FEEG6005 2021-22 Coursework 2 Template

Erik Valls
30405351

Part 1 RANS solution using a medium resolution mesh

Q1

1) Reynolds number: 291620

2) Boundary layer thickness:

$$Bl = 0.37 * x * Re^{-\frac{1}{5}} \tag{1}$$
$$Bl = 0.0298\, m$$

3) Dimensionless maximum boundary thickness:

$$Bl = 0.238 * \left(\frac{\rho^{13}}{u^{.18} * \mu^{13}}\right)^{\left(\frac{1}{25}\right)} \tag{2}$$
$$Bl = \ 1452.39$$

Q2

1) **Present the corrected boundary conditions**:

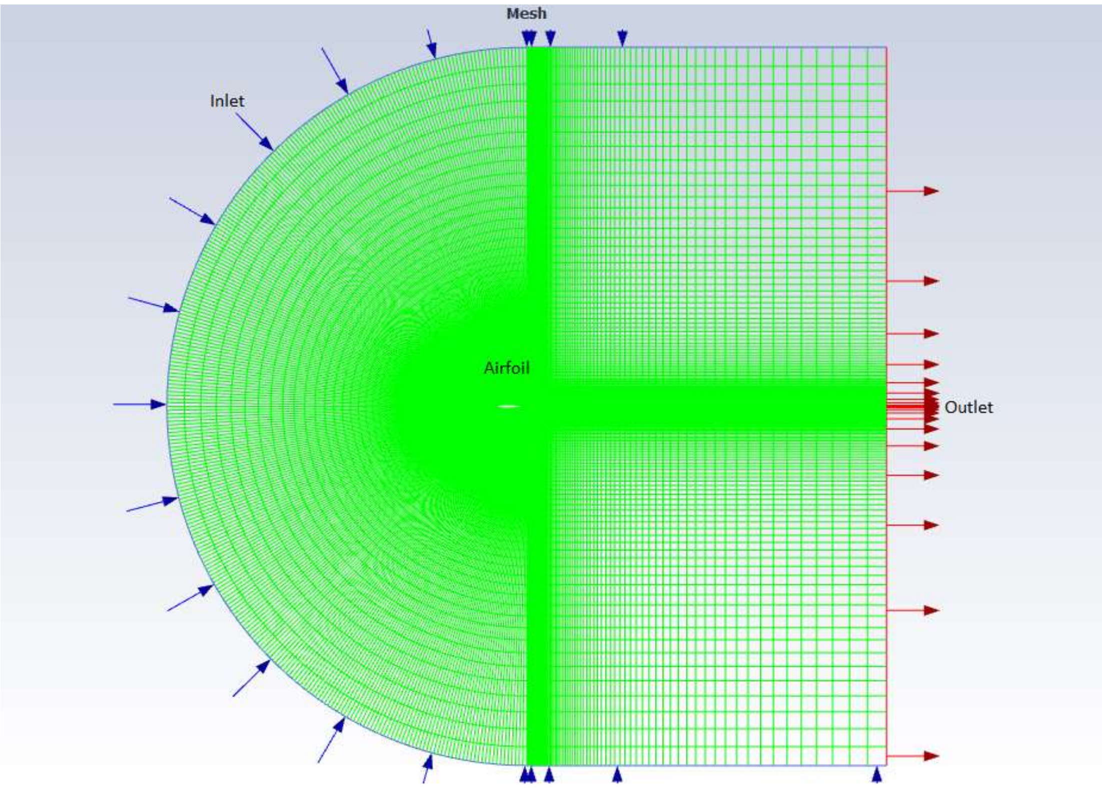


Figure 1. Corrected mesh

2) **A table to details of all the boundary conditions**:

Table 1. Corrected boundary conditions

	Inlet	Outlet	Airfoil	Interior, Proj: above, below, upstream, downstream
Boundary condition type	Velocities	Pressure	Wall	Interior
Parameters	Turbulence Intensity: 0.3% Viscosity ratio: 1	Turbulence Intensity: 0.3% Viscosity ratio: 1	No slip	

Q3

1) y_1^+ at the trailing edge: 21.3832

$$\left(\frac{u\cdot}{U_\infty}\right)^2 = 0.0296 * Re^{-\frac{1}{5}} \quad (3)$$

$$u \cdot = 0.0489ms^{-1}$$

$$y_1 = \frac{h}{2} = \frac{0.003}{2} = 0.0015\, m$$

$$y_1^+ = \frac{y_1 * u\cdot}{v} = 21.3832\, m \quad (4)$$

2) y_1^+ at: (using the same procedure but varying the Reynolds number)

- 25% chord: 24.5628 m
- 50% chord: 22.9818 m
- 75% chord: 22.0073 m

3) **What causes the difference of y1+ over the chord? Give a suggestion how to reduce the effect of variation of y_1^+ on the estimation of mesh resolution for the RANS model selection:**

The variation is caused by the difference in the Reynolds number as we approach the trailing edge.

4) **Turbulence model to be used and explain why it is chosen:**

Due to the resolution achieved, the k-epsilon model has been chosen for the task. Spalart-Allmaras models are suitable for resolutions that are >30 and <5. Since it is a medium resolution mesh, this resolution is reasonable enough to give a decent first approach to the solution.

Q4

1) **Estimate the number of grid points within the boundary layer at the trailing edge:**

$grid\ points = \frac{Bl}{h} = 9.9567 \sim 10\ grid\ points$ (5)

2) Comment whether this number is sufficient to resolve a boundary layer:

To resolve the boundary layer, there should be at least 10 grid points that cover it completely. We do not cover the whole area required to resolve it completely (by a very little), since this is a medium resolution mesh (not refined) where all grid points are equal, it is not sufficient to have a real solution, but an approximation.

Q5

Table 2 Spatial-discretisation schemes, solver, and residuals to be achieved.

Pressure-Velocity coupling scheme	**SIMPLEC - for better convergence**	**No skewness correction**	
Spatial-discretisation schemes	Gradient: least squares cell based	Momentum, Turbulent kinetic energy & dissipation: second order	Pressure: standard
Velocities	x: 0.9945 ms^{-1}	y: 0.1045 ms^{-1}	U_∞= 1ms^{-1}
Turbulence quantities	Inlet Intensity: 0.3% Viscosity ratio: 1	Outlet Intensity: 0.3% Viscosity ratio: 1	
Residuals (planned residuals to be achieved)	Continuity: 10e-5	xvel, yvel: 1e-5	k, epsilon: 1e-5

The SIMPLEC coupling scheme is shown to have a better convergence. - A few tests were performed using the SIMPLE scheme, but the residuals diverged to higher values, so it was not stable.
For the spatial discretization scheme, second order seems to have a much precise values than first order, since first order is used for initialization. While being faster when performing the simulation, it is no interest on using it since it is not a complex calculation, so second order discretization was chosen.
Velocities are normalized to 1 and the corrections are carried by changing the fluid's kinematic viscosity to match the Reynolds number. Since a 6-degree angle of attack is necessary, velocities are divided in the x and y components:

$$U_x = U_\infty * \cos(6)\,; U_y = U_\infty * \sin(6)$$

The turbulent intensity is adequate for external flow airfoils. The viscosity ratio is the lowest from the acceptable values for this type of case (1-10)
Finally, the residuals were chosen arbitrarily, but it was shown how a higher value of convergence for the residuals might lead to fluctuations in the cl and cd values, not giving the program enough iterations to give a stable value. These fluctuations were observed in the continuity range of 1e-4.

Q6

1) Final achieved residuals*:*

Table 3. Final residuals for medium mesh

RESIDUALS					
Iterations	continuity	x velocity	y velocity	k	epsilon
594	9.90E-06	2.06E-08	6.11E-09	1.51E-06	2.10E-06

2) **Compare the lift coefficient *Cl*, drag coefficient *Cd* against the reference data:**

Table 4 Cl and *Cd* of *the NACA 0012 Airfoil Section at a 6-degree angle of attack*

	CFD	Expt. (Re=2x10^6, Mach M=0.15, transition fixed at 5% chord) [1]	Expt. (Re=2x10^6, Mach M=0.15, free transition) [1]	Absolute error	Relative Error
Cl	0.60608	0.6084	0.6250	0.00232	0.38%
Cd	0.02066	0.0134	0.0087	0.00726	54.21%

3) **Discuss on any discrepancy between the numerical data and experimental data. List at least three causes (with evidence) on discrepancy between the numerical data and experimental data.**

It can be appreciated that while the cl error is very small, the cd error is very high, more than 50%. This can be caused by the boundary layer being unresolved, causing the skin drag to give higher values than it should.

Q7

1) **Assessment of *Cp* (e.g., at the stagnation point):**

Stagnation pressure coefficient: 1.00445

2) **Compare the *Cp* distribution against the experimental data, and discuss on any discrepancy:**

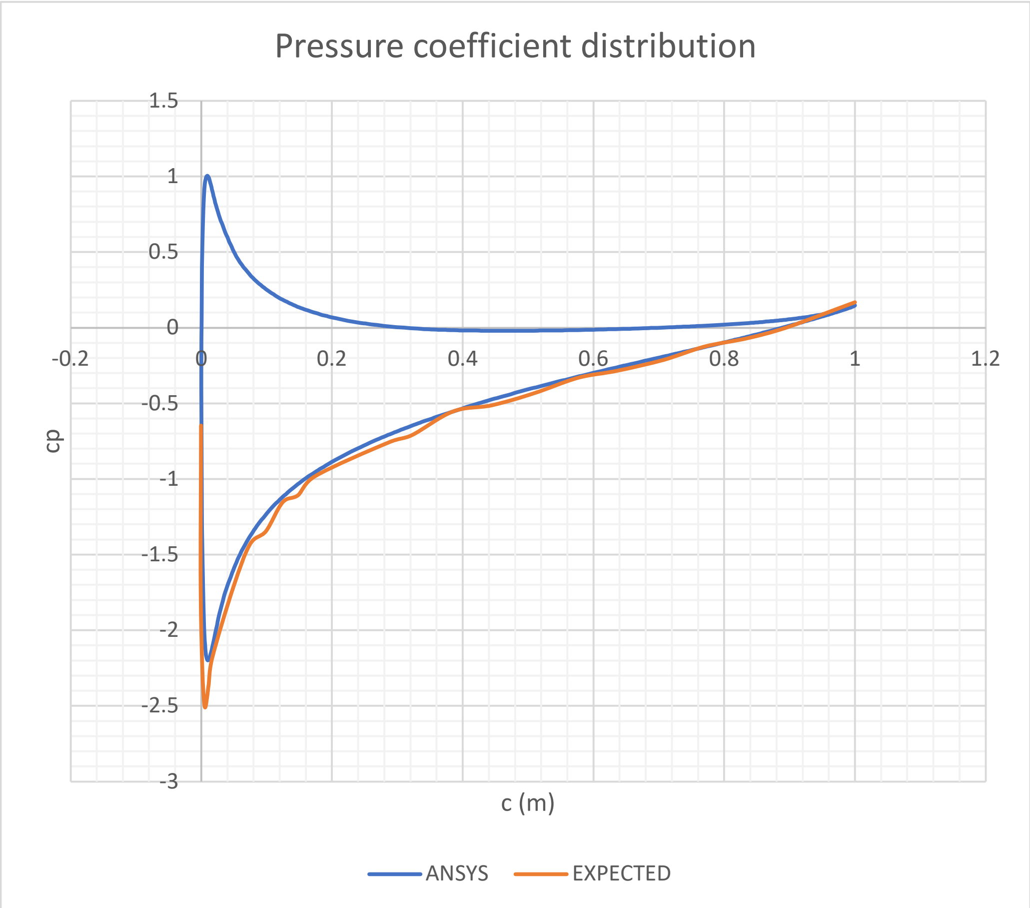


Figure 2. Pressure coefficient distribution achieved compared to the experimental

There is a slight discrepancy on the maximum values. It can be apreciated that the experimental values are slightly higher in magnitude than the ones simulated. This can be caused by differences in the atmospheric conditions when performing the experiments, which can be derived in bad choice of the simulation values.

Besides that, the two curves seem to be very similar, and it is always an error that we expect to have.

3) Plot the C_f distribution, compare the magnitude of *Cp* against that of C_f, and discuss the accuracy, error and confidence of the calculation of the two quantities.

In terms of order of magnitude, the maximum absolute value of cp is 200 times higher than cf.
In terms of the distribution, the upper skin friction coefficient distribution seems to match accurately with the cp upper distribution, but in opposite signs. This is caused by the flow speed in the upper surface being much higher, increasing the friction coefficient but turning the pressure negative causing suction forces. When focusing on the lower surface of the airfoil, the cp distribution goes stays positive, since the velocity here is lower and therefore the skin friction is decreased with respect to the upper pressure skin friction distribution.

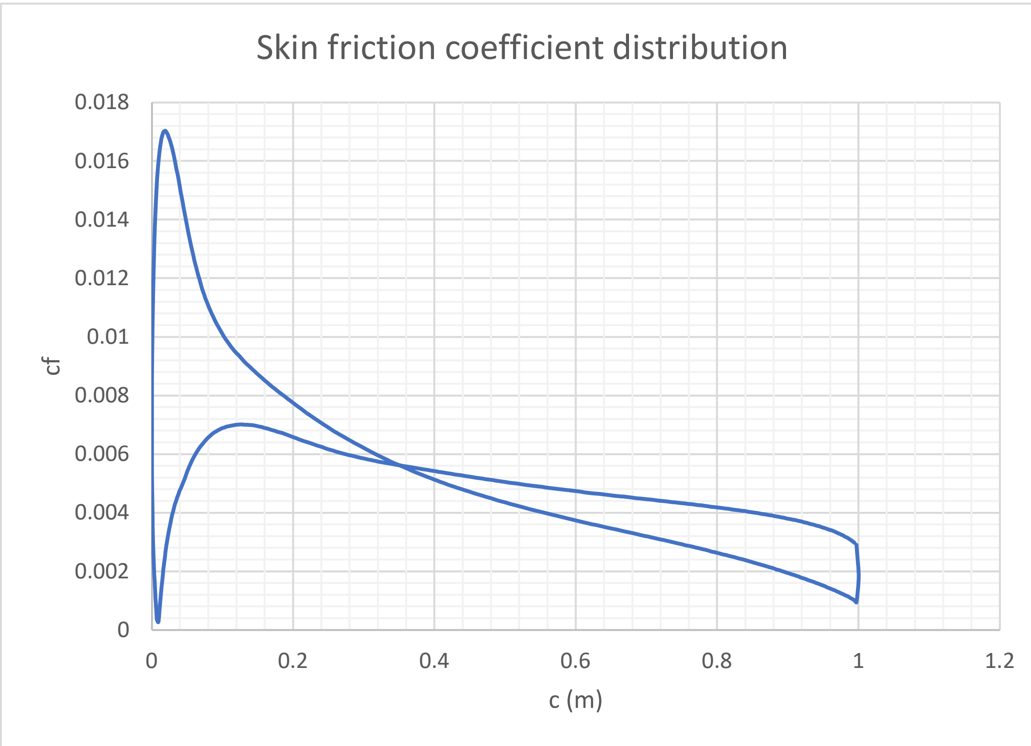


Figure 3. Skin friction coefficient along the chord.

Part 2 RANS solution using a refined mesh

Q8

1) **y_1^+ at the trailing edge of the new mesh**:
Using values from Q3:

$$y_{1\,refined} = \frac{h_{new}}{2} = \frac{0.000818\,m}{2} = 0.000409\,m$$

$$y^+_{1\,refined} = \frac{y_1 * u_*}{v} = 5.83\,m\ at\ trailing\ edge$$

2) **Describe the refinement process and explain why and where the refinement is conducted:**

   The refinement process consisted of a 2-step manual cell register coarsement of 20 cells in each step. This brought down the cell height to 8.18e-4 and the y+ to 4.09e-4 and the resolution got reduced to approximately 6.

3) **Re-check all of the numerical settings, in particular the turbulence model and the inflow conditions. Change these settings as appropriate:**

   The turbulence model has therefore been changed to Spalart-Allmaras, since the resolution is very close to 5. Turbulence intensity was kept to 1 in both the inlet and outlet boundary conditions.

Q9

1) **Final achieved residuals*:***

Table 5. Final achieved residuals for refined mesh

RESIDUALS				
Iterations	**continuity**	X velocity	Y velocity	Nut
3000	2.40E-04	3.87E-09	5.31E-09	8.72E-08

   These final values were achieved after 3000 interations. Even though the continuity did not achieve the convergence minimum of 1e-6, cl and cd values were observed to have enough stability to be approved. This situation is also observed in other cases from other questions.

2) **Compare the lift coefficient *Cl*, drag coefficient *Cd* of the two resolutions:**

Table 6 A comparison of the lift coefficient Cl, drag coefficient Cd of the two resolutions

	CFD with medium resolution	*CFD with refined resolution*
Cl	0.60608	0.60908
Cd	0.020664	0.01773

3) **Discuss quantitatively on any discrepancy of the coefficients between the medium resolution and refined resolution**:

   In terms of the lift coefficient, the refined resolution presented a slightly higher number, which compared to the experimental values, lands slightly closer than the medium resolution value. The opposite happens to the drag coefficient, where the experimental value is lower than the ones achieved using ANSYS, even though the refined still has a better tendency than the medium resolution, just as expected.

Q10

1) **Plot y_1^+ of the refined mesh over the airfoil surface and compare with the analytical solution:**

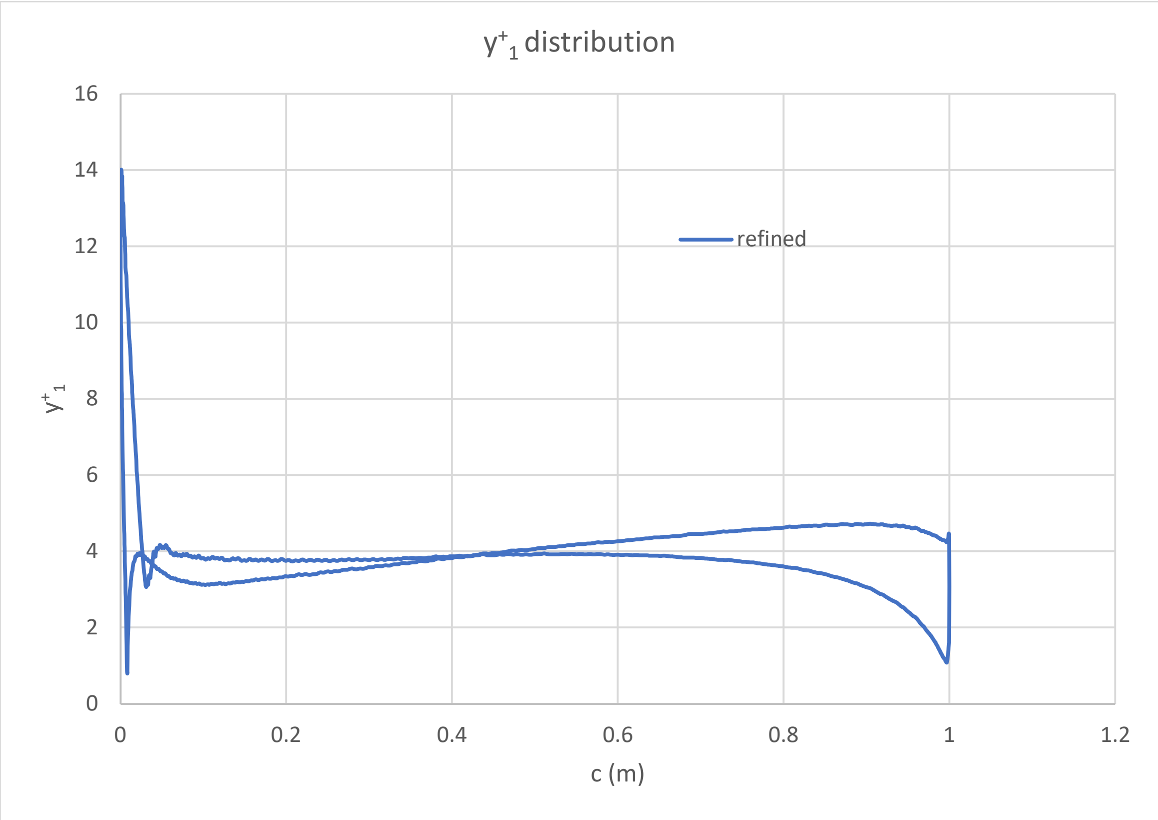


Figure 4. y^+_1 values for the refined mesh along the chord

In can be seen that most values of $y1^+$ lie in the range around 4. For the trailing edge, this value increases up to almost 5, which is reasonably close to the analytical value of 5.83.

2) **Discuss whether the chosen turbulence model is suitable for this range of y_1^+ value:**

The Spalart-Allmaras turbulence model is suitable for resolutions that lie higher than 30, and lower than 5. Therefore, as the $y1^+$ value is roughly 4 this model is suitable to be used.

Q11

1) **Justify whether you will use the medium resolution mesh in Part 1 or the refined mesh for the rest of the simulations, i.e. various angles of attack. You should consider the crucial aerodynamics forces to make the decision:**

   As seen in the results from Q9, the refined mesh gives a much closer result to the experimental value. The residuals tend to oscillate and not converge to the criteria, but they certainly achieve a small enough value to be considered correct. Knowing this, since this problem is not very complicated, calculation time is not a huge concern for us, and therefore there is no point on using the medium resolution for time saving. (it only takes a few minutes to do the simulation)

2) **Perform simulations for angles of attack α =0°, 3°, 9°, 12°, and write down the achieved residuals of the simulated cases**:

Table 7. Residuals at angles of attack 0 to 15 of the refined mesh

Degrees	***X vel***	***Y vel***	***iterations***	***continuity***	***xvel***	***yvel***	***nut***	***cd***	***cl***
0	1	0	663	9.96E-06	3.38E-08	1.15E-08	9.67E-06	0.01398	-0.00021
3	0.99863	0.05234	1648	9.96E-06	1.60E-08	6.24E-09	3.04E-07	0.01486	0.30814
6	0.99452	0.10453	3000	2.40E-04	3.87E-09	5.31E-09	8.72E-08	0.01773	0.60908
9	0.98769	0.15643	4000	1.16E-04	4.38E-09	4.37E-09	1.34E-07	0.02349	0.88780
10	0.98481	0.17365	4000	1.01E-04	1.10E-08	1.02E-08	3.07E-07	0.02650	0.97019
12	0.97815	0.20791	4747	9.71E-06	1.20E-08	6.40E-09	7.10E-07	0.03590	1.10500
13	0.97437	0.22495	5000	4.87E-05	2.55E-08	1.37E-08	1.58E-06	0.04425	1.14320
14	0.97030	0.24192	5730	9.98E-06	3.87E-08	1.90E-08	2.36E-06	0.05981	1.13200
15	0.96593	0.25882	3109	9.89E-06	3.86E-08	1.82E-08	1.05E-06	0.11412	0.92469

3) **Plot the curve of lift coefficient *Cl* versus angle of attack α, including near the stall angle, Calculate the slope of the linear region of the *Cl*- α curve and compare with that obtained from the thin airfoil theory:**

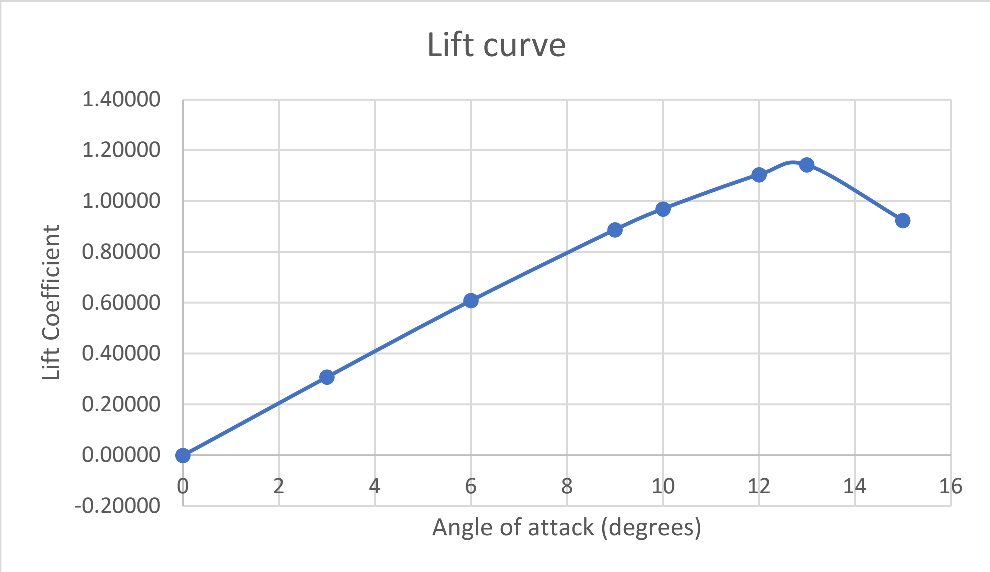


Figure 5. Lift coefficient versus angle of attack, including near the stall point

The near stall angle was set to be around 13.5 degrees. The lift curve slope of the linear region is 5.6462.

Results predicted by thin airfoil theory give a lift curve slope of 2π, but this is only true for airfoils with thickness ratios below 10%. For bigger airfoils (this is the case), the lift curve slope starts approaching 5.73. It can be stated that the simulations are close to the theory predictions.

4) **Plot the curve of drag coefficient *Cd* versus angle of attack α, including near the stall angle:**

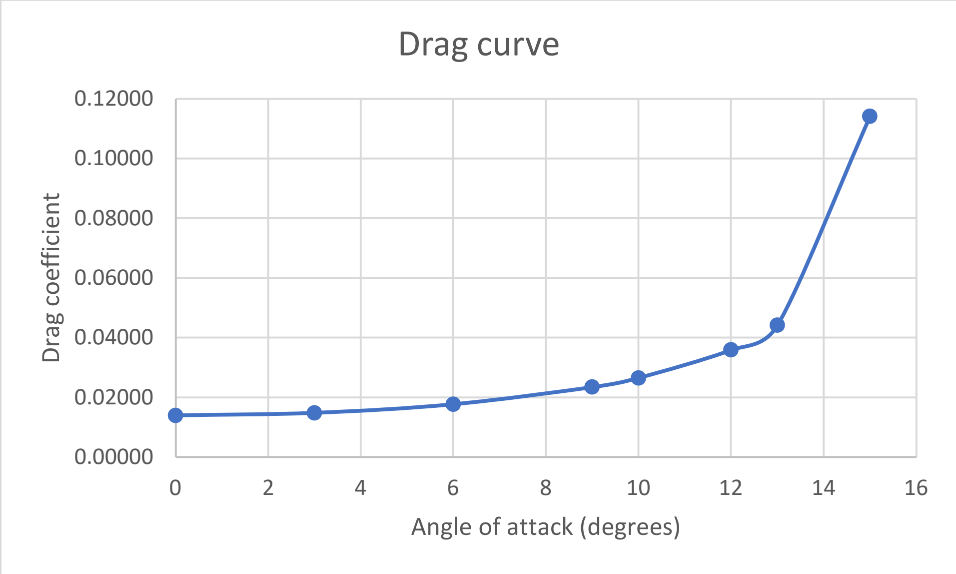


Figure 6. Drag coefficient versus angle of attack

5) **Justify the estimated near-stall angle (simulate more angles of attack as necessary):**

Having a closer look into figure 3, from 13 to 14 degrees of angle of attack, the lift coefficient decreases. This means that the stall angle has to be between these two values.
A good approximation is for it to be 13.5 degrees.

Q12

As the changes required are all related to Reynolds number quantities, to keep the same number while maintaining the given values of velocity and viscosity, the chord length has to be modified. This means that the X-plane of the mesh has to be scaled to a factor of 1.25 to give a chord length of 1.25, instead of 1, as it was originally.

Table 8. A list of the changes of settings compared to Question 8.

	Q8 settings	*Q12 settings*
Change	*c = 1*	*c = 1.25*

Q13

1) **Final achieved residuals*:***

Table 9. Final achieved residuals for new chord requirement

RESIDUALS						
iterations	continuity	xvelocity	Yvelocity	nut	cd	cl
4000	1.99E-05	4.45E-10	5.66E-10	1.71E-08	2.08E-02	7.77E-01

2) **Compare the lift coefficient *Cl*, drag coefficient *Cd* with those in Question 9:**

Table 10. A comparison of the lift coefficient Cl, drag coefficient Cd of the two cases

	Q9	*Q13*
Cl	0.6091	0.7772
Cd	0.0177	0.0207

3) Compare the pressure coefficient *Cp* distribution and y_1^+ distribution along the airfoil:

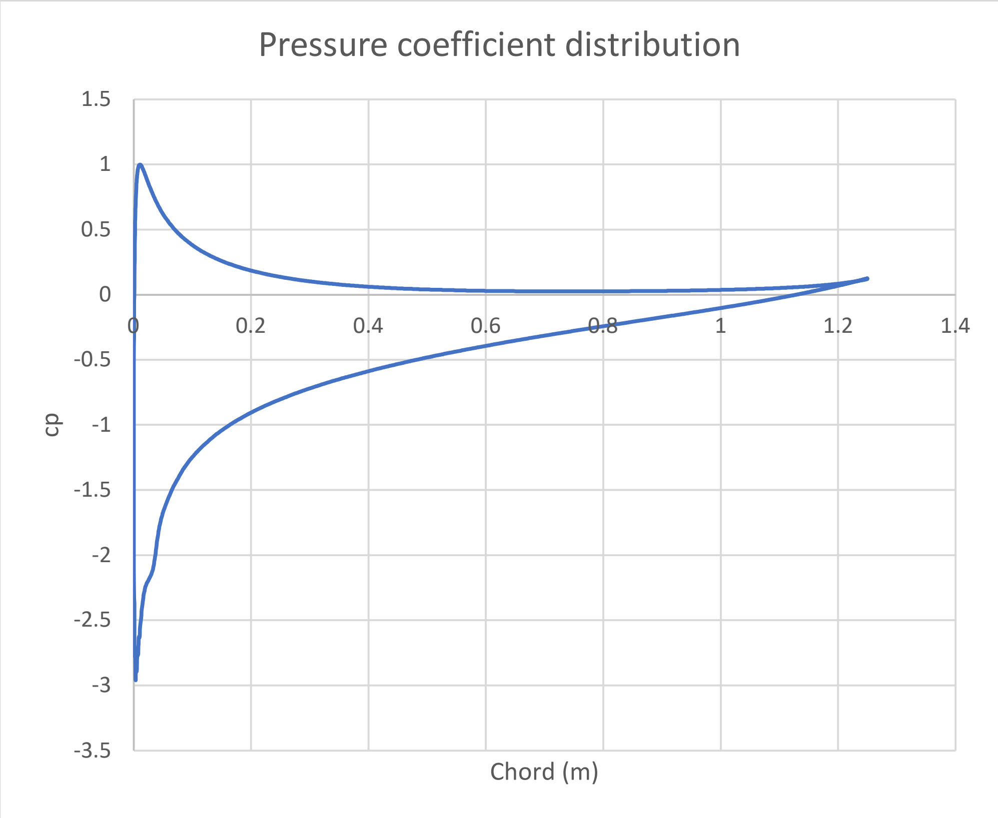


Figure 7. Pressure coefficient distribution along the airfoil for the new chord

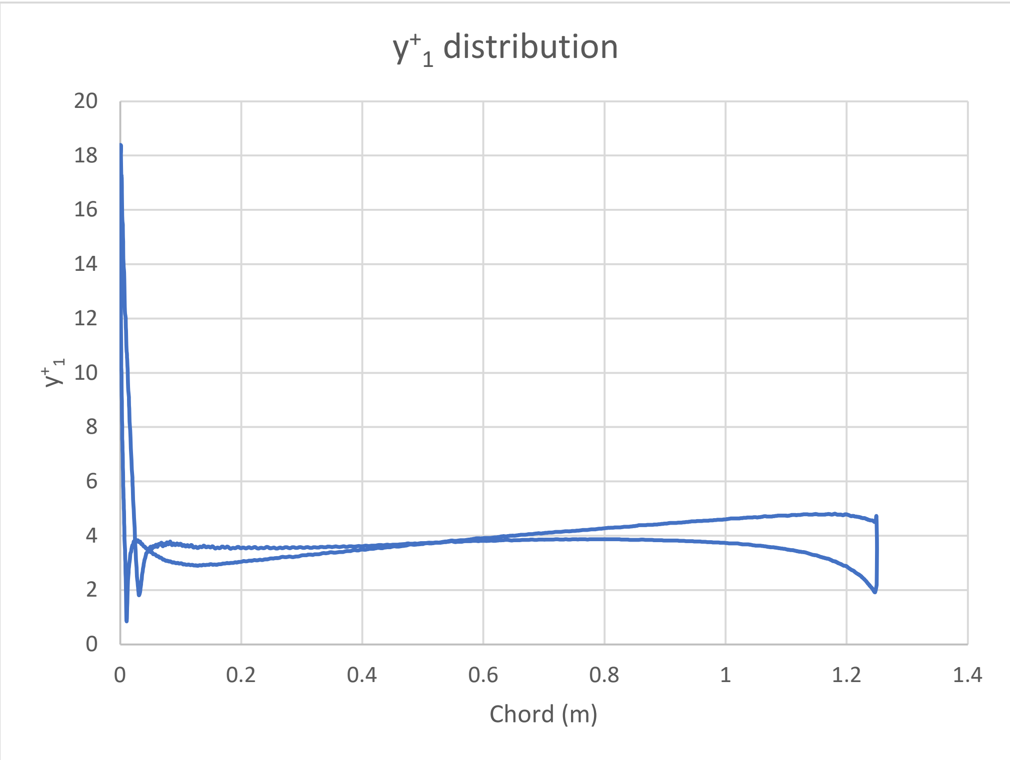


Figure 8. y+ distribution along the airfoil for the new chord

4) **Comment on the output of the comparison between the two cases:**

   Looking at the differences in the lift and drag coefficients, it is clear they present higher values for the scaled mesh. This makes sense because they have bigger areas; and lift and drag forces are proportional to these areas.

Q14

1) **Estimate the maximum boundary thickness at zero-degree angle of attack:**

   Using equation (1): Boundary layer thickness = 0.018847m

2) **Estimate the number of grid points within the boundary layer at the trailing edge and the y_1^+ at the trailing edge:**

   Using equation (4): $y_1^+ = 206.1$

   Using equation (5): *grid points = 5.16 ~* ***5***

3) **Discuss the resolution for this new case with the greater Reynolds number (i.e. ten times your allocated Reynolds number):**

   As the Reynolds number is increase, to keep the same velocity and viscosity (because it is the same fluid) it means the airfoil has to be scaled up 10 times bigger (in chord length). Therefore, the resolution is increased as the size of the cells is also increased.

4) **Assuming you would like to resolve the viscous sublayer and therefore to refine the mesh, give an approximate estimation of the total cell number for this new 2D case, supported by evidence.**

   We need the resolution to be $y_1^+ = 1$ to be able to resolve the viscous sublayer. As the new BL is 0.018m high, we need a first cell height of 8.85e-6 , which means that 1063 cells are needed inside the BL to resolve it. Using manual mesh refinement, a new mesh was achieved with all of these requirements met, composed of 1427714 cells.

Q15

1) **Plot and compare the pressure (in Pa) distribution on the airfoil surface between using the Gauge Pressure 0 Pa and (*Re/30,000*) Pa.**

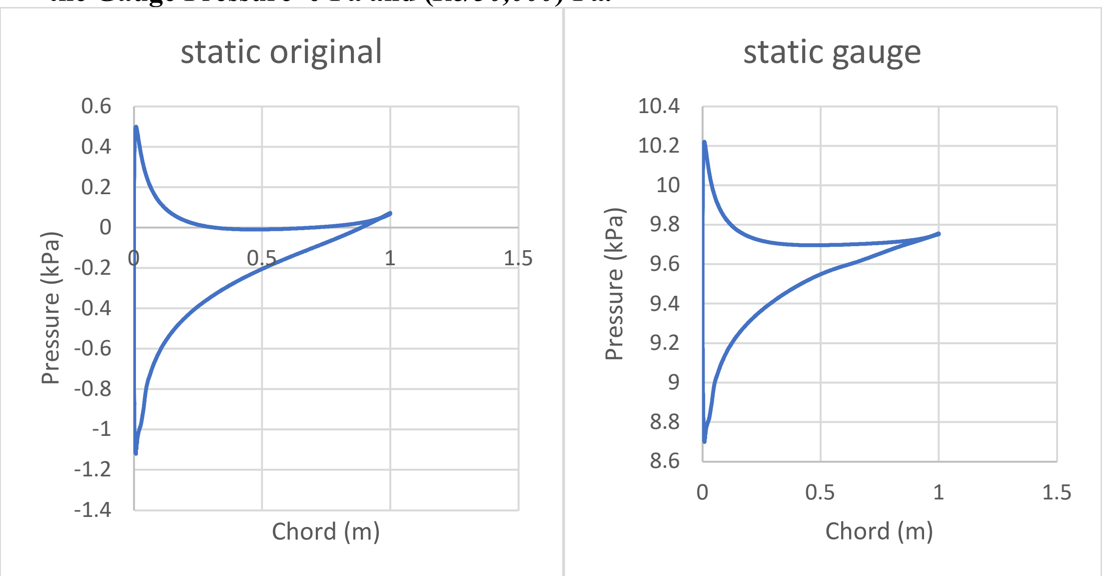
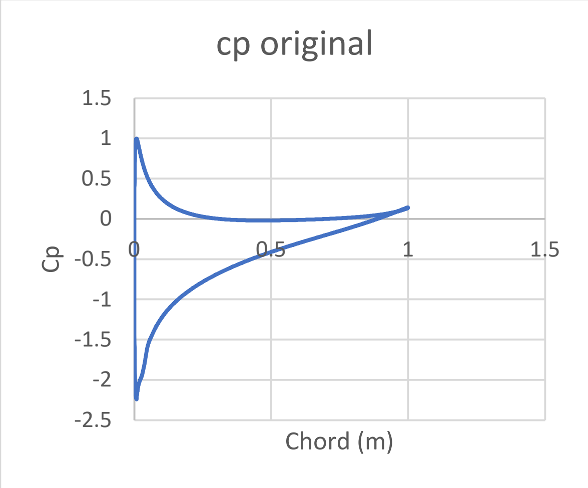


Figure 9. Static pressure distribution comparison for the original and the modified gauge outlet cases

2) **Plot and compare the pressure coefficient distribution on the airfoil surface between using the Gauge Pressure 0 Pa and (*Re/30,000*) Pa.**



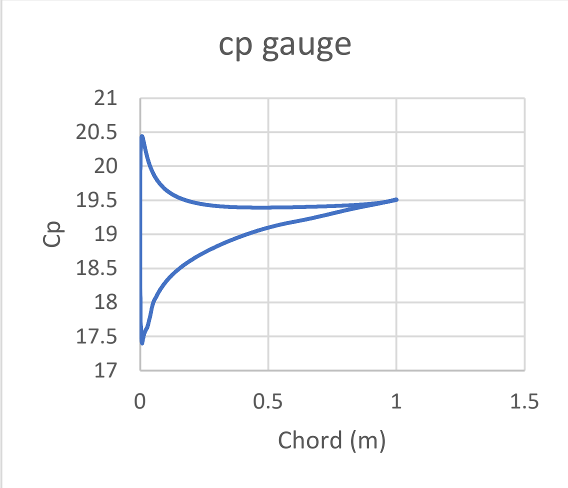


Figure 10. Pressure coefficient distribution comparison for the original and modified gauge outlet

3) **Show and compare the lift and drag coefficients between using the Gauge Pressure 0 Pa and (*Re/30,000*) Pa.**

Table 11. Comparison of Cd and Cl between the modified gauge pressure setting and the original

	Original	*Gauge*
Cl	0.6091	0.5271
Cd	0.0177	0.0178

4) **Brief your findings in Question 15.**
The most significant finding we can extract from the graphs is that the shape of the pressure distributions (both static and coefficient) are kept the same between them two cases. The only thing that changes is the values for the y axis, which in the static pressure changes the same amount as the gauge pressure value selected for the pressure outlet. E.g. if the pressure at point 0 for the original case is p_0, the pressure in the gauge case is $p_1 = p_0 + p_{gauge}$, being p_{gauge} Re/30000.
Something similar happens with the pressure coefficient, where the values of the gauge case have an added constant value that comes from the pressure outlet settings.
Gauge pressure case lift coefficient values are seen to be higher, but not the drag coefficient.

Q16

SIMPLE is based on the pressure correction method. This method solves momentum equations by a guessed pressure field to obtain a velocity field, so iterations are based on achieving new values by small corrections. Since the guessed pressure field in the modified gauge case is changed to a certain value (not 0 as in the normal case), it is also changed when the solver is executed, giving different values.

We are also taking the pressure from the inlet as a reference value, and as

$$cp = \frac{p - p_{\infty}}{\frac{1}{2} * \rho * U_{\infty}^{2}}$$

As the measured pressure in the airfoil is affected by the outlet gauge pressure, which is much bigger than the inlet reference value, we have massive values for the pressure coefficient to the order of 20, but as explained in Q15, if we were to subtract the gauge pressure selected for the outlet to the pressure coefficient values, we would get to approximately the same pressure coefficient values as in the original case with no gauge outlet pressure.

REFERENCES

Xie, D. Z.-T. (2021). *FEEG6005 Aplications of CFD (Part 2).* Southampton: University of Southampton.