SURFACE VEHICLE RECOMMENDED PRACTICE

J2966™ JUN2021

Issued 2013-09
Revised 2021-06

Superseding J2966 APR2017

Guidelines for Aerodynamic Assessment of Medium and Heavy Commercial Ground Vehicles Using Computational Fluid Dynamics

SAENORM.COM : Click to view the full PDF of j2966_202106

RATIONALE

This document has been updated to provide clear requirements and guidance for a successful aerodynamic assesment of medium and heavy commercial ground vehicles using common computational fluid dynamics methods.

1. SCOPE

This document outlines general requirements for the use of CFD methods for aerodynamic simulation of medium and heavy commercial ground vehicles weighing more than 10000 pounds. The document provides guidance for aerodynamic simulation with CFD methods to support current vehicle characterization, vehicle development, vehicle concept development, and vehicle component development. The guidelines presented in the document are related to Navier-Stokes and Lattice-Boltzmann based solvers.

This document is only valid for the classes of CFD methods and applications mentioned. Other classes of methods and applications may or may not be appropriate to simulate the aerodynamics of medium and heavy commercial ground vehicle weighing more than 10000 pounds.

1.1 Purpose

The purpose of this document is to standardize the use of Navier-Stokes and Lattice-Boltzmann CFD for aerodynamic simulation for medium and heavy commercial ground vehicles weighing more than 10000 pounds and to guide CFD users to utilize CFD tools effectively and efficiently. The recommendations are grouped under three main sections: (1) CFD model setup, (2) solver setup, and (3) data processing and communication. The model setup section aims to address issues related to model preparation and physical accuracy. It explains what physical properties need to be modeled and how they need to be configured. The solver setup section deals with appropriate solver selection and different options in turbulence modeling, boundary layer representation, and discretization. The last section, data processing and communication, gives guidance about how the CFD results need to be presented and interpreted. The section also contains recommendations on information collection and processing related to CFD simulations.

1.2 Document Use

This document is structured to service both the novice and the expert user.

SAE Executive Standards Committee Rules provide that: "This report is published by SAE to advance the state of technical and engineering sciences. The use of this report is entirely voluntary, and its applicability and suitability for any particular use, including any patent infringement arising therefrom, is the sole responsibility of the user."

SAE reviews each technical report at least every five years at which time it may be revised, reaffirmed, stabilized, or cancelled. SAE invites your written comments and suggestions.

Copyright © 2021 SAE International

All rights reserved. No part of this publication may be reproduced, stored in a retrieval system or transmitted, in any form or by any means, electronic, mechanical, photocopying, recording, or otherwise, without the prior written permission of SAE.

TO PLACE A DOCUMENT ORDER: **Tel: 877-606-7323 (inside USA and Canada)**
Tel: +1 724-776-4970 (outside USA)
Fax: 724-776-0790
Email: CustomerService@sae.org
SAE WEB ADDRESS: http://www.sae.org

For more information on this standard, visit
https://www.sae.org/standards/content/J2966_202106

2. REFERENCES

2.1 Applicable Documents

The following publications form a part of this specification to the extent specified herein. Unless otherwise indicated, the latest issue of SAE publications shall apply.

2.1.1 SAE Publications

Available from SAE International, 400 Commonwealth Drive, Warrendale, PA 15096-0001, Tel: 877-606-7323 (inside USA and Canada) or +1 724-776-4970 (outside USA), www.sae.org.

SAE J1252 SAE Wind Tunnel Test Procedure for Trucks and Buses

SAE R-177 Hucho, W-H., Aerodynamics of Road Vehicles, SAE International, ISBN: 978-0-7680-0029-0, 1988

2.2 Related Publications

The following publications are provided for information purposes only and are not a required part of this SAE Technical Report.

2.2.1 SAE Publications

Available from SAE International, 400 Commonwealth Drive, Warrendale, PA 15096-0001, Tel: 877-606-7323 (inside USA and Canada) or +1 724-776-4970 (outside USA), www.sae.org.

Bayraktar, I. and Bayraktar, T., "Guidelines for CFD Simulations of Ground Vehicle Aerodynamics," SAE Technical Paper 2006-01-3544, 2006, https://doi.org/10.4271/2006-01-3544.

Duncan, B., Kandasamy, S., Sbeih, K., Lounsberry, T. et al., "Further CFD Studies for Detailed Tires using Aerodynamics Simulation with Rolling Road Conditions," SAE Technical Paper 2010-01-0756, 2010, https://doi.org/10.4271/2010-01-0756.

Heinzelmann, B., Indinger, T., Adams, N., and Blanke, R., "Experimental and Numerical Investigation of the Under Hood Flow with Heat Transfer for a Scaled Tractor-Trailer," SAE Int. J. Commer. Veh. 5(1):42-56, 2012, https://doi.org/10.4271/2012-01-0107.

Holloway, S., Leylek, J., York, W., and Khalighi, B., "Aerodynamics of a Pickup Truck: Combined CFD and Experimental Study," SAE Int. J. Commer. Veh. 2(1):88-100, 2009, https://doi.org/10.4271/2009-01-1167.

Horrigan, K., Duncan, B., Sivakumar, P., Gupta, A. et al., "Aerodynamic Simulations of a Class 8 Heavy Truck: Comparison to Wind Tunnel Results and Investigation of Blockage Influences," SAE Technical Paper 2007-01-4295, 2007, https://doi.org/10.4271/2007-01-4295.

Söderblom, D., Elofsson, P., Hjelm, L., and Lofdahl, L., "Experimental and Numerical Investigation of Wheel Housing Aerodynamics on Heavy Trucks," SAE Int. J. Commer. Veh. 5(1):29-41, 2012, https://doi.org/10.4271/2012-01-0106.

Wood, R., "Reynolds Number Impact on Commercial Vehicle Aerodynamics and Performance," SAE Int. J. Commer. Veh. 8(2):590-667, 2015, https://doi.org/10.4271/2015-01-2859.

2.2.2 AIAA Publications

Available from American Institute of Aeronautics and Astronautics, 1801 Alexander Bell Drive, Suite 500, Reston, VA 20191-4344, Tel: 703-264-7500, www.aiaa.org.

AIAA 2000-2306 Spalart, P.R., "Trends in Turbulence Treatments," June 2000

AIAA 93-2906 Menter, F.R., "Zonal Two Equation k-ω Turbulence Models for Aerodynamic Flows," AIAA Paper 93-2906, 1993

SAENORM.COM : Click to view the full PDF of j2966_202106

AIAA 98-3007 Poirier, D. et.al. "The CGNS System," Fluid Dynamics Conference, 29th, Albuquerque, NM, June 15-18, 1998

AIAA-G-077-1998 Guide for the Verification and Validation of Computational Fluid Dynamics Simulations, ISBN: 978-1-56347-354-8

2.2.3 ASME Publications

Available from ASME, P.O. Box 2900, 22 Law Drive, Fairfield, NJ 07007-2900, Tel: 800-843-2763 (U.S./Canada), 001-800-843-2763 (Mexico), 973-882-1170 (outside North America), www.asme.org.

ASME AMR v.62.4 Alfonsi, G., "Reynolds-Averaged Navier-Stokes Equations for Turbulence Modeling," Applied Mechanics Reviews, Vol. 62, No. 4., 2009, 040802, doi:10.1115/1.3124648

ASME JFE v.130 Celik, B.I. et.al., "Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications, Journal of Fluids Engineering, v. 130, July 2008, doi: 10.1115/1.2960953

ASME V&V 20-2009 Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer, ISBN: 9780791832097

2.2.4 Other Documents

AFOSR 1997 Spalart, P.R., Jou, W.H., Strelets, M., Allmaras, S.R., "Comments on the feasibility of LES for wings, and on a Hybrid RANS/LES approach," 1997 Advances in DNS/LES, Proc. 1st AFOSR Inter. Conf. on DNS and LES, (NY: Greyden Press)

ARFM 1998 Chen, S. and Doolen, G., "Lattice Boltzmann Method for Fluid Flows," Ann. Rev. Fluid Mech., Vol. 30, 1998, pp. 329-364

CF v.24.3 Shih, T.H., Liou, W.W., Shabbir, A., Yang, Z., and Zhu, J., "A New k-epsilon Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation," Computers Fluids, 24(3), 227-238, 1995

DARCY 1856 Darcy, H., "Les Fontaines Publiques de la Ville de Dijon," Appendix - Note D, Dalmont, Paris, 1856

JCP v.62 Issa, R.I., Gosman, A.D., and Watkins, A.P., "The computation of compressible and incompressible recirculating flows by a noniterative implicit scheme," Journal of Computational Physics, Vol. 62, pp. 66-82 (1986)

JHTMT, v15 Patankar, S.V. and Spalding, D.B., "A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows," International Journal of Heat and Mass Transfer, Vol. 15, Issue 10, pp. 1787-1806, Oct. 1972

Landahl MS 1992 Landahl, M.T. and Mollo-Christensen, E., "Turbulence and Random Processes in Fluid Mechanics," Cambridge, 2ed, 1992

NHT v.7 Van Doormaal, J.P. and Raithby, G.S., "Enhancements of the SIMPLE method for predicting incompressible fluid flows," Numerical Heat Transfer, Vol. 7, pp. 147-163 (1984)

OUP 2001 Succi, S., "The Lattice Boltzmann Equation for Fluid Dynamics and Beyond," Numerical Mathematics and Scientific Computation, Oxford University Press, August 2011

S v.301 Chen, H., Kandasamy, S., Orszag, S., Shock, R., Succi, S., and Yakhot, V., "Extended Boltzmann Kinetic Equation for Turbulent Flows," Science, Vol. 301, August 2003, pp. 633-336

SAENORM.COM : Click to view the full PDF of j2966_202106

3. DEFINITIONS AND ACRONYMS

AIAA: American Institute of Aeronautics and Astronautics.

ASME: American Society of Mechanical Engineers.

CAD: Computer-aided design.

CAE: Computer-aided engineering.

C_D: Drag coefficient, normalized drag force.

CFD: Computational fluid dynamics.

CFL: Courant number.

CGNS: CFD general notation system.

DDES: Delayed detached eddy simulation.

DES: Detached eddy simulation.

DRAG: A force that opposes the forward motion of the vehicle. The total force is composed of a pressure drag and a viscous drag component.

GCI: Grid convergence index.

H: Vehicle height based on projection plane in air flow direction.

IEE: Iteration error estimation.

IGES: Initial graphics exchange specification data format.

ISO: International Organization for Standardization.

L: Length of the vehicle.

LES: Large eddy simulation.

LIFT: The aerodynamic force that acts perpendicular in a vertical direction to the forward motion of the vehicle.

MODEL: Geometric representation of a vehicle.

RANS: Reynolds-averaged Navier-Stokes.

SBES: Stress-blended eddy simulation.

SIDE FORCE: Aerodynamic force acts parallel to ground plane and perpendicular to the forward motion of the vehicle.

SIMULATION: Computer reproduction of a physical event.

STEP: Standard for the exchange of product model data (defined in ISO 10303 document).

W: Vehicle width based on projection plane in air flow direction.

y^+: Non-dimensional wall distance for a wall-bounded flow.

YAW ANGLE: Angle between the vehicle body longitudinal axis and the component of the relative air velocity vector in the ground plane.

SAENORM.COM : Click to view the full PDF of j2966_202106

4. CFD MODEL SETUP

As a general principle, numerical simulations should replicate the complete physical system under investigation. However, modeling excessive vehicle geometric details does not necessarily improve the fidelity and quality of the aerodynamic simulation, and it could increase simulation time. Therefore, the vehicle geometric features in an aerodynamic simulation need to be selected carefully based on their relevance to the aerodynamic investigation. For example, simplifying underhood components for an external aerodynamic simulation of a wind deflector can reduce the total mesh count and simulation time without influencing the overall flow field. This section contains guidelines for the aerodynamic analysis of medium and heavy commercial ground vehicles. Users of this document need to consider their own applications when using the guidelines.

The guidelines in this section are divided into two subsections according to the simulation environment. Requirements related to general model setup are listed in 4.1. Specific items related to CFD simulations in a wind tunnel environment are grouped in 4.2.

4.1 General On-Road Simulation and Wind Tunnel Simulations

Generic requirements that could be relevant for either on-road or wind tunnel simulations are grouped under this section. Wind tunnel simulation specific items are listed in the next subsection.

4.1.1 The simulation model shall include all aerodynamically significant physical characteristics of the subject test article.

4.1.2 Deformable parts such as rubber mud flaps, under-hood seals, and recirculation shields shall be modeled to represent their suitable shape in operating condition.

4.1.3 Recommended minimum computational domain size for on-road simulations is depicted in Figure 1. It should be noted that the values of W and H are to be calculated based on the vehicle geometric models' projection area in flow direction and $G = \sqrt{W\ x\ H}$. In order to minimize the influence of the domain boundaries, the domain shall be extended at a minimum 10xG upstream and 30xG downstream of the complete vehicle. The domain width shall be at least 20xG and the height should be at least 10xG, resulting in no more than 0.5% blockage (ratio of the vehicle frontal area to the domain cross section). Each yaw simulation shall meet this maximum blockage requirement.

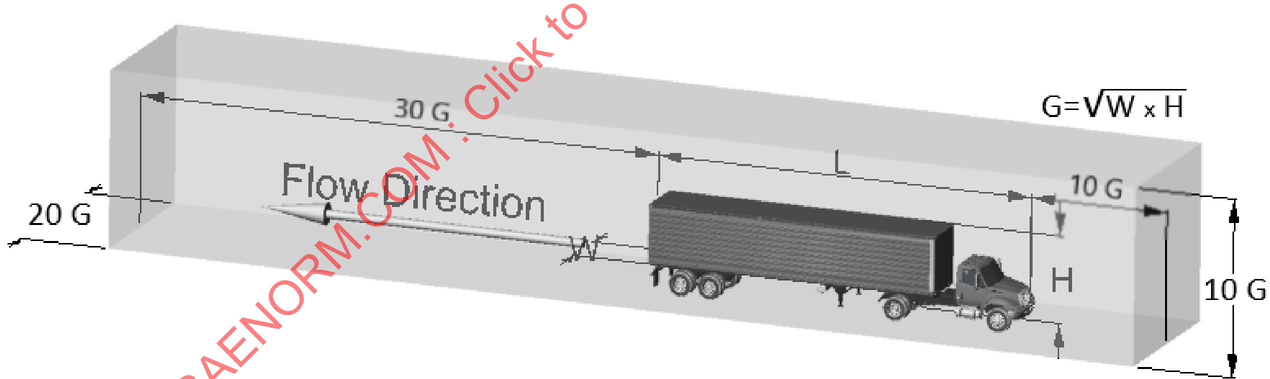

Figure 1 - Recommended minimum computational domain size for on-road simulations; W and H are to be calculated based on the vehicle projection area in wind direction (figure not drawn to scale)

4.1.4 Appropriate boundary conditions should be specified on the outer boundary surfaces of the computational domain. In general, velocity inlet and zero-gradient pressure outlet are acceptable conditions for the inlet/outlet setup. The domain sides and top can be modeled with opening, symmetry, or free-slip (sliding) wall boundary conditions to minimize any far-field influence on the model for on-road simulations. Reynolds number for the numerical model should match the physical model Reynolds number to ensure same environmental conditions.

4.1.5 The simulation model shall match the physical article under investigation and may include the open grill, the cooling package, and detailed underhood components, where applicable. Pressure drop in porous media (cores) in the cooling package can be represented with Darcy's Law (DARCY 1856) or a polynomial function.

SAENORM.COM : Click to view the full PDF of j2966_202106

4.1.6 It is recommended to consider the effect of the cooling fan. Stationary fan geometry or porous jump boundary conditions are acceptable; however, modeling the fan as a rotating (or free-spinning) component is recommended. If the CFD simulation is being done to replicate a wind tunnel test, the cooling fan modeling should be consistent with the wind tunnel setup.

4.1.7 The numerical model shall represent physical system at the test condition for main features like ground clearance, tractor-trailer gap, wheelbase, and roof fairing dimensions.

4.2 Wind Tunnel Simulation

4.2.1 The geometric model for wind tunnel simulation includes the vehicle model and the test section geometry including test section walls, model support system and any pressure relief slots. A general wind tunnel configuration is depicted in Figure 2.

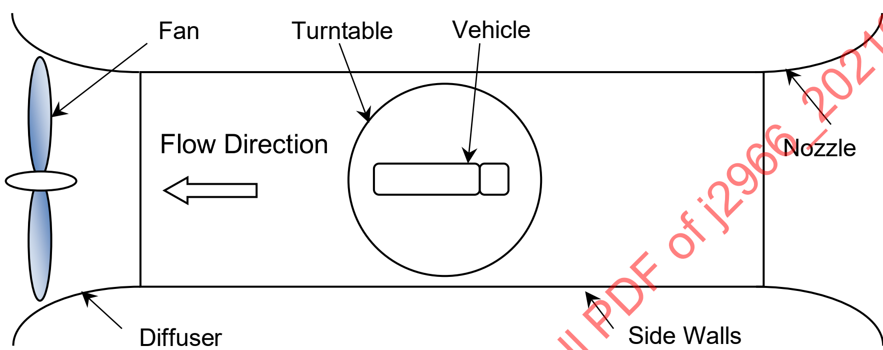

Figure 2 - General view of a wind tunnel test configuration

4.2.2 The vehicle geometric model shall accurately represent the physical properties of the vehicle installed in the wind tunnel to the greatest extent possible. Examples include ground clearance (at all four corners), tractor-trailer gap, tire contact patch, wheelbase, roof fairing dimensions, etc.

4.2.3 Open jet wind tunnel simulations should include upstream nozzle and downstream collector geometry of the wind tunnel facility test section.

4.2.4 Boundary layer state and thickness on both the wind tunnel floor and on the vehicle model surface should be modeled to replicate the tunnel condition. If boundary layer suction or blowing is employed for the wind tunnel floor boundary layer, an appropriate wall boundary condition should be applied into the same area to accurately represent the wind tunnel operational conditions. If there is a moving belt present, moving wall boundary condition needs to be implemented at the same size and at the same location. It should be noted that most CFD simulations assume a fully turbulent flow; therefore, if there is a requirement to model laminar boundary layer or laminar-turbulent transition, a transition turbulence model needs to be selected in the solver setting and verified against wind tunnel measurements where possible.

4.2.5 Supporting structures such as mounting boards, support struts and air bearings shall be included to the geometric model.

4.2.6 Comparisons between wind tunnel data and aerodynamic simulation force coefficients should be made using wind tunnel aerodynamic data that have not been corrected for wind tunnel flow variations such as buoyancy.

4.2.7 An empty-domain CFD simulation should be performed based on the empty wind tunnel calibration when available to adjust inlet velocity boundary condition for the CFD model. If an empty wind tunnel velocity calibration is not available, then velocity measurements from at least one location should be taken during the wind tunnel test to adjust the inlet velocity for the CFD simulation.

4.2.8 It is recommended to specify turbulence intensity and turbulence length scale in the CFD model to match the wind tunnel condition.

SAENORM.COM : Click to view the full PDF of j2966_202106

4.2.9 The wind tunnel aerodynamic simulation shall be performed with the vehicle model positioned at the same location within the wind tunnel test section as that for the wind tunnel test.

5. SOLVER SETUP

The solver setup section covers topics related to appropriate solver selection and different options in turbulence modeling, boundary layer representation, and discretization. In general, the flow field around medium and heavy commercial trucks is unsteady, and unsteady CFD simulations better capture this feature. However, steady-state CFD simulations can also be used successfully in many design iteration studies where short turn-around time is required and the design modifications are not changing or interacting with the unsteady effects. It is recommended to follow published guidelines for solver setup, unless different guidelines are specified in this document. In most cases, the flow field around the truck is assumed to be fully turbulent in the CFD simulations. The guidelines listed in this section currently cover only Navier-Stokes and Lattice-Boltzmann based solvers.

5.1 General Requirements for Solvers

5.1.1 The grid for a full vehicle CFD model shall have at least 60 million volume elements with multiple grid refinement regions at high flow gradient and wake regions (such as front and back of the vehicle, around mirrors, grills, louvers, underbody, tires, and leading and trailing edges, etc.). Including additional refinement regions along the flow separation side of the vehicle when simulating at yaw conditions is also required.

5.1.2 All solver parameters (grid resolution, turbulence model, etc.) should remain identical for each design evaluation.

5.1.3 For steady-state simulations, grid convergence index (GCI) and iteration error estimation (IEE) (as described in ASME JFE v.130) should be calculated at least once for the subject model, or representative geometry, to ensure adequate grid refinement and comparable grid quality between the projects. The calculations need to be repeated when there is a change in the grid density, solver settings, or software package. The GCI and IEE results are reported with the case results. Calculations procedure for GCI and IEE are given in Attachment A.

5.1.4 For unsteady simulations, time averaging is necessary to obtain relevant aerodynamic properties. However, to limit the influence of initial conditions, a sufficient duration of the initial transient data should be discarded before averaging is applied. A consistent methodology should be applied to determine this duration. Beyond this initial transient, the simulation typically needs to run for at least four convective flow passes, defined as the time it takes for air at free stream velocity to traverse the vehicle length. Data should be collected every time step and other than the initial transient, no data may be discarded, i.e., the time averaging window must extend to the end of the simulation (or the end of recorded data). Figure 3 illustrates this time averaging procedure. Statistical tools (such as the 95% confidence interval) may be applied on the data inside the averaging window to obtain the uncertainty associated with the average. Engineering judgment still applies since averaging times can be highly model dependent. Longer run times are recommended when force coefficient time history indicates low frequency oscillations or when the uncertainty is too large.

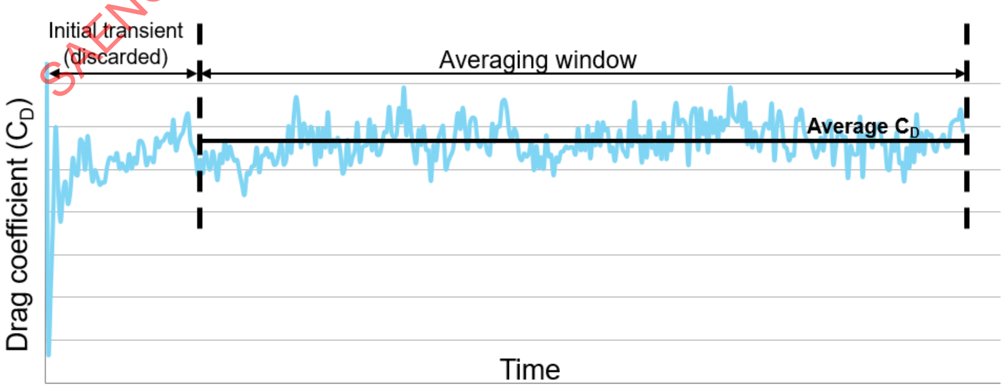

Figure 3 - Example analysis of an unsteady simulation

SAENORM.COM : Click to view the full PDF of j2966_202106

5.1.5 It is recommended to use measured free-stream turbulence quantities for wind tunnel simulations. If they are unavailable, use an inlet turbulence intensity between 0.2% and 1% along with a length scale equivalent to the hydraulic diameter (if there is a mesh at the wind tunnel entrance), or along with a turbulence viscosity ratio between 200 and 500. Zero-pressure gradient is generally sufficient for the outlet boundary condition. For simulating on-road conditions, higher turbulence intensities and length scales may be appropriate depending on the environment under consideration.

5.2 Requirements for Navier-Stokes Based Solvers

Requirements listed in this section are applicable to the Navier-Stokes based solvers. Solver setting values given below are for guidance only, and software vendors might be able to provide more specific values for their software packages.

5.2.1 Minimum second order discretization is required for all Navier-Stokes based solvers.

5.2.2 Appropriate y^+ values depend on Reynolds number and selected wall treatment method. In general, y^+ should remain below 300 on vehicle surfaces (between 30 and 100 is recommended for attached flow on the external surfaces of the vehicle) when a wall function is utilized. When wall functions are not utilized, a y^+ value between one and five is recommended to resolve the viscous sub-layer. General view of a near-wall grid is represented in Figure 4. The y^+ value is determined by the cell height on the wall (H1). It is recommended to keep the growth ratio (H2/H1) below 1.2 in the boundary-layer region on the external surfaces of the vehicle. In general, element aspect ratio (cell width/cell height) should not be larger than 50 away from the walls. Elements close to walls could have much higher aspect ratio (W1/H1) values, especially when the boundary layer is meshed.

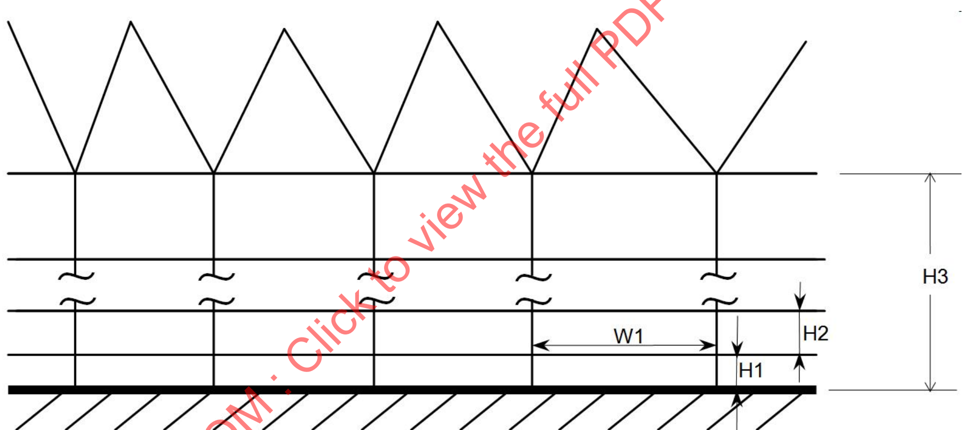

Figure 4 - General view of near wall grid distribution

5.2.3 When the boundary layer is resolved with meshing (i.e., no wall functions are used), the flow field needs to be checked to make sure the boundary layer (H3) is covered with at least ten or more nodes in the normal direction. This could be checked with plotting turbulence viscosity contours, which should show a peak in the middle of the boundary layer.

5.2.4 Requirements for Steady-State Navier-Stokes Based Solvers

Requirements listed in this section are applicable to steady-state Navier-Stokes based solvers.

5.2.4.1 For steady-state calculations, using a coupled solver is recommended primarily for speeding up convergence. For fastest convergence, it is recommended starting with a CFL value between 100 and 150. As the solution progresses it may be possible to raise the CFL up to 250 to aid convergence speed up. If the case is showing signs of convergence issues, lowering CFL <50 at the beginning of iterations or switching to a segregated solver is recommended. Lowering the explicit relaxation factors between 0.25 and 0.5 may also help improve convergence. Pseudo-transient under-relaxation can also be used with a coupled solver. A pseudo transient time-step size equivalent to the characteristic time for the vehicle (vehicle length/ freestream velocity) is a good starting point. This can be reduced or increased based on solution convergence behavior. For segregated solvers, lowering the pressure relaxation factor improves stability. The pressure relaxation factor seldom needs to go below 0.1. If the solution is still unstable, the grid quality should be checked.

SAENORM.COM : Click to view the full PDF of j2966_202106

5.2.4.2 CFD uses an iterative-based solution process to balance the equations of mass, momentum, and energy in any given model. The imbalance, or error, in each equation is assessed iteration to iteration until they drop below a default threshold set by the software manufacturer or by the user. These errors are represented through residuals, and how the residual value changes iteration to iteration is defined as convergence. The residual convergence is usually plotted on a log scale, and the shape of the convergence plot gives a good indication of the quality of the solution being calculated. A solution that is converging well should show decreasing residuals iteration to iteration, asymptoting towards a value several orders of magnitude below its starting residual. Poor residual convergence is characterized by the erratic increases and decreases in the residuals and/or the residuals dropping less than three orders of magnitude. In extreme cases, the residuals increase iteration to iteration, if this continues and the residual value raises above the initial residual the equations become unstable and the model becomes invalid. This is termed divergence. An example figure of good and poor steady-state converge is presented in Figure 5.

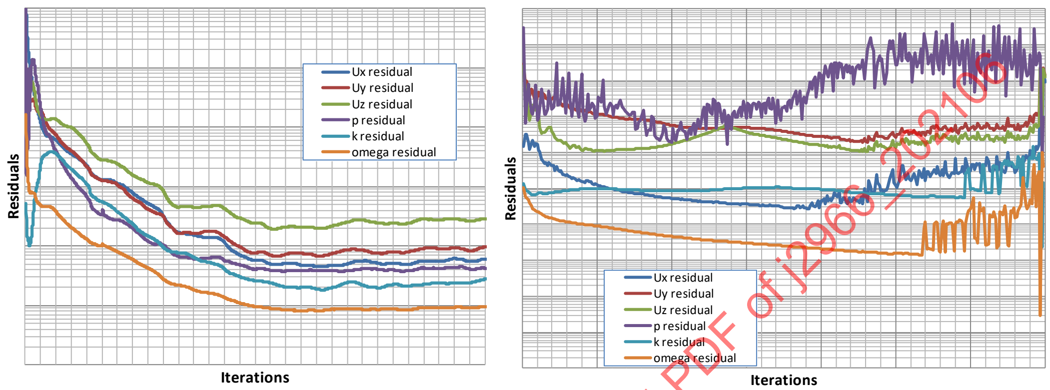

Figure 5 - An example for good (a) and poor (b) residual convergence

5.2.4.3 In addition to looking at the convergence of the residuals, the convergence of the drag force (or the parameter of interest from the results) should also be monitored. Plotting the drag value with iteration number (drag history) will allow the user to determine if the forces have reached a constant value, are oscillating within an acceptable range or are showing large swings. Appropriate averaging range can also be determined from this plot. It may be necessary to run an unsteady simulation to evaluate the drag value for steady-state simulations that show high amplitude of oscillations. Figure 6 presents different drag coefficient convergence behaviors for a steady-state solution.

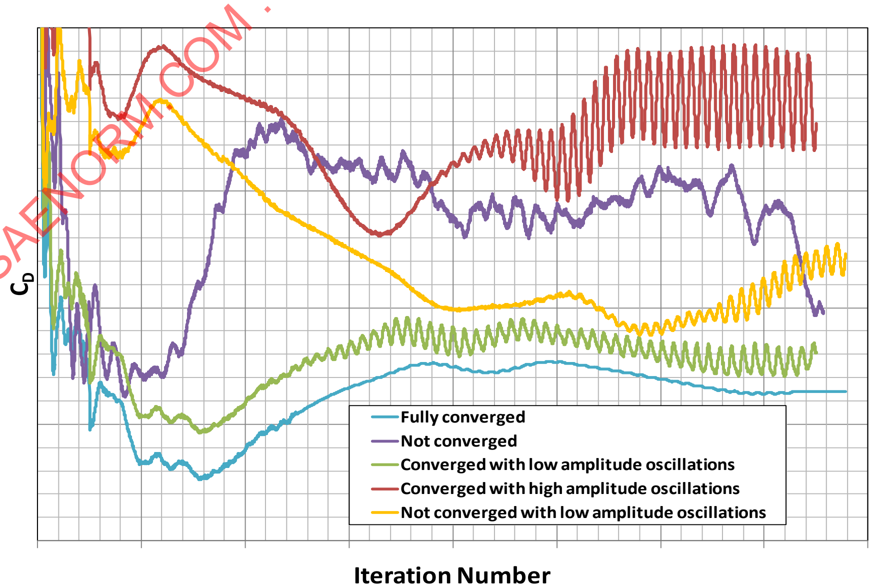

Figure 6 - Drag coefficient convergence examples

SAENORM.COM : Click to view the full PDF of j2966_202106

5.2.4.4 In case of a convergence issue at the beginning of the iterations, verifying grid quality and boundary conditions and initializing the model (where applicable) with a large turbulence dissipation rate (at least two orders of magnitude larger than turbulence kinetic energy) may help to kick start the solution.

5.2.5 Requirements for Unsteady Navier-Stokes Based Solvers

Requirements listed in this section are applicable to unsteady Navier-Stokes based solvers.

5.2.5.1 For unsteady Reynolds-Averaged Navier-Stokes (as described in ASME AMR v.62.4) simulations, either segregated or coupled solver can be used. For segregated solver, the relaxation factors for both momentum and pressure should be increased (to 0.8 or higher) from default values used for the steady simulation. In doing so, the flow field can be solved in as little as three to five sub-iterations, depending partly on the choice of the time step used. If convergence issues prohibit increase of relaxation factors, and more sub-iterations are needed, the coupled solver may have the advantage of faster convergence, and be therefore used.

5.2.5.2 For higher accuracy modeling of wake flows, hybrid RANS-LES models such as DES, DDES, or SBES are recommended. In general, large flow structures in the wake flow are more accurately captured using DES or LES simulations. Either segregated solver or coupled solver can be used for DES or SBES simulation. Vendor specific relaxation factors for momentum and pressure with five to eight sub-iterations can be utilized to improve both convergence rate and stability. If a segregated solver is chosen, pressure-velocity coupling scheme can be either SIMPLE (JHTMT, v15), SIMPLEC (as defined in NHT v.7) or PISO (JCP v.62) is recommended.

5.2.5.3 Recommended grid sizing for a DES (AFOSR 1997) simulation is around 50 to 100 times of the local Kolmogorov (Landahl MS 199) length scale. For k-ε turbulence models the Kolmogorov length scale is defined as $\eta = (\nu^3/\epsilon)^{1/4}$, where ϵ is the average rate of dissipation of turbulence kinetic energy per unit mass, and ν is the kinematic viscosity of the fluid. For k-ω turbulence models the Kolmogorov scale is defined as $\eta = (\nu^3/\beta^*\omega k)^{1/4}$ where β^* is a constant given in the k-ω model, ω is the specific dissipation rate and k is the turbulent kinetic energy. The LES regions that require scale resolved in ground vehicle aerodynamics are typically located in the wake of the wheels and vehicle body. For high fidelity and practical computational cost, to capture the drag of a commercial vehicle, it is recommended that a maximum grid spacing of 6 mm should be used on the external surfaces of the vehicle and outside of the boundary layer in regions of high flow gradients.

5.2.5.4 In general, time step for a DES run can be set to 50 to 100 times the Kolmogorov time scale. For k-ε turbulence models the Kolmogorov time scale is defined as $\tau_\eta = (\nu/\epsilon)^{1/2}$ whilst for k-ω turbulence model it is defined as $\tau_\eta = (\nu/\beta^*\omega k)^{1/2}$. A more conventional approach is to assess the Courant-Friedrichs-Lewy condition by evaluating the CFL (Courant) number as a measure of how well the time scales are captured. CFL is defined as $U^*\Delta t/\Delta x$, where Δx is the smallest cell size outside the boundary layers, and U is the free stream speed and Δt is the timestep size. Grid density variations on the model surface and in the boundary layer should be minimized to avoid convergence and false turbulence transition issues. Also, initial time step for DES can be calculated based on the mesh size away from the boundary layer region (LES part of the flow). For an explicit solver, cell CFL number should be less than one to match spatial and temporal resolutions in the model. However, for an implicit solver, when conducting design studies, a higher CFL number value can be used to reduce turn-around time. For high fidelity studies a CFL of 5 or lower is recommended to capture the drag of a vehicle. A rule of thumb for time step is $dt \approx dx/U$, where dx is the smallest cell size outside the boundary layers, and U is the free stream speed. The time step value from the CFL equation above should be used as a baseline value for the DES time-step selection process.

5.3 Requirements for Lattice-Boltzmann Based Solvers

Requirements listed in this section are applicable to the Lattice-Boltzmann based solvers. Solver setting values given below are for guidance only, and software vendors might be able to provide more specific values for their software package.

5.3.1 A minimum of 19 velocity state discretization is required for all Lattice-Boltzmann based solvers.

5.3.2 In general, near wall lattice size should not exceed 6 mm in regions of high flow gradients and smaller geometry features when a wall function is utilized. It is recommended that this cell size is applied to regions including front end tractor regions with high curvature, the a-pillar, the visor region, and the side mirrors.

SAENORM.COM : Click to view the full PDF of j2966_202106

5.3.3 When the boundary layer is resolved with meshing (i.e., no wall functions are used), the near wall resolution needs to be checked. A y^+ value between one and five is recommended to resolve the viscous sub-layer.

5.3.4 It is recommended that all geometry and wake region are contained within cells 24 mm in size or smaller. For non-zero yaw angles, care should be taken to also include the leeward wake of the tractor and trailer.

5.3.5 The exterior surfaces should be facetized with an adequate number of triangular elements to preserve curvature and details that are present on the test article being modeled. The chordal deviation along an edge, defined as the distance from the edge to the original CAD curve, should be less than 0.1 mm for the entire full-scale vehicle. This should be reduced to 0.05 mm for flow-critical areas, such as the leading edge of the hood and side mirror housings. Under-hood parts may be facetized using a wrapping mesh algorithm; however, it is recommended that all geometric details are preserved to achieve high accuracy in the prediction of underhood airflow resistance. For a fully detailed model, the number of facets will typically be greater than 6 million.

5.3.6 Lattice Boltzmann simulations are inherently unsteady simulations. Identification of when monitored values have reached a statistically stationary value should be performed using the time averaging procedure outlined in 5.1.4.

5.3.7 In Lattice-Boltzmann simulations, the air flow is usually simulated using a compressible flow approximation. Typically, a simulation will be run at an artificially high Mach number when compared to the actual Mach number on the road to improve simulation performance. For aerodynamic simulations, the simulated Mach number of the incoming flow should not exceed 0.35 in order not to degrade aerodynamic results. Similarly, the highest local Mach number in the flow should not exceed approximately Mach 0.5.

6. DATA PROCESSING AND COMMUNICATION

CFD simulations generate substantial amounts of data. Extracting the correct values using appropriate methods and creating a standard to compare CFD results from multiple resources reduces user error and improves confidence in evaluation efforts. This section talks about how results can be presented and interpreted. It also contains recommendations on information collection and processing related to CFD simulations.

6.1 Aerodynamic simulations are dependent on an accurate representation of the test article geometry. Acceptable geometric sources include CAD/CAE files and scanned 3D model data. The geometric resolution of the model shall be reported with the CFD results. Models for deformable parts and add-on devices that may influence vehicle aerodynamics shall be modeled in their intended design shape.

6.2 STEP and IGES are acceptable file formats to transfer CAD models between different sources when original CAD model is not available.

6.3 Vehicle geometric model should be exported to the applicable CFD software package with highest accuracy possible.

6.4 Each report needs to include basic information about the simulation:

6.4.1 Vehicle make, model, and year information, along with a description of geometry of the test article.

6.4.2 Short description of the simulation.

6.4.3 Official name/title, date, version number, and vendor information of the software product.

6.4.4 Type of the software code (such as Navier-Stokes or Lattice-Boltzmann).

6.4.5 Description of simulation procedure.

6.4.6 Aerodynamic data along with reference values for frontal area and velocity used for normalizing the force data. If wind averaged drag coefficient needs to be calculated according to the guidelines in SAE J1252, the effective ground speed and wind speed shall be documented.

6.5 It is recommended to document the results in CGNS (as defined in AIAA 98-3007) compatible format where native file formats are not readable.

SAENORM.COM : Click to view the full PDF of j2966_202106

6.6 Certain variables are required to be reported with each simulation:

6.6.1 Averaged force and moment coefficients (drag, lift, side force, yaw moment, etc.) per case. For steady state simulations, the number of iterations over which the data is averaged shall be stated. For unsteady simulations, both the length of time discarded for the initial transient and the length of time over which the data is averaged shall be stated. Where appropriate, the method used to determine the length of the initial transient shall be stated.

6.6.2 Contours, streamlines, vectors, and iso-surfaces for velocity and pressure as required. It is recommended to use averaged values for creating plots for known unsteady flow regions (such as vehicle wake and tractor-trailer gap).

6.6.3 Report pressure and force coefficient development along the vehicle-model's longitudinal axis. The maximum pressure coefficient for the stagnation region should remain below 1.1.

6.6.4 Plot y^+ values on the model surface to confirm grid density and its applicability on the selected turbulence model and wall function.

6.6.5 Report maximum turbulence viscosity ratio with an iso-surface plot. In general, turbulence viscosity ratio values may be above 1000, but they need to remain below 1000 in the free stream region.

6.6.6 Check and report maximum velocity in the computational domain. In general, the maximum velocity needs to be below 150 m/s in most cases.

7. NOTES

7.1 Revision Indicator

A change bar (l) located in the left margin is for the convenience of the user in locating areas where technical revisions, not editorial changes, have been made to the previous issue of this document. An (R) symbol to the left of the document title indicates a complete revision of the document, including technical revisions. Change bars and (R) are not used in original publications, nor in documents that contain editorial changes only.

PREPARED BY THE SAE TRUCK AND BUS AERODYNAMICS AND FUEL ECONOMY COMMITTEE

SAENORM.COM : Click to view the full PDF of j2966_202106