

Subsonic Body CFD Lab

AERO 455 - CFD for Aerospace Applications

Instructor: Dr. Ziad Boutanios
TA: Ms. Christina Kolokotronis
Concordia University - MIAE

23/02/2022 and 09/03/2022

1 Introduction

This CFD lab is the first one in the series using Star-CCM+ and is meant to get the student acquainted with the software interface in general and the workflow within Star-CCM+. The present lab is based on the Introduction tutorial from the Star-CCM+ documentation, which shows how to create a polyhedral 3D computational mesh around a blunt body, and how to set up a transonic flow simulation.

We will follow the instructions in the official tutorial up to the last part of section *Selecting the Physics Models* where we will enter air material properties that correspond to freestream Mach number $M_\infty \approx 0.42$, Reynolds number $Re = 10^6$ and freestream static temperature $T_{s,\infty} = 300$ K. Appropriate boundary and initial conditions will be provided as well. After that, we will demonstrate the effect of the prismatic cell layer on the turbulence $y+$ parameter. We will also study the parallel efficiency of the solver for two different mesh sizes.

2 Solution Set-up and Initial Run

1. Prismatic layer meshing

Follow the instructions of the official tutorial up to the part of section *Generating the Volume Mesh* where you set the number of prismatic layers and use only 1 layer, instead of the tutorial value of 5. Set the *Prism Layer Stretching* parameter to 1.2 and continue following the official instructions.

2. Surface controls

When implementing the *Target Surface Size* for the Inner.wall boundary use *Relative to base* with a percentage value of 10% instead of *Absolute*. Do the same for *Minimum Surface Size* with a percentage value of 1%.

3. Air material properties

Follow the instructions of the official tutorial up to the last part of section *Selecting the Physics Models* where you should use the following air material properties instead of the default ones,

- Dynamic viscosity $\mu_a = 1.36 \cdot 10^{-5}$ Pa.s
- Thermal conductivity $k_a = 0.019$ W/m.K
- Turbulent Prandtl number $Pr = 0.7$

4. Discretization accuracy

Go to *Continua > Physics 1 > Models > Coupled Flow > Discretization* and choose *1st-order*. Do the same for *Continua > Physics 1 > Models > Realizable K-Epsilon Two-Layer > Convection*. These are very robust settings that will ensure good convergence. Continue with the tutorial instructions.

5. Initial conditions

In the following section on *Specifying the Initial Conditions for the Simulation* use 150 m/s for the freestream velocity value. This will give a freestream Mach number of approximately 0.42 at a static temperature of 300 K.

6. Boundary conditions

For the Inlet use a gauge total pressure value $P_t = 13241$ Pa, and a total temperature value of $T_t = 339.4$ K. Keep the official tutorial values for the remaining variables and continue with the tutorial instructions.

7. Visualization

Follow the instructions in section *Visualizing the Solution* to set up a Mach number scene. Then repeat the same procedure to create a y^+ scene. You will find the y^+ parameter as *Wall y^+* in the function item under *Scalar Field*. Display the y^+ only on the Inner-wall boundary and keep the scene default settings. Continue with the tutorial instructions.

8. Setting stopping criteria

Use 250 steps instead of the tutorial's 500. This is more than enough to ensure residual convergence of more than 4 orders of magnitude for the subsonic flow conditions we implemented.

9. Solver elapsed time report

Right-click on the *Reports* tab and select *New Report > System > Total Solver Elapsed Time*. We will run this report after simulation completion to get the simulation time for the parallel performance study.

Save the simulation as *bluntBody-subsonic-base* and run it to make sure all the settings are fine. After 250 steps the residuals should all be below 10^{-13} and the force coefficient should have stabilized to about $1.7 \cdot 10^{-3}$. The wall y^+ should fall between 131 and 433. Write down the final values

of the force coefficient and the min/max y^+ as shown in the legend of the wall y^+ scene. Run the elapsed time report and write down the elapsed time shown in the log window.

Clear the solution in the top menu item *Solution* *Clear Solution*, save and close the simulation.

3 Prismatic Layer Sensitivity

In this section we will assess the effect of increasing prismatic layers on the variation of the wall y^+ and the force coefficient. The turbulence model we are using is valid for all y^+ values all the way down to 1. It is also recommended to keep the wall y^+ values below 100. Since the wall y^+ is proportional to the height of the first cell at the wall we will reduce it by increasing the number of prismatic layers, which in turn reduces the height of the wall cell.

1. Open the simulation file *bluntBody-subsonic-base*, change the number of prismatic layers in the mesh to 3. Regenerate the volume mesh and save the simulation as *bluntBody-subsonic-order1-layers*. Run the simulation and note the values of the final force coefficient and the min/max y^+ as shown in the legend of the wall y^+ scene. Note the final residual orders of magnitude as well. The maximum value of the wall y^+ should still be above 100 so we will need to add more layers and run again.
2. Clear the solution, increase the number of prismatic layers to 5, regenerate the volume mesh and run the simulation again. Note the values of the final force coefficient and the min/max y^+ as shown in the legend of the wall y^+ scene. Note the final residual orders of magnitude as well. If the maximum value of the wall y^+ is still above 100 we will need to add more layers and run again.
3. The maximum wall y^+ value obtained with 5 layers should be very close to 100 so we will only add one more prismatic layer, making the total number 6, regenerate the volume mesh and run the simulation again. This time the maximum wall y^+ value should be comfortably

below 100. Note the values of the final force coefficient and the min/max y^+ as shown in the legend of the wall y^+ scene. Note the final residual orders of magnitude as well.

This completes our prismatic layer refinement exercise. It showed us how we can improve the quality of a turbulent simulation by reducing the wall y^+ values below 100 as we increase the number of prismatic layers, reducing the size of the computational cell at the wall. You will plot the min and max wall y^+ values, as well as the force coefficient against the number of prismatic layers for your report.

4 Parallel Performance

In this section we will assess the parallel performance of the Star-CCM+ by increasing the number of cores used to run the simulations and recording the elapsed solution time. When you increase the number of cores Star-CCM+ divides the mesh almost evenly between the cores and each will solve its own part of the problem. Some points and faces will be shared between the cores at the edges of the mesh blocks and there will be communication between the cores to solve the Riemann problems at these faces. The cores will need to communicate solution values to each other as well to set up the Riemann fluxes and finite difference schemes at the shared faces.

Logically, when you increase the number of cores the solution time should decrease proportionally. However, there is always overhead due to communication between the cores since they cannot compute while waiting to get data from each other. There is also the quality of the parallelisation of the solver's algorithms. Each formulation and each solver have different parallelisation efficiencies and they vary by computational mesh as well. Therefore they must be assessed on a case-by-case basis. We will do this here for the 1-prismatic layer mesh already used for the previous sensitivity study, and then we will refine the mesh and repeat the exercise.

1. Start Star-CCM+ with 2 processes selected as well as file *bluntBody-subsonic-base* as shown in figure 4.1.
2. Run the simulation for 250 steps.
3. Run the elapsed time report and write down the elapsed time.
4. Close Star-CCM+.

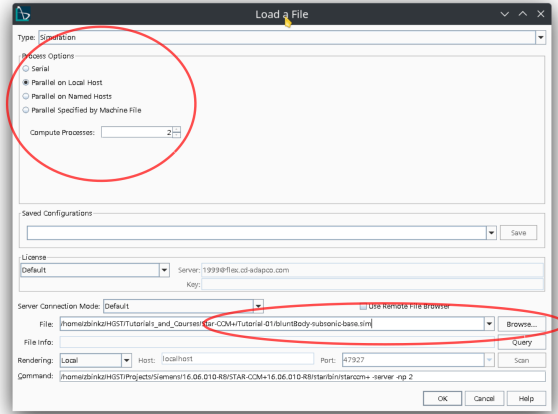


Figure 4.1: Selection window with 2 processes.

5. Repeat the 4 steps above process with 4 processes.
6. Then repeat the 4 steps above with 8 processes.

You now have elapsed runtimes for 3 parallel runs and the original serial 1-process. You need to plot the simulation acceleration factor $A(p)$ against number of processes p defined as,

$$A(p) = \frac{ET_s}{ET_p}, \quad (4.1)$$

where ET_s is the serial elapsed solution time, and ET_p is the parallel elapsed solution time for p processes. The 1:1 line (or $y = x$) corresponds to perfect scaling where a p -times increase in number of processes results in p -times acceleration in the elapsed solution time. The line you plotted will be below the perfect scaling line and should be diverging more from perfect scaling as we increase the number of processes.

Now we repeat the procedure with a finer mesh, which has more cells.

1. Close Star-CCM+ and re-open file *bluntBody-subsonic-base* in serial mode.
2. Change the automated mesh *Base Size* to 0.005 m, that is half of what we started with, and generate the mesh.
3. Save the simulation file as *bluntBody-subsonic-fine*.
4. Run the simulation for 250 steps and get the elapsed time.
5. Close Star-CCM+.
6. Open *bluntBody-subsonic-fine* and repeat the procedure above in parallel with 2, 4 and 8 processes.

Calculate and plot $A(p)$ for the fine mesh on the same plot as the base mesh. Which plot diverges from perfect scaling first? Why do you think that is? The answer lies in the reasons stated previously for parallel inefficiency and which one of these reasons worsens with mesh size.