
40 C.F.R. § 1037.532

Using computational fluid dynamics to calculate drag area (C_dA).

This section describes how to use commercially available computational fluid dynamics (CFD) software to determine C_dA values, subject to the provisions of §§ 1037.525 through 1037.527. This is considered to be an alternate method for both tractors and trailers.

(a) For Phase 2 vehicles, use SAE J2966 (incorporated by reference in § 1037.810), with the following clarifications and exceptions:

- (1) Vehicles are subject to the requirement to meet standards based on the average of testing at yaw angles of $+4.5^\circ$ and -4.5° ; however, you may submit your application for certification with CFD results based on only one of those yaw angles.
- (2) For CFD code with a Navier–Stokes based solver, follow the additional steps in paragraph (d) of this section. For Lattice–Boltzmann based CFD code, follow the additional steps in paragraph (e) of this section.
- (3) Simulate a Reynolds number of 5.1 million (based on a 102-inch trailer width) and an air speed of 65 mi/hr.
- (4) Perform an open-road simulation (not the Wind Tunnel Simulation).
- (5) Use a free stream turbulence intensity of 0.0%.
- (6) Choose time steps that can accurately resolve intrinsic flow instabilities, consistent with good engineering judgment.
- (7) The result must be drag area (C_dA), not drag coefficient (C_d), based on an air speed of 65 mi/hr.

This document is only available to subscribers. Please [log in](#) or [purchase access](#).

[Purchase Login](#)