirts, lowered front bumper, roof fairings and rounded edges trailer were 2.37%, 6.21%, 1.12%, 3.96%, and 11.04%, respectively. The optimum curvature radius of the rounded trailer edges were found to be 125 mm, with an arc length of 196.3 mm. In side skirts configuration (Case-2), higher aerodynamic drag benefit was observed with lower clearance of side skirts between wheels and ground. In lowered front bumper configuration (Case-3), higher aerodynamic drag benefit was observed by covering the entire front-side view of the vehicle underhood components. In modified roof fairing and side extenders configuration (Case-4), aerodynamic drag benefit was observed by covering the entire front-side view of the trailer.

2. Aerodynamic Drag Optimization of M3 Medium-duty Truck

Abstract

In this work, external aerodrag analysis and underhood cooling simulations were carried out using STAR-CCM+V8[®] commercial software.

A significant reduced aerodrag is observed with modified roof, trailer box and side extenders and side skirts. The performed analysis shows side deflectors increases the drag by 1.5%. 10% drag reduced with modified roof, which is aligned to the height of the trailer box. 5% reduced drag observed with flat bed and smooth surface trailer box. 10% drag reduced with rounded smooth corners in a trailer box. The overall improved design saves 14% of the fuel as compared to the base case.

2.1. Vehicle Configuration Studied

In addition to 9600 medium-duty delivery truck in Chapter 1, several simulations were performed to obtain optimized aerodynamic drag. The provided base case configuration consists of several drawbacks in terms of fuel economy.

M3 Truck

Figure 2.1. Base Case truck configurations

As seen Figure 2.1, the main external truck configurations consist of cabin, trailer, mirrors, side extenders, tires, front bumper, axles and cabin roof.

Numerical experiments were performed in a wind tunnel with a specified inlet, outlet, floor and wall surfaces in addition to the truck configuration as described in Section 1.3. To

capture vortex at trailer rear end wind tunnel considered with dimensions of 90m x 40 m x 30m as similar to Figure 1.2.

2.2. Boundary Conditions

As seen in Figure 2.1 and Table 1.1, pressure is imposed at outlet boundary. Periodic boundary conditions are applied for the wind tunnel side walls. Air flow velocity specified at inlet. Also, no slip at the wind-tunnel floor and roof are imposed.

2.3. CAD models and Mesh

The main challenge was to import the base case Computer Aided Design (CAD) geometries and cleaning the solid surfaces to generate the good quality mesh. The quality mesh provides a reliable numerical solution for a specified problem specification. The solid surface cleaning consists of mainly to closes the open surfaces with surface wrapping operation in standard meshing tools. Later, the surface remeshing was performed before generating control volume mesh in the wind tunnel.

For aerodynamic drag wind tunnel simulations Polyhedral mesh was generated. The minimum cell edge in undehood was used 3 mm and maximum size was 45 mm. Two prism layers were considered with thickness of 4.5mm. Residuals were dropped three orders of magnitude using an underrelaxation factor for velocity components, pressure, energy and fan are considered 0.7, 0.3, 0.9 and 0.3, respectively.

2.4. Results and Discussion

Aerodrag analysis performed for several combination of modified configuration as shown in Table 2.1.

As seen in Figure 2.2, 14% fuel economy predicted with improved design as compared to the base case in a Yaw angle of 0° . Note, the fuel economy is assumed to be half of the aerodynamic drag reduction. The improved design consists of flat bed and trailer, modified cabin roof, side extenders and rounded trailer corners (see Table 2.1). The sharp drag rise at trailer end (8.5m) is mainly due to vortex region rear to the trailer. In improved design, the sharp drag rise between cabin and trailer is due to modified roof which aligned with the trailer height.

Figure 2.2. Drag coefficient along the length of the M3 truck

Table 2.1. Base Case Vs. Improved design

Base Case	Improved Design
Rough surface Bed & Trailer	Flat Bed & Trailer
Base case roof extender	Modified roof extender
No side extenders	Side extenders
Sharp corners trailer box	Rounded corners trailer box

2.5. Conclusions

A significant reduced aerodrag was observed with modified roof, trailer box, side extenders and side skirts. The performed analysis shows 10% drag reduced with modified roof, which was aligned to the height of the trailer box. 5% reduced drag observed with flat bed and smooth surface trailer box. 10% drag reduced with rounded smooth corners in a trailer box. The overall improved design saves 14% of the fuel as compared to the base case.

3. Comparison of Aerodynamic Drag and Underhood Thermal Analysis of Two Heavy-Duty Vehicles

Abstract

External aerodynamics and underhood thermal analyses of two heavy-duty vehicles, (i) cab-over engine (European style flat-front) and (ii) conventional truck (North American class-8-style front engine), were being studied to assess their aerodynamic and underhood cooling performances. Both vehicles have similar geometrical dimensions, i.e. width, height, and length.

3D Computational Fluid Dynamics (CFD) modeling and simulations were performed using StarCCM+® commercial software. Aerodynamic drag analysis was carried out separately for each configuration. Significant fuel savings were observed in a conventional truck when compared to cab-over engine configurations at all yaw angles. The major reduction in fuel consumption is observed mainly from the cabin designs alone (up to 7.5%).

Apart from the aerodynamic drag analysis, underhood thermal analysis was also performed to compare both heavy-duty vehicle configurations. Based on coolant temperatures, 12% and 10% thermal efficiency improvements were observed in conventional trucks when compared to cab-over engine for charge air cooler (CAC) and radiator, respectively.

3.1. Introduction

In the automotive industry, optimization of a vehicle's aerodynamic efficiency and thermal management of an engine cooling system and underhood components is highly desirable to achieve durable and fuel efficient designs that meet today's energy demand [1,9,13,18,19]. The CFD analysis offers an inexpensive and fast alternative to experiments. Commercially available CFD simulation software such as StarCCM+® can be used to investigate and assess the various factors that affect the aerodynamic drag and underhood thermal vehicle performance [18]. The aerodynamic and thermal optimization of vehicles can reduce fuel consumption and vehicle emission while improving vehicle durability.

Aerodynamic analysis of two optimized heavy-duty vehicle models were performed and compared in this study. These two vehicle models were previously optimized [18-19] using the following design and modeling sequences: (i) a comprehensive study to determine efficient design of the external body of the vehicle, (ii) development of a model of the vehicle for aerodynamic studies, (iii) locating the components of the vehicle that are generating the significant drag, (iv) modifying the components to improve the overall drag coefficient of the vehicle, (v) simulating the vehicle in multiple configurations, and (vi) conducting a complete set of aerodynamic drag simulations to improve the drag coefficient with all modified components [13].

In this project, different configurations were modeled for both cab-over engine and conventional truck models and simulated separately. The modeled configurations included cabin only, cabin plus flatbed, and cabin plus trailer (base case) configurations including or

excluding side skirts. In addition, wheel cover and closed cabin-grill simulations were performed for the base case configuration (cabin plus trailer). An aerodynamic drag sensitivity analysis for the cabin-trailer gap was also performed for the base case / full-vehicle simulations.

Apart from the aerodynamic drag analysis, an underhood thermal efficiency assessment was performed for both heavy-duty vehicle models to evaluate charge air cooler (CAC), radiator, and fan performance under similar operating conditions.

3.2. Vehicle configurations studied

The conventional and cab-over engine truck configurations for which the aerodynamic drag performance assessments were made are shown in Figure 3.1. There are two significant differences between conventional and cab-over engine models, the front cabin design and the larger cabin-trailer gap in the cab-over engine model.

(i) Conventional heavy-duty vehicle: a) cabin, b) cabin plus flatbed, c) base case, and d) base case plus side skirts

(ii) Cab-over engine heavy-duty vehicle: a) cabin, b) cabin plus flatbed, c) base case and d) base case plus side skirts

Figure 3.1. Heavy-duty vehicle configuration of (i) conventional (G7) and (ii) cab-over engine (M7)

3.3. Model Development

The configurations in Figure 3.1 and their variants were analyzed in StarCCM+ CFD software, which uses the Finite Volume (FV) discretized method to solve the Navier-Stokes equations numerically for fluid flow [14].

Three dimensional incompressible, turbulent, isothermal, and steady-state simulations were performed using segregated-flow solver for aerodynamic drag analysis. The air turbulence was modeled using k- ϵ model with standard parameters. A similar approach was applied for underhood thermal simulations except non-isothermal condition was implemented.

Numerical studies were performed for a wind tunnel configuration with a specified inlet, outlet, and floor and wall surfaces surrounding the vehicle. To capture vortex shedding behind the trailer, a large wind tunnel was modeled with dimensions of 90 m x 40 m x 30 m. The recommended practice guidance for the use of CFD in evaluating the commercial vehicle aerodynamic performance can be found elsewhere [20].

Figure 3.2. Wind tunnel numerical setup

A pressure boundary condition was specified at the outlet, and periodic boundary conditions were applied for the wind tunnel side walls to mimic the conditions at non-zero yaw angles. An air flow velocity was specified at the inlet, and no-slip boundary conditions were imposed at the wind tunnel floor and roof.

As shown in Table 3.1, the following geometry and operating conditions were considered to simulate the two vehicles.

Table 3.1. Aerodynamic drag simulation operating conditions

Operating conditions	
Air velocity inlet [mph]	55
Yaw angle [deg]	0, 6 and 12
Outlet pressure [atm]	1
Wind tunnel dimensions [m ³] [Length x Width x Height]	90 x 40 x 30
Number of polyhedral cells [millions]	30

In numerical simulations, the main challenge is generation of a good quality mesh [1]. This is done through creating a quality model by importing the base case Computer Aided Design (CAD) geometries as well as cleaning the solid surfaces. The quality mesh is vital to get a reliable numerical solution for a given vehicle configuration. In this study, the surface wrapping operation of StarCCM+ meshing tool was used to close the open surfaces. Later, surface re-meshing was performed before generating a volume mesh in the wind tunnel.

The polyhedral topology mesh was used in all vehicle configurations. The minimum and maximum cell edges in the vehicle configuration were 3 mm and 45 mm, respectively. A volumetric control block was used to control the mesh size around the vehicle geometry. This produces a finer mesh with a fairly constant cell size, helping to prevent numerical instabilities. This produces a maximum cell edge size of 60mm in the volumetric control region. Four prism layers with thickness of 20 mm were modeled to capture the turbulence near the vehicle surface. Standard wall functions were used by keeping the y-plus value within an acceptable range from 20 to 200 [16]. The overall residuals were dropped by three orders of magnitude using an under-relaxation factor for velocity, pressure, and turbulence parameters (k and ϵ) considering 0.7, 0.3, 0.8, and 1, respectively.

3.4. Results and discussion

3.4.1. Aerodynamic analysis

Grid/mesh convergence study was performed for a cab-over engine full vehicle configuration considering both coarse and fine mesh across the computational domain with 30 and 40 million volume cells, respectively. The observed changes for the aerodynamics drag values were negligible ($\pm 0.05\%$) for both coarse and fine mesh.

Aerodynamic drag analyses were performed and compared for the following conventional and cab-over engine heavy-duty vehicle configurations: (i) cabin only; (ii) cabin plus flatbed, with or without side skirts; (iii) base case, with or without side skirts; (iv) base case, with or without wheel covers. In addition, cabin-trailer gap sensitivity analyses were performed.

Significant Yaw Averaged Aerodynamic Drag (YAD) reduction was observed for all the above mentioned designs in a conventional truck when compared to a cab-over engine

configuration, as shown in Figure 3.3(i). YAD was based on the average value of three different yaw angles, 0° , 6° , and 12° , as discussed in Appendix A.1. In a conventional truck, fuel savings were observed to be up to 9% in the base vehicle (Figure 3.3(ii)). Fuel savings are roughly half of the YAD reduction (Appendix A.2). The details of each vehicle configuration are discussed in subsequent sections. All aerodynamic drag results were normalized based on cab-over engine base case at 0° yaw angle.

(i) YAD

(ii) Fuel savings

Figure 3.3. Conventional truck performance over cab-over engine configuration (i) YAD and (ii) fuel savings

3.4.2. Cabin only

As shown in Figure 3.4, cab-over engine model drag coefficient was higher than that for conventional model at all yaw angles due to the front cabin design. As shown in Figure 3.5, the blunt frontal face of the cab-over engine model was the main contributor to the drag for the cabin only model. Since the fuel economy is roughly half of the aerodynamic drag reduction, this makes a significant difference in fuel efficiency. All aerodynamic drag results were normalized based on cab-over engine base case at 0° yaw angle. Note, C_D is normalized cumulative aerodynamics drag coefficient.

Figure 3.4. Normalized drag comparison of conventional and cab-over engine cabin over different yaw angles

Figure 3.5. Normalized drag coefficient along the length of the (i) conventional and (ii) cab-over engine cabin

3.4.3. Cabin plus flatbed, with or without side skirts

Figure 3.6 shows that aerodynamic drag was consistently reduced with flatbed side skirts in both conventional and cab-over engine cabin plus flatbed simulations at all yaw angles.

Figure 3.6. Normalized drag comparison of conventional and cab-over engine cabin-flatbed, over different yaw angles

Figure 3.7 illustrates the smooth air flow observed at the front side of the cabin in the conventional truck. This was due to the North-American class-8 front cabin design, which is different from the cab-over engine, European flat-type cabin design. In a cab-over engine cabin, a large vortex zone was observed at the front side, due to the flat-type design. This large vortex zone significantly increases the aerodynamic drag.

Figure 3.7. Velocity field in central vertical plane of (i) cab-over engine and (ii) conventional truck [yaw: 0°]

As shown in Figures 3.8-3.10, the cabin was the major contributor to aerodynamic drag in both heavy-duty vehicles. In both conventional and cab-over engine models, positive pressure coefficients were mainly in the frontal areas. The positive pressure coefficients contributed to aerodynamic drag and are shown in the figures as red zones. These red zones are mainly due to the impact of air velocity on external vehicle surfaces. At all yaw angles, the rear-axle wheels also contributed significantly to aerodynamic drag, as shown in Figures 3.8 and 3.9. In Figure 3.10 (i) and (iii), the rear-axle wheels showed significant drag increases at 6° yaw in comparison to 0° yaw angle case. This justifies spreading the high pressure coefficient zone at the rear-axle wheels in Figure 3.8(ii) and 3.9(ii).

Figure 3.8. Cab-over engine pressure coefficient contours (i) yaw: 0° and (ii) yaw: 6°.

Figure 3.9. Conventional truck pressure coefficient contours (i) yaw: 0° and (ii) yaw: 6°.

As shown in Figure 3.10 (ii) and (iv), aerodynamic drag reduction was observed with underbody side skirts as compared to the base case in 6° yaw. This reduction was caused by the deflection of air flow away from the flatbed underbody with side skirts, as illustrated in the comparisons in Figures 3.11 and 3.12.

Figure 3.10. Normalized drag coefficient along the length of the conventional and cab-over engine cabin-flatbed vehicles using side skirts

Figure 3.11. Cab-over engine underbody velocity field in horizontal plane at center of the wheels (i) without side skirts and (ii) with side skirts [yaw: 6°]

Figure 3.12. Conventional underbody velocity field in horizontal plane at center of the wheels (i) without side skirts and (ii) with side skirts [yaw: 6°]

3.4.4. Base case, with or without side skirts

Aerodynamic drag reduction was observed with trailer side-skirts for the base configurations at all yaw angles, as shown in Figure 3.13. Again, this was mainly due to deflection of air flow away from the trailer underbody, and similar observations were found in cabin plus flatbed configurations.

Figure 3.13 Normalized drag comparison of conventional and cab-over engine base case (cabin plus trailer) over different yaw angles

As shown in Figure 3.14, a sizable aerodynamic drag increase was observed in the cabin-trailer gap and the rear-side of trailer in both conventional and cab-over engine models. This rise was due to air recirculation in those regions, as highlighted in Figure 3.15.

Figure 3.14. Normalized drag coefficient along the length of the conventional and cab-over engine vehicle (i) conventional and (ii) cab-over engine

Figure 3.15 shows two significant vortex zones identified in the cabin-trailer gap and at the trailer rear side in the base case / full-vehicle simulations. In these two vortex zones, a steep rise in the aerodynamic drag was observed, as shown in Figure 3.14.

Figure 3.15. Velocity field in central vertical plane (i) cab-over engine base case and (ii) conventional base case [yaw: 0°]

As shown in Figure 3.16, high pressure coefficient values were observed at the front cabin, at the trailer front, and at rear-axle wheels. Also, the high pressure coefficient zone spread with higher yaw angles.

Figure 3.16. Conventional truck pressure coefficient contours (i) yaw: 0° without side-skirts, (ii) yaw: 6° without side-skirts, and (iii) yaw: 6° with side skirts.

As shown in Table 3.2, aerodynamic drag value increased with yaw angles in both the (a) cab-over engine and (b) conventional heavy-duty vehicles.

Table 3.2a. Cab-over engine vehicle normalized aerodynamic drag

	0°	6°	12°	Ref. Area [m ²]
Base case	1.0	1.369	1.879	10
Base case with side skirts	0.963	1.233	1.721	10
Cabin	0.955	0.996	1.094	10
Cabin plus flatbed	0.936	1.044	1.263	10
Cabin plus flatbed with side skirts	0.932	1.014	1.159	10

Table 3.2b. Conventional vehicle normalized aerodynamic drag

	0°	6°	12°	Ref. Area [m ²]
Base case	0.853	1.227	1.413	10
Base case with side skirts	0.824	1.063	1.284	10
Cabin	0.747	0.796	0.862	10
Cabin plus flatbed	0.793	0.947	1.096	10
Cabin plus flatbed with side skirts	0.728	0.856	1.001	10
Closed cabin-grill plus base case	0.849	1.197	1.401	10

3.4.5. Base case, with and without wheel covers

As shown in Figure 3.17, a 1.1% decrease in aerodynamic drag coefficient was observed with wheel covers as compared to the base case configuration at 6° yaw angle. In contrast, fuel efficiency dropped by 0.5% with wheel covers at yaw angle of 0°. The fuel efficiency drop was due to deflected air flow from drive-axle wheels, which directly hit the trailer rear-axle wheels as highlighted in Figure 3.18(ii). Note: the base case does not include the wheel covers.

Figure 3.17. Normalized drag coefficient along the length of the conventional vehicle using wheel covers (i) yaw: 0° and (ii) yaw: 6°

Figure 3.18. Conventional truck underbody velocity vector in horizontal plane at center of the rear wheel axles (i) base case [yaw: 0°] and (ii) wheel covers [yaw: 0°]

Overall drag was seen to increase with wheel covers at 0° and 6° yaw angles as compared to models without wheel covers simulations, as shown in Figure 3.19. On the other hand, drag reduced at 12° yaw (Table 3.3). At all yaw angles, aerodynamic drag dropped in drive-axle wheels but immediately increased in the wheel gaps at rear-drive axles.

Figure 3.19. Normalized drag coefficient along the length of the cab-over engine vehicle using wheel covers (i) yaw: 0° and (ii) yaw: 6°

Table 3.3. Cab-over engine vehicle drag comparison with and without wheel covers

	0°	6°	12°
Wheel Covers	1.0	1.369	1.879
No Wheel Covers	0.966	1.338	1.883
% drag rise	3.4	2.3	-0.1

3.4.6. Cabin-trailer gap sensitivity aerodynamic drag analysis

Figure 3.20. Cab-over engine gap sensitivity analysis between cabin and trailer (i) large gap (ii) reduced gap.

Figure 3.20 shows the two different simulations performed to analyze cabin-trailer gap sensitivity on the aerodynamic drag performance.

From Table 3.4 and Figures 3.21 and 3.22, a significant drop in aerodynamic drag was observed with reduced cabin-trailer gap at all yaw angles. In the cab-over engine model,

aerodynamic drag slope decreased with reduced gap as compared to large gap between cabin and trailer. On the other hand, slope increase in cab-over engine model could have been due to the difference in the trailer bed, floor clearance, and cabin side skirts.

Figure 3.21. Cab-over engine and conventional cabin-trailer gap sensitivity on normalized aerodynamic drag over different yaw angles

(i) Yaw: 0°

(ii) Yaw: 6°

Figure 3.22. Cabin-trailer gap sensitivity on normalized aerodynamic drag along the length of the cab-over engine vehicle (i) yaw: 0° and (ii) yaw: 6°

Table 3.4. Cab-over engine gap sensitivity normalized aerodynamic drag analysis

	0°	6°	12°
1.36 m cabin-trailer gap	1.0	1.369	1.879
0.563 m cabin-trailer gap	0.936	1.214	1.518
% drag drop	6.3	11.3	19.24

As shown in Table 3.5, cab-over engine and conventional base case vehicle dimensions are comparable but a deviation in cabin-trailer gap is present.

Table 3.5. Conventional and cab-over engine dimensions

	Cab-over engine, large gap (base case)	Cab-over engine, reduced gap	Conventional, base case
Box Length [m]	12.43	13.367	12.57
Cabin & box clearance [m]	1.36	0.563	0.563
Cabin height [m]	3.828	3.828	3.686
Cabin Length [m]	6.964	6.964	7.08
Cabin alone [m]	2.59	2.59	4.08

3.5. Underhood thermal analysis

As shown in Figures 3.1 (i)a and (ii)a, conventional and cab-over engine cabin geometries were simulated to analyze the thermal performance of the two vehicles. The main underhood components in these geometries are charge air cooler (CAC), radiator, fan, and engine.

As shown in Figure 3.23, significant differences were seen in thermal performances for both conventional and cab-over engine configurations. In this study, both thermal simulations were carried out at an ambient temperature of 27.3 °C, an inlet velocity of 20 mph, and constant heat rejection values of CAC and radiator, 48.5 kW and 110 kW, respectively. In both simulations, fan speed was considered to be 1400 rpm. The CAC and radiator were modeled using a single-duty heat exchanger with constant inertial and viscous porous resistances in both conventional and cab-over engine models. In these simulations, inertial and viscous resistance values were considered to be 67 kg/m⁴ and 415 kg/m³.s⁻¹, respectively.

As shown in Table 3.6 and Figure 3.23, an overall thermal efficiency of 12% was observed in conventional CAC as compared to cab-over engine truck. Also, a 10% increase in efficiency was observed in a conventional truck radiator. Smoother and more uniform air passage through underhood, which leads to higher mass flow rates through both CAC and radiator, was seen as the main cause of this increase. Also, high mass flow rates were driven by the fan, as shown in Table 3.6. Note: thermal efficiencies were calculated based on the average temperature of the CAC and radiator.

Table 3.6. Conventional and cab-over engine calculated thermal performance data of CAC, radiator and fan

	Conventional		Cab-over engine	
	Avg. temperature [°C]	Mass Flow rate [kg/s]	Avg. temperature [°C]	Mass Flow rate [kg/s]
CAC	45.08	1.773	50.38	1.398
Radiator	74.90	2.468	82.49	2.409
Fan	-	3.198	-	2.626

Figure 3.23. Thermal performance comparison of conventional and cab-over engine (i) front side of conventional truck-CAC, (ii) front side of cab-over engine-CAC, (iii) rear side of conventional truck-radiator, and (iv) rear side of cab-over engine-radiator.

Velocity profiles were found to be more uniform in the conventional truck than in the cab-over engine, as shown in Figure 3.24. This may have been due to the following three reasons: (i) differences in cabin-grill openings in both models, (ii) off-centered fan location in cab-over engine, and (iii) air flow restriction due to engine in cab-over engine. In CAC models, low velocity zones are mainly due to support material of the cabin-grill, which obstructs the air flow. These low velocities can lead to high temperatures as shown in Figure 3.23 (i) and (ii).

Figure 3.24. Comparison of velocity in conventional and cab-over engine cooling packages (i) front side of conventional truck-CAC, (ii) front side of cab-over engine-CAC, (iii) Rear side of conventional truck-radiator, and (iv) rear side of cab-over engine-radiator

As shown in Figure 3.25(iii) and (iv), underhood air flow was highly streamlined in conventional truck models when compared to cab-over engine model. This increased the thermal efficiencies of the cooling package, i.e. CAC and radiator. Similar observations were also found in the vertical plane velocity fields (Figure 3.25(i) and (iii)).

Figure 3.25. Comparison of underbody velocity field in conventional truck and cab-over engine: (i) conventional truck: fan centered horizontal plane, (ii) cab-over engine: fan centered horizontal plane, (iii) conventional truck: fan centered vertical plane, and (iv) cab-over engine: fan centered vertical plane

3.6. Conclusions

Aerodynamic drag analyses were performed for two heavy-duty vehicles, namely conventional and cab-over engine. Significant fuel savings were observed in a conventional truck when compared to cab-over engine configurations at all yaw angles. The major reduction in fuel consumption was observed to come from the cabin designs alone.

In both vehicles, aerodynamic drag performance was improved with side skirts at all yaw angles due to deflection of air flow away from the underbody. In a closed cabin-grill simulation, reduced aerodynamic drag was observed due to reduced air flow through the underbody.

In the analysis of wheel cover configuration, a drop in aerodynamic drag was observed at the drive wheels; however, overall drag increased due to the rear wheels. Cab-over engine gap sensitivity aerodynamic drag analysis showed significant aerodynamic drag reduction when the cabin-trailer gap was decreased.

In underhood thermal simulations, better cooling performance was obtained for conventional trucks in CAC and radiator when compared to cab-over engine truck. This was mainly due to less restrictive underbody air flow in conventional truck model.

4. Boat-Tail Influence on Tractor-Trailer Aerodynamic Drag

Abstract

The boat-tail reduces the aerodynamic drag upto 12%. The major influence on the aerodynamics drag is due to slant angle and width of the plates. In this project, the aerodynamics optimization was performed for several boat-tail configurations. The Optimized plate width found to be 1.2m with side-skirts configuration. On the other hand the optimized plate width was found to be 0.6m without side-skirts.

4.1. Introduction

Several researchers reported fuel savings were upto 7.5% using boat-tails in heavy-duty trucks. The improved aerodynamics was dependent on the length of the boat tail. Wind-tunnel and road tests have demonstrated that a boat tail with a length of 24 to 32 inches is optimal for drag reduction purposes and typical length restrictions. As with side skirts, the interaction of boat tails with other devices is important for optimization (<https://www.tc.gc.ca/eng/programs/environment-etv-menu-eng-2939.html>). Van Leeuwen (2009) found that the optimized slant angle was 10° and it represents the boat tail geometry where the flow stays attached to the boat tail. Further increasing slant angle will cause the flow to separate from the boat tail.

The aerodynamic drag reduction drops significantly by reducing vehicle speed [11]. The relationship between aerodynamic drag reduction, fuel economy, and vehicle speed were shown in Wood and Bauer [11], Hoerner [21], Himeno et al. [22], Lachman [23].

Most of the published literature on boat-tails is for heavy-duty vehicles. However, the optimized parameters like slant angle and boat-tail plate width may valid for medium-duty trucks but there is no evidence. In this project, we performed several numerical simulations for aerodynamic optimization of slant angle, boat-tail and vehicle/wind-speed.

4.2. Modeling

Figure 4.1. Medium-duty delivery truck (i) no side-skirts and boat-tail (base case) and (ii) side-skirts boat-tail.

As seen in Figure 4.1, medium-duty delivery truck considered for investigation of aerodynamic drag influence using boat-tails. The medium-duty delivery truck length, width and height are 12 m, 2.55m and 3.77, respectively.

Figure 4.2. Boat tail plate width; (i) 0.3 m , (ii) 0.6m and (iii) 1.2m

As seen in Figure 4.2, three different boat-tail plate width configurations were studied for various slant angles of 10° , 15° and 25° .

As seen in Table 4.1, modeling and simulations were performed for a given computational domain dimensions [Length x width x height] and operating conditions. Air velocity and pressure were specified at inlet and outlet-boundaries, respectively. Periodic boundary condition applied for side walls. No-slip wall conditions were applied for the rest of the wall boundaries, i.e. floor and roof.

Table 4.1. Vehicle operating conditions

Operating conditions	
Air Inlet velocity [kph]	70, 90 and 110
Yaw angle [deg]	0-12
Outlet pressure [Pa]	0
Turbulent intensity [-]	0.05
Number of trimmer cells [millions]	20 to 30
Computational Domain [L x W x H], m ³	80 x 40 x 30

The presented results are grid independent. Two different grids were studied to justify the grid independency on aerodynamic drag results for no-traffic vehicle. The predicted aerodynamic drag values are consistent in both fine and coarse mesh with marginal deviations ($<0.1\%$). Also, there are no differences in steady-state and transient and RNG k-epsilon and k-omega.

4.3. Results

Modeling and simulations were performed to analyze the boat-tail influence on aerodynamic drag. The following combinations of boat-tail configurations were studied; (i) with and without side-skirts, (ii) Boat-tail plate width of 0.3, 0.6 and 1.2m , (iii) Boat-tail slant angle of 10° and 15° and (iv) wind speed of 70, 90 and 110 kph.

As seen in Figure 4.3, higher aerodynamic drag benefit was observed with side-skirts when compared to no side-skirts configuration. Also, higher benefit observed when using lower boat-tail plate towards the ground compared to no lower boat-tail plate (Figure 4.4). The Optimized plate width found to be 1.2m with side-skirts configuration (Figure 4.3i). On the other hand the optimized plate width was found to be 0.6m without side-skirts (Figure 4.3ii).

Figure 4.3. Aerodynamic drag for different boat-tail plate width for wind velocity of 90 kph and slant angle of 15° ; (i) with side-skirts and (ii) without side-skirts.

Figure 4.4. Aerodynamic drag comparisons of with and without boat-tail plate towards ground side for wind velocity of 90kph.

As seen in Figure 4.5, boat-tail reduces the aerodynamic drag mainly at rear-side of the trailer in both yaw angles (0 and 6°).

Figure 4.5. Aerodynamics drag coefficient along the vehicle length for boat-tail plate width of 1.2m and wind velocity of 90kph (no side-skirts); (i) Yaw=0° and (ii) Yaw=6°

As seen in Figure 4.6, lower flow separation observed at trailer rear side in a boat-tail configuration when compared to no boat-tail configuration. It should be noted that the boat-tail consists of four surfaces attached to the back of the trailer, thereby streamlining the trailer, delaying separation and thus reducing the wake size. Flow separation plays an important role in the total drag of vehicles (bluff bodies) since pressure drag is a result of flow separation.

Figure 4.6. Velocity field at trailer rear side (horizontal plane across the center of the vehicle height)

As seen in Figure 4.7, aerodynamic drag drop increases with increase of yaw angle upto 10°. Further increasing the yaw angle, it may diminish or even negative effect can be observed.

Figure 4.7. % of aerodynamic reduction Vs. yaw angle

As seen in Figure 4.8 and Table 4.2, clearly demonstrates the advantage of boat-tail over no boat-tail configuration.

Figure 4.8. Comparison of aerodynamic drag force with and without boat-tail vehicle configuration

Table 4.2. Comparison of Aerodynamic drag Force at different yaw angles [Boat-tail plate width: 1.2m, Wind speed: 90 kph]

Yaw /Force [N]	0°	2°	4°	6°	8°	10°	12°
No boat-tail [N]	2064.75	2082.78	2129.81	2197.55	2285.82	2400.55	2503.52
Boat-tail [N]	1953.65	1956.01	1985.40	2013.30	2063.81	2140.98	2233.86

As seen in Figure 4.9, no significant changes were observed in fuel savings for both slant angle of 10° and 15° . Note, the fuel savings are assumed to be roughly half of the aerodynamic drag gains/improvements.

Figure 4.9. Comparison of fuel savings for two different boat-tail slant angles for vehicle speed of 90kph and with side-skirts.

4.4. Conclusions

No significant difference was observed in fuel economy at 10° and 15° slant angle. Further increasing slant angle aerodynamic drag raises (not shown). Aerodynamic drag drops with lower boat-tail plate. The optimized plate width found to be 1.2m with side-skirts configuration. On the other hand, the optimized plate width found to be 0.6m without side-skirts configuration.

5. Fan Shroud Optimization Using Adjoint Solver

Abstract

Fan and fan-shroud design is critical for underhood air flow management. The objective of this work is to demonstrate a method to optimize fan-shroud shape in order to maximize cooling air mass flow rates through the heat exchangers using the Adjoint Solver in STAR-CCM+[®]. Such techniques using Computational Fluid Dynamics (CFD) analysis enables the automotive/transport industry to reduce the number of costly experiments that they perform. This work presents the use of CFD as a simulation tool to investigate and assess the various factors that can affect the vehicle thermal performance.

In heavy-duty trucks, the cooling package includes heat exchangers, fan-shroud, and fan. In this work, the STAR-CCM+[®] solver was selected and a java macro built to run the primal flow and the Adjoint solutions sequentially in an automated fashion. In this analysis, the primal flow provides flow information (e.g. velocity, temperature and pressure) and the Adjoint solver provides surface morphing details w.r.t. max or min of the specified objective function. In the present work, the fan-shroud surface morphing was performed based on the maximum air mass flow rate through the heat exchanger outlet w.r.t. the spatial positions of the fan-shroud surface. An overall 1.4% increase in cooling air mass flow was observed in the heat exchanger with the optimized/morphed fan-shroud surface. The identified main optimized locations were mainly at the sharp edges of the original manufactured fan-shroud. Further evaluation is underway to maximize the cooling air mass flow benefit.

5.1. Introduction

Tighter emissions requirements and underhood packing due to higher power demands from engines requires more optimized cooling packages. Vehicle designers have to meet cooling performance requirements while minimizing fan power, noise and overall aerodynamic drag. Fan and fan-shroud design is critical for underhood air flow management. Fan shrouds funnel cooling air sucked by fan, which is passed through heat exchangers. Shroud design impacts both the air flow and noise generated by fan.

There is a great deal of literature focusing on overall front-end cooling thermal performance [1-3] but there are very few works which focus on fan and fan-shroud design. Only Srinivasa et al. [28] and Mesravran et al. [29] studied the impact of fan shroud shape on cooling air flow. Srinivasa et al. [28] did an optimization study on the fan shroud assembly and investigated the effect of fan immersion ratio, fan to core distance, and also shroud chamber length on fan airflow and power, as well as airflow uniformity. However, his shroud geometry was very simplified. Mehravaran and Zhang [29] studied different shapes of fan shrouds and different installation factors like tip clearance and fan immersion to maximize air flow. They concluded convex shaped shrouds resulted in maximum air flow.

One of the earliest works on the Adjoint approach for optimization is presented by Giles and Peirce [30]. They discussed the Adjoint approach as applied to Aeronautical applications, and discussed optimization of business jets. Hinterberger and Olesen [31] applied the Adjoint

optimization method to exhaust systems. Han et al. [32] studied aerodynamic shape optimization using the Adjoint method and pressure gradient method. They concluded that the shape sensitivities based on the Adjoint method are explicit and more consistent than the pressure gradient method.

Most recently, Rainer et al. [33] studied shape optimization of 2D duct flow using and Adjoint equation derived from the near wall $k - \zeta - f$ turbulence model. Magazoni et al. [34] performed aerodynamic shape improvement for driver side view mirror of a hatchback vehicle. They demonstrated the potential of the optimization technique, which can be applied in the early stages of projects and extended to optimize several surfaces of a vehicle body.

Based on the above references, the Adjoint optimization has received much attention as an efficient sensitivity analysis method for aerodynamic shape and pressure drop evaluation. The objective of this work is to demonstrate a method to optimize fan-shroud shape in order to maximize cooling air mass flow rates through the heat exchangers using the Adjoint Solver in STAR-CCM+[®]. Such techniques using Computational Fluid Dynamics (CFD) analysis enables the automotive/transport industry to reduce the number of costly experiments that they perform. This work presents the use of CFD as a simulation tool to investigate and assess the various factors that can affect the vehicle thermal performance.

5.2. Vehicle Underhood Compartment

The vehicle underhood compartment consists of engine and cooling package. As shown in Figure 5.1(i), the cooling package includes the heat exchanger, fan-shroud, and fan.

Figure 5.1. Geometry (i) vehicle underhood compartment and (ii) computational domain

As shown in Figure 5.1(ii), the modeling and simulations were performed for a given computational domain with dimensions of 17.5m x 5m x 6m [Length x width x height]. A

periodic boundary condition was applied at the side walls. Cold air velocity and pressure (0 Pa) were specified at inlet and outlet-boundaries, respectively. No-slip wall conditions were applied for floor and roof boundaries.

A Polyhedral mesh was used with prism layers at the surface of the engine, fan blades, and fan hub to capture the near-wall effects. A total of six prism layers were used with 10mm thickness over the engine block. Six prism layers were used for fan blades with a refined thickness of 0.6mm. Six prism layers were used over the shroud surface with thickness of 5mm.

As shown in Figure 5.2, a finer grid was generated around the fan-shroud and fan using volumetric refinements to accurately predict the effects of the fan and Adjoint in the area of interest. Further, to minimize the cell count, a coarser mesh was implemented towards the inlet, outlet, side-walls, ground floor and roof sides. The entire computational domain consists of a total of 10 million cells.

The maximum and minimum target surface size for the engine block was 30mm and 15mm, respectively. For the fan-blades, the maximum and minimum target surface size was 3mm and 0.6mm, respectively. Over the fan-shroud, the maximum and minimum target surface size was 6mm and 3mm, respectively. The porous body heat exchangers used a maximum and minimum target surface size of 15mm and 7.5mm, respectively. For the computational domain boundaries (floor, inlet, outlet, sides and roof), the maximum and minimum target surface size was 120mm and 60mm, respectively.

A volumetric control block was used to control the mesh size around the shroud and fan geometry. This produces a finer mesh with a fairly constant cell size in the area of interest, helping to prevent instability in the Adjoint solve, as well as capturing the flow effects of the fan. This produces a maximum cell edge size of 6mm in the volumetric control region.

Figure 5.2. Computational Mesh, (i) side view and (ii) top view

Table 5.1. Operating Conditions

Fan speed [rpm]	1400
Inlet velocity [kph]	20
Inlet Temperature [K]	300
Pressure outlet [Pa]	0

5.3. Modeling

The optimization process is accomplished using the Adjoint solver and mesh morpher in STAR-CCM+.

First, a well converged primal solution is obtained on a polyhedral mesh. The primal solution is solved steady-state, using a Coupled solver approach and k-epsilon turbulence. Since a steady-state approach is used, actual movement of the fan geometry is not modeled; rather, a Moving Reference Frame approach is used to simulate the effects of the moving geometry. The primal solution is driven to convergence by successively increasing the Coupled Solver Courant Number. Convergence is judged by monitors of residuals, which are driven as low as possible, and of mass flow rate through the radiator in front of the fan, which is driven to an asymptotic value.

Using the converged primal solution, a cost function of mass flow rate through the radiator with respect to fan shroud position is solved on pre-defined control points (Figure 5.3) situated near the fan shroud surface to obtain a solution to the Adjoint equation. The solution to the Adjoint is used to calculate sensitivities of the primal flow to changes in control point position, resulting in a field of Adjoint sensitivity vectors on these points, seen in Figure 5.4. These Adjoint sensitivity vectors indicate in which direction movement of the control points, and by extension, the underlying geometry, would increase the mass flow rate, and in which direction movement would decrease the mass flow rate.

Figure 5.3. Control points over fan-shroud

Figure 5.4. Adjoint position sensitivity vector (2nd Iteration)

These vectors are then scaled using the following definition: $\{\text{MaxDisplacement}/(\{\text{MaxAdjointReport}\})$ in order to limit the maximum displacement of the morpher, thus producing a smoother transition to the optimal shape. The scaled sensitivity vector values are then used as displacements for the morpher. The morphing of these control points results in a deformation of the mesh, and ultimately, in the underlying fan shroud geometry. In this work, the 'MaxDisplacement' was set at 5mm and 'MaxAdjointReport' was obtained from a report of the maximum 'Adjoint of Mass Flow w.r.t. Position'.

This process of solving for the primal solution, Adjoint solution, Adjoint sensitivities, and mesh deformation is then repeated multiple times to generate a cumulative displacement of the control points and the shroud geometry in order to increase the mass flow rate through the radiator. Mesh quality metrics, such as cell quality and volume, are reported after each morph of the control points to ensure that the morphing procedure has not degraded the mesh to a point that simulation results would suffer. If the mesh quality falls below a pre-determined value, the driving java macro automatically exports the current morphed geometry, replaces the original CAD model, and re-meshes the domain, resulting in a new, higher quality mesh on the morphed geometry. The process then continues on this new mesh. Automation of this process is accomplished through the use of a java macro in STAR-CCM+.

As shown in Figure 5.1(i), the heat exchangers were modeled using a porous body with an inertial and viscous resistance. The heat exchangers, Charge Air Cooler (CAC), and radiator inertial and viscous resistance values were given in Table 5.2. Moreover, heat transfer in the heat exchangers was not modeled, and compressibility effects were included.

Table 5.2. Heat exchanger porous modeling

Air Flow Resistance	CAC	Radiator
Inertial [kg.m ⁻⁴]	91.67	98.41
Viscous [kg.m ⁻³ .s ⁻¹]	-31.434	312.07

As shown in Figure 5.5, the convergence plot shows reasonable residual drops for the primal flow and Adjoint solution. The velocity variables residuals were dropped around 3 orders of magnitude. Adjoint variables residuals were dropped around 15 orders of magnitude.

Figure 5.5. Convergence/residuals plot (un-normalized)

5.4. Results

Fan-shroud optimization was carried out using the coupled flow solver, Adjoint solver, and mesh morpher in STAR-CCM+ as discussed in modeling section. The optimized solution was based on 20 iterations of primal flows and Adjoints. Mehravaran and Zhang [29] have performed CFD simulations with different shroud shapes. Contrary, we performed a simulation using the original shroud and optimized its design using the Adjoint solver without any externally modifications of the fan shroud. Note, the fan insertion depth is kept at constant and the only expected fan-shroud changes were in radial direction

In Figure 5.6, the original shroud surface consists of sharp edges when compared to the morphed/optimized surfaces. The highlighted surface edges show the modified/optimized edges which provide lower pressure loss and higher mass flow rates through the heat exchangers.

Figure 5.6. Comparison of (i) original and (ii) optimized fan-shroud

As shown in Figures 5.4 and 5.7, the position sensitivities are the main determining factor used to optimize the fan-shroud surface. The higher the position magnitude is, the larger the morpher displacement will be on this surface. The position vector direction guides the surface changes towards inner or outer movements of the surface. The significant position sensitivities drop was observed with an increase of adjoint iterations (see 6th and 2nd iteration) and consequently it leads to drop in % of fan-shroud morphing.

Figure 5.7. Position sensitivity magnitudes at 2nd and 6th iteration [surface optimizes @ higher position sensitivities]

As shown in Figure 5.8(ii), the static pressure over the shroud surface is more uniform when compared to the original fan-shroud results (Figure 5.8i). The main difference between the original and optimized fan-shroud is variation in the shape of the surface edges, as discussed above in Figure 5.6.

Figure 5.8. Pressure distribution, (i) shroud: original and (ii) shroud: optimized

As shown in Table 5.3, a 1.4% increase in air mass flow rate was observed in the heat exchangers with fan operating at 1400 rpm when compared to original fan-shroud model. To be noted, the simulation residuals are dropped to 10^{-8} for flow variables and 10^{-10} or lower for Adjoint position sensitivities. In Table 5.3, the simulation data shows that the optimized fan-shroud has a higher cold air flow rate compared to the original design. The original shroud has several sharp surfaces, leading to flow separation and a reduced air mass flow rate through the heat exchangers, as discussed above.

Table 5.3. Heat exchanger cooling air mass outflow rate

Original fan-shroud	3.800
Optimized fan-shroud	3.852 (~1.4%)

Figure 5.9 shows the pressure distribution over a vertical plane through the center of the fan. The pressure distribution is more uniform in the optimized fan-shroud than in original fan-shroud.

Figure 5.9. Pressure distribution, (i) original and (ii) optimized flow.

As shown in Figure 5.10, the velocity flows away smoothly from fan. In underhood compartments, there are often stagnant flow zones which are very common between fan and engine block.

Figure 5.10. Velocity contours

5.5. Conclusions

Modeling and simulation were performed for fan-shroud optimization in STAR-CCM+® using the Adjoint solver. The flow variables, velocity, pressure and turbulence parameters were solved by using the Coupled Solver. Consequently, the Adjoint solver provides the position sensitivities around the fan-shroud surface w.r.t. heat exchanger outlet air mass flow rates. The fan-shroud surface morphing was carried out using the calculated position sensitivities from the Adjoint solver.

A total 1.4% increase in cold air flow was observed with an optimization of the fan-shroud surface. The improvement gain was mainly due to smoothing of the fan-shroud surface edges, rather than the sharp edges which are present in the original fan-shroud model from the manufacturer. The air flow around the optimized surface edges was much smoother, as they act to reduce pressure losses and ultimately led to a higher mass flow rate through the heat exchanger. Further evaluation is underway to maximize the cooling air mass flow benefit.

6. Investigation of Aerodynamic Influence on Truck Platooning

Abstract

This project investigates the aerodynamic influence of multiple on-highway trucks in different platooning configurations. Complex pressure fields are generated on the highways due to interference of multiple vehicles. This pressure field causes an aerodynamic drag to be different than the aerodynamic drag of a vehicle in a no-traffic condition. In order to study the effect of platooning, three-dimensional modeling and numerical simulations were performed using STAR-CCM⁺® commercial Computational Fluid Dynamics (CFD) tool. The aerodynamic characteristics of vehicles were analyzed in five different platooning configurations with two and three vehicles in single and multiple lanes.

A significant Yaw Averaged Aerodynamic Drag (YAD) reduction was observed in both leading and trailing vehicles. YAD was based on the average result of three different yaw angles at 0°, -6° and 6°. In single-lane traffic, YAD reduction was up to 8% and 38% in leading and trailing vehicles, respectively. In both leading and trailing vehicles, YAD reduction effects dropped with higher separation distances. In contrast, a trailing vehicle following the leading vehicle in the immediate next separate lane showed a YAD reduction up to 3% in the leading truck at lower separation distances. In further increases of the separation distance, no YAD benefit was observed in the leading vehicle. On the other hand, the trailing vehicle YAD was reduced up to 15% with increases in the separation distance. In side-by-side vehicle configurations, negative YAD impact was observed in the leading vehicles. Three different side-by-side configurations were studied and results are presented in the form of both YAD and fuel savings.

6.1. Introduction

The objective of this work is to evaluate the fuel savings potential of on-highway platooning vehicles. Tractor-trailers operate in a heterogeneous aerodynamic environment that includes influences of multiple surrounding vehicles. Platooning involves the reduction of aerodynamic drag by grouping vehicles together and decreasing the distance between them in single and/or multiple lanes.

Recent research studies have demonstrated that platooning can be effective at reducing the aerodynamic drag on all of the vehicles in the platoon, even in the lead vehicles [35-37]; however, the highest aerodynamic drag reduction occurs for the vehicles between the first and last truck. Moreover, fuel consumption depends on vehicle spacing, vehicle speed, vehicle position, and vehicle mass [38].

Romberg et al. [39] had shown experimental aerodynamics of NASCAR in drafting and passing situations. Zabat et al. [40] and Bogdan and Fred [41] performed experiment analysis of three identical minivans in misaligned platooning configuration. Alam et al. [42] presented his work on fuel reduction in heavy-duty vehicle platooning with two identical trucks in real on-road conditions.

Modeling and simulations can identify potential aerodynamic drag reduction in traffic conditions. Several research groups are focused on primary fuel savings by improving the vehicle external design and the engine performance including rejected heat utilization. Further fuel savings can be obtained by aerodynamic aid devices like side skirts, boat tail, etc. [18-19] and also through vehicle platooning [35-37].

In this project, 3D Computational Fluid Dynamics (CFD) modeling and simulations are performed in STAR-CCM+® commercial tool [18]. Aerodynamic drag numerical results were analyzed for multiple vehicle interactions occurring while the vehicles were under the influence of the traffic that may be impacted by the overall aerodynamic drag of the leading and the trailing vehicles in both single and multiple lanes.

6.2. Vehicle Configuration Studied

In this study, identical medium-duty delivery trucks were considered for platooning in single-lane and multi-lane traffic, as shown in Figure 6.1. The truck dimensional length, width, and height were considered to be 12 m, 2.55 m, and 3.72 m, respectively.

Figure 6.1. Medium-duty delivery truck.

As shown in Figure 6.2 (i-ii), the aerodynamic drag performances of identical medium-duty vehicles in drafting were investigated. Firstly, vehicle-to-vehicle (V2V) configurations were studied considering single and multiple lanes for several separation distances between both the leading and the trailing vehicles. Secondly, an aerodynamic drag analysis was performed for multiple vehicles influenced in side-by-side configurations as shown in Figure 6.2 (iii-v). In multi-lane traffic, a lateral separation distance between vehicles was considered to be 1.2 m.

Figure 6.2. Configuration of different truck positions for investigation; (i) **Configuration 1:** V2V in single lane, (ii) **Configuration 2:** V2V in multiple lanes, (iii) **Configuration 3:** Two side-by-side vehicles [LV1-LV2] (iv) **Configuration 4:** Three side-by-side vehicles [LV3-LV1-LV2] and (v) **Configuration 5:** Two side-by-side vehicles and one trailing vehicle [LV1-LV2-TV1].

6.3. Modeling

Incompressible turbulent 3D isothermal steady state simulations were performed using segregated-flow solver. Typically, the gas-phase turbulence was modeled using $k-\epsilon$ turbulence model with standard parameters. This software used the Finite Volume (FV) discretized method to solve the Navier-Stokes equations numerically for the fluid behavior. Partial Differential (PD) model equations of continuity, momentum, and turbulence can be found elsewhere [14-43].

Numerical simulations were performed in a computational domain with a specified inlet, outlet, and floor and wall surfaces, using the vehicle configuration as shown in Figure 6.3. To capture vortex at a trailer's rear, a large computational domain with dimensions of 250 m x 500 m x 200 m was considered. Note that the computational domain length varied between 100 m to 250 m based on the vehicle configuration and separation distance [20].

Figure 6.3. Numerical setup.

(i)

(ii)

Figure 6.4. Mesh top view; (i) 150 ft V2V separation distance and (ii) cabin.

A pressure boundary condition was specified at the outlet, periodic boundary condition was applied for the side walls to capture the influence of non-zero yaw angles, air flow velocity was specified at the inlet, and no-slip boundary condition was imposed at the floor and roof.

In all configurations, the trimmer mesh was implemented as shown in Figure 6.4. The minimum and maximum cell edges in the vehicle configuration were 7.5 mm and 40 mm, respectively. Four prism layers with thickness of 20 mm were considered to capture the turbulence near the vehicle surface. The prism layers were used to provide an accurate estimation of the velocity near the wall in high Reynolds number external flows ($\sim 5.7E6$). Standard wall functions were used by keeping the y-plus value within an acceptable range from 20 to 200 [16]. The overall residuals were dropped by three orders of magnitude using an under-relaxation factor for velocity, pressure, and turbulence parameters (k and ϵ) considering 0.7, 0.3, and (0.8 and 1), respectively. The recommended practice guidance for the use of CFD in evaluating the commercial vehicle aerodynamic performance can be found elsewhere [20].

As shown in Table 6.1, the following geometry and operating conditions were considered to simulate the multiple vehicle configurations at constant vehicle speed value of 55 mph. Results in the following sections are presented after normalization with respect to Yaw Average Aerodynamic Drag (YAD) of a standalone vehicle. It is noted that YAD is based on the average result of three different yaw angles, 0° , -6° and 6° , as discussed in Appendix A.1.

Table 6.1. Aerodynamic drag simulation operating conditions

Operating conditions	
Air velocity inlet [mph]	55
Yaw angle [deg]	0, -6 and 6
Outlet pressure [atm]	1
Computational domain [m^3] [Length x Width x Height]	250 x 500 x 200
Number of trimmer cells [millions]	30 to 200

6.4. Results and Discussion

6.4.1. Two Vehicle Configurations

6.4.1.1. Vehicle-to-Vehicle (V2V) simulations in single-lane traffic

Figure 6.5. Configuration 1

In this section, aerodynamic characteristics were studied for configuration 1 (Figure 6.5). As shown in Figure 6.6, Yaw Averaged Aerodynamic Drag (YAD) reduction drops with higher separation distances in both leading and trailing vehicles. Drag benefit of the leading vehicle was observed only for separation distances of less than 90 ft. This was mainly due to significant influence of the trailing vehicle's pressure wave on the leading vehicle's wake region as shown in Figure 6.7.

Figure 6.6. Yaw Averaged Aerodynamic Drag (YAD) reduction of leading and trailing vehicles [averaged values of three different yaw angles 0° , -6° , and 6°].

Figure 6.7. Static pressure distribution in 0° yaw angle (i) no-traffic, (ii) 30 ft, and (iii) 90 ft

Figure 6.6 shows that the higher YAD reduction was observed in the trailing vehicle when compared to the leading vehicle. This is due to reduced oncoming air flow velocity from the leading to the trailing vehicles, which leads to a lower drag in the trailing vehicle. In the trailing vehicle, aerodynamic drag drops were observed with the higher separation distances. Here, the YAD was based on the average result of three different yaw angles at 0° ,

-6°, and 6°. As shown in Figure 6.8, estimated fuel savings dropped with separation distance. It is noted that estimated fuel savings are roughly half of the YAD reduction (Appendix A.2).

Figure 6.8. Estimated fuel savings in leading and trailing vehicles [averaged values of three different yaw angles 0°, -6°, and 6°].

As shown in Figure 6.7, at higher separation distances, negligible trailing vehicle influence on the leading vehicle was observed. In lower separation distances, the stagnation pressure from the trailing vehicle communicated with the wake pressure of the leading vehicle causing an increased base pressure on the leading vehicle and an overall drag reduction.

Figure 6.9, using a superimposed vehicle front view, shows that oncoming air flow velocity from leading vehicle to trailing vehicle increased with the separation distance. At 6° yaw angle, vortex location shifted from one side to the other side and oncoming air flow velocity raised by increasing the separation distance leading to a higher aerodynamic drag (see Table 6.2).

Figure 6.9. Velocity contours at 3 ft in front of a trailing vehicle in different vehicle separation distances at 6° yaw angle.

Table 6.2 shows that aerodynamic drag increased with the yaw angle in the leading and trailing vehicles. Drag values were normalized by no-traffic condition at 0° yaw angle. A significant drag reduction was observed with lower separation distances in both the leading and trailing vehicles. This resulted mainly from interference of the pressure wave from the trailing vehicle to the leading vehicle, and also reduced oncoming air flow velocity from leading vehicle to trailing vehicle, as discussed above.

Table 6.2. Normalized aerodynamic drag of leading and trailing vehicles in a single lane

	Yaw angle	Separation Distance			
		30 ft	60 ft	90 ft	150 ft
Leading Vehicle [Drag]	0°	0.922	0.984	0.993	0.99
	6°	1.0	1.07	1.08	1.1
Trailing Vehicle [Drag]	0°	0.506	0.606	0.67	0.704
	6°	0.780	0.860	0.870	0.950

6.4.1.2. Vehicle-to-Vehicle (V2V) simulations in multi-lane traffic

Figure 6.10. Configuration 2

In this section, aerodynamic characteristics were studied for configuration 2 (Figure 6.10), in which the leading vehicle was in one lane and the trailing vehicle was following behind and in the next lane. The lateral vehicle separation distance was kept at a constant value of 1.2 m, while longitudinal vehicle separation distances varied. As shown in Figure 6.11, YAD reduction was observed in both leading and trailing vehicles. At higher separation distances, lower fuel savings were observed in the leading vehicle (Figure 6.12). This is due to insignificant influence of the trailing vehicle pressure wave on the leading vehicle. A similar explanation can be found as discussed in Figure 6.6; however, unlike the single lane vehicles shown in Figure 6.8, higher fuel savings were observed in the trailing vehicle due to lower oncoming air flow velocity from the leading vehicle at higher separate distances as shown in Figure 6.13.

Figure 6.11. Yaw Averaged Aerodynamic Drag (YAD) reduction of leading and trailing vehicle in multi-lane traffic [averaged values of three different yaw angles at 0° , -6° , and 6°].

Figure 6.12. Estimated fuel savings in leading and trailing vehicle in multi-lane traffic [averaged values of three different yaw angles at 0° , -6° and 6°].

Figure 6.12 illustrates that estimated fuel savings rose with separation distance. Estimated fuel savings were found to be roughly half of the YAD reduction (Appendix A.2).

Figure 6.13. Velocity contours at 3 ft in front of trailing vehicle at 0° yaw angle; (i) 90 ft, (ii) 60 ft and (iii) 30 ft.

Table 6.3 shows that aerodynamic drag values increased with the yaw angle in both leading and trailing vehicles. Drag values were normalized by no-traffic condition at 0° yaw. In the leading vehicle, a significant drag reduction was observed with the lower separation distances. Interestingly, higher aerodynamic drag reduction was predicted for the trailing

vehicle at 6° yaw angle when compared to -6° and 0° yaw. This was mainly due to shedding of the lower air velocity zone towards the trailing vehicle, as shown in Figure 6.14.

Table 6.3. Normalized aerodynamic drag of leading and trailing vehicles in multiple lanes

	Yaw angle	Separation Distance		
		30 ft	60 ft	90 ft
Leading Vehicle[Drag]	0°	0.95	0.97	0.99
	6°	1.05	1.10	1.10
	-6°	1.09	1.10	1.10
Trailing Vehicle[Drag]	0°	0.98	0.96	0.95
	6°	0.94	0.84	0.82
	-6°	1.06	1.03	1.01

Table 6.3 shows lower aerodynamic drag reductions were observed in -6° yaw angle when compared to 0° and 6° yaw angles. This is mainly due to the air stream velocities in both leading and trailing vehicles were similar as compared to single vehicle in no-traffic condition.

Figure 6.14. Velocity magnitude of 30 ft vehicle separation; (i) 0° yaw, (ii) 6° yaw and (iii) -6° yaw.

The shedding effects shown in Figure 6.14 were much stronger in 6° yaw when compared to 0° and -6° yaw. At higher separation distances, the lower air velocity zone further increased towards the trailing vehicle and consequently led to a higher aerodynamic drag reduction in all yaw angles, as shown in Table 6.3. Figure 6.15 shows the lower static pressures that were observed around the trailing vehicle with increased separation distances. It is clear that lower static pressure leads to lower aerodynamic drags.

Figure 6.15. Static pressure distribution in 6° yaw angle (i) 30 ft, (ii) 60 ft and (iii) 90 ft.

6.4.1.3. Side-by-side vehicle simulations in multi-lane traffic

Figure 6.16. Configuration 3

The observed YAD performance of configuration 3 (Figure 6.16), where two vehicles are side-by-side, penalty was up to 11% (Figure 6.17). This penalty is mainly due to the limited clearance to escape airflow from front cabins. This condition leads to higher static pressure at the front portion of the cabin when compared to the no-traffic condition illustrated in Figure 6.18.

Figure 6.17. Yaw Averaged Aerodynamic Drag (YAD) of multiple vehicles in multi-lane traffic [averaged values of three different yaw angles at 0°, -6° and 6°].

Figure 6.18. Static pressure distribution in 6° yaw angle (i) no-traffic and (ii) two vehicles in configuration 3.

6.4.2. Three Vehicle Configurations

6.4.2.1. Multiple side-by-side vehicle simulations in multi-lane traffic

Figure 6.19. Configuration 4

When vehicles were in configuration 4, as shown in Figure 6.19, the observed YAD penalty was 23% in center vehicle and 17% in side vehicles, as shown in Figure 6.20. This is mainly due to the limited clearance to escape airflow, which leads to higher static pressure at the front cabins. This is discussed above in the side-by-side vehicle simulations in multi-lane traffic section (configuration 3).

Figure 6.20. Yaw Averaged Aerodynamic Drag (YAD) of multiple vehicles in multi-lane traffic [averaged values of three different yaw angles at 0° , -6° , and 6°].

Aerodynamic drag rises in the center lane vehicle (LV1) more than side-vehicles LV2 and LV3, as shown in Figure 6.21. The increased drag is due to low clearance between vehicles, which leads to higher velocity till the trailer front when compared to no-traffic condition. On the other hand, the observed aerodynamic drag values are similar in both side vehicles due to the symmetrical vehicle geometry at 0° yaw. In this configuration (configuration 4), the rise of the drag occurred mainly in the cabin when compared to no-traffic condition.

Figure 6.21. Normalized cumulative aerodynamic drag along the vehicle at 0° yaw angle for three vehicles side-by-side in multiple lanes (LV3-LV1-LV2).

6.4.2.2. Two side-by-side leading vehicles and one trailing vehicle simulations in multi-lane traffic

Figure 6.22. Configuration 5

When vehicles were in configuration 5, as shown in Figure 6.22, the observed YAD penalty was up to 6% in leading vehicles. On the other hand, a 42% YAD reduction was observed in the trailing vehicle due to lower oncoming air flow from the leading vehicles (Figures 6.23 and 6.24). In this configuration, the leading and trailing vehicles were separated by a 30 ft distance.

Figure 6.23. Yaw Averaged Aerodynamic Drag (YAD) of multiple vehicles in multiple lanes [averaged values of three different yaw angles at 0°, -6° and 6°].

(i) Configuration 1: one leading vehicle

(ii) Configuration 5: two leading vehicles

Figure 6.24. Velocity magnitude in 0° yaw for (i) configuration 1: one leading and one trailing vehicle and (ii) configuration 5: two side-by-side leading vehicles and one trailing vehicle (LV1-LV2-TV1).

As shown in Figure 6.23, the higher YAD benefit was observed in the trailing vehicle when compared to configurations 1 and 2 (Figures 6.6 and 6.11). This is due to higher reduction of oncoming air flow from two leading vehicles when compared to one leading vehicle configuration (configuration 1), as shown in Figures 6.24 and 6.25.

(i) Configuration 1: 6° Yaw

(ii) Configuration 5: 6° Yaw

(iii) Configuration 5: -6° Yaw

Figure 6.25. Velocity magnitude in different yaw angles (i) configuration 1: 6° yaw (ii) configuration 5: 6° yaw and (iii) configuration 5: -6° yaw.

As shown in Figures 6.24(ii) and 6.25(ii-iii), the lower oncoming air flow velocity zone was observed in trailing vehicle at 0° and -6° yaw angles when compared to 6° yaw angle. Table 6.4 shows that lower aerodynamic drag values were observed in the trailing vehicle at 0° and -6° yaw angles when compared to 6° yaw angle. This is mainly due to spreading of the lower velocity zone towards a trailing vehicle from the leading vehicles in 0° and -6° yaw angles (Figures 6.24(ii) and 6.25(iii)).

Table 6.4. Normalized aerodynamic drag of two leading and one trailing vehicles in multiple lanes [30 ft separation]. Drag values were normalized by no-traffic condition at 0° yaw.

	Yaw angle	LV1	LV2	TV1
Aerodrag	0°	1.09	1.04	0.51
	6°	1.11	1.17	0.91
	-6°	1.20	1.09	0.42

6.5. Conclusions

External aerodynamic drag analyses were studied for different vehicle platooning positions in single and multiple lanes, and the following conclusions were made based on this study.

1. Configuration 1, one vehicle behind another in the same lane. This is the most common for platooning in single-lane traffic and had the highest benefit for Yaw Averaged Drag (YAD) reduction and team fuel savings, up to 23% (Figures 6.6 and 6.8). YAD benefit was reduced with increase in separation distance for both leading and trailing vehicles (Figure 6.6).
2. Configuration 2, one vehicle in one lane and the trailing vehicle behind and in the next lane. In configuration 2, fuel savings potential was minimum at a separation distance of 30 ft (Figure 6.12). Higher YAD benefits were observed in the trailing vehicle with higher separation distance due to vortex shredding by leading vehicle (Figure 6.11).
3. Configurations 3 and 4, side-by-side vehicle configurations. Negative YAD impact was observed due to rise in static pressure at the front portion of the cabin. Team fuel consumption was increased up to 11% in two vehicle and 29% in three vehicle configurations.
4. Configuration 4, three vehicles side-by-side. In configuration 4, the center vehicle experienced most of the YAD (LV1 in Figure 6.21).
5. Configuration 5, two vehicles side-by-side and another vehicle following behind. In configuration 5, negative YAD impact was on the leading vehicle with fuel economy penalty up to 6%. The trailing vehicle was observed to experience most of the YAD benefit (Figure 6.23).

7. Investigation on Underhood Thermal Analysis of Truck Platooning

Abstract

A 3D CFD underhood thermal simulations were performed in two different vehicle platooning configurations; (i) single-lane and (ii) two-lane traffic conditions. The vehicle platooning consists of two identical vehicles, i.e. leading and trailing vehicle.

In this work, heat exchangers were modeled by two different heat rejection rate models. In the first model, a constant heat rejection rates were considered as similar to no-traffic vehicle condition. In the other model, a varied heat rejection rates were implemented by considering an aerodynamic influence on fuel consumption rates.

In a constant heat rejection rate model, the trailing vehicle thermal performance is significantly dropped in single-lane traffic due to reduced oncoming cold mass air flow velocities from leading vehicle. Also, the similar observations were found in two-lane traffic but at higher vehicle separation distances.

In a varied heat rejection rate model, significant drop in temperature raise was observed in both leading and trailing vehicles when compared to a constant heat rejection rate model in the single-lane and the two-lane traffic conditions. In leading and trailing vehicles, the varied heat rejection rates were obtained from fuel consumption rates.

7.1. Introduction

In the automotive and transport industry, aerodynamic thermal improvements are highly desirable in the design of fuel-efficient vehicles since the expected *global fuel consumption* to grow at a compound annual growth rate (CAGR) of 2.1 percent over the period 2014-2019 [44]. Over the last two decades, continuous efforts to build aerodynamic designs by modifying external vehicle surfaces and through add-on aerodynamic drag reduction devices are growing rapidly [9,11,16,20]. In transport vehicles, external air-flows are heterogeneous and it can have a positive or even negative impact on both aerodynamics and thermal management, depending on the influence of vehicle traffic and vehicle position. Computational Fluid Dynamics (CFD) modeling offers an excellent alternative to experiments in modeling and designing of aerodynamics vehicles. Today's available High Performance Computing (HPC) allows the building of very robust aerodynamic and thermal fuel-efficient vehicles with minimal amounts of time and cost.

On-road highways, vehicle platooning reduces a significant amount of fuel consumption in both leading and trailing vehicles. Most of the recent studies had shown the

aerodynamic considerations in vehicle platooning [1,35,36]. In platooning configurations, lower cold mass air flow rates were observed through the cooling package particularly in trailing vehicles by assuming a constant heat rejection rates for fuel condenser, Charge Air Cooler (CAC) and radiator.

Vegendla et al. [1] had shown the significant aerodynamic drag reduction at lower vehicle separation distance in single-lane traffic. The following observations were also made in their recent work. The aerodynamic drag reduction reduces with an increase of vehicle separation distance in both leading and trailing vehicles. The aerodynamic drag reduction in trailing vehicle was mainly due to the lower air velocities towards the front side of the cabin, which drastically reduces the static pressure at trailing vehicle cabin and thus leads to a lower aerodynamic drag values when compared to no-traffic vehicle. In two-lane traffic, trailing vehicle aerodynamic drag reduction raise was observed with an increase of vehicle separation distance. This was mainly due to the shedding of low velocity zone from leading to trailing vehicle. In leading vehicles, the drop in aerodynamic drag reduction was observed with an increase of vehicle separation distance due to negligible influence of trailing vehicle on leading vehicles.

In this work, underhood thermal performances were modeled by (i) assuming a constant heat rejection rates in both leading and trailing vehicle cooling packages and (ii) varying the heat rejection rates were considered to mimic the on-road vehicle platooning conditions based on vehicle fuel consumption rates. It should be noted that the fuel consumption rates varies with an external aerodynamics drag predictions compared to no-traffic vehicle.

7.2. Vehicle Modeling

In this study, two identical medium-duty delivery vehicles were considered for platooning in single-lane and two-lane traffic, as shown in Figure 7.1. The vehicle dimensional length, width, and height were considered to be 12 m, 2.55 m, and 3.72 m, respectively.

Figure 7.1. Medium-duty Delivery Truck.

As shown in Figure 7.2 (i-ii), the underhood thermal performances of identical medium-duty vehicles in platooning were investigated. In these two configurations, the aerodynamic drag results were presented in Vegendla et al. [1]. In the present work, the vehicle-to-vehicle (V2V) configurations were studied for a several separation distances between both the leading and the trailing vehicles in single-lane and two-lane traffic conditions. In two-lane traffic, a lateral separation distance between vehicles was considered to be 1.2 m.

Figure 7.2. Platooning configurations; (i) Single-lane traffic and (ii) two-lane traffic [6]

As shown in Figure 7.3(i), the following boundary conditions were implemented in a computational domain; the air flow velocity specified at inlet, the pressure boundary specified at outlet, the wall boundary specified at floor and opposite side of the floor wall and the periodic boundary condition specified at side walls.

Figure 7.3. Computational domain; (i) numerical setup and (ii) mesh top view

Table 7.1. Operating conditions

Operating conditions	
Air Inlet velocity [mph]	55
Yaw angle [deg]	0, -6 and 6
Outlet pressure [atm]	1
Computational domain [m ³] [Length x Width x Height]	250 x 500 x 200
Number of trimmer cells [millions]	30 to 150

Vehicle underhood thermal simulations were modeled using StarCCM+® software. In this work, the following physics continua were selected; (i) non-isothermal steady-state solver, (ii) segregated-flow solver, (iii) k-epsilon turbulence model with standard parameters [14,43], (iv) porous region for heat exchangers and (v) rigid body rotating motion for a cooling fan.

The volume mesh was generated by choosing a polyhedral mesh in StarCCM+® software. In this model, the minimum cell edge was kept at 1.5 mm for front cabin grill and the maximum cell edge was allowed up to 40 mm. Moreover, the volume controls were selected around the vehicles surface to keep a maximum edge size was up to 40 mm as shown in Figure 7.3(ii). The numerical wind-tunnel dimensions and near wall region turbulence were modeled as suggested by J2966 SAE technical paper [20] and Connor et al [16].

7.3. Results

7.3.1. Underhood Thermal Analysis by a Constant Heat Rejection Rate Model

In this paper, underhood thermal analyses were performed in two different vehicle platooning configurations; (i) single-lane and (ii) two-lane traffic conditions. As shown in Figure 7.4, the heat exchangers were modeled using a porous body with an inertial and viscous resistance. The CAC, radiator and fuel condenser porous resistances values were shown in Table 7.2. The total heat rejection rates were considered to be 24.25, 55 and 11 kW for CAC, radiator and fuel condenser, respectively. The fan rotates at 1400 rpm for all configurations.

Table 7.2. Heat exchanger porous modeling

	CAC	Radiator	Fuel Condenser
Inertial [kg.m^{-4}]	67	330	66
Viscous [$\text{kg.m}^{-3}.\text{s}^{-1}$]	415	335	66
Heat rejection rate [kW]	24.25	55	11

Figure 7.4. Vehicle underhood components; (i) vehicle top view and (ii) cooling package

7.3.2. Single-lane traffic

In single-lane traffic, one vehicle behind another in the same lane (Figure 7.2i). As shown in Figure 7.5(i), the Trailing Vehicle (TV) cooling package temperature raise drops with an increase of vehicle separation distance when compared to no-traffic vehicle. This was mainly due to the amount of cold mass air flow rates raises with an increase of the vehicle separation distance. At lower vehicle separation distance of 30 ft, around 20% (equivalent to 25% temperature raise) drop in mass flow rates were observed. This was mainly due to the low velocity vortex region between both leading and trailing vehicles. On the other hand, there was no significant temperature raise in the leading vehicle when compared to no-traffic vehicle. It should be noted that the average temperatures were extracted from all heat exchangers cold mass air flow rates ($Q=m C_p \Delta T$) and at three different yaw angles 0° , -6° , and 6° .

As shown in Figures 7.5(ii-iv), the significant temperature rise was observed in trailing vehicle heat exchangers, i.e, fuel condenser, CAC and radiator. In this model, the constant heat rejection rates were considered for both leading and trailing vehicles as shown in Table 7.2. The raise in temperature was mainly due to the lower oncoming air flow from leading vehicle as described above. As shown in Figure 7.6, the significant hot regions were observed in trailing vehicle when compared to leading vehicle.

Figure 7.5. Averaged temperature rise in heat exchangers of trailing vehicle at a constant fan speed; (i) cooling package, (ii) fuel condenser, (iii) CAC, and (iv) radiator [averaged values of three different yaw angles 0° , -6° , and 6°].

Figure 7.6. Temperature distribution in 0° Yaw angle at 30ft vehicle separation distance [horizontal plane at fan center position].

As shown in Figure 7.7, the temperature raise drops with an increase of vehicle separation distance from 30 ft to 150 ft in all heat exchangers, i.e. radiator, CAC and fuel condenser. In trailing vehicle, significant temperature raise was observed when compared to no-traffic vehicle (Figures 7.7a-c(i)).

Figure 7.7a. Radiator temperature in yaw 0 deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.7b. CAC temperature in yaw 0 deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.7c. Fuel condenser temperature in yaw 0 deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

7.3.3. Two-lane traffic

In two-lane traffic, one vehicle in one lane and the trailing vehicle behind and in the next lane (Figure 7.2ii). As shown in Figure 7.8(i), the raise in temperature was almost independent of the vehicle separation distance in two-lane traffic condition. This was mainly due to the average cold mass air flow velocities were almost constant around the trailing vehicle cooling package. In the leading vehicle, no changes were observed in temperatures when compared to no-traffic vehicle. Also, the similar observations were found in single-lane traffic condition.

As shown in Figures 7.8(ii-iv), the temperature rise was observed in trailing vehicle heat exchangers, i.e, fuel condenser, CAC and radiator. In this model, the constant heat rejection rates were considered in both leading and trailing vehicles as similar to the single-lane traffic condition. The raise in temperature was mainly due to the lower oncoming air flow from the leading vehicle as explained above in Figure 7.5. As shown in Figure 7.5, the highest temperature rise was observed at 60ft vehicle separation distance. This might be due to the following two reasons; (i) wind flow direction and (ii) higher temperature stream from the leading vehicle. As shown in Figure 7.9, the lower temperature raise was observed when compared to single-lane traffic trailing vehicle (Figures 7.5-7.7).

Figure 7.8. Averaged temperature rise in heat exchangers of trailing vehicle at a constant fan speed; (i) cooling package, (ii) fuel condenser, (iii) CAC, and (iv) radiator [averaged values of three different yaw angles 0° and 6°].

Figure 7.9. Temperature distribution in 0° Yaw angle at 30ft vehicle separation distance [horizontal plane at fan center position].

As shown in Figure 7.10, the temperature raises with an increase of vehicle separation distance from 30 ft to 150 ft in all heat exchangers, i.e, radiator, CAC and fuel condenser. In trailing vehicle, the significant temperature raise was observed when compared to no-traffic vehicle (Figures 7.10a-c (i)).

Figure 7.10a. Radiator temperature in yaw 0 deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.10b. CAC temperature in yaw 0 deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.10c. Fuel condenser temperature in yaw 0 deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

7.4. Underhood Thermal Analysis by Varying Heat Rejection Rate Model with an Influence of Aerodynamic Drag on Fuel Consumption

In this section, the varied heat rejection rates were obtained from an aerodynamic drag influence on heat rejection rates (heat rejection is proportional to the fuel consumption) as shown in the following equation. It should be noted that, the amount of fuel consumption drop or raise is roughly equal to an half of the amount of aerodynamic drag drop or raise, respectively.

$$\dot{Q} = \dot{Q}_{no-traffic} \left[1.0 - \frac{\left(1.0 - \frac{C_{d,i}}{C_{d,no-traffic}^{yaw=0^\circ}} \right)}{2} \right],$$

where $i = 0, 6$ and -6°

where, $\dot{Q}$ is heat rejection rates and C_d is vehicle aerodynamic drag coefficient.

7.4.1. Single-lane traffic

In single-lane traffic, leading and trailing vehicle thermal simulations were performed. Vengdla et al. [1] had shown the significant aerodynamic drag reduction and thus leads to a significant drop in cold mass air flow rates in trailing vehicle at lower vehicle separation distances as shown in Figure 7.11. The reduction in aerodynamic drag and thermal performance drop reduces with an increase of vehicle separation distance. On the other hand, the cold mass air flow rates increases slightly in leading vehicle.

Figure 7.11. Yaw averaged cooling air flow performance; (i) Leading vehicle and (ii) Trailing vehicle

As shown in Figure 7.12, temperature rise was observed in trailing vehicle (TV) for all heat exchangers, i.e. fuel condenser, CAC and radiator. The temperature raise was drops with an increase of vehicle separation distance, except at 150ft. Interestingly, the significant drop in temperature raise was observed when compared to a constant heat rejection rate model (see Figure 7.5). This was mainly due to the varied heat rejection rates based on the fuel consumption. On the other hand, lower temperatures were observed in the leading vehicle (LV) due to a drop in the heat rejection loads based on the fuel consumption rates. In leading

vehicle, the only limiting factor was heat rejection rates since the changes in cold air mass flow rates were almost negligible as shown in Figure 7.11.

Figure 7.12. Yaw averaged temperature rise over no-traffic vehicle in heat exchangers; (i) fuel condenser, (ii) CAC and (iii) radiator package [averaged values of three different yaw angles 0° , -6° , and 6°].

As shown in Figure 7.13, the temperature raise drops with an increase of vehicle separation distance from 30 ft to 150 ft in all heat exchangers, i.e., radiator, CAC and fuel condenser. Also, the similar observations were found in a constant heat rejection rate model as discussed above in Figure 7.7.

Figure 7.13a. Radiator temperature in yaw 0° deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.13b. CAC temperature in yaw 0 deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.13c. Fuel condenser temperature in yaw 0 deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

7.4.2. Two-lane traffic

In two-lane traffic, two vehicles were separated with lateral distance of 1.2m and at different longitudinal distances of 30ft, 60ft, 90ft and 150ft.

Figure 7.14. Yaw averaged cooling air flow performance; (i) Leading vehicle and (ii) Trailing vehicle

As shown in Figure 7.15, the temperature rise was observed in the trailing vehicle radiator. On the other hand, lower temperatures were observed in the leading vehicle radiator. The changes in the fuel condenser and CAC were negligible. However, the drop in cold air mass flow rates (Figure 7.14) doesn't seem to be effective on temperatures since the heat rejection loads varies with an fuel consumption criteria. Interestingly, the significant drop in temperature was observed when compared to a constant heat rejection rate model (see Figure 7.8).

Figure 7.15. Yaw averaged temperature rise over no-traffic vehicle in heat exchangers; (i) fuel condenser, (ii) CAC and (iii) radiator package [averaged values of three different yaw angles 0° , -6° , and 6°].

As shown in Figure 7.16, the temperature raise was observed with an increase of vehicle separation distance from 30 ft to 150ft in all heat exchangers, i.e, radiator, CAC and fuel condenser. Similar observations were found in a constant heat rejection rate model as discussed above in Figure 7.10. However, the significant drop in temperature was observed when compared to a constant heat rejection rate model (see Figure 7.10).

Figure 7.16a. Radiator temperature in yaw 0° deg ; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

Figure 7.16b. CAC temperature in yaw 0 deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft.

Figure 7.16c. Fuel condenser temperature in yaw 0 deg; (i) no-traffic, (ii) TV at 30ft, (iii) TV at 90ft and (iv) TV at 150ft

7.5. Conclusions

A 3D CFD underhood thermal simulations were performed in two different vehicle platooning configurations; (i) single-lane and (ii) two-lane traffic conditions. In this work, heat exchangers were modeled by two different heat rejection rate models. In the first model, a constant heat rejection rates were considered. In the other model, a varied heat rejection rates were implemented by considering an aerodynamic influence on fuel consumption rates.

In trailing vehicle, significant thermal performance drop was observed in both single-lane and two-lane traffic conditions due to lower cold mass air flow rates. A significant temperature raise was observed in the constant heat rejection rate model when compared to the varied heat rejection rate model. On the other hand, the temperature raise was significantly dropped in the varied heat rejection rate model due to an aerodynamic influence on vehicle fuel consumption rates.

In leading vehicle, no changes were observed in the constant heat rejection rate model for both single-lane and two-lane traffic conditions. Contrary, lower temperatures were observed in the varied heat rejection rate model at lower vehicle separation distances due to lower fuel consumption.

8. Investigation of Aerodynamic Drag on Tractor-Trailer in Underground Road Traffic

Abstract

Computational Fluid Dynamics (CFD) simulations in 3D were performed to analyze the aerodynamic drag performance of a truck for two different on-highway road-tunnel dimensions. The objective of this work was to investigate the aerodynamic drag performance of single and multiple-vehicle platooning configurations in single- and multi-lane one-way road traffic condition.

Significant aerodynamic drag benefit was observed in no-traffic vehicle with an increase of gap between vehicle and road-tunnel. In the vehicle platooning, the predicted aerodynamic drag benefit was up-to 23% at lower vehicle separation. The aerodynamic drag benefit gradually drops in one-lane traffic and negative performance was observed from 60ft separation distance onwards. In two-lane traffic where one vehicle was behind the other without a lateral separation, the predicted aerodynamic drag benefit was up to 60% and 44% at vehicle separation distance of 30ft and 150ft, respectively. The aerodynamic drag changes were negligible with an increase of vehicle separation distance where two vehicles were separated laterally with 1.2m distance. In all configurations, the leading vehicle aerodynamic drag changes were negligible irrespective of the vehicle separation distance.

8.1. Introduction

In transportation industry, significant fuel savings were achieved in vehicle platooning configurations as described in Vegendla *et al.* [1]. Additional fuel savings can be obtained by aerodynamically improved road designs, which includes valleys and underground tunnels.

On-highways, road tunnels are very common and have significant aerodynamic drag influence on tractor-trailer due to narrow tunnel dimensions of width and height apart from the ventilation and safety. Worldwide, there are 100's of road-tunnels with tunnel length of 100 m or above. For example, in US, the Maryland and the New York Holland road-tunnels are one of the busiest tunnels and everyday 1000's of vehicles pass through these tunnels. Most of these tunnels are designed for one-way traffic with single- or multi-lanes.

To author's knowledge, a little information is available in open literature for full scale 3D Computational Fluid Dynamics (CFD) vehicle aerodynamic of road-tunnel simulations. Jang and Chen [45] had shown ventilation effects including vehicle aerodynamic drag values which were calculated based on the data of the dynamic traffic as well as of the traffic-induced wind speed measured in the 1.8-km-long highway tunnel of Fu-De, Taipei City. Leong *et al.* [46] evaluated ventilation effects in Hsuehshan tunnel, Taipei by 1D modeling assuming no cross wind flows. Collela [47] and Koc [48] extensively studied ventilation effect including fire and cold flow using advance models like 3D-CFD and neural networks. Most of the underground transportation research community has mainly focused on fire and ventilation flows. In this paper, the aerodynamics fuel consumption or fuel saving potentials

are analyzed for multiple road-tunnel dimensions (especially different dimensions of width and height) including influence of multiple vehicles in single and multi-lane traffic.

Most of the published literature of vehicle aerodynamics is for open-road and wind-tunnel based modeling and experiments [49-50]. Moreover, the aerodynamic drag benefit is due to second generation devices like trailer skirts, trailer boat and gap reduction between tractor-trailer devices which were evaluated with track testing and CFD simulations from Surcel and Shetty [50]. Similarly, Van Raemdonck [51] had shown the aerodynamics performance of a relatively short rear-end drag reduction device for a semi-trailer. In open road conditions, multiple vehicles in platooning configuration have shown a significant amount of aerodynamic drag variations [1,35-36]. Recently, the importance of Reynolds number for CFD aerodynamics modeling was discussed in Wood [49].

This study demonstrates the vehicle and road-tunnel configurations following with results and discussion section. In results section, the aerodynamic drag results are analyzed for combination of single and multiple vehicles in single- and multiple-lane traffic condition. It should be noted that the cross-wind is assumed to be negligible due to closed side walls in long and narrow tunnels.

8.2. Vehicle and road-tunnel configurations

As in Figure 8.1, a medium-duty delivery truck was considered for road-tunnel simulations. The measured vehicle length, width and height are 12 m, 2.55m and 3.77, respectively.

Figure 8.1. Medium-duty delivery truck

In this project, two different road-tunnel configurations are investigated. The width and height of the tunnel dimensions are shown in Table 8.1. The tunnel inlet and outlet influence on vehicle aerodynamic drag is assumed to be negligible by considering the very long tunnel dimensions as suggested by SAE J2966 paper [20] (Table 8.1).

Table 8.1. Road tunnel dimensions [LV: Leading Vehicle; TV: Trailing Vehicle]

	Width [m]	Height [m]	Length from LV cabin to tunnel Inlet [m]	Length from TV rear end to tunnel outlet [m]
One-lane	5	5	3 times of vehicle length (36m)	10 times of vehicle length (120 m)
Two-lane	8.775	5	[J2966]	[J2966]

Figure 8.2. Vehicle platooning configuration top view, (i) single lane, (ii) vehicles are in same lane but two-lane traffic and (iii) vehicles are in two separate lanes.

In this work, the flat tunnel-roof was selected. In general most of the highway tunnel roofs are in hemispherical shape. Nonetheless, the main objective of this work is to provide an insight of fuel consumption or savings in different tunnel dimensions and multiple vehicle influence on aerodynamic drag performance.

3D CFD modeling and simulations were performed using StarCCM+® commercial software. The velocity and turbulence equations were solved using steady-state isothermal segregated flow solver. The considered computational domain replicates the actual road-tunnel dimensions as in Figure 8.2.

The aerodynamic characteristics were studied for configurations of Figures 8.2(ii-iii). In Figure 8.2ii, the leading and trailing vehicles are in the same lane without a lateral separation. In Figure 8.2iii, the leading vehicle was in one lane and the trailing vehicle was following behind and in the next lane. The lateral vehicle separation distance was kept at a constant value of 1.2 m, while longitudinal vehicle separation distances varied.

The volume mesh was generated to simulate the configurations shown in Figure 8.2. The minimum volume cell edge size was 7 mm over the vehicle surface and the maximum cell edge size was allowed up to 60 mm away from the vehicle surface. A volumetric control block was used to control the mesh size around the vehicle geometry. This produces a finer mesh with a fairly constant cell size, helping to prevent numerical instabilities and the maximum cell edge size was kept at 40mm in the volumetric control region. A total of four prism layers were selected over the vehicle surfaces with a total thickness of 20mm. In Re-Normalized Group (RNG) k-epsilon turbulence modeling, wall functions are considered to avoid the high resolution at vehicle surfaces [16]. The total number of mesh cells varied from 20 to 50 million, based on number of vehicles and its platooning configuration. Vegendla *et al.* [2] validated the aerodynamic drag by two different commercially available softwares StarCCM+® and PowerFLOW®. The predicted aerodynamic drag values were consistent in both softwares. The finite volume and discrete particle method was used to solve the Navier-Stokes equations in StarCCM+® and PowerFLOW®, respectively.

As seen in Tables 8.1-8.2, modeling and simulations were performed for given computational domain dimensions [Length x width x height]. Air velocity and pressure were specified at inlet and outlet-boundaries, respectively. No-slip wall conditions were applied for the rest of the wall boundaries, i.e. sides, floor and roof.

Table 8.2. Vehicle operating conditions

Operating conditions	
Air Inlet velocity [mph]	55
Yaw angle [deg]	0
Outlet pressure [Pa]	0
Turbulent intensity [-]	0.05
Number of trimmer cells [millions]	17 to 50

As seen in Table 8.3, the following blockage ratios were considered to model and analyze the results for no-traffic vehicle and platooning configuration in one and two-lane traffic conditions.

Table 8.3. Blockage Ratio [vehicle cross sectional area: 9.478 m²]

Inlet cross sectional area [width x height]	Blockage Ratio %
5m x 5m (single lane)	37.912
5m x 5.5m (single lane)	34.465
8.75m x 5m (two lane)	21.664
8.75m x 5.5m (two lane)	19.694

The simulation results processed after required convergence criteria for velocity, pressure and turbulence variables were met. The velocity, pressure and turbulence parameters (k-epsilon) residuals were dropped around four orders of magnitude.

In the following section, the presented results were grid independent. Two different grids were studied to justify the grid independency on aerodynamic drag results for no-traffic vehicle and platooning configuration of 90ft vehicle separation distance in single-lane traffic. As seen in Table 8.4, the predicted aerodynamic drag values were consistent in both fine and coarse mesh with marginal deviations.

Table 8.4. Comparison of aerodynamic drag for fine and coarse grid

Grid Count [Million]	Aerodynamic Drag [-]	
	Leading Vehicle	Trailing Vehicle
No-traffic vehicle		
17.4	1.953	N/A
23.9	1.98	N/A
Platooning [90ft separation]		
39.4	1.978	1.971
54.4	1.982	1.972

8.3. Results and Discussion

The aerodynamic drag was analyzed for single vehicle and multiple vehicles in both single- and two-lane traffic. In the vehicle platooning, two identical vehicles were considered for both leading and trailing vehicles as shown in Figure 8.2. Two different vehicle platooning configurations were considered in two-lane one-way road traffic; (i) the leading and trailing vehicles were in same lane and (ii) the leading and trailing vehicles were in two separate lanes.

8.3.1. Single/No-traffic Vehicle

As seen in Figures 8.3 and 8.4, aerodynamic drag was compared for two different height clearances between vehicle and road-tunnel in one- and two-lane configurations. Note, the normalized aerodynamic drag was based on one-lane configuration and with a height clearance of 1.23m between the vehicle and the tunnel. A significant amount of aerodynamic drag reduction was observed with an increase of height clearance in both one- and two-lane traffic. This is mainly due to escape tendency of the air mass flow velocity which rises with an increase of tunnel height. On the other hand, the lower aerodynamic drag was observed in two-lane configuration when compared to one lane (Figure 8.4ii). This is mainly due to the minimum impact of the air mass flow velocity from side tunnel walls on vehicles in two-lane configuration.

Figure 8.3. Comparison of normalized aerodynamic drag for two different height clearance between vehicle and road tunnel.

Figure 8.4. Comparison of normalized aerodynamic drag along the vehicle length; (i) different road-tunnel height in one-lane and (ii) one- and two-lane for 1.23m height clearance between vehicle and road tunnel.

8.3.2. Multiple Vehicles

The aerodynamic drag performances of both leading and trailing vehicles were compared in one- and two-lane traffic condition. It should be noted that the normalized aerodynamic drag was based on single vehicle in one-lane configuration and with a height clearance of 1.23m between the vehicle and the tunnel, as discussed above. In this section, the air blockage ratio for single-lane and two-lanes are approximately 37.9% and 21.6%, respectively (Table 8.3).

As seen in Figure 8.5i, 23% aerodynamic drag performance improvement was observed in trailing vehicle at lower vehicle separation distance (~30 ft) in single-lane traffic (Figure 8.2i). In trailing vehicle, the aerodynamic drag benefit gradually drops and negative performance was observed from 60ft separation distance onwards. The predicted negative performance in one-lane traffic was mainly due to the restricted air mass flow rates and high blockage ratio (~37.9% vs 21.6%) than in two lanes. The air mass flow velocities from leading vehicle directly hits on the tunnel side walls and subsequently reflects on the trailing vehicle surface from 60ft vehicle separation distance onwards. In one- and two-lane traffic, there was no-significant impact observed on leading vehicle for all vehicle separation distances (Figures 8.5i-iii).

In Figure 8.5ii, aerodynamic characteristics were studied for configuration of Figure 8.2ii, in which the leading and trailing vehicles were in the same lane without a lateral separation in two-lane. The highest benefit was up-to 63% in trailing vehicle when compared to no-traffic vehicle in single-lane (Figure 8.5ii). Similar observations were found in leading vehicle but constant value (up-to 43%) for all vehicle separation distances. This was mainly due to the negligible influence of the pressure wave from the trailing vehicle on leading vehicles as discussed in Vegendla *et al.* [1]. The trailing vehicle aerodynamic drag performance drops with an increase of vehicle separation distances due to the raise of the oncoming airflow as shown Figure 8.6ii.

Figure 8.5. Aerodynamic drag performance of leading and trailing vehicles at different vehicle-to-vehicle (V2V) separation distances; (i) One lane (Figure 8.2i), (ii) Two lane: Leading and Trailing Vehicles are in same lane (Figure 8.2ii) and (iii) Two lane: Leading and Trailing Vehicles are in two separate lanes (Figure 8.2iii).

In Figure 8.5iii, aerodynamic characteristics were studied for configuration of Figure 8.2iii, in which the leading vehicle was in one lane and the trailing vehicle was following behind and in the next lane. The lateral vehicle separation distance was kept at a constant value of 1.2 m, while longitudinal vehicle separation distances varied. The aerodynamic drag benefit was up-to 42% compared to no-traffic vehicle in one lane. There was no significant change with separation distance in both leading and trailing vehicle. This was mainly due to the negligible influence of the leading vehicle generated vortex (low-velocity zone) on trailing vehicle as shown in Figure 8.6iii.

Figure 8.6. Velocity fields across the center of the truck height (1.27m) at separation distances of 30ft, 90ft and 150ft; (i) One lane, (ii) Two lane: Leading and Trailing Vehicles are in same lane and (iii) Two lane: Leading and Trailing Vehicles are in a separate lane.

As seen in Figure 8.6i, the observed lower velocity zone disappears with an increase of vehicle separation distance, particularly at front portion of the trailing vehicle cabin. The lower velocity zone leads to lower aerodynamic drag and this statement is valid only up to 60ft vehicle separation distance. Further increasing the separation distance, the negative aerodynamic drag effect was observed due to restricted air mass flow rates in one-lane configuration, as discussed in Figure 8.5.

As seen in Figures 8.5ii and 8.6ii, the maximum aerodynamic drag benefit was observed at lower vehicle separation distance due to lower velocities in a two-lane configuration, where both leading and trailing vehicles were in the same lane as discussed in Vegendla *et al.* [1]. The aerodynamic drag benefit drops with an increase of vehicle separation distances. In Figure 8.6iii, the aerodynamic drag benefit was independent of the vehicle separation distance due to negligible influence on each other.

(i)

(ii)

Figure 8.7. Static pressure across the center of the truck height (1.27m) at different separation distances of 30ft, 90ft and 150ft; (i) One lane (Figure 8.2i), (ii) Two lane: Leading and Trailing Vehicles are in same lane (Figure 8.2ii) and (iii) Two lane: Leading and Trailing Vehicles are in two separate lanes (Figure 8.2iii).

As seen in Figures 8.7(i-ii), the static pressure increases with vehicle separation distances and thus leads to higher aerodynamic drag values due to impact of the higher velocities on vehicle surfaces (Figure 8.6i). Similar observations were discussed in Figures 8.5i and 8.6i. In Figure 8.7iii, the static pressure was independent (negligible change) at front portion of the trailing vehicle cabin and thus leads to a negligible change in aerodynamic drag values with an increase of vehicle separation distances. In all configurations, the leading vehicle doesn't show any significant effect with an increase of vehicle separation distances.

Figure 8.8. Comparison of normalized aerodynamic drag along the trailing vehicle length for two different vehicle-to-vehicle (V2V) distances of 30 and 150 ft; (i) One lane (Figure 8.2i), (ii) Two lane: Leading and Trailing Vehicles are in same lane (Figure 8.2ii) and (iii) Two lane: Leading and Trailing Vehicles are in two separate lanes (Figure 8.2iii).

As seen in Figure 8.8, lower aerodynamic drag values were observed in trailing vehicle at lower vehicle separation distances. The lower aerodynamic drag values were mainly from vehicle cabin due to low static pressure and low air mass flow velocities. The higher drop in aerodynamic drag values in two-lane when compared to one-lane was due to lower blockage ratio in two lanes (Figure 8.8ii). Furthermore, higher aerodynamic drag values were observed in Figure 8.8iii when compared to Figure 8.8ii (two-lane configuration). This is due to vehicles following in two separate lanes that mitigates the influence on each other (leading and trailing vehicle).

8.4. Conclusions

A medium-duty tractor-trailer aerodynamic drag results were analyzed for two different road-tunnel dimensions including one and two-lane road traffic conditions. This study included the aerodynamic drag performance for both single and multiple vehicles.

In a single vehicle, the aerodynamic drag drop was observed with an increase of height clearance between vehicle and road-tunnel in both single- and two-lane traffic.

In vehicle platooning, the aerodynamic drag analysis was performed for two identical medium-duty vehicles, i.e. leading and trailing vehicle. In the trailing vehicle, a significant aerodynamic drag drop was observed at lower vehicle separation distances in both single- and two-lane traffic. Further, with an increase of the vehicle separation distances, the aerodynamic drag savings dropped significantly in one-lane traffic and negative performance was observed from 60ft separation distance onwards.

In two-lane traffic where one vehicle is behind the other without a lateral separation, the predicted aerodynamic drag benefit was up to 60% and 44% at vehicle separation distance of 30ft and 150ft, respectively. The aerodynamic drag changes were negligible with an increase of vehicle separation distance where two vehicles were separated laterally with 1.2m distance. In all configurations, the leading vehicle aerodynamic drag changes were negligible irrespective of the vehicle separation distance.

9. References

- [1] Vegendla, S.N.P., Sofu, T., Saha, R., Kumar, M.M and Hwang, L.K., “Investigation of Aerodynamic Influence on Truck Platooning,” 2015, SAE International, 2015-01-2895.
- [2] Vegendla, P., Sofu, T., Saha, R., Hwang, L.K. Comparison of Aerodynamic drag and Underhood Thermal Analysis of Two Heavy-duty Vehicles. *Int. Journal of Aerodynamics*, Vol. 5, No. 2, 2016; pp:105-124.
- [3] Lee, S., “Heavy Vehicle Systems Optimization Program,” *Annual Progress Report, US DOE*, 2007.
- [4] Sirenko, V., Pavlovsky’s and Rohatgi, S.U., “Methods of Reducing Vehicle Aerodynamic Drag,” ASME Summer Heat Transfer Conference, Puerto Rico, USA, 2012, BNL-98039-2102-CP.
- [5] Song, K.S., Kang, S.O., Jun, S.O., Park, H.I., Kee, J.D., Kim, K.H and Lee, D.H., “Aerodynamic Design Optimization of Rear Body Shapes of a Sedan for Drag Reduction,” *International Journal of Automotive Technology*, 13(6): 905–914, 2012.
- [6] Muyl, F., Dumas, L and Herbert, V., “Hybrid Method for Aerodynamic Shape Optimization in Automotive Industry,” *Computers & Fluids* 33 (5-6): 849–858, 2004.
- [7] Al-Obaidi, A.Sh.M, and Sun, L.C., “Calculation and Optimization of the Aerodynamic Drag of an Open-Wheel Race Car,” *Journal of Engineering Science and Technology EURECA* 2013 (8):1-15, 2014.
- [8] Ahmed, H. and Chacko, S., “Computational Optimization of Vehicle Aerodynamics,” *Annals of Daaam for 2012 & Proceedings of the 23rd International Daaam Symposium*, 23, (1), issn 2304-1382.
- [9] Lee, S., “Advanced Vehicle Technology Analysis and Evaluation Activities and Heavy Vehicle Systems Optimization Program,” *Annual Progress Report US DOE*, 2008.
- [10] Liao, Y.G and Quail, A.M., “Experiment and Simulation of Medium-Duty Tactical Truck for Fuel Economy Improvement,” *Energies*, 4:276-293; 2011, doi: 10.3390/en4020276.
- [11] Wood, R.M and Bauer, S.X.S., “Simple and Low-Cost Aerodynamic Drag Reduction Devices for Tractor-Trailer Trucks,” *SAE International*, 2003-01-3377, 2003.
- [12] Chowdhury, H., Moria, H., Ali, A., Khan, I., Alam, F and Simon Watkins, “A Study on Aerodynamic Drag of a Semi-Trailer Truck,” *Procedia Engineering* 56: 201-205, 2013.
- [13] Driant, T., Fellouach, H., Moreau, S., Desrochers, A., Remaki, L., “Numerical Simulation And Wind Tunnel Measurements On A Tricycle Wheel Sub-System,” *IJESMS*, 2013.5:159-167.
- [14] STAR-CCM+-v9.06.011, 2014, User Guide.
- [15] PowerFLOW-4.4d, 2013, Release Notes. EXA.
- [16] Connor, C., Kharazi, A., Walter, J. and Martindale, B. (2006) ‘Comparison of Wind Tunnel Configurations for Testing Closed-Wheel Race Cars: A CFD Study’. *SAE Paper 2006-01-3620*.
- [17] Ross, M., Salari, K., Ortega, J., Castellucci, P., Pointer, D., Browand, F., Ross, J., Storms, B.’DOE project on heavy vehicle aerodynamic drag’. *Annual Progress Report US DOE*, 2007.

- [18] Sofu, T. (2013) ‘Aerodynamics and underhood thermal analysis of heavy/medium vehicles’, US DOE Annual Report for Vehicle Technology Program, DOE/EE-1023. energy.gov/sites/prod/files/2014/04/f15/2013_vsst_report.pdf
- [19] Vegendla, S.N.P., Sofu, T. and Lee, S (2014). ‘Aerodynamics and underhood thermal analysis of heavy/medium vehicles’ US DOE Annual Report -Vehicle and System Simulation and Testing R&D, DOE-EE-1165. 2014, pp: 345-349. energy.gov/sites/prod/files/2015/08/f25/vehicle_systems_annual_report_080415.pdf.
- [20] Guidelines for aerodynamic assessment of medium and heavy commercial ground vehicles using computational fluid dynamics, SAE Paper, 2013-09-17, J2966.
- [21] Hoerner, S. F.: Fluid Dynamic Drag. 1965
- [22] Himeno, R., Fujitan, K., Nishikawa, J., Watanabe, Y.,and Michitoshi, T.: Practical Economic Aspects of Tractor/Trailer Aerodynamics. SAE 760103, 1976.
- [23] Lachman, G. V.: Boundary Layer and Flow Control.Vol. I Pergamon Press, N. Y., 1961.
- [24] P.M. Van Leeuwen, Computational Analysis of Base Drag Reduction Using Active Flow Control. MS thesis, Aerospace Engineering, Delft University of Technology, 2009.
- [25] Davis, F., Veling, T., Caltrider, J., and Madhavar, R., “Impact of Computer Aided Engineering on Ford Motor Company Light Truck Cooling Design and Development Processes,” SAE Technical Paper 932977, 1993, doi:10.4271/932977.
- [26] Fellague, K., Hu, S., and Willoughby, D., “Determination of the Effects of Inlet Air Velocity and Temperature Distributions on the Performance of an Automotive Radiator,” SAE Technical Paper 940771, 1994, doi:10.4271/940771.
- [27] Hsu, I. and Schwartz, W., “Simulation of the Thermal Environment Surrounding an Underbody Fuel Tank in a Passenger Vehicle Using Orthogonally Structured and Body-Fitted Unstructured CFD Codes in Series,” SAE Technical Paper 950616, 1995, doi:10.4271/950616.
- [28] Srinivasa, V., S, R., and Shome, B., "Design of Experiments Enabled CFD Approach for Optimizing Cooling Fan Performance," SAE Technical Paper 2014-01-0658, 2014, doi:10.4271/2014-01-0658.
- [29] Mehravaran, M. and Zhang, Y., "Optimizing the Geometry of Fan-Shroud Using CFD," SAE Technical Paper 2015-0101336, doi:10.4271/2015-01-1336.
- [30] GILES, M. B.; PIERCE N. A., An Introduction to the Adjoint Approach to Design, Flow, Turbulence and Combustion, 65, 393-415, 2000.
- [31] Hinterberger, C. and Olesen, M., "Automatic Geometry Optimization of Exhaust Systems Based on Sensitivities Computed by a Continuous Adjoint CFD Method in OpenFOAM," SAE Technical Paper 2010-01-1278, 2010, doi:10.4271/2010-01-1278.
- [32] Han, T., Kaushik, S., Gin, R., and Bot, E., "Adjoint Method for Aerodynamic Shape Improvement," SAE Technical Paper 2012-01-0167, 2012, doi:10.4271/2012-01-0167.
- [33] Rainer, M., Haase, G., Basara, B., and Offner, G., "Shape Optimization by an Adjoint Solver based on a near-wall Turbulence Model," SAE Technical Paper 2015-01-1358, 2015, doi:10.4271/2015-01-1358.
- [34] Magazoni, F., Buscariolo, F., Maruyama, F., Alves, J. et al., "Aerodynamic Shape Improvement for Driver Side View Mirror of Hatchback Vehicle using Adjoint

- Optimization Method," SAE Technical Paper 2015-36-0156, 2015, doi:10.4271/2015-36-0156.
- [35] Smith, J., Mihelic, R., Gifford, B., Ellis, M., "Aerodynamic Impact of Tractor-Trailer in Drafting Configuration," SAE technical paper, 2014, DOI:10.4271/2014-01-2436.
- [36] Lammert, MP., Duran, A., Diez, J., Burton, K., et al., Effect of Platooning on Fuel Consumption of Class 8 Vehicles Over a Range of Speeds, Following Distances, and Mass," SAE technical paper, 2014, DOI:10.4271/2014-01-2438.
- [37] Johansson, KH., "Cooperative driving for road goods transportation: Optimization and control," The 33rd Chinese Control Conference, July 28-30, 2014, Nanjing.
- [38] Lammert, M., Gender, J., Reducing fuel consumption through semi-automated platooning with class 8 tractor trailer combinations. Automated Vehicle Symposium, 2014, San Francisco, CA.
- [39] Romberg, G., Chianese, F., Lajoie, R., "Aerodynamics of Race Cars in Drafting and Passing Situations," SAE Paper 1971-02-01, 1972, doi:10.4271/710213
- [40] Zabat, M., Stabile, N., Frascaroli, S., Browand, F., 1995, The aerodynamic performance of platoons; A final report", PATH Report UCB-ITS-PRR-95-35
- [41] Bogdan, M., Browand, F., "The Aerodynamic Forces on Misaligned Platoons," California PATH Research Report UCB-ITS-PRR-98-4, 1998, ISSN: 1055-1425.
- [42] Alam, Al A., Gattami, A., Johansson, K., "An Experimental Study on the Fuel Reduction Potential of Heavy Duty Vehicle Platooning," presented at 13th International IEEE Conference on Intelligent Transportation Systems (ITTSC), 2010, doi:10.1109/ITSC.2010.5625054.
- [43] Vegendla, S.N.P., Heynderickx, G.J., Marin, G.B., "Micromixing effects on series-parallel and autocatalytic reactions in a turbulent single-phase gas flow" Chemical Engineering Science, 2010, pp:4621-4629.
- [44] <http://www.prnewswire.com/news-releases/global-fuel-consumption-market-for-transportation-industry-2015-2019-300056696.html>
- [45] Jang, HM and Chen, F. (2002) 'On the determination of the aerodynamic coefficients of highway tunnels'. *Journal of Wind Engineering and Industrial Aerodynamics*. Vol.90 No.8, pp:869-896.
- [46] Leong, JC., Tsai, CH and Li, CT. (2009) 'Development of training software for Hsuehshan tunnel operators'. <http://www.iiis.org/CDs2008/CD2009SCI/MEI2009/PapersPdf/M083HY.pdf>
- [47] Colella, F. (2010) Multi scale modeling of tunnel ventilation flows and fires. PhD Thesis, Dipartimento di Energetica, Politecnico di Torino, Italy.
- [48] Koc, G. (2012) Utilization of neural networks for simulation of vehicle induced flow in tunnel systems. PhD Thesis, Mechanical Engineering Department, Middle East Technical University, Turkey.
- [49] Wood, R. (2015) 'Reynolds Number Impact on Commercial Vehicle Aerodynamics and Performance" SAE Int. J. Commer. Veh. Vol.8 No.2, pp:590-667.
- [50] Surcel, M and Shetty, M. (2015) 'The impact of design, position and combination of aerodynamic devise on drag and fuel consumption'. SAE technical paper, 2015-01-2894.
- [51] Van Raemdonck, G. (2015) 'The aerodynamic performance of a relatively short rear-end drag reduction device for a semi-trailer'. SAE-2016 Commercial Vehicle Engineering Congress.

Acknowledgments

This work was conducted under CRADA contract (C1201001) and supported by the U.S. Department of Energy, Office of Energy Efficiency and Renewable Energy, under contract # DE-AC02-06CH11357. Thanks to US DOE Vehicle System Optimization Program Manager 'Lee Slezak' for his support throughout this work.

We gratefully acknowledge the computing resources provided on Blues and/or Fusion, a high-performance computing cluster operated by the Laboratory Computing Resource Center at Argonne National Laboratory.

Appendix A

A.1. Yaw Averaged Drag (YAD)

$$YAD = \frac{\sum_{i=1}^N C_{D,i}^{platooning}}{\sum_{i=1}^N C_{D,i}^{no-traffic}}$$

$$YAD (\%) = [1 - YAD]. 100\%$$

where, N and C_D is the number of Yaw angles and aerodynamic drag coefficient, respectively. In the present study, YAD was based on the average result of three different yaw angles at 0°, 6° and 12°.

A.2. Estimated fuel savings

The estimated fuel savings were assumed to be roughly half of the YAD reduction. In the present study, the vehicle speed and the torque were kept at constant.

$$\text{Fuel savings (\%)} = \frac{[1 - YAD]. 100\%}{2}$$

A.3. Aerodynamic Drag Force Equation

$$F_D = \frac{1}{2} \rho u^2 C_D A$$

where, F_D , ρ , u , C_D , A are drag force, mass density, flow velocity and reference Area, respectively

Appendix B

As seen in Table B.1, the predicted aerodynamic drag results are grid independent for both yaw angles [0 and 6°].

Table B.1. Comparison of aerodynamic drag for coarse and fine grid.

Grid Count [Million]	Normalized Aerodynamic Drag [-]	
	Yaw = 0 deg	Yaw = 6 deg
28	1.000	1.1137
35	0.999	1.1139

Energy Systems Division

Argonne National Laboratory

9700 South Cass Avenue, Bldg. 208

Argonne, IL 60439

www.anl.gov

Argonne National Laboratory is a U.S. Department of Energy
laboratory managed by UChicago Argonne, LLC