AERO 97951 Applied Computational Aerodynamics

2019-20: Coursework Assignment 2

This second coursework is a joint work for groups of 2 students. It consists of tasks that should be addressed separately by each individual student and joint tasks that are to be addressed by the entire group.

The overall coursework consists of three parts. The weights of Part A, Part B, and Part C are 10%, 35%, and 55%, respectively.

Part A on Imperial College CFD codes

Joint task:

Fours codes were presented during the Special Session on "In house CFD codes": *pyFR, Pantharei, Incompact3D and Nektar++*. In no more than a page, describe the types of flow they simulate, the turbulence models they use, the spatial and temporal discretisation algorithms they employ, and a practical application of the code.

Part B on 2D Aerofoil

The simulations in this part should all be run with the following settings:

Parameter	Value
Aerofoil geometry	AG16
Chord length (cref)	1.0m
Mesh	Polygonal with prism layers
Mach number (M)	0.3
Reynolds number (Re)	$1*10^6$
Turbulence model	RANS, k-omega SST
Transition model	None (fully turbulent)
Target wall y+	< 1
Equation of state	Ideal gas
Solver	Steady, segregated (enthalpy)
Flow incidence angle	As specified in individual tasks

Set up a simulation in *STAR-CCM+* for the AG16 aerofoil, using the file *AG16_aerofoil.csv* provided on *BlackBoard Learn*. Specify a *Freestream* boundary on the circular far-field and a *Wall* boundary on the aerofoil surface.

For the reference and boundary conditions in *STAR-CCM+*, assume standard atmospheric conditions at sea level. Calculate the velocity and the dynamic viscosity, so that your settings correspond to the specified Reynolds and Mach numbers. (You can manipulate the dynamic viscosity in the model tree under *Continua -> Physics -> Models -> Gas -> Air*.)

Insert a table stating the reference and boundary conditions in your report.

Provide a ‘Summary Report’ for each individual task (2 in total). These can be extracted from *STAR-CCM+* in HTML format.

Use the coordinate file *AG16_aerofoil.dat* as an input for the *XFOIL* simulations.

Individual task (a):

Perform a mesh refinement study for an angle of incidence of 6 degrees, going through the following steps:

i. Calculate the boundary layer thickness and the thickness of the first element at the wall. Find a good value for the number of prism layers, too. Briefly explain your approach and provide the resulting values.
ii. Set the base size to 0.01m and the surface growth rate to 1.15. Using a custom control, set the target surface size of both the far-field and the flow domain to 1.0m. Keeping all other parameters as default, vary the size of the circular domain. Investigate the effect on integral values of lift and drag coefficients, as well as the pressure coefficient profile. Produce a plot showing the variation of the lift and drag coefficients. Given the set mesh parameters, explain why it is more difficult to converge the drag coefficient. State which domain size you take forward for the next steps and explain your rationale.
iii. Keeping all parameters as above, vary the base size of the individual cells. (Here, you might change the thickness of the prism layer mesh if it increases the mesh quality.) Investigate the effect on integral values of lift and drag coefficients, as well as on the pressure coefficient profile. Produce a plot showing the variation of pressure coefficient over the chord and another plot showing the variation of the lift and drag coefficients. Are the results improving and do you observe convergence? State which base size you would take forward and explain your rationale. Use detailed plots of the mesh at the leading and trailing edge to support your arguments.

Extract the lift and drag coefficients of the AG16 aerofoil for angles of incidence from 0 to 10 degrees from *XFOIL* using the same flow settings, but with a free transition model.

Individual task (b):

Create a mesh using the boundary layer settings as in task (a). Further set the radius of the circular domain to 50m, the base size to 0.005m, and the surface growth rate to 1.15. Using a custom control, set the target surface size of both the far-field and the flow domain to 1m.

Investigate angles of incidence, α, of 0, 2, 4, 6, and 8 degrees. How did you initialise the flow field for the different simulations? Extract the lift and drag coefficients of the aerofoil for each angle from *STAR-CCM+*. For an angles of incidence α = 4 degrees, present a monitor plot of the default residuals (energy, momentum, …) and a plot of the convergence history of the two force coefficients (lift and drag). Explain which criteria you have applied to assess whether the solution has converged.

Try to simulate an angle of incidence of 9 degrees or higher. What do you observe and how do you explain it?

Compare the pressure coefficient profile for an angle of incidence α = 4 degrees with the results obtained from *XFOIL,* using a single plot. Comment on the agreement, or lack thereof, between the two curves.

Joint task (c):

Plot your obtained values for lift and drag coefficients (from task b) against the range of incidence angles in a single plot. Further insert the lift and drag coefficients as obtained from *XFOIL* in the same plot.

Interpret the different results of the lift and drag coefficient curves:

- Comment on the suitability of using a single mesh for multiple angles of incidence. Is the target y+ value still suitable?
- How could you further refine the mesh for flows with high incidence angles?
- Comment on the suitability of the applied *STAR-CCM+* physics models.
- Which simulations (*XFOIL*, *STAR-CCM+*) do you trust more and why?

Part C on 3D Wing

In this task, each of your individual tasks is based on the same wing geometry, the RAE Wing A. This was designed as a benchmark case for transonic flow, see the report *RAE_WingA.pdf* on *BlackBoard Learn* for more details.

We have already prepared the geometry and refinement curves and surfaces for you, which you can find on *BlackBoard Learn* as *RAE_WingA.sim*. Display the wing and identify its position with reference to the given frame of reference.

The simulations in this part should all be run with the following settings:

Parameter	**Value**
Wing geometry	RAE Wing A
Mean aerodynamic chord (MAC)	0.15244m
Mesh	Polyhedral with prism layers
Mach number (M)	0.4 and 0.8
Reynolds number (Re)	$1*10^6$ for the viscous case
Turbulence model	RANS, k-omega SST
Transition model	None (fully turbulent)
Target wall y+	< 1
Equation of state	Ideal gas
Solver	Steady, coupled
Coupled Implicit Solver Settings:	
Courant number	30/M (rule of thumb for a good-quality mesh)
Expert initialisation	Grid Sequencing
Solution driver	Expert Driver
Flow incidence angle	2 degrees
Radius of half-spherical domain	12.0m
Target number of cells (±5%)	300 000 (coarse) and 600 000 (fine)

To create your mesh, keep most settings as their default, apart from the following:

Mesh Parameter	Value
Surface Remesher	
Minimum Face Quality	0.2
Polyhedral Mesher	
Enable Growth Rate	active
Optimisation Cycles	4
Quality Threshold	0.8
Prism Layer Mesher	
Minimum Thickness Percentage	5.0
Layer Reduction Percentage	10.0
Boundary March Angle	85.0
Default Controls	
Maximum Cell Size	2.0m
CAD Projection	active
Custom Controls	
Far Field & Symmetry Plane	
Target Surface Size	1.0m
Discontinuity (TE, LE & Tip Curves and Wing Tip Surface) Refinement	
Target Surface Size	25% of base size

MESH INSTRUCTIONS: Start with the provided geometry and create a coarse mesh. Follow the specifications given above and in addition vary <u>only</u> the following parameters: base size, number of prism layers, thickness of the prism layer mesh, and thickness of the first cell of the prism layer. Create a second finer mesh by adapting <u>only</u> the same 4 parameters. For each of the two meshes, provide a single scene showing both the surface mesh on the xz-plane and on the wing surface. Justify your choice of mesh parameters.

PROCESSING INSTRUCTIONS: For both meshes and Mach numbers, show a plot of the iteration history for the lift and drag coefficients and extract their converged values. Also plot the default residuals (energy, momentum, …) for one of the four cases. Further extract sectional pressure coefficient distributions corresponding to the pressure measurements of the *RAE Wing A* experimental setup at y/b = 0.4, and 0.925 for all four cases. Create one pressure coefficient plot of the two spanwise positions showing the data of all four cases.

<u>Individual task (a)</u>

Investigate low-speed and transonic flow over the wing using an **inviscid** flow model:

- Create two meshes as explained in the MESH INSTRUCTIONS.
- Set the Mach number to 0.4 and 0.8 and the *STAR-CCM+* physics model to *Inviscid***.** State which settings for flow speed, temperature, and pressure you have set and how you have obtained them.

- Follow the PROCESSING INSTRUCTIONS.

<u>Individual task (b)</u>

Investigate low speed and transonic flow over the wing using a **viscous** flow model:

- Create two meshes as explained in the MESH INSTRUCTIONS.
- Set the Mach number to 0.4 and 0.8 and the *STAR-CCM+* physics model to V*iscous*. State which settings for flow speed, temperature, pressure, and viscosity you have set and how you have obtained them.
- Follow the PROCESSING INSTRUCTIONS.

<u>Joint task (c)</u>

On *BlackBoard Learn* you can find digitised experimental data of the RAE Wing A (*RAE_WingA_Exp_cp.csv*).

On *BlackBoard Learn* you can also find an *AVL* configuration file of the wing (*RAE_WingA.avl*) and a data file containing distributions of pressure coefficient deltas (dCp) at different spanwise locations (*RAE_WingA_AVL_dCp.dat*).

Create two plots in which you gather the results of the two individual tasks and interpret and discuss the results:

- A first plot containing the data sets of the pressure coefficient deltas (dCp) for Ma=0.4 with the fine mesh. Further plot the corresponding AVL results and experimental data into the same figure. (From the pressure coefficient distributions obtained from *STAR-CCM+*, calculate dCp as the difference between the values of Cp on the upper and the lower surface along the chord. You will probably need to use a sensible interpolation of the extracted data points and *constrained plane* sections to do this.)
- A second plot containing the data sets of the pressure coefficients (Cp) for Ma=0.8 with the fine mesh. Further plot the corresponding experimental data into the same figure.

Discuss the effects of mesh resolution on your results:

- Comment on the mesh resolution and its influence on resolving shocks in the flow.
- Comment on the different prism layer mesh requirements for inviscid and viscous flows. How did you ensure the y+ target was met?
- Investigate the entropy for the inviscid solution near the wall boundary and comment on its behaviour.

Provide a ‘Summary Report’ for each individual task (2 in total, one inviscid, one viscous). These can be extracted from *STAR-CCM+* in HTML format.

General remarks

Create a joint report per group containing all individual and the joint tasks, *clearly identifying the author of each individual task*. Submit the joint report via *BlackBoard Learn* *before 11pm on Friday 31st January 2020*.

Create a ZIP-file containing the 4 *STAR-CCM+* 'Summary Reports' and upload them, too.

Follow the best practices outlined in the Department of Aeronautics Report Style Guide when writing your report. Keep your answers short. The length of the overall report, including graphs and tables, ***should not exceed 11 pages*** using 11pt font and 15mm margins.

Use the version of *STAR-CCM+* (13.04.011, mixed precision) available from the *Imperial Software Hub,* because a different version might lead to different results.

The plots should be presented in an appropriate scale and be adequately captioned, numbered, and labelled. Do not use a dark background for the plots.

Plan your time wisely! A single 3D simulation might take more than one hour to converge.