\_\_

## Virtual Wind Tunnel, 2d Cars

### Brief description

This simulation acts as a “virtual wind tunnel” by allowing you to understand the approximate flow features down the centerline of different vehicles. Freestream velocity and car bodies can be easily changed, allowing for comparisons and flow sweeps.

### Learning objectives

At the end of this exercise you should be able to ….

1. Understand the relationship between drag, lift, Cd, and Cl on car type, overall body shape, and freestream velocity.

### Steps you need to take

1. Open STAR-CCM+.
2. File, load simulation.
3. Where appropriate, enter additional licensing options if needed/wanted (Power on Demand, put in your PoD key, select parallel, etc.).
4. Browse, go to the folder containing the sim file for this course, select virtual\_wind\_tunnel\_2d\_cars.sim. Select Open, OK.
5. Select the Create/Open Scenes icon at the top of the screen (), then select Open All Scenes. Note that multiple tabs will open above, you can click these tabs to switch between what is shown in the viewing window.
6. To manipulate the viewing of a scene:
   1. Panning: To pan the image in the visualization display, hold down the right mouse button and drag the mouse in the direction where you want to move the object. The image appears to change its view angle as you pan. This effect is because the view is in the Perspective projection mode, which depicts three-dimensional space. In the Parallel projection mode, the view angle stays the same.
   2. Rolling: To roll the scene, hold down the <Ctrl> key, and click and drag the mouse upward or downward. Using the <Ctrl> key rotates the object along an axis perpendicular to your screen.
   3. Rotating (not enabled for 2d simulations): STAR-CCM+ uses a point on a part to rotate. If you select a point somewhere other than on a part, STAR-CCM+ uses the depth to that last point; the depth is normal to the view plane. The scene can be rotated around various axes, vertical, horizontal, or arbitrary, depending on the direction you move the mouse. To rotate, choose a point on or near the object in the visualization display, and click and drag (holding down the left mouse button) from that point. This technique rotates the object around the axis perpendicular to the direction of your drag. You set the location of the axis when you choose a point to begin clicking and dragging.
   4. Zooming: There are two ways to zoom with the mouse:
      * 1. Choosing a focal point: Choose a point inside the display, then click and drag downward using the middle mouse button. This action brings your view closer to that point. Moving the mouse straight up has the opposite effect.
        2. Using the mouse wheel: Turning the mouse wheel toward you brings your view closer to the visualization display; the opposite wheel movement takes your view farther away.
      1. The mouse pointer must be within the visualization display for this operation to work. Its particular position within the display determines the direction of this type of zooming.
   5. Hot Keys:
      1. Side View, S key
      2. Top View, T key
      3. Front View, F key
      4. Flip vertical, C key
      5. Reset View, R key
   6. Animations: Click  to play and  to stop. In this simulation, animations are enabled for the vector scenes only.
7. Select the Create/Open Plots icon at the top of the screen (), then select Open All Plots. These plots will help you to compare drag, lift, Cd, and Cl numerical results for the body in use. When changes are made to the body or freestream velocity, you will also be able to use these plots to numerically compare the results of the new analysis compared to the old one. For the purpose of this exercise you can ignore the residuals tab.
8. With the plots tabs now open at the top of the screen, there are a number of tabs to sort through. To see the tabs that are hidden to the left/right of the screen, use the arrows () at the top right of your screen.
9. Expand the Operation node, select the node named “What body do you want to use? (Boolean Subtract)”, select the area to the right of “Input Parts”, deselect the body you do not want to use, and select the body you want to use. Click the  icon at the top of the screen. Browse to the java file meshrun.java and select Open.
10. This java macro will automatically mesh and then continue to run your simulation. While the simulation runs you will see the current results in the scalar scenes, vector scenes, and plots. Let the simulation run until the force and/or force coefficient plots converge to a value. This should happen in approximately additional 500 solve iterations. To zoom in/out on the plots, use the middle mouse wheel or this icon . To reset the view hit R on the keyboard or the icon shown here in the center .

### Discussion questions

1. What was the drag and Cd for each car? What geometric features led to higher/lower drag and Cd? Why did these features produce more/less drag?
2. In the design of cars, why should drag be avoided? If you were trying to reduce the drag on each car, what changes could you make to the geometry to reduce this? What geometric features would you add?
3. What was the downforce and -Cl for each car? What geometric features led to higher/lower downforce and Cl? Why did these features produce more/less downforce?
4. In the design of cars, why should downforce be encouraged? If you were trying to increase the downforce on each car, what changes could you make to the geometry to improve this? What geometric features would you add?
5. Explain what ground effect is and illustrate an example of how one of the cars employ this concept to improve the aerodynamic performance of the car. What aerodynamic performance parameter (lift, drag, etc) is this improving? How is this helping the car practically (for example: fuel consumption)?
6. What would be the ideal centerline car shape if one were trying to both minimize drag and maximize downforce? Considering realistic constraints (room for passengers, engine, and storage, chassis, etc) could this profile be used in car manufacturing? Why or why not?
7. Give an example to illustrate why changes in geometry change both drag and downforce. Why are the two related? For example: If F1 cars are trying to maximize downforce and minimize drag, why do they produce a large amount of downforce and a large amount of drag?

Disclaimer:

The focus of this lesson is on the approximate flow features and not realistic 3d simulation of the actual cars which requires a higher cell count and run time. This simulation is meshed in 2d to allow a larger number of computers the ability to run these calculations. With this in mind, the numerical results obtained in these simulation are not a representation of the actual performance of these car.