Department of Aeronautics, Imperial College London

AERO97051 Applied Computational Aerodynamics

Coursework Assignment 2020-21

Name	Iaroslav Kazakov	CID No	01334502

Answer the questions using the template below

Part A: 2D Aerofoil

XFOIL

1. Common flow features (4%)

a. At what location on the upper surface is the pressure gradient adverse?

Adverse pressure gradient corresponds to the region of the graph where $-C_p$ starts decreasing. This is where the static pressure starts increasing, and the dynamic pressure drops. As per the plots below, at $\frac{x}{c} \approx 0.05604$, $-C_p$ is largest at the suction peak, hence the adverse pressure gradient is at $\frac{x}{c} \geq 0.05604$, more specifically at $\frac{x}{c} \in (0.05604, 1)$.

b. What are the differences in the C_p distribution between the viscous and inviscid solutions? Where are they located? What do the differences indicate? (125 words max.)

The difference in C_p prevails because the inviscid simulation does consider viscous boundary layer existence meaning that the skin friction is not accounted for. From the plots below, we see that $C_{p,invisc} \approx C_{p,visc}$ up until the point of separation on the upper surface. $C_{p,visc}$ becomes more negative due to the separation bubble on the upper surface. The main discrepancy starts occurring after separation bubble is encountered at $\frac{x}{c} \approx 0.5452$ on the upper surface, whereby $C_{p,visc}$ increases insignificantly but $C_{p,invisc}$ increases substantially. After the point of transition at $\frac{x}{c} \approx 0.7879$, $C_{p,visc}$ increases dramatically until the point of reattachment at $\frac{x}{c} \approx 0.8691$. At the point of reattachment, the turbulent flow decelerates with $C_{p,invisc} > C_{p,visc}$ at the trailing edge; this could happen due to associated higher energy losses in a turbulent viscous wake. On the bottom surface, both solutions are similar until $\frac{x}{c} \approx 0.8691$ where the trailing edge vortices of viscous solution start decreasing C_p.

2. Description of relevant flow phenomena at Re = 10^5 (15%)

a. Is there flow separation and/or reattachment? Is there a separation bubble? If so, mention where each phenomena is/spans based on the relevant features you can distinguish from the indicators (C_p, C_f, δ and θ plots). How does each observed phenomena affect the relevant indicator? (250 words max.)

The separation and reattachment start occurring at $\frac{x}{c} \approx 0.5452$ and $\frac{x}{c} \approx 0.8691$ respectively with the separation bubble being at $\frac{x}{c} \in [0.5452, 0.8691]$. After the separation bubble is encountered $-\frac{dC_p}{dx} \approx -0.0020$ and the magnitude of $C_{p,visc}$ is larger than of $C_{p,invisc}$, indicating that the separation bubble has occurred. C_f $vs \frac{x}{c}$ plot indicates that $C_f = 0$ at $\frac{x}{c} \approx 0.53$ (with $\frac{dC_f}{dx} < 0$) and becomes negative further downstream, clearly demonstrating that the flow has separated and de-attached from the aerofoil surface. $\frac{dC_f}{dx} > 0$ and $C_f = 0$ are at $\frac{x}{c} \approx 0.8432$ showing that the reattachment occurred after the transition. From δ^* vs $\frac{x}{c}$ plot one would expect to see a reduction in displacement thickness at the point of transition until the point of reattachment. At $\frac{x}{c} \approx 0.8691$ the flow attaches recovering the boundary layer meaning that δ^* grows. As per graphs below δ^* starts growing at $\frac{x}{c} \approx 0.87$ which again proves that the boundary layer reattachment occurred at the right location. θ started increasing at $\frac{x}{c} \approx 0.7879$ where the flow moved from laminar to turbulent, indicating the flow reattached and the boundary layer momentum started increasing due to higher momentum flux of turbulent fluids.

b. Is there transition to turbulence? If so, discuss its effect on the relevant indicators and state the transition location. (75 words max.)

The transition occurred at $\frac{x}{c} \in (0.7870, 0.7945)$, i.e. where $\frac{dC_p}{dx}$ increases due to large velocity gradients. From $\theta\ vs\ \frac{x}{c}$ plot an increase in momentum thickness on the upper surface occurs at $\frac{x}{c} \approx 0.7879$ (point of transition) as the velocity in a boundary layer changes. Turbulence increases the energetic capacity of the flow leading to flow reattachment. $C_f\ vs\ \frac{x}{c}$ plot indicates transition where a sudden increase in C_f on the upper surface occurs at $\frac{x}{c} \approx 0.7989$ and $\frac{dC_f}{dx}$ becomes positive.

3. Aerofoil performance for Re = 300 900 and XFOIL prediction (12%)

a. Write down the following parameters from the *XFOIL* simulation and experimental data:

i. $dC_l/d\alpha|_{Xfoil} = 0.0931\ \frac{1}{deg}$; $dC_l/d\alpha|_{exp} = 0.0864\ \frac{1}{deg}$

ii. $C_{l0}|_{Xfoil} = 0.2952$; $C_{l0}|_{exp} = 0.1944$

iii. $\alpha_0|_{Xfoil} = -3.3128°$; $\alpha_0|_{exp} = -2.2482°$

iv. $(L/D)_{max}|_{Xfoil} = 70.2415$; $(L/D)_{max}|_{exp} = 53.8125$

b. Comment on the scope and accuracy of *XFOIL*'s prediction ability. (150 words max.)

XFoil overestimates all the performance parameters and the relative errors for $dC_l/d\alpha$, C_{l0}, α_0, $(L/D)_{max}$ are 7.75%, 51.85%, 47.35% and 30.53% respectively. This discrepancy could occur due to the linear vorticity stream function panel method and the idealised model flow XFoil uses. Wake is modelled using panels, from an inviscid solution, hence wake viscous effects that decrease lift and increase drag are omitted. XFoil is unable to output realistic predictions at negative α where the near stall behaviour can persist at relatively low Re. However, $dC_l/d\alpha$ XFoil predictions are realistic for the non-stall regimes in the region of $\alpha \in (0°, 10°)$, whereby the gradients of $C_l\ vs\ \alpha$ plot run almost in parallel.

Another uncertainty could have appeared due to the quality of experimental data. If it was taken from a wind tunnel, the wall interference could minimize the boundary layer sensitivity. Measuring drag forces in a wind tunnel accurately could presented a challenge, leading to uncertainty in C_D.

4. Plots (8%)

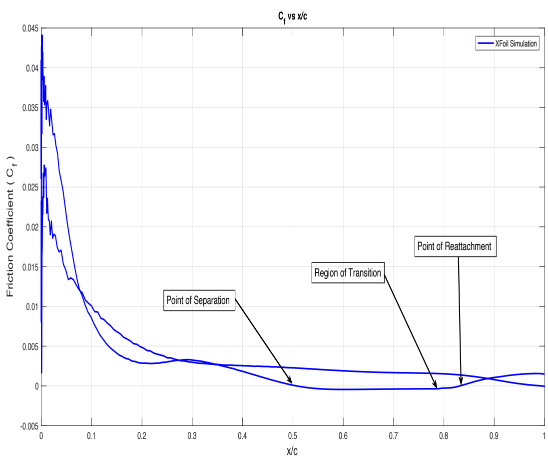


Figure 1: Friction Coefficient vs x/c

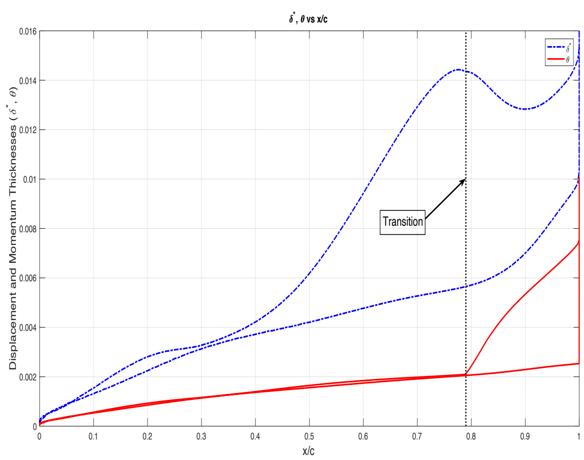


Figure 2: $\delta^*, \theta\ vs\ \frac{x}{c}$

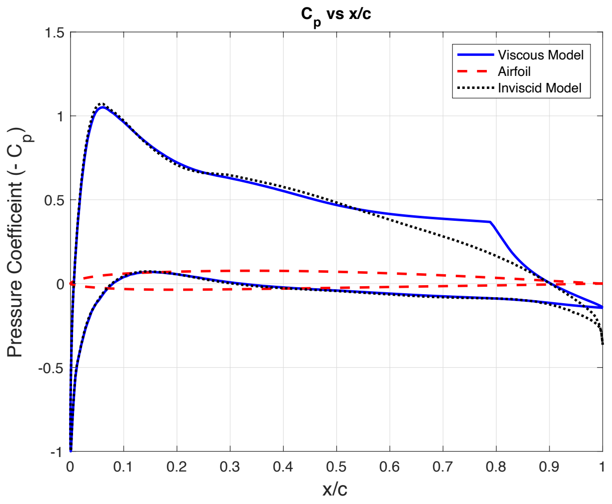


Figure 3: Pressure Coefficient vs x/c

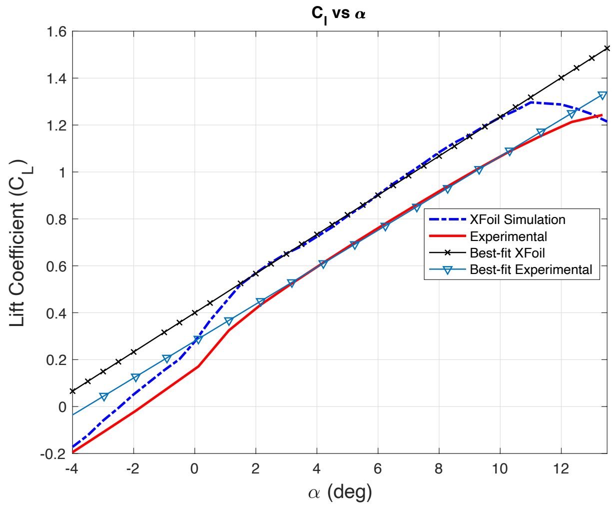


Figure 4: Lift Coefficient vs AoA

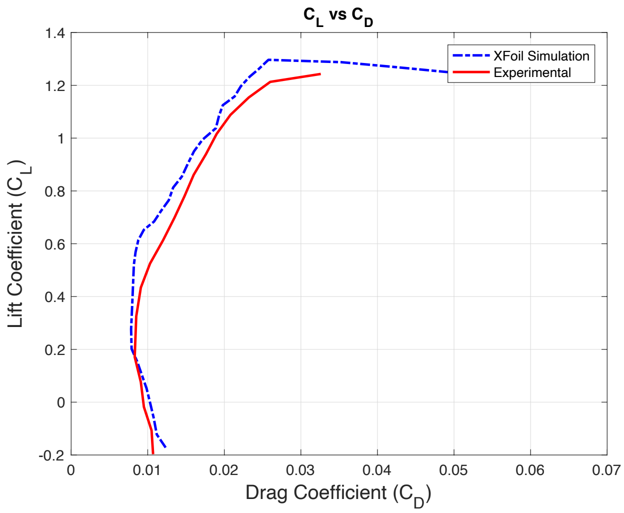


Figure 5: Drag Polar

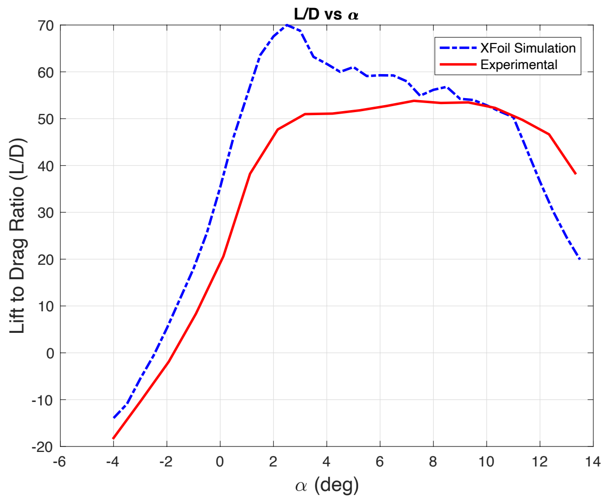


Figure 6: L/D vs AoA

2D *STAR-CCM+*

1. **Mesh Refinement Study** (15%)

a. What prism layer count, boundary layer thickness and first prism layer thickness did you use? How did you arrive at these values? (250 words max.)

Prism Layer Count	Boundary Layer Thickness (δ)	First Prism Layer Thickness ($2^*\Delta s$)	Base size
25	2.97e-2 (m)	6.40e-5 (m)	0.025

Table 1: Mesh parameters

First, the BL thickness of the flow had been calculated using the Blasius solution for a flat plate as per following $\delta = \frac{0.37*x}{(Re_x)^{0.2}}$. The following parameters were defined:

Dynamic Viscosity (ν)	Coefficient of Friction	Velocity	Characteristic Length	Density (ρ)	Targeted Wall (y^+)
4.16e-4 (Pa-s)	4.65e-3	102.09 (m/s)	1.0 (m)	1.225 (kg/mˆ3)	1.0

Table 2: Flow Parameters

Using the following set of equations, the value of Δs was estimated:

$$\Delta s = \frac{y^+\,\nu}{u_\tau}, Re = \frac{\rho U x}{\mu}, \tau_w = \frac{C_f U^2 \rho}{2}$$

After one iteration, the result yielded $\Delta s = 6.98e - 5$. Using the geometric series and relating the growth rate $g \approx 1.16$, the first prism layer thickness Δs and boundary layer thickness δ, the prism layer count n can be found as per below (sum of geometric series):

$$\delta = \sum_{i=1}^{n} \Delta s\, g^{i-1} = \frac{\Delta s(g^n - 1)}{g - 1}$$

The formula resulted in 25 prism layers. Due to limitations of the CPU, the base size had to be set to a reasonable value of 0.025 whilst still ensuring fine mesh surrounding the aerofoil. To ensure the targeting wall value y is met, through iterations Δs value was decreased until y^+ became slightly less than 1. From there Δs arrived to 6.40e-5 m.

b. Provide the two plots for the mesh refinement study of the Cp, and the lift and drag coefficients.

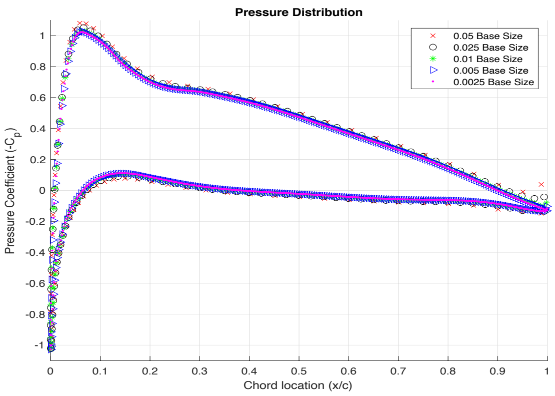


Figure 1: Pressure Distribution

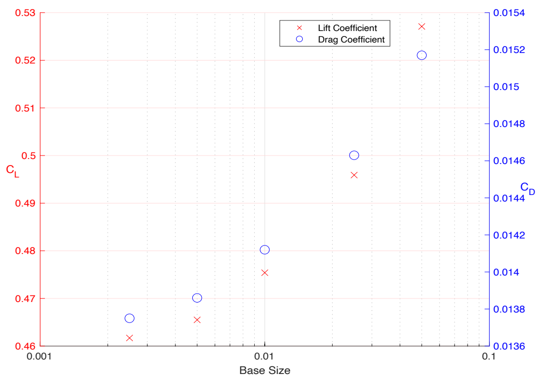


Figure 2: C_D *and* C_L convergence

c. Do the results improve? Do you see convergence? Based on your results, what base size would you select to use for further work and why? (125 words max.)

Yes, the results improve, and the convergence criterion is fulfilled. As per the pressure distribution plot, the most significant differences can be observed at the trailing and leading edge. C_L and C_D show convergence as the base size decreases. 0.005 and 0.01 Base Size are good choices because C_P demonstrates reasonable convergence at these values. Drag and lift coefficients at 0.005 base size are only 1% and 1.4% larger than that of 0.0025. However, the computational gain between the last two scenarios is more significant than the benefit in coefficients estimations. For 0.005 base size, the mesh at the trailing and leading edges look smooth and is not too coarse.

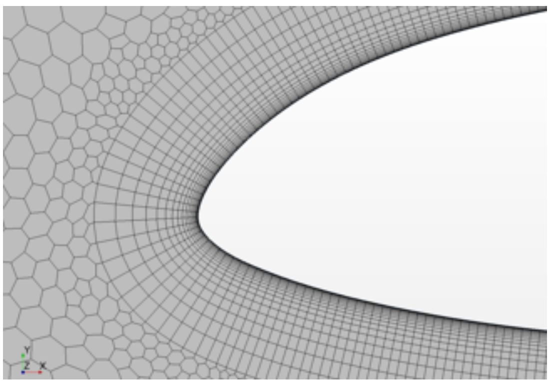


Figure 3: Leading Edge Mesh

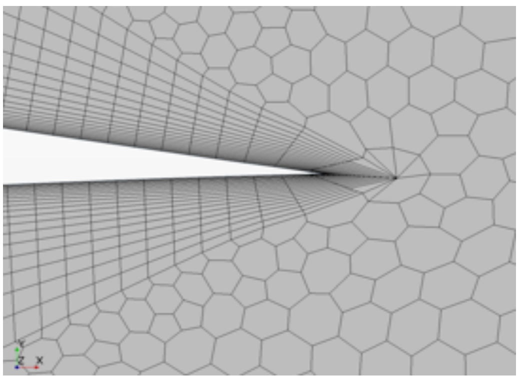


Figure 4: Trailing Edge Mesh

Anything below base size 0.005 leads to no substantial convergence improvements but adds substantial computational weight.

2. Simulation and Results (20%)

a. Provide the convergence history of the lift/drag coefficients and the residuals monitor and write down the converged coefficient values:

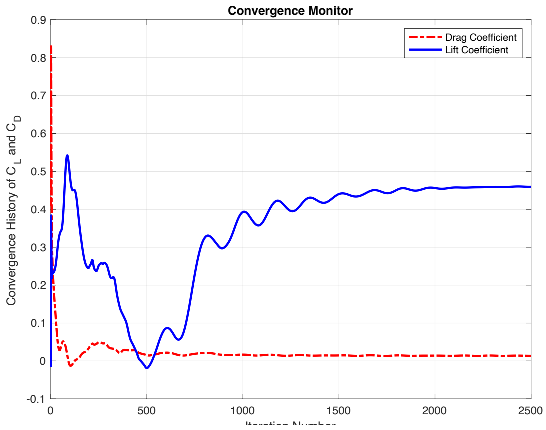


Figure 5: Coefficients Convergence at $\alpha = 2°$

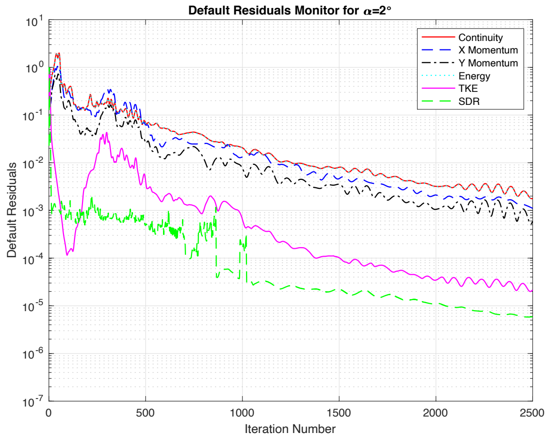


Figure 6: Residuals Convergence at $\alpha = 2°$

i. $C_l = 0.45941$

ii. $C_d = 0.01349$

b. What are the criteria you have used to assess convergence? (75 words max.)

The convergence was assessed based on the change in 4 significant figures of each coefficient. If the coefficients remained unchanged within 4 s.f. for approximately 100 consequent iterations, then the convergence criterion is met. After 2000 iterations, drag and lift coefficients values stop varying substantially. The STAR-CCM converged values were also cross-checked by the value comparison between the experimental results and XFoil from the previous sections.

c. Briefly comment on the suitability of using a single mesh for multiple incident angles. Would the y+ target still be satisfied at different incidences? (125 words max.)

If one controls on the mesh parameter and increase incident angles, the upper boundary on y+ is increased. Therefore, for a mesh configuration, at TE and LE where the value of y+ is less than 1 at higher incidences, this value can overshoot 1 if α is too high. Based on the simulation, the proximity of y+ to 1 is encountered at TE and LE only, and the targeted y+ is satisfied at other locations for higher incidences. The STAR-CCM simulation at higher AoAs and fixed mesh parameter showed that the targeted y+ exceeded 1 at multiple points, thus making the mesh unsuitable to predict flow characteristics at those regions. Hence, to capture the behaviour well the mesh needs to be refined further at higher AoAs.

d. What happens when you try to simulate a high incidence flow (9 degree or higher angle of attack), why do you think this happens? (125 words max.)

From the velocity profile, flow separation on the upper surface was encountered at AoA larger than 9 degrees. The default residuals and lift and drag coefficient could not converge through 6000 iterations because of the encountered strong wake and early separation at higher α. This happens because the drag coefficient increased dramatically due to an increase in pressure drag at high AoA. Unsteady and nonlinear behaviour from which flow separation originates could not be captured well by a simple steady solver and hence no convergence can be achieved. The nonlinear behaviour might have originated from the separation bubble creation, flow attachment and transition.

e. In order to accurately simulate the flow, how could you further refine the mesh for flows with high incidence angles? (100 words max.)

The most complex behaviour of the flow occurs at the near-wall region at TE and LE where nonlinearities due to viscosity could affect the simulation accuracy. Hence, refining the near-wall mesh such that y+ is always smaller than 1 would simulate the flow more accurately. To achieve that, the first layer prism thickness needs to be reduced, increasing the count of the prism layers (>25) such that the prism growth rate $g \geq 1.15$. Increasing thickness of prism layer can be a good addition because the boundary layer gets thicker at higher AoAs due to transition and flow separation.

f. Comment on the suitability of the applied *STAR-CCM+* physics models? (125 words max.)

The best environment for STAR-CCM+ is when the flow is fully turbulent and the flow is attached, i.e. no separation is encountered because STAR-CCM uses a fully viscous and turbulent model. STAR-CCM+ can yield inaccurate results for cases when the boundary layer is laminar or/and transition is present. At high Reynolds (Re > 1e6) boundary layer is usually turbulent, and the outputted results can be accurate. At low Re, when the boundary layer is laminar STAR-CCM is not suitable either as it overestimates drag due to higher C_f of turbulent flows. As evident from the plots, STAR-CCM is not suitable when there is separation leading to transition (at high α).

STAR-CCM+ vs XFOIL Comparison

1. Comparative Plots (11%)

a. Provide the pressure coefficient profile plot, the lift/drag vs. α plot and the drag polar plot:

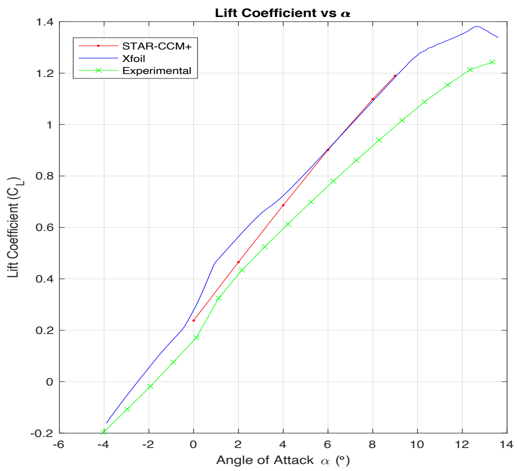


Figure 7: Lift Coefficient vs AoA

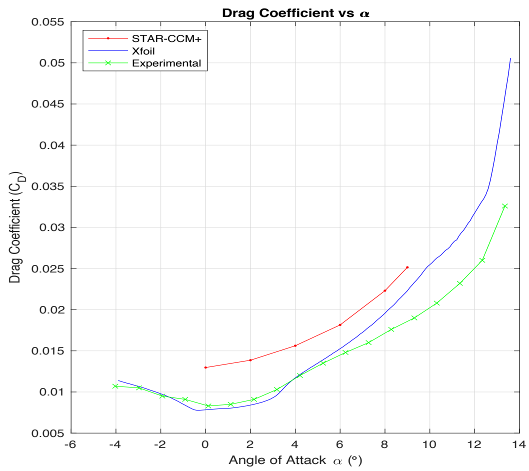


Figure 8: Drag Coefficient vs AoA

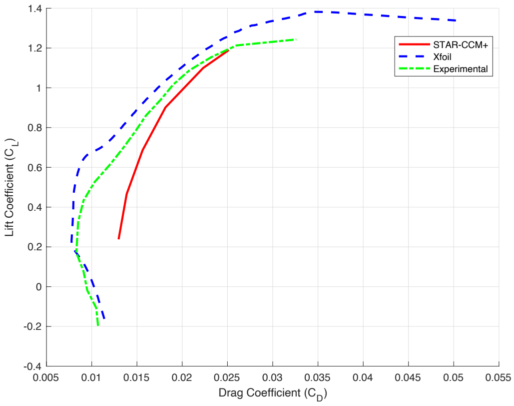


Figure 9: Drag Polar

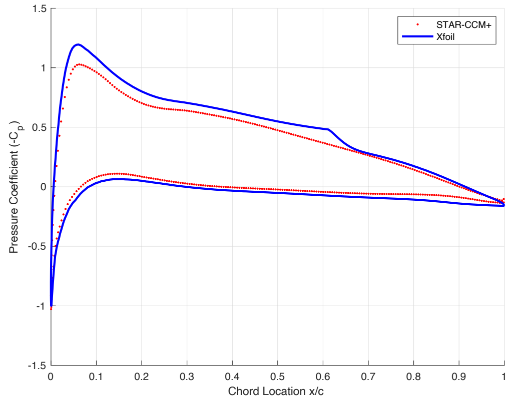


Figure 10: Pressure Distributions at $\alpha = 2°$

Comparative Study (15%)

2.

a. Interpret and comment on the Cp results of *STAR-CCM+* and *XFOIL*, including any agreements or differences. (125 words max.)

The resemblance between the two plots is recognisable. The deviation is noticeable on the top surface at $x \in [0.025, 0.65]$. The difference could occur due to different modelling approaches. STAR-CCM models the fluid as a fully-turbulent flow adding more drag and increasing static pressure, whilst XFoil uses a free transition model. Before the transition at x=0.62 XFoil uses laminar flow model and hence yields higher -Cp values while after the point of transition, XFoil uses turbulent model. Thus, both STAR-CCM and XFoil results agree with each other after x=0.62. However, STAR-CCM does not agree with XFoil before x=0.62 because it still uses the turbulent model there. Because the flow on the lower surface is laminar the disagreement between two models is significant from x=0 to x=c.

b. Identify the differences of the drag polar as well as the lift and drag coefficient results between *STAR-CCM+* and *XFOIL* and the experimental results. What is the reason behind the deviations? (400 words max.)

The lift coefficient is overestimated in both simulations. Yet because the boundary layer is relatively thin at low angles of attack, the match between the three lines is strong. Although STAR-CCM uses a fully turbulent model, it still considers viscous effects of the wake which reduce lift and increase drag. As a result, STAR-CCM agrees with experiment results at $\alpha \in [0,2]$ but starts deviating at higher AoA. For low flow speed segregated flow model seem reasonable even though STAR-CCM's model is turbulent. Applying the same mesh to a range of angle of attacks is not adequate for the reasons associated with y+ described above. STAR-CCM would have produced better results had the variable mesh been used for every α. XFoil cannot be improved for higher AoA because the outer flow inviscid solver can never capture turbulent regimes, therefore at high α the deviation between the experiment and XFoil increases.

STAR-CCM overestimates the drag coefficient because of the turbulent modelling assumption while XFoil matches the experimental drag well for $\alpha < 6$. XFoil uses a free-transition model, and STAR-CCM uses a full turbulent model hence the drag for a turbulent boundary layer will be higher than that of laminar nature. Shear stress and hence C_f is higher for turbulent flows by definition. XFoil fits the low angle experimental data very accurately but starts deviating at $\alpha > 6$ because of the free-transition assumption and the flow separation for which drag cannot be modelled well. XFoil is limited in predicting flow separation that can occur at higher AoAs because it uses a viscous solver at the boundary layer only but an inviscid solution for the outer flow field.

Because of the aforementioned least constraining assumptions, XFoil mismatch is less severe making the drag polar estimations fit the experimental data better than that of STAR-CCM.

Note that in order to better judge on STAR-CCM+ fit, a wider range of α should have been tested because as seen from the plots, STAR-CCM has the least amount of data points available out of the three.

c. Which 2D simulation (*XFOIL* or *STAR-CCM+)* do you prefer to use and why? (125 words max.)

XFoil was simpler and more intuitive to understand in terms of users' inputs. Additionally, XFoil yielded more accurate C_L, C_D and pressure distribution at low angles of attack and computed the results quicker for wider ranges of incidence angles. One could even see where the separation occurred because of free-transition model XFoil uses.

STAR-CCM+ was less user friendly and required more competence to start using its tools adequately. STAR-CCM, unlike XFoil, takes a broader range of inputs thus allowing to specify flow characteristics more precisely. However, because of the physics model described above, STAR-CCM results were less accurate in the given flight conditions.

Thus, for a 2D flow in the given flight operation, I would choose XFoil.

Part B: 3D Wing

AVL

1. **Wing geometry** (2%)

a. What is the nominal wing area, the tip chord, the root chord, and the x-coordinate of the tip section?

$$S_{ref} = 0.13935;\ c_{tip} = 0.0762;\ c_{root} = 0.2286;\ x_{tip} = 0.34016$$

2. **Drag Polar** (5%)

a. Show the *RAE101* drag polar as obtained from *XFOIL*, and your data-fitted polar that *AVL* will operate on. (Disregard the two branches of the polar that are extrapolated beyond the stall angles.)

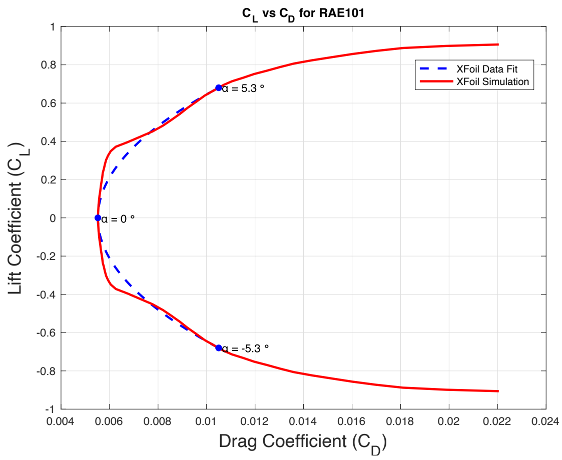


Figure 1: Drag Polar

b. Give your chosen parameters of the keyword `CDCL` in *AVL*'s format.

$$CD_1 = 0.0105;\ CL_1 = -0.680; CD_2 = 0.00552;\ CL_2 = 0; CD_3 = 0.0105;\ CL_3 = 0.681$$

3. **Vortex Lattice Convergence** (12%)

a. Show the vortex-lattice convergence plots using the lift and induced drag coefficients (calculated from Trefftz plane) at a 2° angle of attack with a uniform panel distribution.

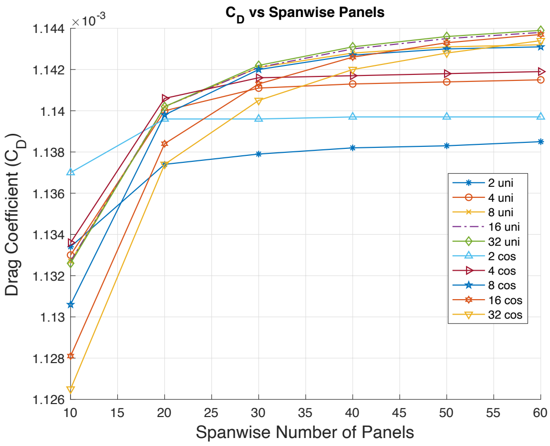


Figure 2: Drag vs Number of Panels

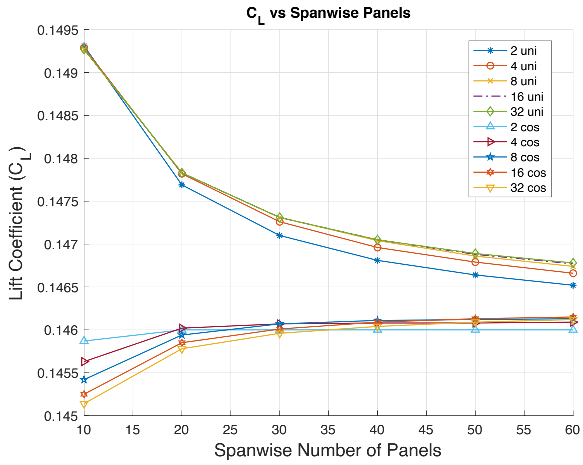


Figure 3: Lift vs Number of Panels

b. State which vortex-lattice spacing distributions you chose as the most efficient and briefly explain why. (125 words max.)

The nonlinearities occurring at the trailing edge and the leading edge, such as suction peak flow acceleration of the aerofoil require additional panels to describe such phenomena better. Hence the cosine spacing was chosen in a chordwise direction because this distribution has a higher density of vortices at the edges and fewer in the middle.

Root fuselage interference and wingtip vortex shedding occurring over the wing requires finer mesh at these regions. The resulting vortices from the sweep and flow distribution would be best to model with finer spacing at the root and the tip. Hence the cosine spacing distribution was chosen in a spanwise direction for the same reason as above.

c. Based on the new selected distributions, which number of span and chordwise vortex-lattices will you use for the remaining questions? Make a very brief comment on the comparison of these numbers to those you obtained with the uniform distribution. (50 words max.)

25 spanwise and 4 chordwise panels were chosen to balance out the computational expenses of higher number of panels. These values were within the asymptotic limit of C_L and C_D. For C_L all chordwise panels converge well, but the difference between 2 and 4 is substantial for C_D convergence, 0.25% against 0.15% tolerance relative to the asymptotic value of 1.114e-3 and the computational time difference is negligible. Uniform panels for C_L converge to higher values while C_D convergence requires more chordwise panels.

4. Lift Distribution Results (10%)

a. Show, in one plot, the lift distribution and sectional lift coefficients, both from the Trefftz plane and the experimental data.

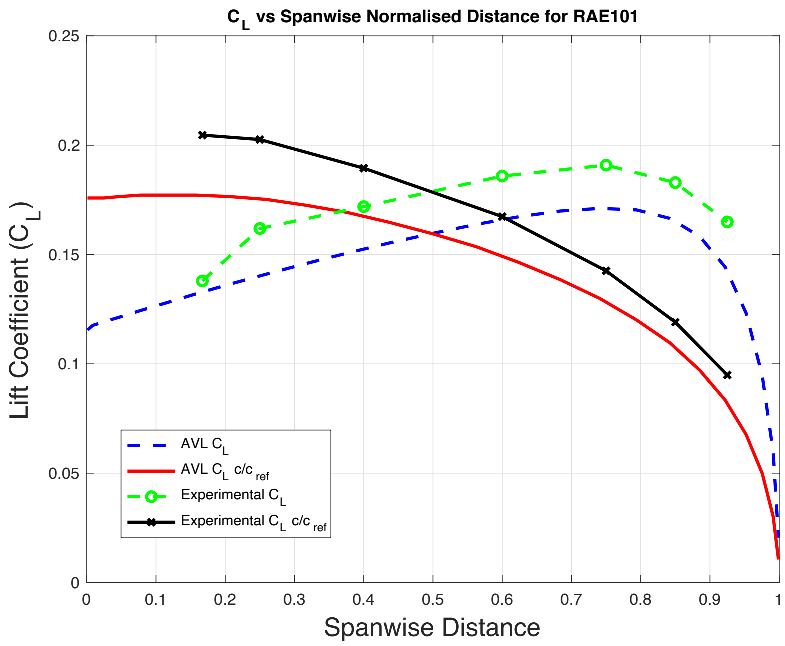


Figure 4: Lift Distribution

b. Briefly discuss the results and comment on how well the calculated and experimental data match and explain any discrepancies. (275 words max.)

The outputted AVL results yielded elliptic lift distribution because the wing is tapered. Elliptic lift distribution is an optimal performance parameter because it minimises induced drag. However, AVL underestimates the lift coefficient in both normalised and total manners as per the above plot. First, Trefftz plane that AVL uses to evaluate performance coefficients does not take into consideration the real 3D effects of the wing. Shed vortices coming off the tip reduce the incoming AoA and decrease C_L as a result. The simulation does not account for the fuselage interference as $y/s \to 0$. The drop in experimental C_L demonstrates that aerodynamic interference can be significant. The experimental and simulated lift distributions tend to converge towards each other as $y/s \to 1$ because of the selected cosine distribution. The mean squared error in C_L between the two frameworks is 3.7e-4 and 3.2e-4 for total and chord normalised C_L respectively. In percentage terms, the differences yield 17.3% and 14.4% with respect to the experimental values for a given $y/s \in [0.167, 0.925]$. The discrepancy might occur due to the interpolation of the aerofoil's drag in the range of $\alpha \in [0,8]$ instead of interpolating at $\alpha = 2°$ directly. The relative differences imply that the AVL simulation and the experimental lift distribution results behave in close agreement for the midsection of the wing. Lastly, the provided data could have been obtained from the wind tunnel where the wall effects influenced the experimental data representativeness.

5. **Determining the maximum lift over drag ratio** (5%)

a. Provide the plot showing the lift over drag ratio L/D for the specified range of angles of incidence.

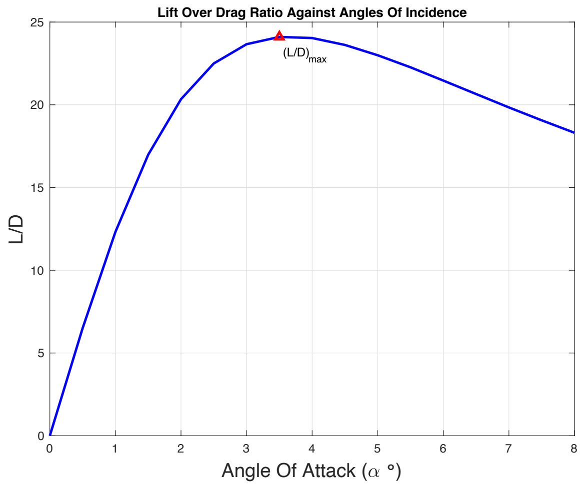


Figure 9: L/D vs AoA

b. What is the optimal angle of incidence for this wing and which L/D ratio can it achieve?

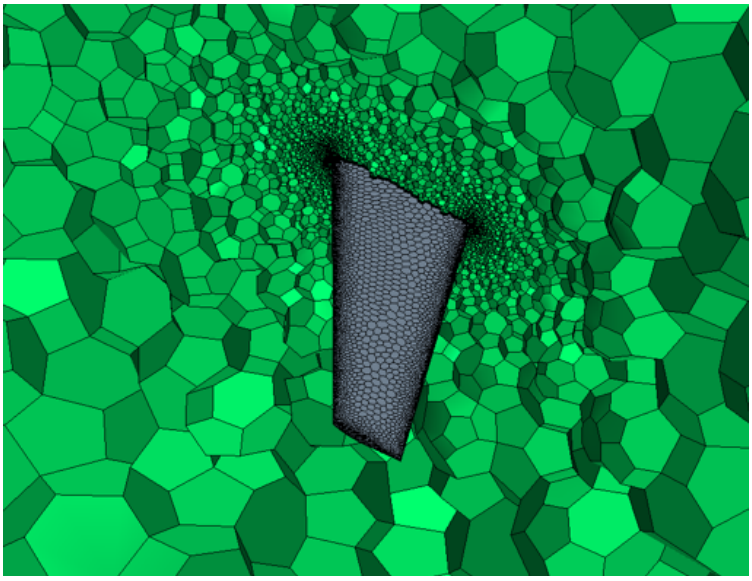
$\alpha_{opt} = 3.5°$; $L/D_{opt} = 24.1$.

6. Wing Configuration file (5%)

3D STAR-CCM+

1. Meshing (12%)

a. Provide the mesh scene image showing the symmetry plane and wing surface mesh.



b. List your mesh parameters and justify your choice – how did you arrive at the values you used? (250 words max.)

The method that was used for 2D STAR-CCM has also been used in this section to identify mesh parameters. The only changes were that flow velocity was set to 270 m/s to satisfy M=0.8 requirement, and Re increased to 1000000. The following parameters were identified:

Prism Layer Count	Boundary Layer Thickness (δ)	First Prism Layer Thickness (2*Δs)	Base size
38	3.58e-3 (m)	1.01e-6 (m)	0.01

Table 1: 3D Mesh Parameters

The boundary layer thickness was calculated using the Blasius solution. Then iterative simulation runs were performed to select the correct base size for a target total cell number. The base size was set to 0.01 precisely such that the total number of cells would be around 617000 because this number guaranteed good mesh refinement at the targeted zones. At that base size, the leading was well modelled with a reasonable curvature instead of a sharp indent. The mesh was then inspected with particular attention to problematic areas, i.e. trailing and leading edges to ensure smooth curvatures and fine meshing. The selected number of cells satisfied the outlined criterion of fine meshing at these zones.

c. How did you ensure the y+ target was met? Were there any issues (minor or otherwise) encountered in this task, if so, what was it? Do you expect it to significantly affect the results? (150 words max.)

Y+ is larger for the transonic regime M=0.8 than for M=0.4 hence the mesh parameter was selected based on the former simulation as it is more constraining. Because of the shock interaction with the boundary layer, the prism layering at the transonic regime was an important factor to consider too. So, the simulation was instantiated at M=0.8 with the initially estimated parameters of δ, Δs, base size and layer count. To ensure the targeted wall value y is met, through iterations Δs value was changed until y^+ became slightly less than 1 for the case of M=0.8.

2. Physics Settings (5%)

a. What values have you set for flow speed, temperature, pressure and viscosity:

i. $U_\infty|_{M=0.4}$ = 141.2 m/s ; $U_\infty|_{M=0.8}$ = 270.3 m/s

ii. $T|_{M=0.4}$ = 310 K ; $T|_{M=0.8}$ = 284 K

iii. $P|_{M=0.4}$ = 61600 Pa ; $P|_{M=0.8}$ = 23700 Pa

iv. $\mu|_{M=0.4}$ = 1.67e-5 Pa-s; $\mu|_{M=0.8}$ = 1.49e-5 Pa-s

b. How did you obtain the values set? (50 words max.)

The velocities are found using $M\sqrt{\gamma RT}$. The static temperatures were derived from the total temperature (320 K) which was given. Viscosity and pressure were found from the ISA standard atmospheric model at 4000 m and 10500 m for Mach=0.4 and Mach=0.8 respectively.

3. Results (19%)

a. Provide the iteration history of the lift/drag coefficients and residuals plots and write down the obtained converged coefficients:

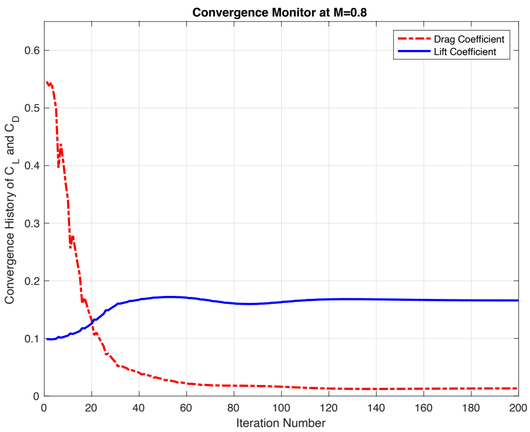


Figure 1: Coefficients Convergence at M=0.8

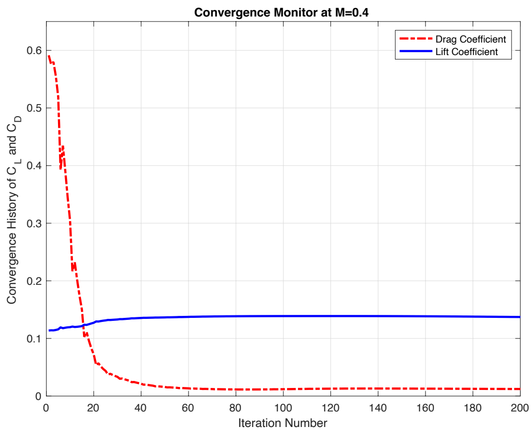


Figure 2: Coefficients Convergence at M=0.4

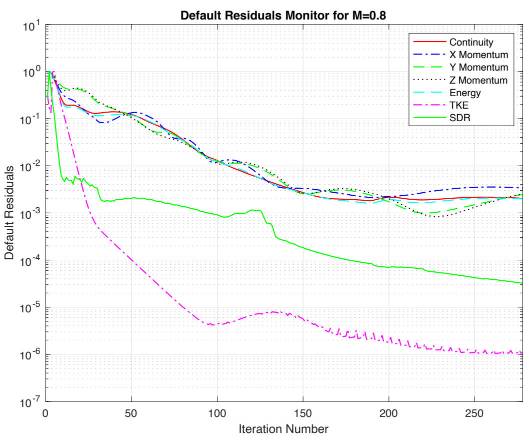


Figure 3: Residuals Convergence at M=0.8

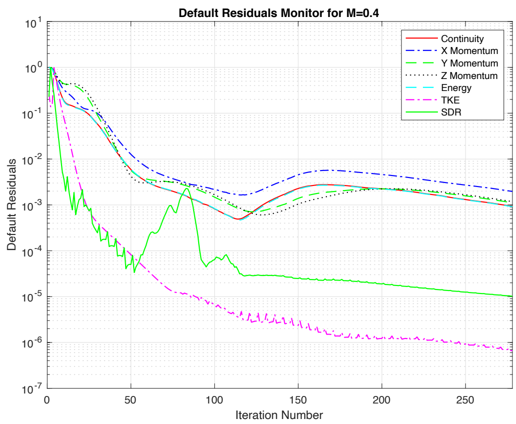


Figure 4: Residuals Convergence at M=0.4

i. $C_L|_{M=0.4}$ = 0.13479 ; $C_L|_{M=0.8}$ = 0.16527

ii. $C_D|_{M=0.4}$ = 0.01165 ; $C_D|_{M=0.8}$ = 0.01285

b. Provide the Cp vs. x plots as mentioned in ‘PROCESSING INSTRUCTIONS’:

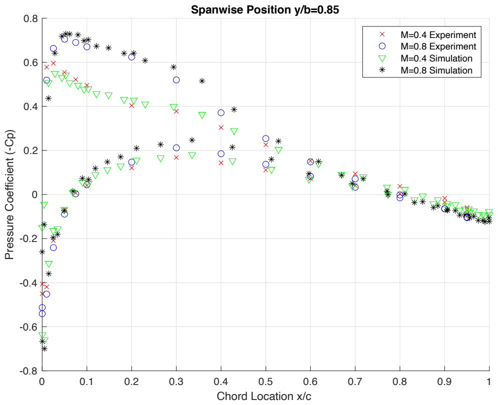


Figure 5: Pressure Distribution at y/b=0.85

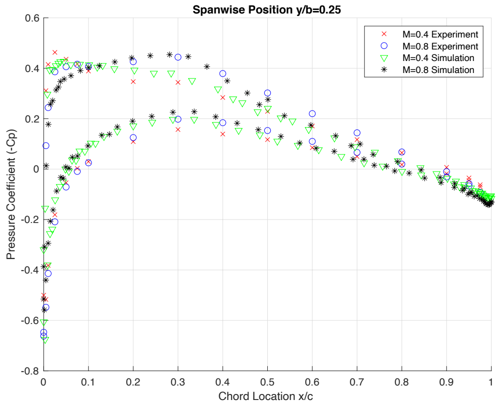


Figure 6: Pressure Distribution at y/b=0.25

c. Discuss the Cp vs. x results of *STAR-CCM+* and the experimental data. Interpret any notable differences or agreements. (150 words max.)

The agreement between STAR-CCM and the experimental data is strong because of the turbulence model that STAR-CCM implements for this high Re case. The -Cp values for M=0.4 agree with the experimental points well, but at the suction peak, STAR-CCM tends to slightly overestimate Cp, i.e. $C_{p,exp\ ,suction} < C_{p,STAR,suction}$. This could have happened due to the lack of finer grid at the leading edge where the flow acceleration occurs. For both Mach regimes because of the sweep, the suction peak at y/b=0.25 is closer to the TE and is not as distinct compared to y/b=0.85. This happened because the sweep pushed the suction peak towards LE by shifting the pressure centre making the pressure gradients larger at the tip. STAR-CCM models this feature well because of the assumptions inherited in the turbulent model. At y/b=0.25 there is a region of approximately constant pressure distribution, i.e. a shock wave occurs, and STAR-CCM predicts it well too.

Summary Report (10%)

Make sure to submit your Summary Report for this simulation file zipped alongside this document to BBL. **Do not include the Summary Report in this document**.

STAR-CCM+ vs AVL Comparison

1. Comparative Plots (4%)

a. Provide the comparative pressure coefficient deltas (dCp) plot from the two subsonic simulations (*XFOIL* and *STAR-CCM+*) with the experimental data:

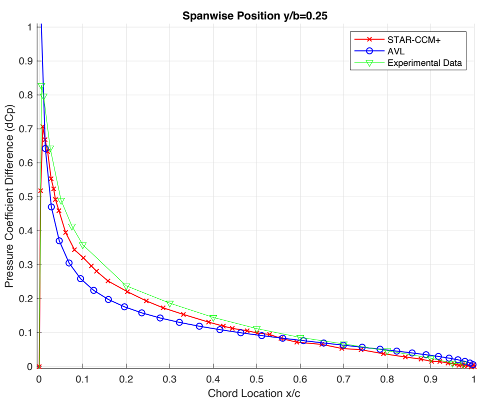


Figure 7: dCp at y/b=0.25

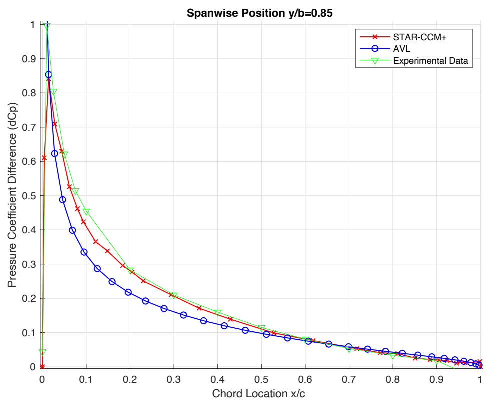


Figure 8: dCp at y/b=0.85

2. Comparative Study (11%)

a. Interpret the dCp results of the experimental data, *STAR-CCM+* and *AVL* as well as any differences between the 3 datasets. (175 words max.)

The level of agreement between STAR-CCM and experimental data is remarkable. At the problematic areas of the leading and trailing edges, STAR-CCM was able to produce accurate results at both spanwise locations. The better agreement can be justified by the turbulence model that STAR-CCM+ uses, and since the flow is at Re=1e6, the accuracy of STAR-CCM is high. AVL results do somewhat agree with the experimental data, but the deviation at $\frac{x}{c} \in [0, 0.55]$ is substantial. This could have happened due to the Vortex Lattice Method designed for low Re, assuming quasi-steady flow, therefore neglecting unsteady vortices shedding (Thomason and Richardson, 2012). Lastly, the dCp value at x=0 and x=c has to be 0 as at this location, the upper and lower surfaces meet each other by definition. STAR-CCM meets this constraint, but AVL's dCp at x=0 started from a much higher dCp=2.20 at y/b=0.25 and dCp=3.05 at y/b=0.85. The above-mentioned discussion concludes better suitability of STAR-CCM for 3D wing predictions not only in terms of experimental data agreement but also constraints fulfilment.

b. List a total of at least four advantages and disadvantages of *STAR-CCM+* over *AVL*. Which software would you use when designing a wing and why? (200 words max.)

Advantages	Disadvantages
Model Complexity Aerodynamics phenomena, such as turbulence, viscoelasticity, multiphase flows.	**Computational Cost** Takes a lot of RAM to model a flow past a 3D wing with a fine mesh
Complex Geometry Not only standard aerofoils can be used, but more complex geometry can also be defined with detailed boundary layers. Both 2D and 3D flows can be modelled.	**Software Costs** STAR-CCM is a commercial software and costs money, unlike free license AVL.
Accuracy Improved drag, lift, pressure and moment estimations.	**High Competence Required** Requires a good level of understanding of software tools and user's interface.
Variable Control Real physical conditions can be defined more precisely as more parameters can be defined.	**No Automated Grid** The grid needs to be tuned manually and iteratively, for instance, to satisfy y+ condition.

Table 2: Comparative analysis of STAR-CCM+ over AVL

The choice of the software depends on the application and the regime at which the wing needs to be modelled. In the commercial aircraft industry where lives of passengers, commercial profitability and durability are at stake STAR-CCM is a required choice for its higher accuracy. For preliminary designs of wing prototypes/quick results/general understanding, AVL may be a better choice due to its simplicity and availability. AVL does not require high competences from users and can yield relatively accurate results when time is a constraint.