\_\_

## Race Car Design Exploration

### Brief description

This exercise will allow you to design a race car so that downforce is maximized and drag is minimized. This will be done in a pre-setup simulation where alterations to geometry (overall car body, diffuser, undertray, and spoiler), meshing, and results visualization/reporting can be changed/explored in seconds.

### Learning objectives

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

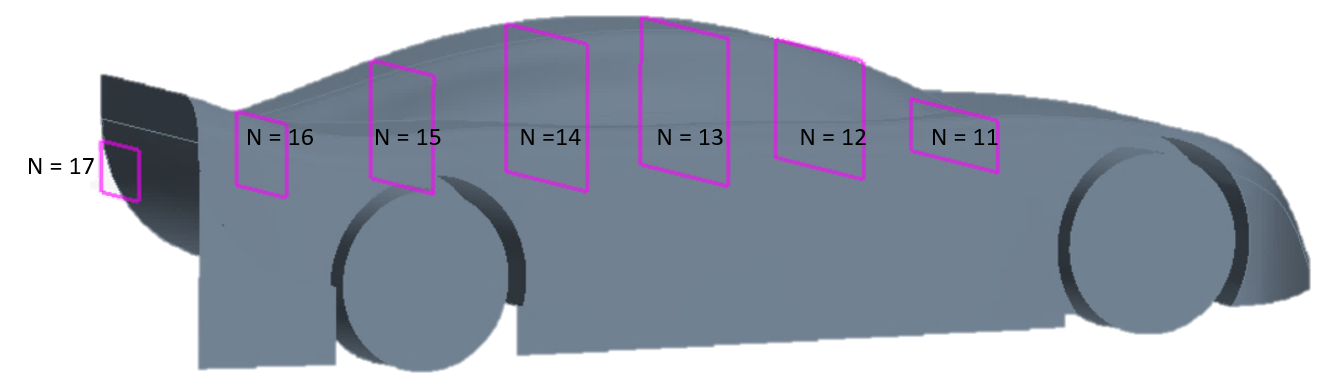
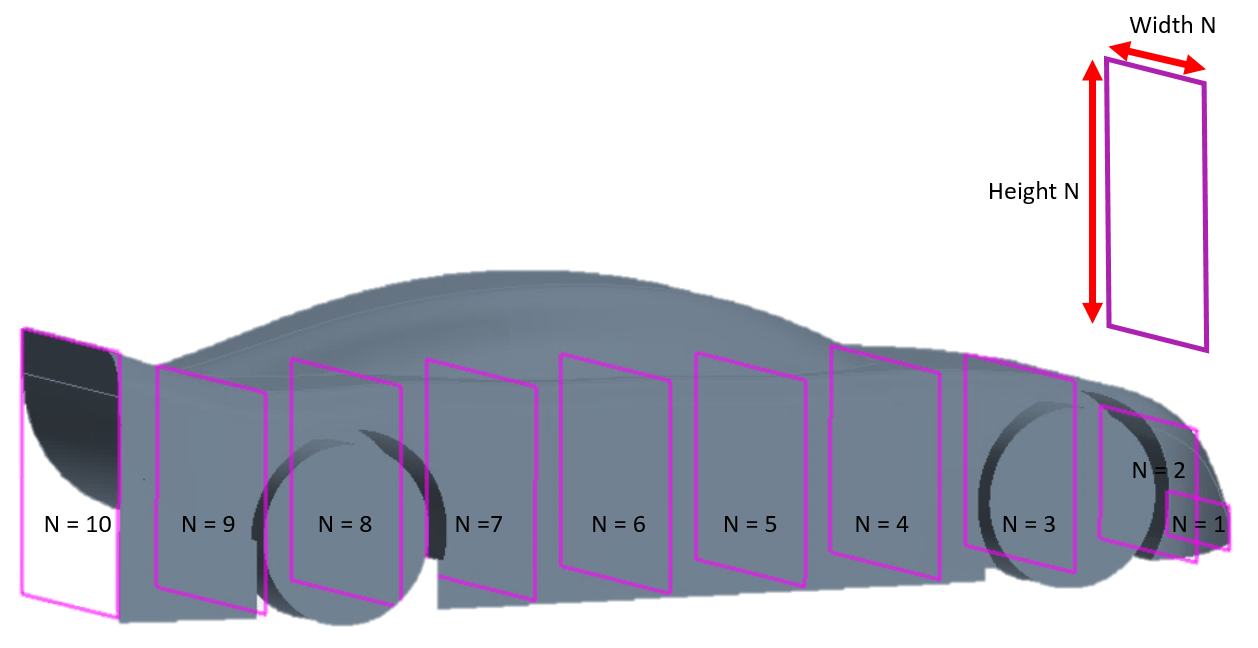
1. Iterate the process of changing exposed geometric design parameters, selecting re-mesh, selecting run, and visualizing the results.
2. Compare new results to previous design iterations to understand cause/affect relationships between geometric design and aerodynamic performance.

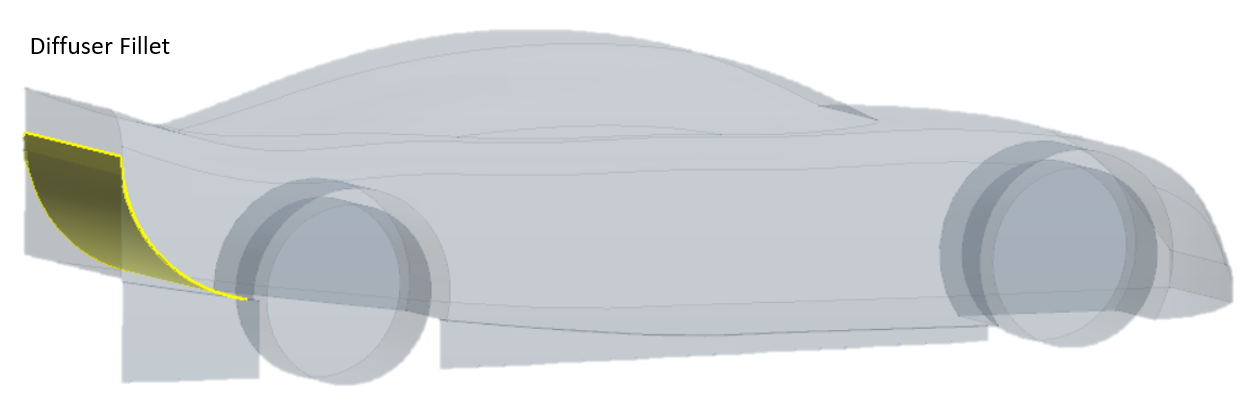
### Problem and Design Objectives

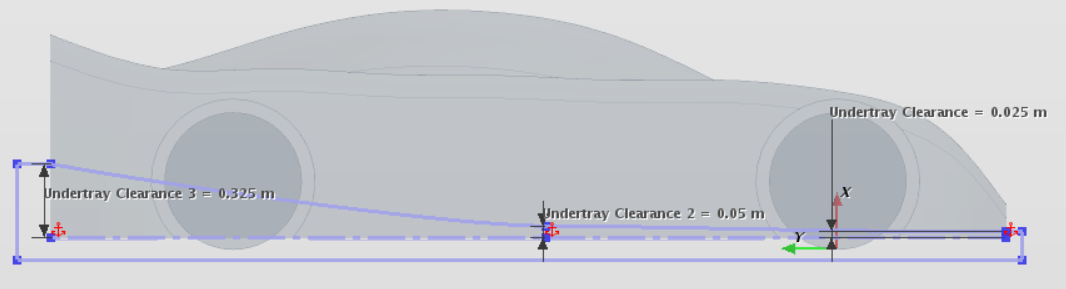
You are the aerodynamicist for a racing team and you need to improve the aerodynamic performance of last year’s car. Using STAR-CCM+ you will start by redesigning the overall shape of the car, the undertray, and the diffuser. For a set speed of 70 mph, you must reduce drag below 96 lbs and increase downforce above 177 lbs. The simulation used here is a half car simulation (to reduce cell count and run time) so in the simulation you must reduce drag below 48 lbs and increase downforce above 88.5 lbs.

### 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 race\_car\_design\_exploration\_cleared.sim. Select Open, OK.
5. The mesh and solution was cleared on this to reduce the file size. Select the Generate Mesh icon (). Meshing should take 1.5 to 3 minutes. Once the meshing process has finished (), select the Run icon (). Let this run for approximately 30 seconds (just to get an initial solution so the viewing of scenes makes more sense in the following steps). Then select the Stop icon ().
6. Select the Create/Open Scenes icon at the top of the screen (), then select Open All Scenes. Note that two tabs will open above, one named “Results” and another named “Geometry with Constraints”. You can click these tabs to switch between what is shown in the viewing window.
7. To familiarize yourself with how scenes works, first click the Results scene tab and then click the scene/plot tab on the upper left of the screen. Under displayers you will notice a number of displayers showing different elements of your results (streamlines for example). You will notice that two of these displayers are not pale in color which indicates that these displayers are being used to show what is current seen in the results scene. The other displayers that are pale in color have had their visibility toggled off. To toggle visibility on/off, right click a displayer and select Toggle Visibility.
8. 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: 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 this applies to the streamline and vector displayers only.
9. 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/downforce numerical results for each component of the car. When you change the car geometry, mesh, and complete your next analysis you will also be able to use these plots to numerically compare the performance of your new designs to old designs. For the purpose of this exercise you can ignore the residuals tab.
10. Switch to the Geometry with Constraints scene. In the simulation tree expand Geometry, 3D-CAD Models, 3D-CAD Model 1, and Design Parameters.
11. Here you will notice the design parameters, the following images outline what each parameter is:







1. To make changes to the geometry and view its layout, first make the desired changes to the mentioned parameters. When done, go to Operations and expand this, right click Subtract, and select Execute. You should notice that the geometry will change. Please note that in the Geometry with Constraints scene, the red surface cannot pierce the car surface.
2. When you are satisfied with the new geometry, select the Generate Mesh icon (). Meshing should take 1.5 to 3 minutes.
3. Once the meshing process has finished (), select the Run icon ().
4. Monitor the drag and downforce plots values to ensure they all either come to a single value or oscillate about some value. Please ensure you let the simulation run enough solve iterations to meet this criteria, if you stop the simulation prematurely you will get misleading results. Depending on the scenario, the simulation should be finished in 250 – 500 solve iterations.
5. Review the results (scenes and plots), make changes to the geometry as needed, mesh, and run the simulation as mentioned in the above steps till you reach the desired performance.

Disclaimer:

The focus of this lesson is on quick design exploration. Therefore, the mesh count required for this simulation needs to be reduced to approximately 500,000 cells to allow a larger number of computers to run these calculations. With this in mind, we cannot guarantee validation level results in the provided simulation.