Flow Design :: Best Practices for Automotive Applications

For automotive applications, Flow Design provides an early understanding of flow characteristics around a vehicle. This helps inform the design direction early in vehicle development and provides the car body designer with feedback they normally do not have access to when they first begin their work. It can show regions where wakes will form and how large they will be, where high and low pressure regions are on the body, and where recirculation will occur. In addition to this qualitative feedback, Flow Design currently offers an estimation of total vehicle drag. The techniques below typically provide a nice balance of speed and accuracy when estimating drag:

- Ensure the model is correctly scaled using the dimensions in canvas as a reference.

- Use a reasonable mesh setting. 150% is normally plenty for conceptual studies.

- Results will develop quickly and deliver insight in a short amount of time, however, allow the drag results to level off before taking a “final number” on the drag coefficient. The Cd and average Cd should be flat and in line with one another.

- Size the tunnel appropriately for an automotive wind tunnel test. The default tunnel sizes are generic and, while broadly usable, they are not tailored specifically for automotive. Typically we see a nice balance of simulation speed and accuracy on vehicle models with:
    - The floor aligned with the bottom of the tires
    - The wind tunnel should have room for at least 2x vehicle lengths upstream of the front bumper
    - The wind tunnel should have room for at least 3x vehicle lengths downstream of the rear bumper
    - The wind tunnel height should be at least 3x the height of the vehicle
    - The wind tunnel width should be at least 4x the width of the vehicle

      *Note: these are good for typical vehicles. The more blunt the vehicle is (for example a very short upright car with a flat front) the bigger the wind tunnel should be.

In its current form, Flow Design is not intended to provide a direct means of measuring very small changes in automotive drag coefficients. Aerodynamicists and wind tunnel engineers spend considerable time and money making small changes to automotive trim and surfacing which, taken together, may impact the drag coefficient by 10% or more. Examples include grill shutters, fences, subtle breaks in the body panels, lip spoilers, etc. Flow Design is not an efficient tool for capturing the effects of these small changes due to the very high mesh resolution it requires for highly detailed studies. For this level of study we would recommend a full engineering CFD simulation product like Simulation CFD.