Post-processing workshop instructions

MTF236 – Road Vehicle Aerodynamics

Task

This session serves as a preparation for the wind tunnel laboratory work, in which you will work with one of four models. Here, you will analyse the flow field around your model and identify problematic areas, describe them, and support your findings with some figures. These findings will help you determine what geometrical changes to test in the wind tunnel.

Connecting to remote StuDAT computers

https://it.portal.chalmers.se/itportal/GenStud/UsingStudentComputersFromHome

The recommendation is to use Linux, but it is Ok to use Windows workstations, just make sure that there is Star-CCM+ installed on the Windows computers you connect to. This can be checked here:
https://www.studat.chalmers.se/studat/studatsearch

Note: Remember to Sing out properly, not just closing the Remote Deasktop window.

Retrieving files

The links for the tutorial files and simulation files ready for post-processing should be available in Canvas.

Getting started

For Windows workstations, you will find Siemens Star-CCM+ in the Windows Start menu (simple double-clicking the *.sim file should work as well, however it will run in a single-core mode).
For Linux, one should open a terminal and type "starccm+".
If you are experiencing license issues, please make sure the Power Session tick is Off.

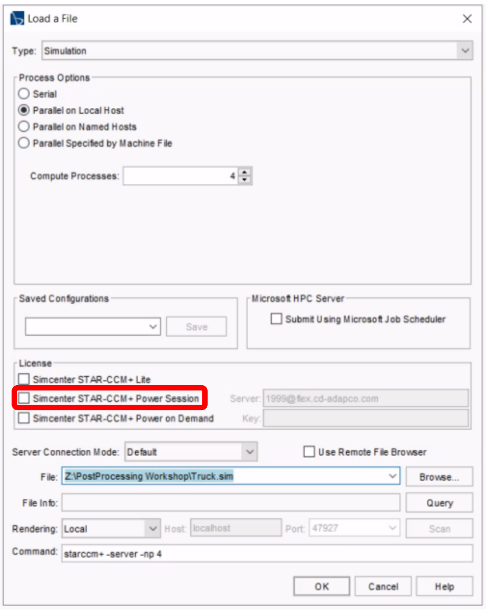

Figure 1. Loading a simulation file

Note: Remember to save your files from time to time not to lose the work!

Tutorials and Help

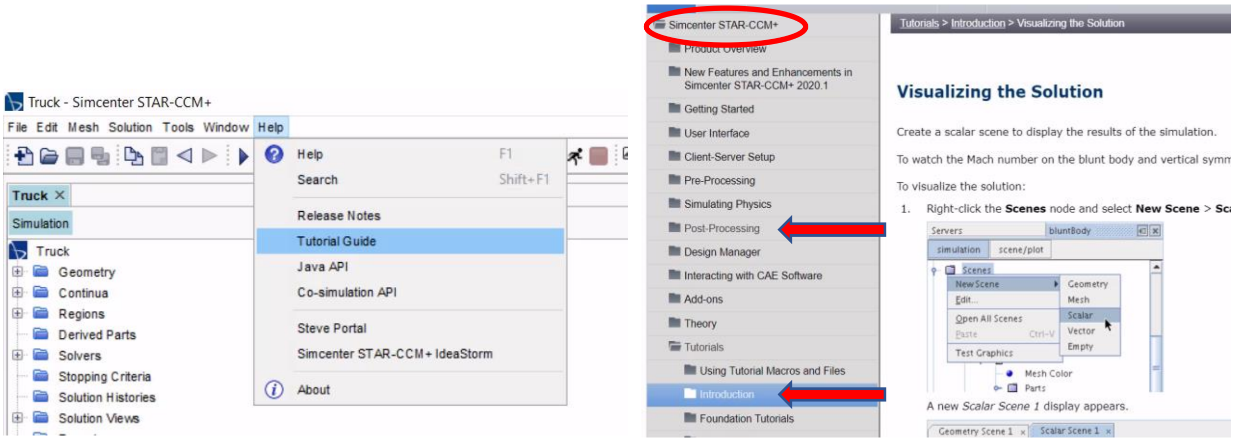

Figure 2. Tutorials location

One should start with the Introduction tutorial, at least the post-processing part of it. Initial and final simulation files for this tutorial can be found in the folder “StarTutorialFiles/introduction”.
You can find the tutorial instructions under Help -> Tutorial Guide -> Siemens StarCCM+ -> Tutorials -> Introduction. If for any reason it is not available, the "backup" folder contains the similar instructions in pdf. However, it is for slightly older version so some of the menus can be different.

Recommendation: when creating a new simulation or opening an existing simulation use the parallel mode and all of the CPUs that are available.

Important subsections of the Post-Processing help section:

- Visualising the solution
- Reporting results
- Plots

Note: Star-CCM+ is a one of the leading commercial CFD solver, so one can find plenty of materials online as well.

Common post-processing image examples

Here are a few examples commonly used in industry and research when evaluating aerodynamic performance of vehicles.

Surface *pressure* coefficient

The surface pressure coefficient provides a good overview on the main aerodynamic forces acting on a vehicle. Stagnation and acceleration regions can easily be identified by plotting the surface pressure coefficient as shown in Figure 3 below.

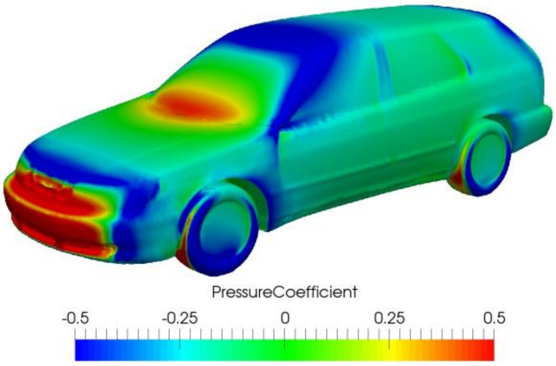

Figure 3. Surface pressure coefficient

It's important to adjust the scale for the area of interest when post-processing. The image on the left in Figure 4 uses a pressure coefficient scale commonly used for the entire vehicle, however when examining the base pressure the scale needs to be adjusted.

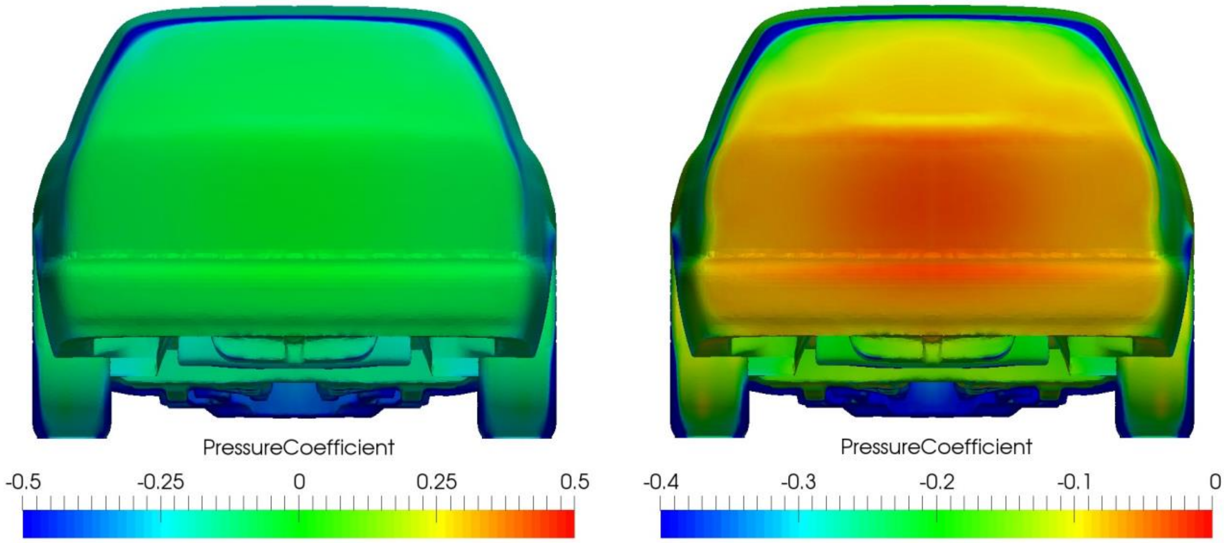

Figure 4. Base pressure in different scales

Note: remember that for Pressure coefficient function to work properly in Star-CCM+ one needs to set a correct reference velocity(inlet velocity) under Tools -> Field Functions -> Pressure Coefficient.

Velocity field around the vehicle

Looking at figures of the velocity field around the vehicle, such as Figure 5, can give insights into the general airflow and wake structure.

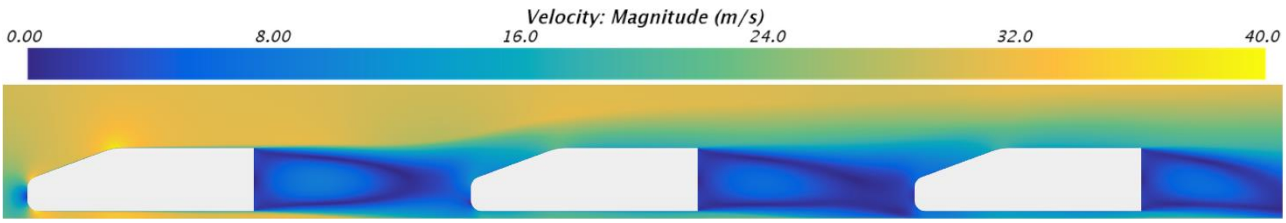

Figure 5. Velocity field in a platooning scenario

Total pressure

Iso-surfaces of total pressure equal to zero, such as Figure 6 corresponds to regions with high losses and can thus be used as a tool to identify wakes. In Star-CCM+ such surfaces can be created as a Derived part.

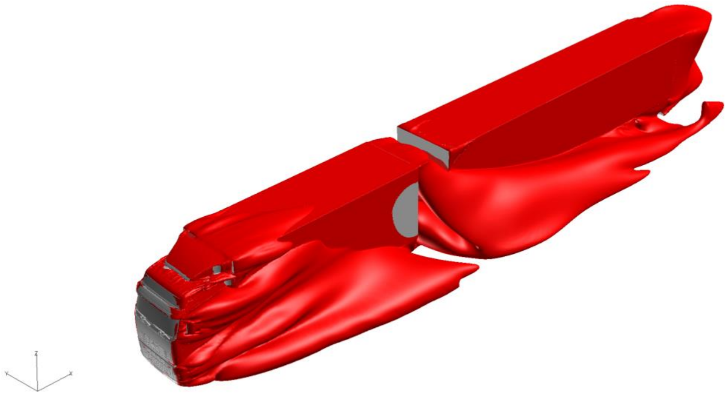

Figure 6. Iso-surfaces of total pressure equal to zero for a truck under side wind conditions

The same quantity can be visualised in different ways to reveal different details depending on the area of interest. For example, by visualising the total pressure in planes, as seen in Figure 7, it is possible to gain information about the total pressure near the surface while also giving (some) information about the overall shape.

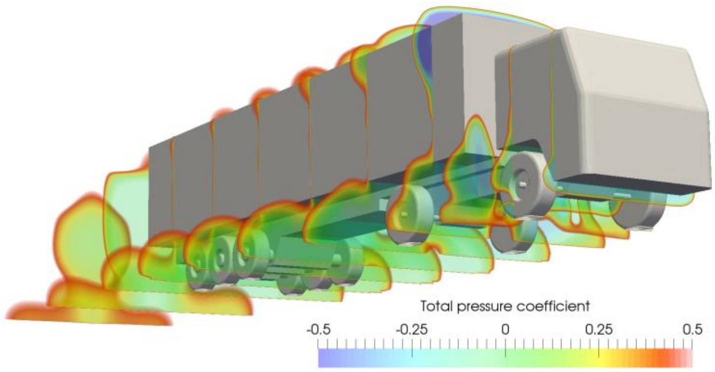

Figure 7. Wake structures

Streamlines

Streamlines can be used to visualize flow structures such as vortices and regions of separation as well as to trace the airflow.

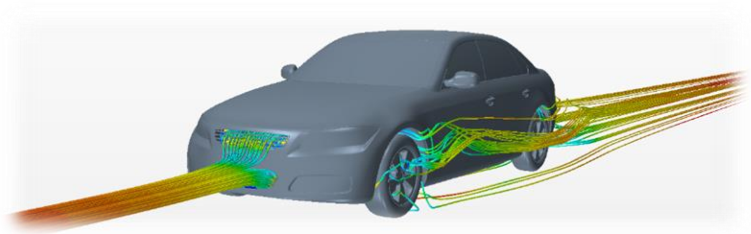

Figure 8. Streamlines showing air exiting the engine bay

Vector scenes

Vector scenes can be done in several different ways, for example using Line Integral Convolution showing not only the direction of the airflow but also other properties, e.g. it's velocity.

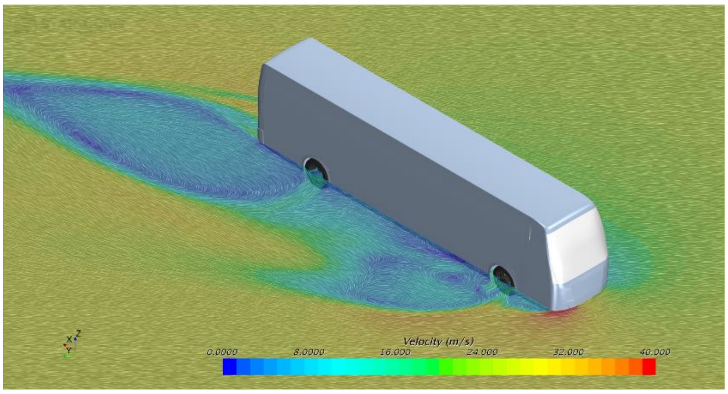

Figure 9. LIC vector scene

Accumulated Forces

Using Star-CCM+ it is also possible to split the vehicle in sections and look at contributions of different parts as well as producing an accumulated forces graph:

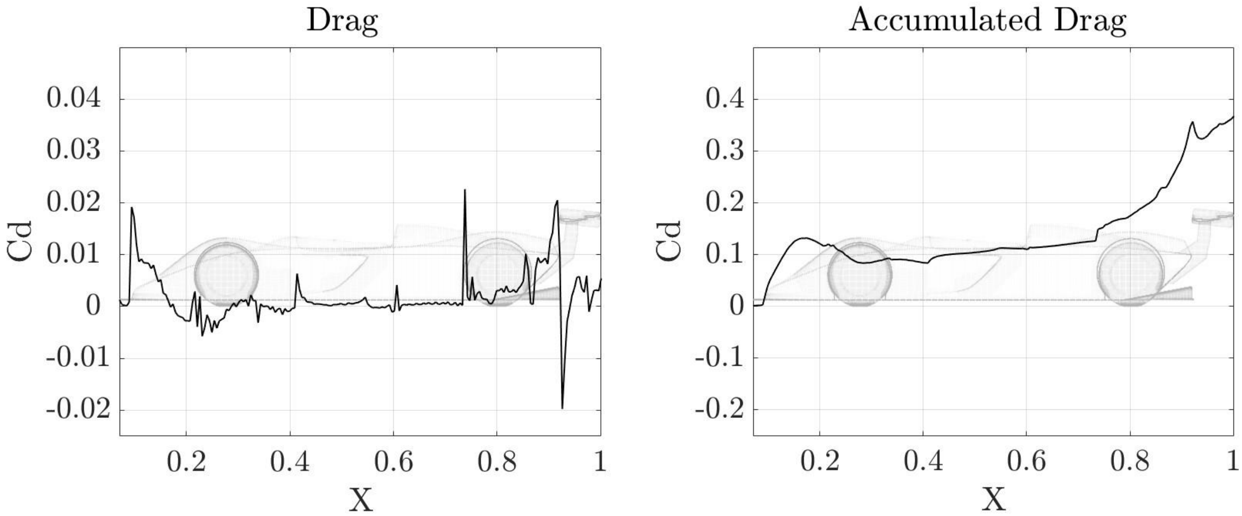

Figure 10. Main drag contributors and an accumulated drag graph

Figures presented here were post-processed in MATLAB, after exporting the data from Star-CCM+. Nevertheless, similar figures can be obtained using Plots and Accumulated Force Table functionality within the software.

Advanced post-processing

Scenes in Star-CCM+ can be much more complicated with animations and volume rendering, see example in Figure 11, so feel free to use your imagination and play around with options.

Note: be careful with the memory consumption for advanced rendering.

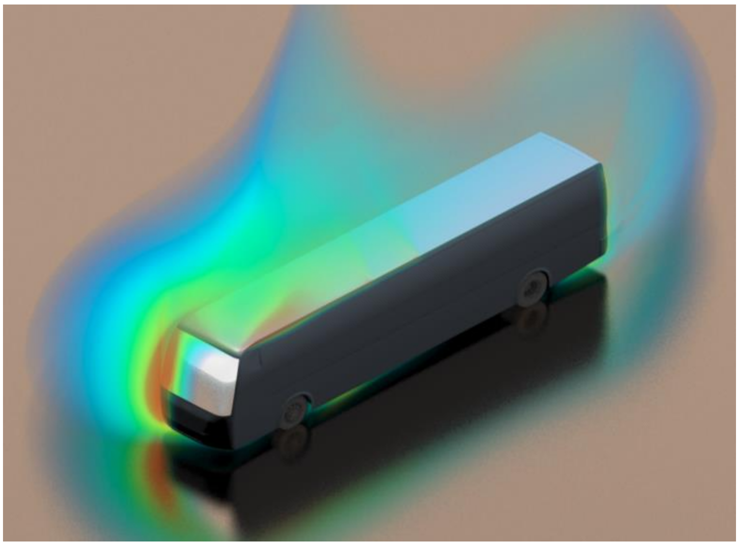
Figure 11. Volume rendering with Advanced Rendering enabled

Optional content

Lastly, for those of you who want to invest more time in learning Star-CCM+ there is an “optional” folder with more tutorials and slides that you can take a look at. Alternatively, just look online, there is plenty of material that is much more up-to-date.