ANSYS-FLUENT Tutorial: High Reynolds Number Flow over NACA0012 Airfoil

This document describes how to set up a CFD (Computational Fluid Dynamics) simulation of airflow around a NACA0012 airfoil using ANSYS-FLUENT.
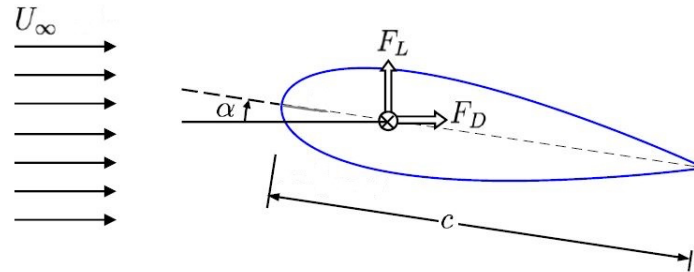
Problem Specifications

Figure 1. Representation of the flow around an airfoil with an angle of attack α and chord length of c. The freestream velocity is directly horizontally with a magnitude U_∞. The resultant drag and lift forces are given by F_D and F_L. Importantly, the direction of the drag force is parallel to the freestream velocity vector; while the lift force is perpendicular to the freestream velocity vector.

Consider the high Reynolds turbulent flow around an airfoil at the angle of attack α, as illustrated in Figure 1. The following are the characteristics of the simulation:

- Airfoil type: NACA0012

- Airfoil chord length: c = 4 in = 0.1016 m

- Angle of attack: $\alpha = 10°$

- Velocity (x-component): U_∞ = 21.566 m/s

- Fluid: Air

- Fluid density: ρ = 1.225 kg/m^3

- Fluid dynamic viscosity: μ = 1.7895e-5 kg/m·s

- Reynolds number, based on chord length: $Re_c = \rho U_\infty c / \mu$ = 150,000

Pre-analysis and Start-up

Before opening FLUENT, the computational domain and boundary conditions are explained here.

Solution Domain:

For an external flow problem like this, the fluid domain must be determined. A *C-type* fluid domain will be used. To minimize the boundary effects on the flow field around the airfoil, the outer boundary will be positioned at a far distance from the airfoil, about 15 times larger than the airfoil size (chord length), as shown in Figure 2.
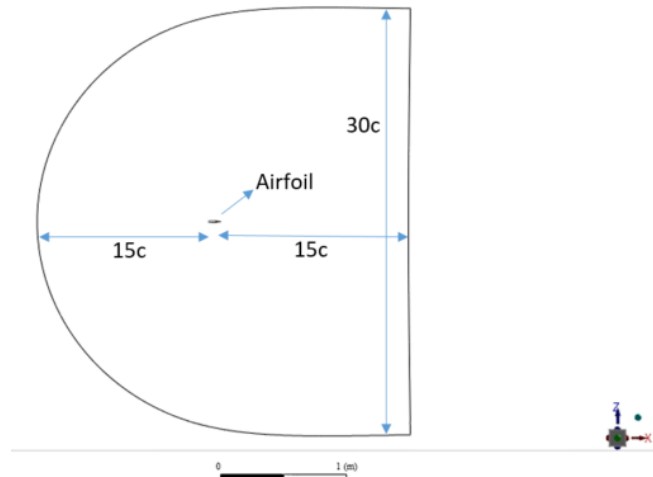
Figure 2. Representation of the Numerical Domain.

Boundary Conditions:

An inlet velocity of 21.566 m/s in the x-direction (streamwise direction) is applied at the inlet. Next, the right edge of the domain will be defined as an outlet with pressure outlet boundary condition with a gauge pressure of zero Pa. Finally, the airfoil will be defined with a wall no-slip boundary condition. These boundary conditions are drawn in Figure 3.
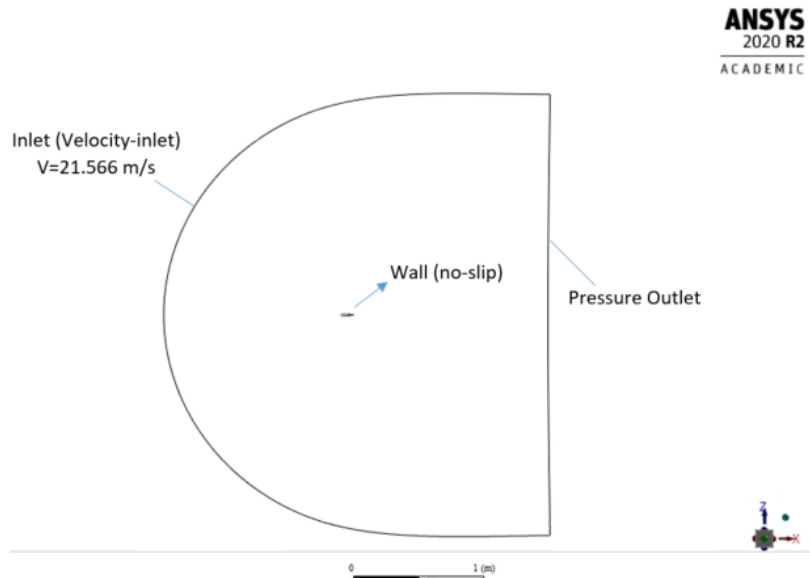

Figure 3. Representation of the Boundary Conditions for the Domain and the Square.

Mesh Independence study:

- One of the essential steps in any CFD simulation is called the "Mesh Independence" study. This step is performed to ensure that the results, such as drag/lift coefficients, are independent of the mesh size. It means that the mesh resolution is fine enough that using a finer resolution (smaller mesh size) does not change the results very much. The mesh case at which results are not changing much after that is called the mesh independent.

- The objective here is to plot drag and lift coefficients as a function of the number of mesh elements. Then, according to the observed pattern, choose the mesh resolution at which results are mesh independent. After finding the mesh independent case, it will be used for any further study.

- In this study, four different mesh resolutions, corresponding to n=50, 100, 400, and 800 mesh nodes on the airfoil, are utilized. Increasing the mesh nodes on the airfoil results in a finer mesh resolution over the entire domain. The final results for the lift and drag coefficients at the angle of attack $\alpha = 10°$ are given in the table below. During lab, you will perform the CFD simulation for the case of **n=50 mesh nodes** on the airfoil to make sure that you can generate the correct results.

File name	n (# of mesh nodes on airfoil)	# of mesh elements in entire domain	C_D at $\alpha = 10°$	C_L at $\alpha = 10°$
NACA0012_n50	50	52,001	0.0354	0.860
NACA0012_n100	100	59,451	0.0300	0.911
NACA0012_n400	400	104,151	0.0294	0.919
NACA0012_n800	800	163,751	0.0294	0.9209

- After obtaining the drag and lift coefficient results for all four mesh resolutions, it is instructive to plot the results. The figure below shows the lift and drag coefficients as a function of total number of mesh cells in the entire domain. You can see that the coarsest mesh overestimates the drag and underestimates the lift considerably. You can also observe that the drag and lift tend to reach a plateau after a critical number of mesh nodes has been reached. For this problem, the lift and drag coefficient appear to be independent of the number of mesh nodes in the domain for the case where n=400.

- You will use the mesh independent case n=400 (i.e., "NACA0012_n400") to complete the CFD simulations required for the assignment as written in the Handout. However, for purposes of this tutorial, you will use the case n=50 (i.e., "NACA0012_n50"), since the simulation runs faster with fewer nodes.
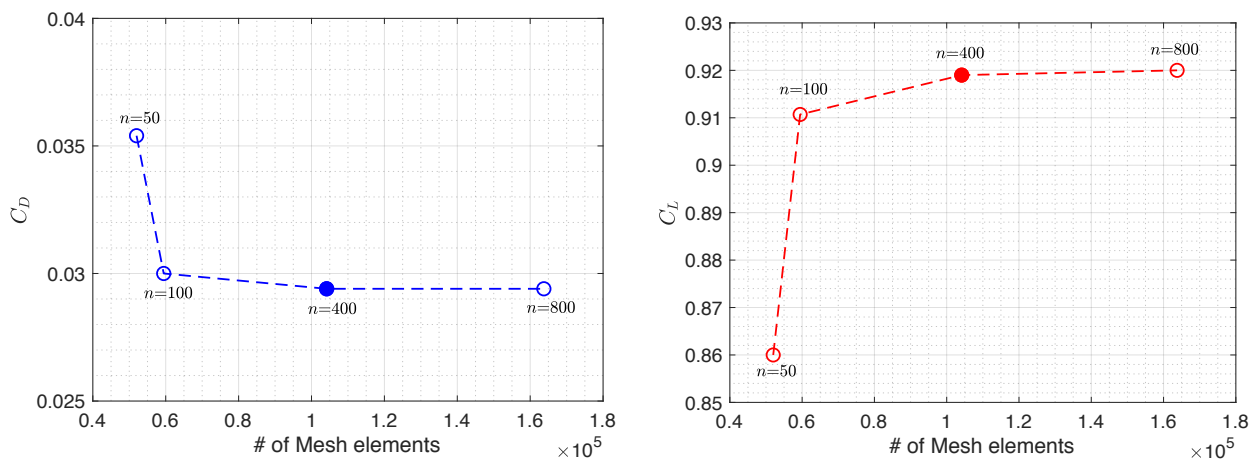

Figure. Results from the mesh independece study. The solid marker indicates the mesh size that should be used for this problem in order to obtain the most accurate results.

1. OPENING ANSYS WORKBENCH AND SELECTING FLUENT SOLVER

1.1. For Windows operation systems, search for "Workbench" in the **search bar** and click on **Workbench 2020 R2**.

    OR

1.2. For UNIX machines, type "Workbench" into the **terminal window**.

1.3. Expand the **"Component Systems"** tab and then drag (or double click) "**Fluent**" into the **Project Schematic** window as shown in Figure 4. Change the project name at the bottom of **menu A** from "Fluent" to "airfoil" by double-clicking on that.
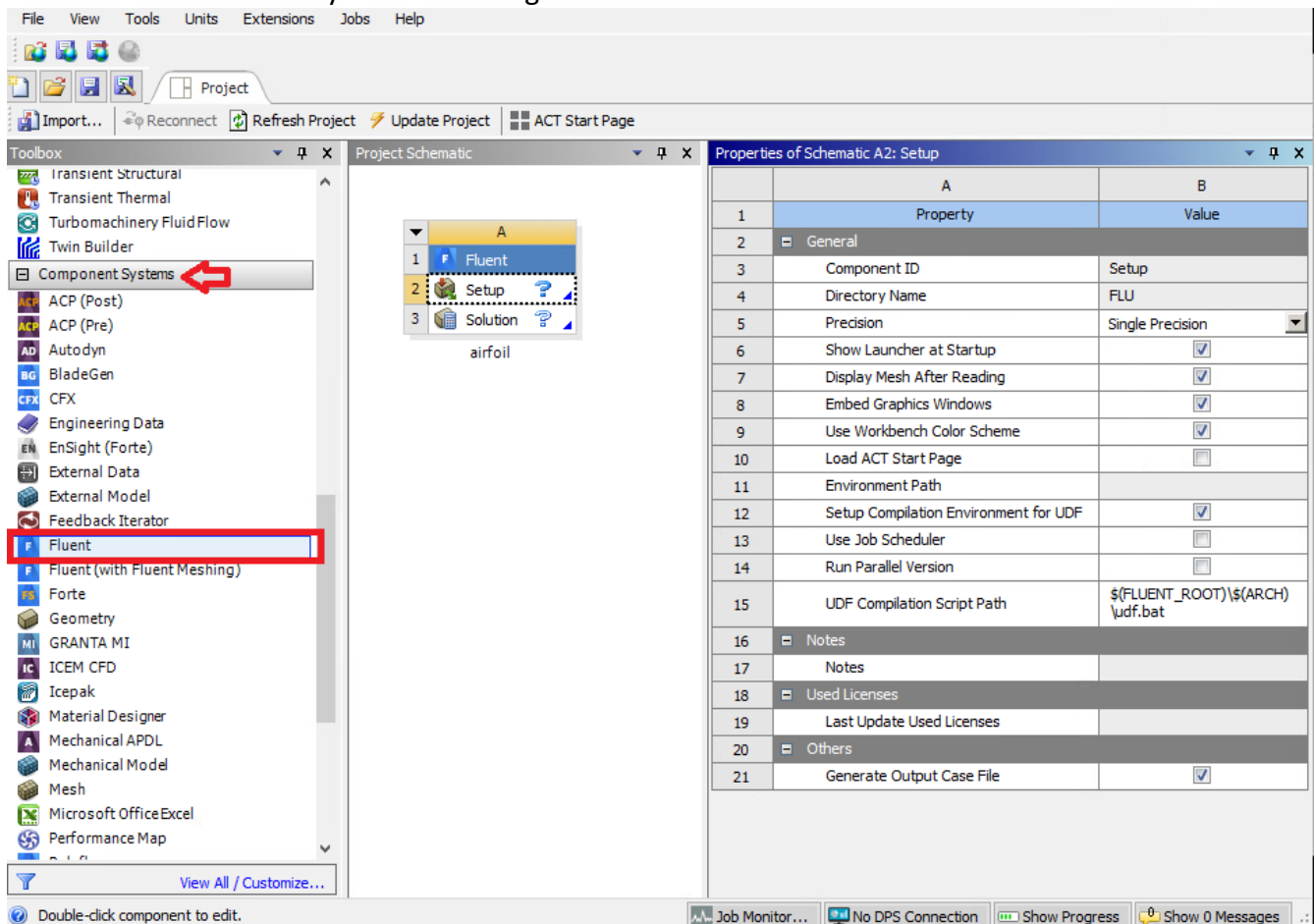

Figure 4. Main ANSYS Menu.

1.4. Under the **File** menu, select **Save As** and save the project as "airfoil" in a folder directory of your choice.

*NOTE – when a file is saved in ANSYS, a file and a folder are created. For instance, in the folder where the project was just saved, a file called "airfoil.wbpj" and a folder called "airfoil_files" should appear. In order to reopen the ANSYS files in the future, both the ".wbpj" file and the folder must be present.

2. SETTING UP THE PHYSICS

2.1. From the main **Workbench** program, in the **Project Schematic** window, double-click **Setup.**

2.2. In the **Fluent Launcher** window, select **Double Precision** and **Parallel** solver with **2 processors**. Verify that the other selections are set as shown in Figure 5. Then click **OK**.
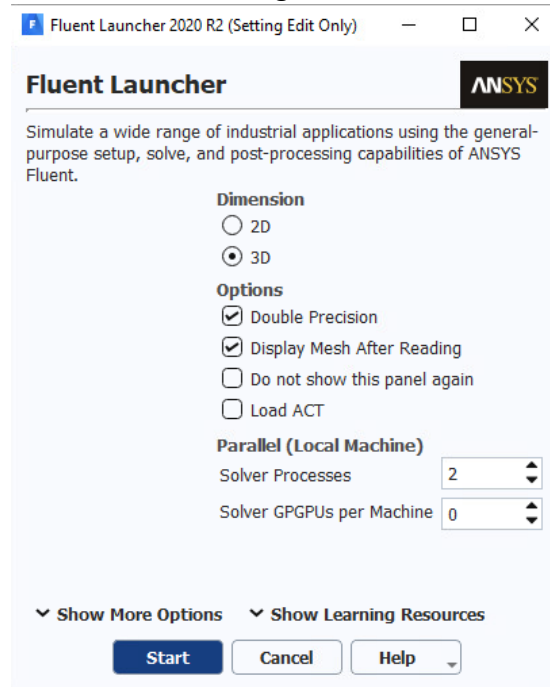

Figure 5. FLUENT Launcher window.

2.3. From the top left, select **File> Import> Case** and select the provided Fluent case file for n=50 available on Canvas ("NACA0012_n50.cas"). This case file includes the geometry and mesh needed for the CFD simulation. Note, you need to go to the directory where you saved these files and import the Case file from there.

2.4. On the top menu bar, select the **Domain** tab. In the **Mesh** section, click **Info > Size** and verify that the output in the **Console** pane (bottom right) states that there are 52001 cells if NACA0012_n50 is imported or 104151 if NACA0012_n400 is imported, as shown in Figure 6.

2.5. Most of the CFD setups are prepared in advance to save your time. You just need to check the settings are as indicated.
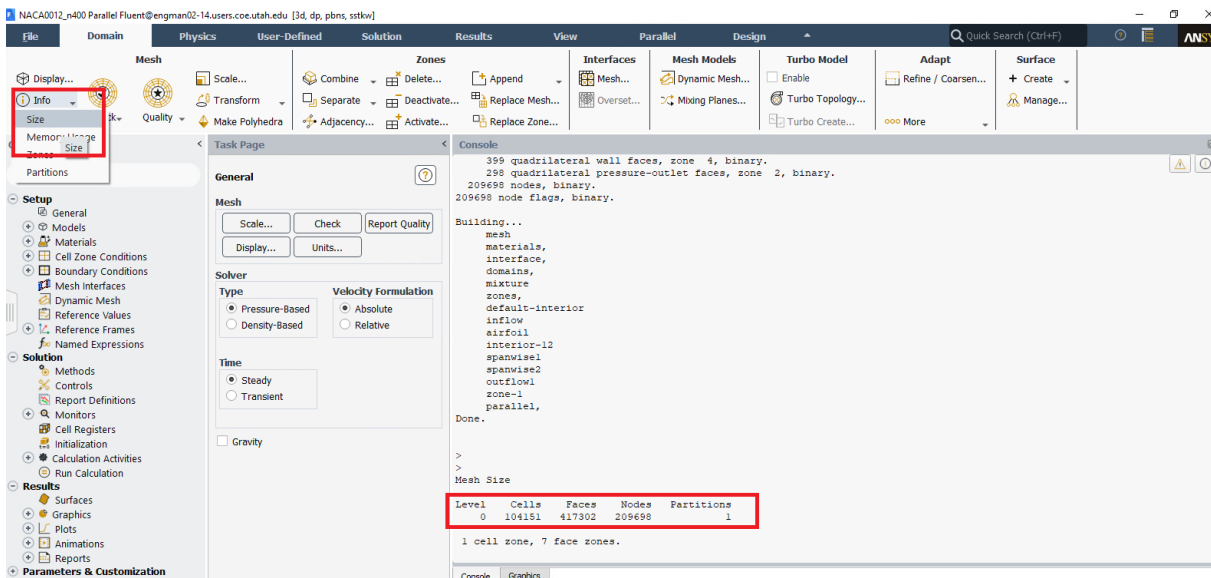
Figure 6. Fluent window showing "Mesh Size" information in the Console pane, for NACA0012_n400 mesh case. Note, this value should be 52001 for the NACA0012_n50 mesh case.

2.6. SETUP MODEL

2.6.1. In the **Outline View** pane (upper left side of window), expand the **Setup** heading, and double-click on **Models,** then double-click on **Viscous** (SST k-omega), as shown in Figure 7.
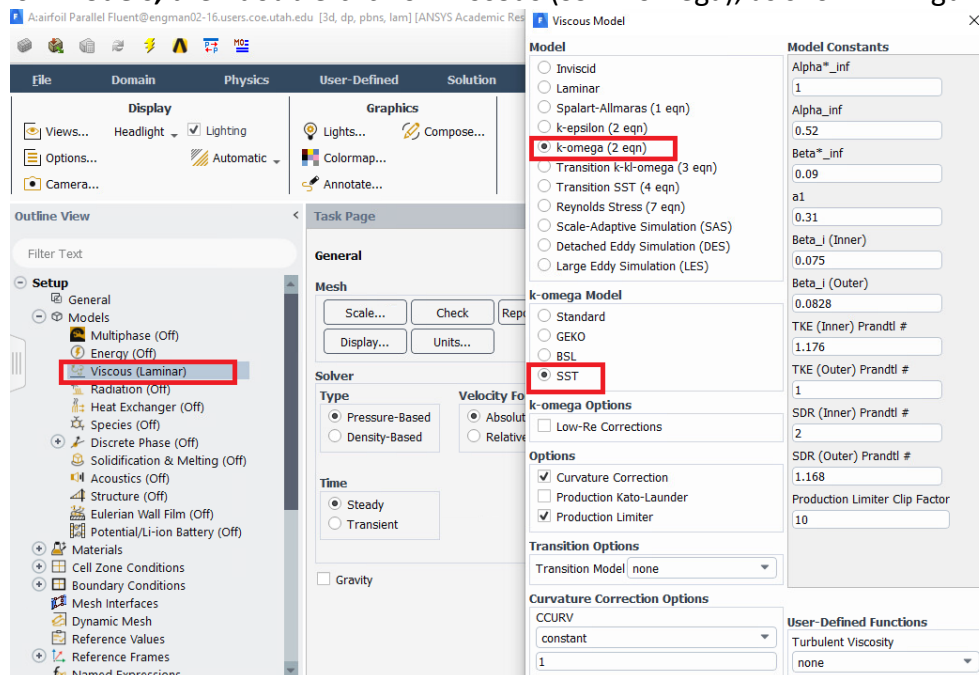

Figure 7. Setting up the fluid-flow models.

2.6.2. Change the fluid-flow model from **Viscous (Laminar)** to **k-omega (2 eqn)**.

2.6.3. Set the k-omega model to **SST** and click **OK.**

2.6.4. In the **Outline View** pane, double-click on **Models** to hide those options.

2.7. SETUP MATERIAL PROPERTIES

2.7.1. In the **Outline View** pane, expand the **Setup** heading, and double-click on **Materials > Fluid > Air**.

2.7.2. Make sure the fluid properties are as following, and then close the window:

- **Name** = "*air*"

- **Material Type** = **Fluid**

- **Density** = **1.225 kg/m³**

- **Viscosity** = **1.7894e-5 kg/m·s**

NOTE – Considering the inlet flow speed of 21.566 m/s and airfoil chord length of 0.1016 m, the Reynolds number will be Re=150,000.

2.7.3. In the **Outline View** pane, double-click on **Materials** to hide those options.

2.8. SET CELL ZONE CONDITIONS

2.8.1. In the **Outline View** pane, under **Setup**, double-click on **Cell Zone Conditions**.

2.8.2. In the **Task Page** pane, double-click on the **zone-1**. In the **Material Name** box, make sure "air" is selected in Material Name, as shown in Figure 8, then close.

2.8.3. In the **Outline View** pane, double-click on **Cell Zone Conditions** to hide those options.
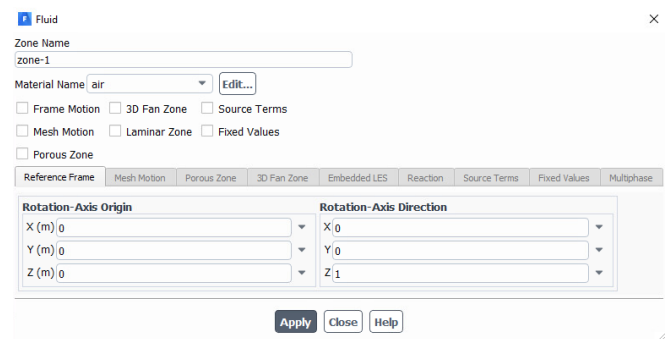
Figure 8. Setting zone material.

2.9. SET BOUNDARY CONDITIONS

2.9.1. In the **Outline View** pane, under **Setup**, double-click on **Boundary Conditions**.

2.9.2. In the **Task Page** pane, click on **inflow**. Make sure the <u>Type</u> is set to "velocity-inlet". Click on the [Edit...] button. In the **Velocity Inlet** popup window, make sure the **Velocity Magnitude** set to <u>21.566 m/s</u>, as shown in Figure 9. Click **Apply** and Close.
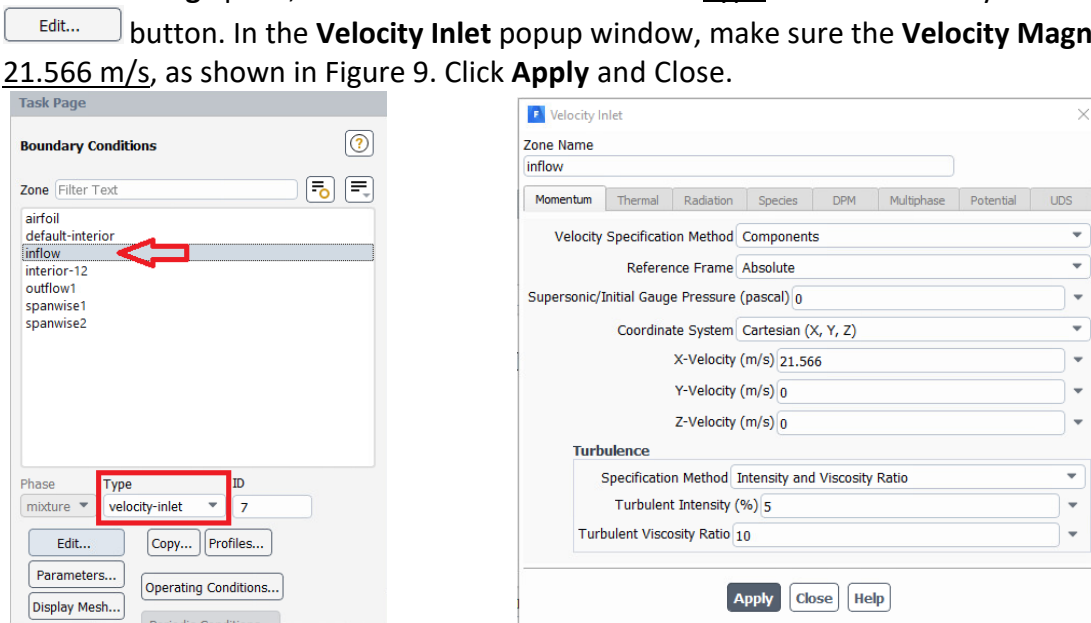

Figure 9. Setting the inlet boundary condition.

2.9.3. In the **Task Page** pane, click on **outflow1**. Make sure the <u>Type</u> is set to "pressure-outlet". Click on the [Edit...] button. In the **Pressure Outlet** popup window, set the **Gauge Pressure** to <u>0 Pa</u> (i.e. the pressure will be kept at ambient pressure at the outlet), as shown in Figure 10. Click **Apply** and Close.
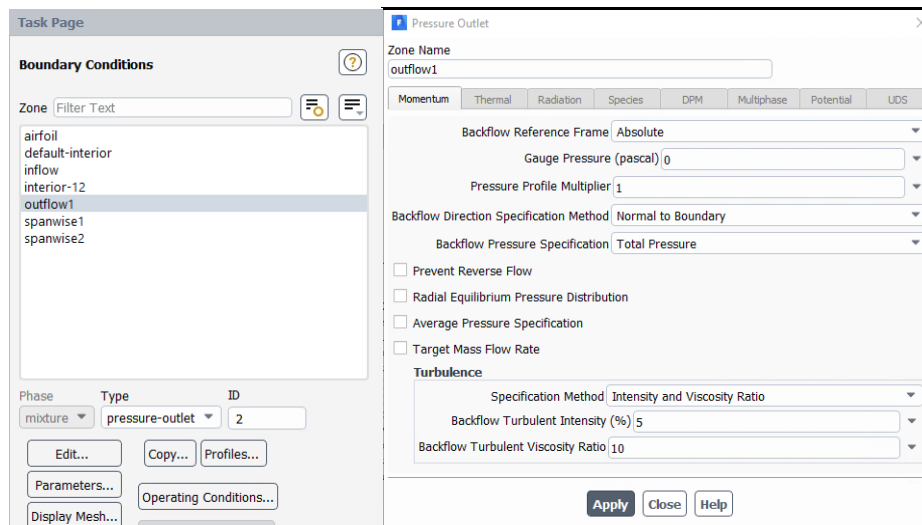

Figure 10. Setting the outlet boundary condition.

2.9.4. In the **Task Page** pane, click on **airfoil**. Make sure the <u>Type</u> is set to "wall". Click on the [Edit...] button. In the **Wall** popup window, the <u>Wall Motion</u> option should be set to "Stationary Wall" and the <u>Shear Condition</u> option to "No Slip", as shown in Figure 11. Click **Apply** and Close.
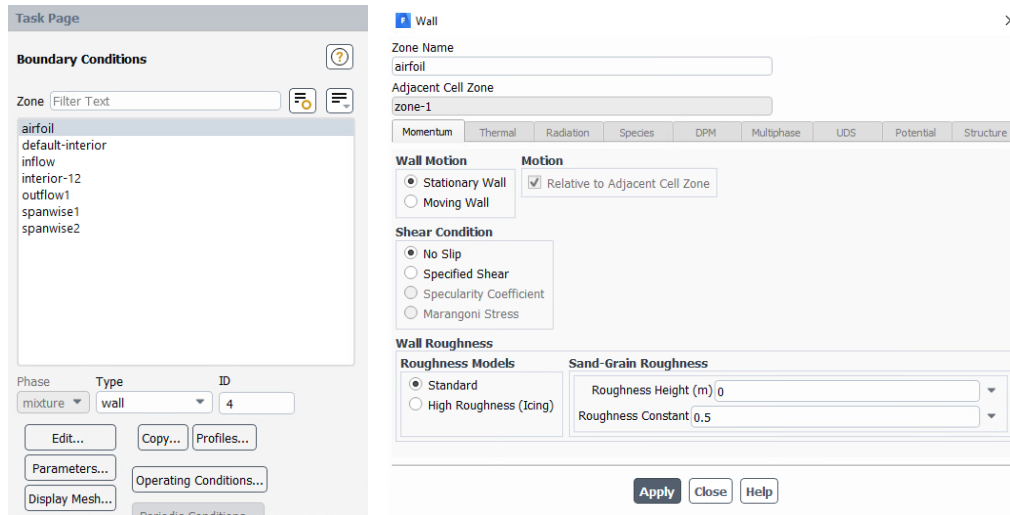
Figure 11. Setting up the wall boundary condition.

2.9.5. Verify that the **spanwise1** and **spanwise2** are set to "symmetry" boundary condition Type.

2.9.6. In the **Outline View** pane, double-click on **Boundary Conditions** to hide those options.

2.10.   SET REFERENCE VALUES

2.10.1.   In the **Outline View** pane, under **Setup**, double-click on **Reference Values**.

2.10.2.   In the **Task Page pane**, set Compute from to **inflow**, as shown in Figure 12.

2.10.3.   Verify the rest of the parameters (e.g., density, velocity, viscosity) and update them if necessary. The values should be as in Figure 12.

NOTE – The reference values are the parameters that FLUENT uses when calculating the drag/lift coefficients. Note, the area is the chord length, c=0.1016 m, multiplied by a unit depth of 1 m in the spanwise direction.
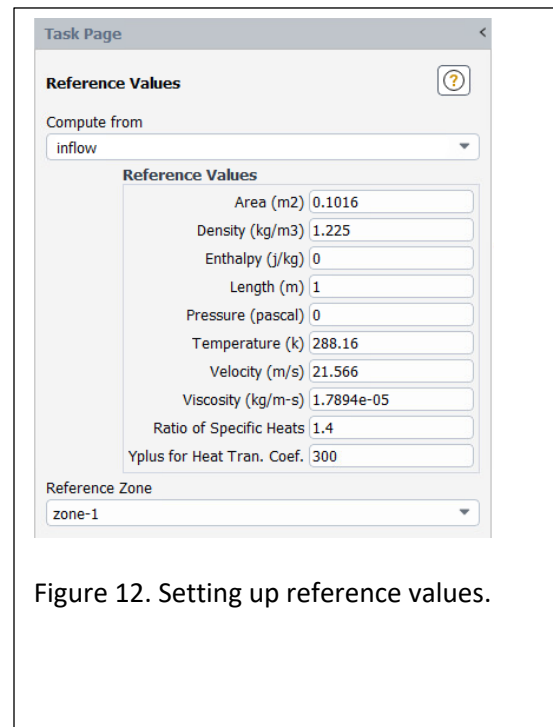

Figure 12. Setting up reference values.

2.11. On the top menu bar, click **File**. Select **Save Project**.

3. SETTING UP THE SOLUTION PARAMETERS

NOTE – Turbulent problems are transient (unsteady) by nature. However, to decrease the simulation's computational time, a steady-state simulation will be run in this exercise. The steady-state solver computes the *time-average* flow field but does account for the turbulence effect on the mean flow (through a parameterized model). Because of this, we will not be able to obtain details about vortex shedding, for example. However, we will be able to capture the dominant flow patterns, such as overall size of the wake and separation point, as well as get a reasonable estimate of the drag and lift coefficients.

3.1. In the **Outline View** pane, expand the **Solution** heading and double-click on **Methods**. Set the parameters in the **Task Page** pane to the values displayed in Figure 13 (left image), if it's different.

3.2. In the **Outline View** pane, under **Solution**, expand the **Monitors** subheading and double-click on **Residual**. In the **Residual Monitors** popup window, set the values as displayed in Figure 13 (right image). Click OK.
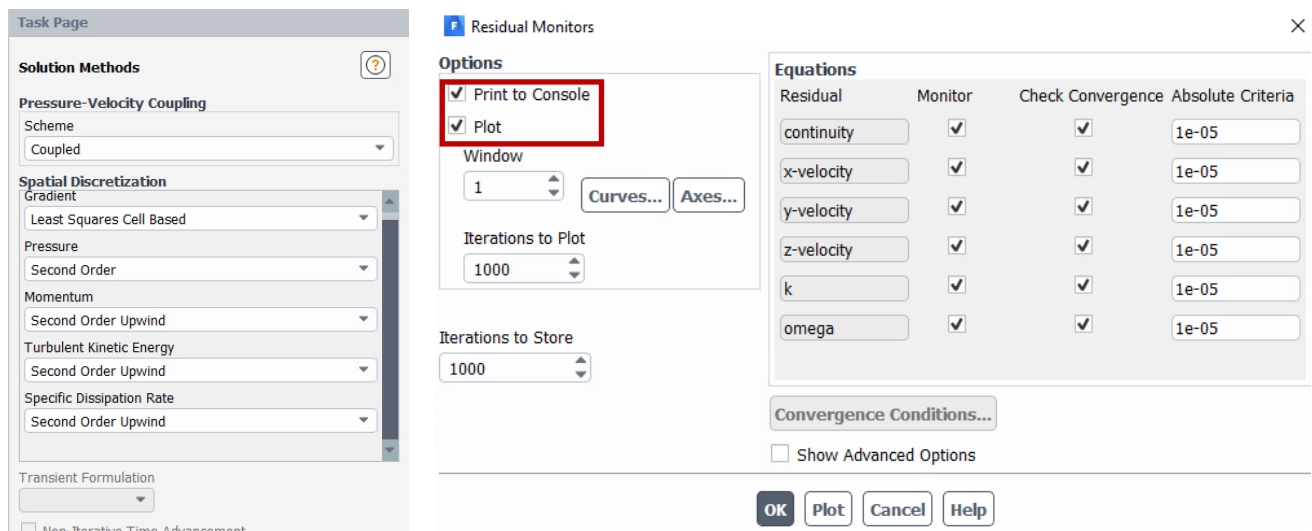

Figure 13. (left) Solver settings. (right) Residual Monitors settings.

3.3. SET INITIALIZATION PARAMETERS

3.3.1. In the **Outline View** pane, under **Solution**, double- click on **Initialization**.

3.3.2. Set the **Initialization Methods** to "Standard Initialization", as shown in Figure 14.

3.3.3. Set **Compute from** to "inflow". This will set the "X Velocity" to the inlet velocity value of 21.566 m/s.

3.3.4. Ensure that the values for the "Gauge Pressure", "Y Velocity", and Z Velocity" are set to 0, as shown in Figure 14.

3.3.5. Click the Initialize button. Note, This will set the value of the velocity everywhere in the domain to the same value of 21.566 m/s.

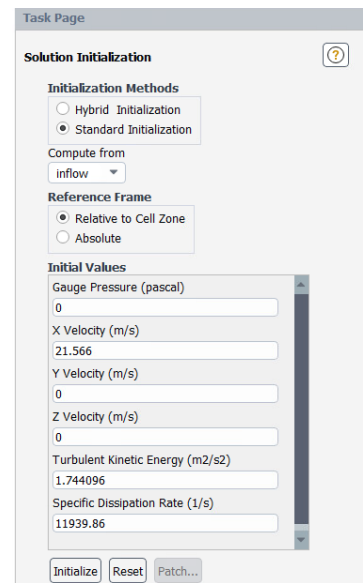Figure 14. Setting up the initialization.

3.4. SET REPORT DEFINITIONS

NOTE – since we are interested in calculating the drag and lift coefficients on the airfoil, we will define output files for these parameters, so that FLUENT calculates the parameters at any iteration, plot the value over iterations, and save the file as an output.

3.4.1. In the **Outline View** pane, under **Solution**, double-click on **Report Definitions**.

3.4.2. Under the Report Definitions box, click **New**. Then select **Force Report > Drag**, as shown in Figure 15 (left image). This will open a new popup window called Drag Report Definition.
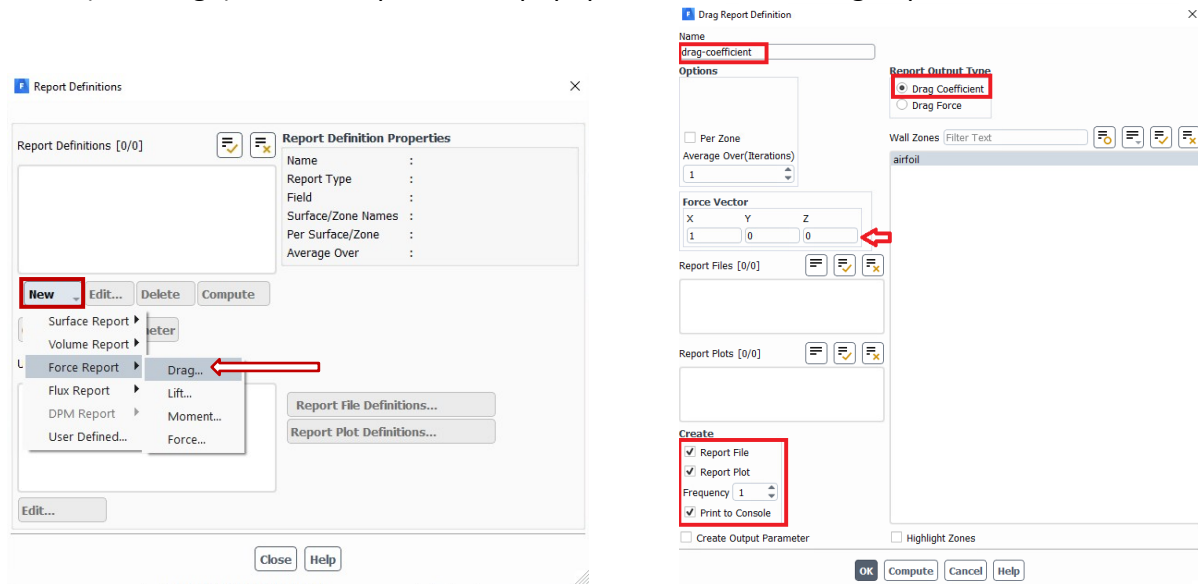

Figure 15. Creating the definition for the drag coefficient report.

3.4.3. In the **Drag Report Definition** popup window, ensure the following settings are selected, as shown in Figure 15 (right image):

- Under the <u>Wall Zones</u> box, click "airfoil".

- Under the <u>Report Output Type</u> heading, select "Drag Coefficient".

- Under the <u>Create</u> box, check "Report File", "Report Plot", and "Print to Console".

- In the <u>Name</u> box, type "drag-coefficient".

- Under the <u>Force Vector</u> box, verify that "X" is set to 1, "Y" and "Z" are set to 0. This means the drag coefficient is calculated based on the x-component of the force, which is correct in our case since the freestream velocity is in the x-direction.

- Click ⬚.

3.4.4. Repeat the previous steps to define lift coefficient. Select **NEW> Force Report > Lift**, as shown in Figure 16 (left image). This will open a new popup window called **Lift Report Definition**.
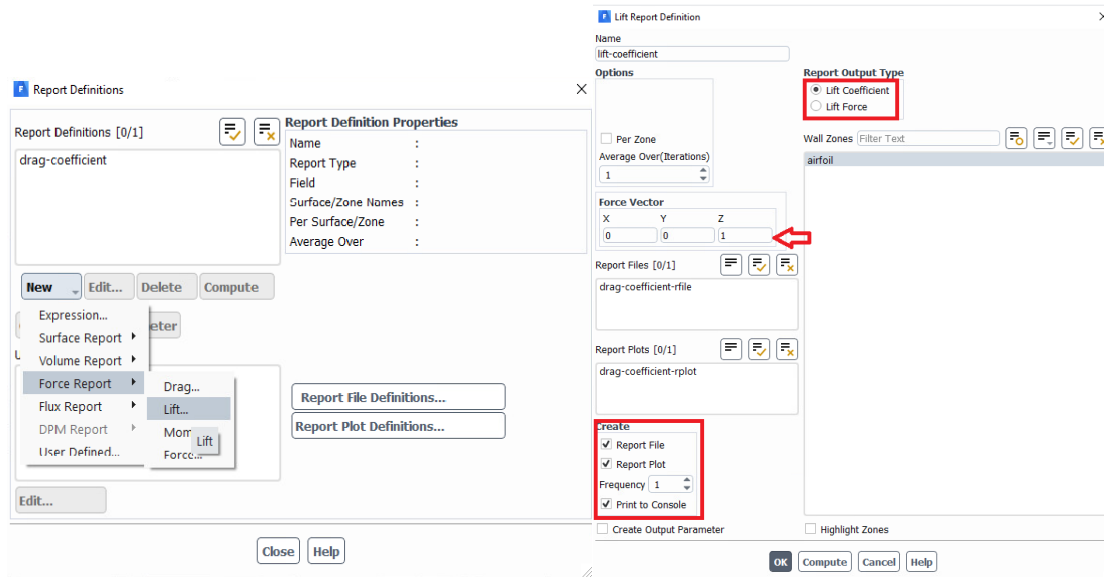
Figure 16. Creating the definition for the lift coefficient report.

3.4.5. In the **Lift Report Definition** popup window, ensure the following settings are selected, as shown in Figure 16 (right image):

- Under the Wall Zones box, click "airfoil".

- Under the Report Output Type heading, select "Lift Coefficient".

- Under the Create box, check "Report File", "Report Plot", and "Print to Console".

- In the Name box, type "lift-coefficient".

- Under the Force Vector box, verify that "X" is set to 0, "Y" is set to 0, and "Z" is set to 1. This means the lift coefficient is calculated based on the z-component of the force.

- Click ⊞.

3.4.6. Close the **Report Definitions** popup window by clicking Close.

3.5. SET MONITORS

3.5.1. **NOTE** – while you wait for the simulation to run, it is instructive to monitor the behavior of the residuals, drag, and lift coefficient in real time. These will indicate whether the solution is tending toward convergence or diverging.

3.5.2. In the **Outline View** pane, under **Solution**, expand the **Monitors** subheading, and then expand the **Report Plots** sub-subheading. Double-click on "drag-coefficient-rplot". The **Edit Report Plot** popup window will appear, as shown in Figure 17 (left image).
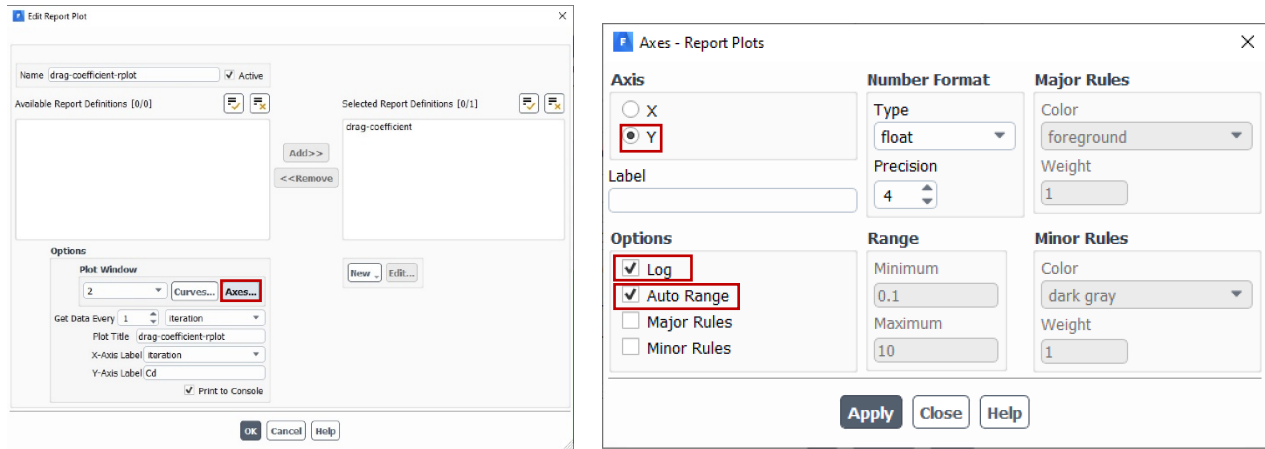
Figure 17. Setting the Axes Properties on the drag coefficient report plot.

3.5.3. In the **Edit Report Plot** popup window, under the Options > Plot Window box, click [Axes...] . A new popup window called **Axes – Report Plots** will appear, as shown in Figure 17 (right image).

3.5.4. In the **Axes – Report Plots** popup window, manually set the Y-axis range, as shown in Figure 17 (right image):

- Under the Axes box, select "Y".

- Under the Options box, select "Log" and select "Auto Range".

- Click [Apply] . Then click [Close].

3.5.5. Back in the **Edit Report Plot** popup window, click [OK].

3.5.6. Repeat the steps 3.5.2 till 3.5.5 for the "lift coefficient-rplot".

3.6. Set angle of attack

3.6.1. Before starting the CFD simulation, we rotate the whole domain including the airfoil, so that airfoil is set at the desired angle of attack. Note, the freestream is fixed and aligned with x-direction, and rotating the domain along "y" direction will change the airfoil angle of attack.

3.6.2. In the top **Domain** tab, under **Mesh**, click on **Transform> Rotate**. In the popup Rotate Mesh window, set "Rotation Angle (deg)" to 10, and "**Rotation Axis**" to X 0, Y 1, Z 0 as shown in Figure 18.

3.6.3. Click [Rotate], then click [Close].

3.6.4. Save your work by going to the top menu bar and clicking **File > Save Project**.
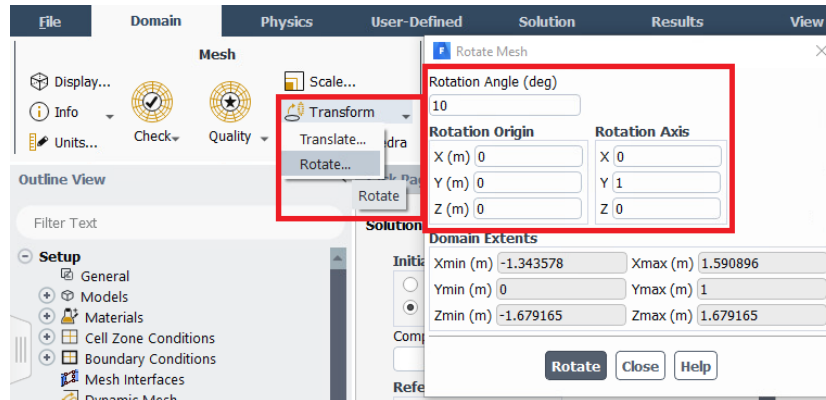
Figure 18. Rotate the domain, and set the angle of attack.

3.7. Display domain

3.7.1. In the **Domain** tab, click on **Display**, select all Surfaces in the Mesh Display window, and click on Display and then click Close as shown in Figure 19 (left image). This will show the computational domain with inlets and outlets in the Graphics Pane as shown in Figure 19 (right image). Make sure the **Graphics** tab in the bottom left corner of the Graphics Pane is selected. To show the view normal to x-z plane, click on the y-axis in the coordinate system triad shown in the lower right corner of the Graphics Pane. Note, arrows are drawn normal to the domain edges.
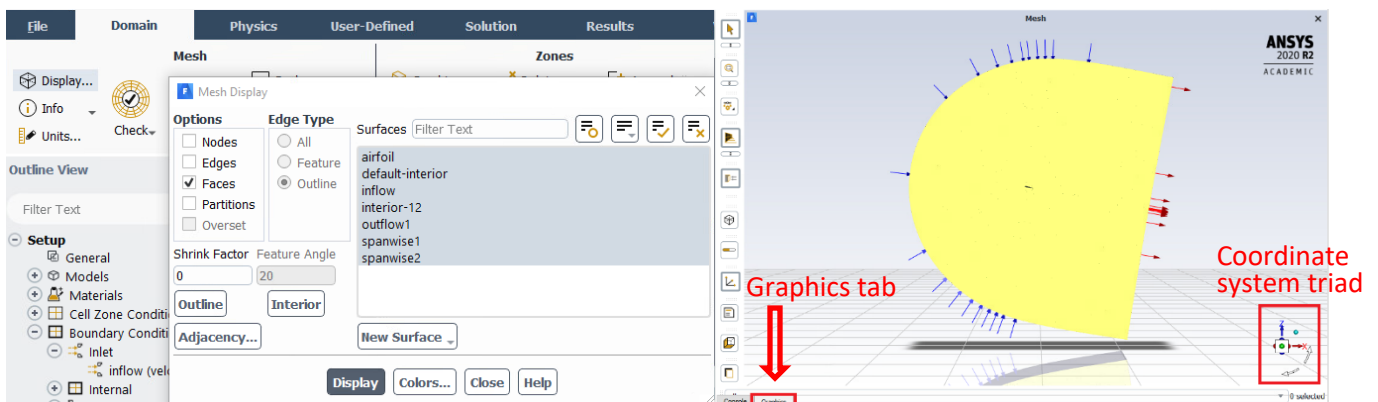

Figure 19. Display fluid domain.

3.8. RUN STEADY-STATE SOLVER

3.8.1. In the **Outline View** pane, under **Solution**, double-click on **Run Calculation**.

3.8.2. In the **Task Page** pane, set the **Number of Iterations** to **500**, as shown in Figure 20.

3.8.3. Click the Calculate button. The blue windows wheel should start spinning.

3.8.4. Toggle between the "Console View" and the "Graphics View", by selecting the **Graphics** tab at the *bottom* of the Graphics Pane. Click individually on the tabs at the *top* of the Graphics Pane labeled "Scaled Residuals", "drag-coefficient-rplot", and "lift-coefficient-rplot". You should see plots similar to the ones shown in Figure 21, though the pattern and values might be different. And, your plots should only show 500 iterations or less (if the solver converges before reaching 500 iterations).
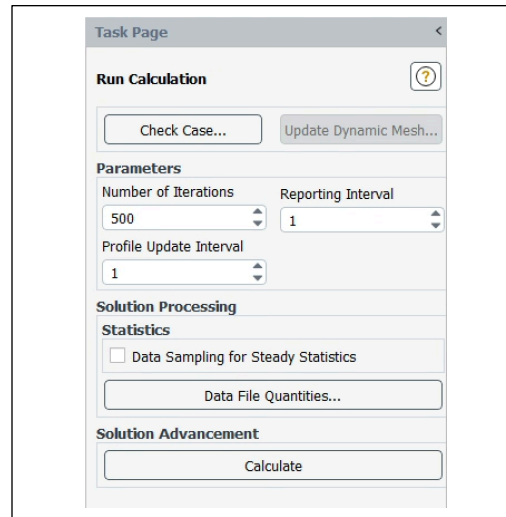
14

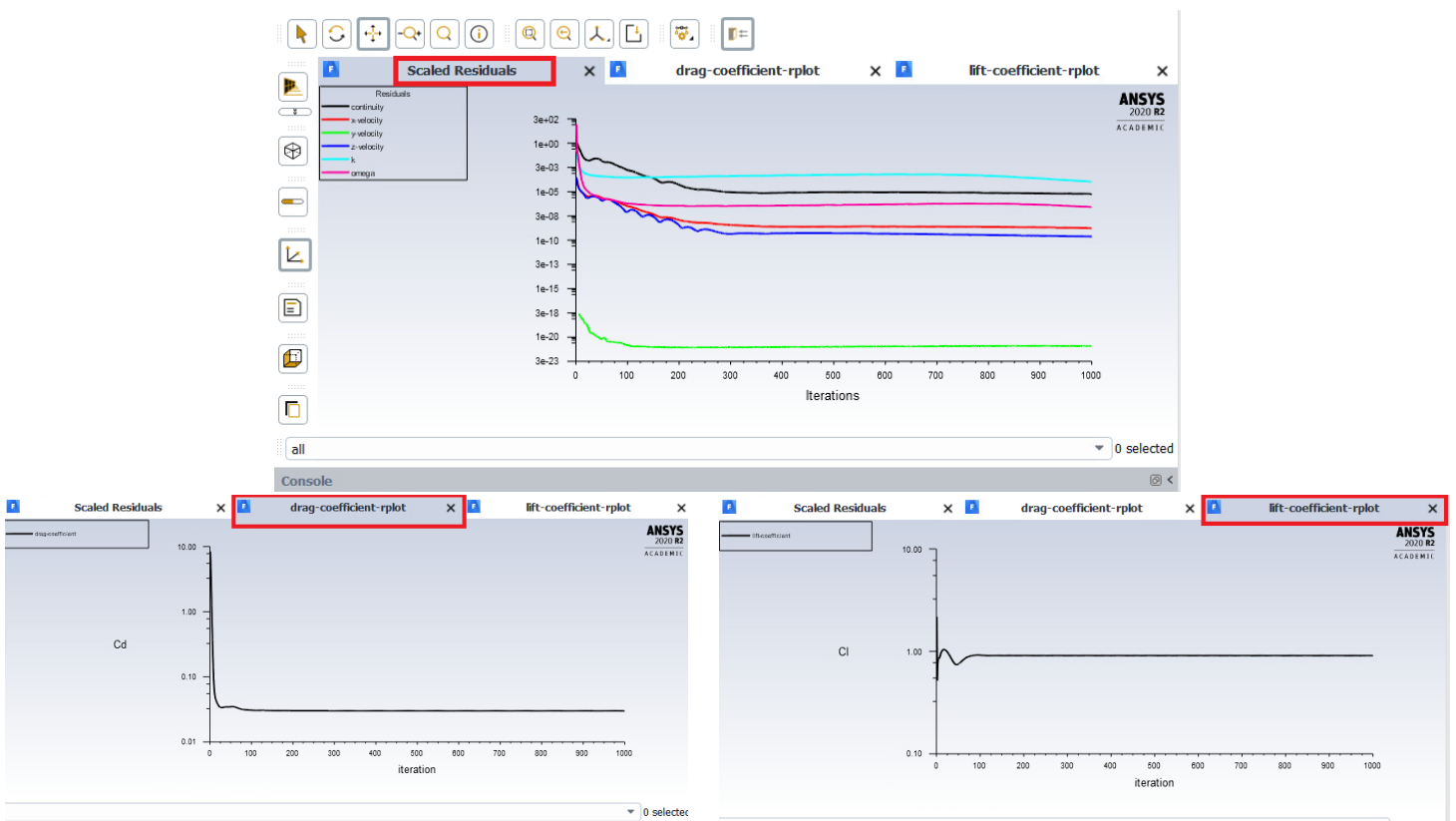Figure 20. Running the steady-state solver.

Figure 21. Plot of the (Top) residuals, (left) drag coefficient, and (right) lift coefficient for 1000 iterations of the steady-state solver.

3.8.5. The solver will require around 10 minutes to complete 500 iterations, depending on the computational speed. Continuity is usually the most challenging condition to satisfy and usually takes the longest to converge. After 500 iterations (or less) of the steady-state solver, all residuals should have reached a value less than 1e-5. A residual value less than 1e-5 is typically desirable.

15

3.8.6.   On the top menu bar, select **File > Save Project**.

3.8.7. In the **Console** view of the **Graphics** pane, observe the value of the drag and lift coefficients after 1000 iterations of the steady-state solver. The drag and lift coefficient plots show the patterns over the iterations. The related output files are saved in ~\airfoil_files\dp0\FLU\Fluent directory, named as drag-coefficient.out and lift-coefficient.out. You could use these files to plot the pattern of drag and lift coefficients versus iteration yourselves or do any other analysis.

4.   RESULTS: OBTAINING THE DRAG AND LIFT FORCES/COEFFICIENTS

4.1. In the **Outline View** pane, expand the heading **Results** > **Reports**. Double-click on the "Forces" heading as indicated in Figure 22.

4.2. In the "Force Report" pop-up window, set the options as indicated in Figure 23. Be sure to set the **Direction Vector** to "X=1", "Y=0", and "Z=0", as shown in Figure 23. Then click Print , and click Close .

4.3. Go to the **Console** view by selecting the **Console** tab at the bottom of the **Graphics** pane. Fluent will calculate the x-component of the forces acting on the airfoil, including the following:

- Pressure force in Newtons
- Viscous force in Newtons
- Total force in Newtons
- Pressure force coefficient
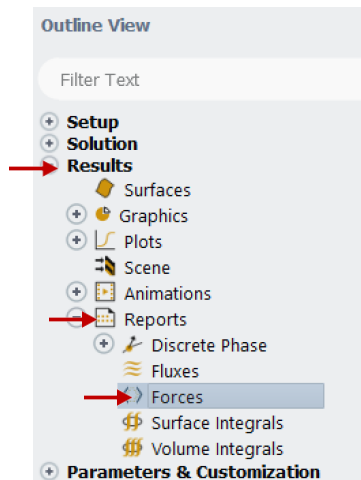- Viscous force coefficient
- Drag force coefficient

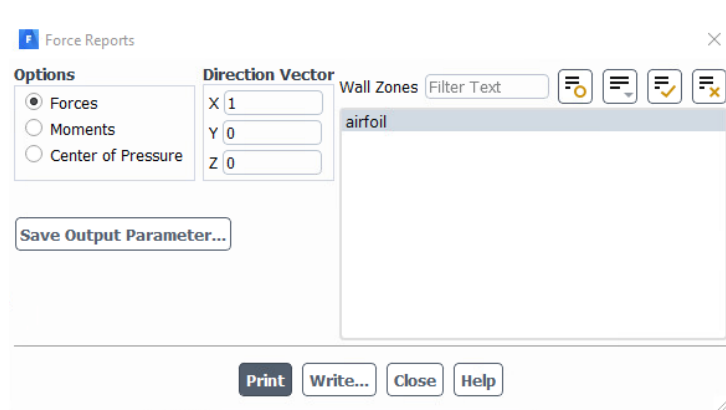Figure 22. Selecting the Forces Report.

Figure 23. Drag Force Report settings.

4.4.   The display should look like that shown in Figure 24, with possibly bit different values. Note, the value shows under Coefficients/Total, is the drag coefficient, including both pressure and viscous effects.

```
Forces - Direction Vector (1 0 0)
                       Forces (n)                                 Coefficients
Zone                   Pressure         Viscous          Total    Pressure         Viscous          Total
airfoil                0.80129674       0.22479031       1.0260871 0.027685628      0.007766737      0.035452365
----------------------- ---------------- ---------------- ---------------- ---------------- ---------------- ----------------
Net                    0.80129674       0.22479031       1.0260871 0.027685628      0.007766737      0.035452365
```

Figure 24. Force report showing the dimensional drag forces in Newton as well as the drag coefficients.

4.5. To get the lift force and lift coefficient, similar steps are followed with different direction vector. Double click on **Forces** to open up "Force Reports" again. Be sure to set the **Direction Vector** to "X=1", "Y=0", and "Z=0", as shown in Figure 25. Click Print , and click Close .

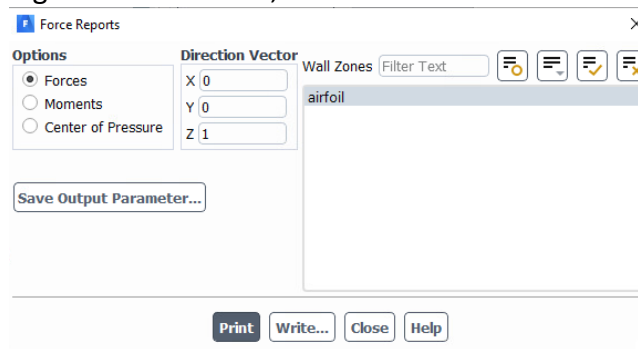Figure 25. Lift Force Report settings

4.6. Then, the total lift force and lift coefficient are shown in the Console window, similar to Figure 26.

```
Forces - Direction Vector (0 0 1)
                       Forces (n)                                 Coefficients
Zone                   Pressure         Viscous          Total    Pressure         Viscous          Total
airfoil                24.870616        0.028331694      24.898948 0.85930542       0.00097888922    0.86028431
----------------------- ---------------- ---------------- ---------------- ---------------- ---------------- ----------------
Net                    24.870616        0.028331694      24.898948 0.85930542       0.00097888922    0.86028431
```

Figure 26. Force report showing the dimensional lift forces in Newton as well as the lift coefficients.

4.7. COMMENT ON THE NUMERICAL SOLUTION

- After finishing this simulation, you should double the number of mesh elements in your domain and repeat the simulation again. You should do this twice more to complete the Mesh Independence study. Fluent Case files are provided for the other three mesh cases ("NACA0012_n100.cas", "NACA0012_n400.cas", and "NACA0012_n800.cas").

- After obtaining the drag and lift data for all the mesh resolution case studies, you should plot both drag and lift coefficient versus the number of mesh elements, similar to the figure provided earlier on page 3. Observe the pattern to determine the mesh independent case.

- Once the mesh independent case has been determined, then you can move on to **validating** the numerical simulation.

5. VALIDATING THE SIMULATION

It is essential to validate your numerical simulations by comparing the results against experimental data under the same conditions. In literature [1], for the flow around a NACA0012 airfoil at a Reynolds number of 1.5e5 and angle of attack $\alpha = 10°$, the drag and lift coefficients from experimental data are respectively: $C_D = 0.024$, and $C_L = 0.167$. The present Fluent simulation (for the mesh independent case) yielded a drag coefficient of about 0.029, which is close to the experimental value. However, the CFD simulation lift coefficient shows a much larger value compared to the experimental data. This is a large difference that can be due to the mesh quality or because the steady solver did not accurately capture the transient/turbulent flow patterns in the wake of the airfoil. One must make an engineering judgment as to whether the difference between the simulation results and experimental values is acceptable. If not, then the simulation solver and/or mesh must be further revised to improve the simulation accuracy.

Figure 28 shows the experimental data from the literature for the drag and lift coefficient of flow around NACA0012 airfoil at different Reynold numbers. The simulation data using the mesh-independent case (n=400) is shown for comparison over a range of angles of attack from 0º to 20º.
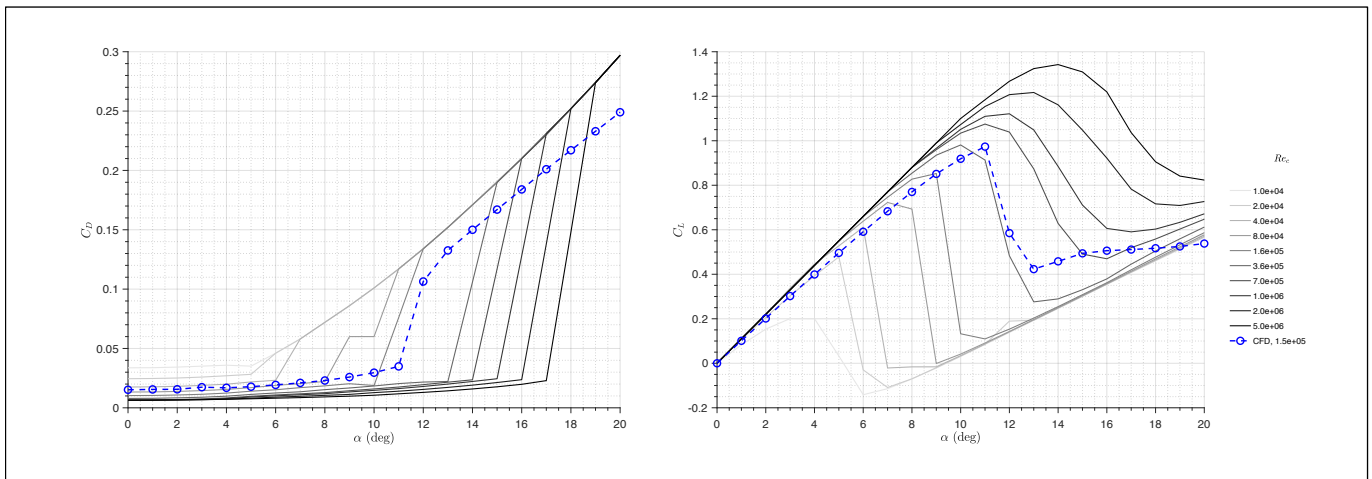

Figure 28. Drag coefficient (left image) and lift coefficient (right image) for NACA0012 airfoil versus angle of attack at different Reynolds numbers. The graylines are experimental data from [1]; while the blue circles represent the CFD simulation results using the mesh case of n=400.

[1]- Sheldahl, Robert E., and Paul C. Klimas. Aerodynamic characteristics of seven symmetrical airfoil sections through 180-degree angle of attack for use in aerodynamic analysis of vertical axis wind turbines. No. SAND-80-2114. Sandia National Labs., Albuquerque, NM (USA), 1981.

6. RESULTS: VISUALIZING THE VELOCITY AND PRESSURE FIELDS

We will visualize the results by plotting the velocity and pressure contours as well as the streamlines. We will also calculate the drag forces and drag coefficient since that is our primary interest.

6.1. Generate a contour plot of the <u>velocity magnitude</u>.

6.1.1. In the **Outline View** pane, expand the **Results** heading and expand the **Graphics** heading. Then, click on **Contours** as shown in Figure 29.

6.1.2. In the Contours window (see Figure 30) set the Contours of textbox to "Velocity …" and the dropdown menu to "Velocity Magnitude". The default settings for the other options should be as shown in Figure 30.
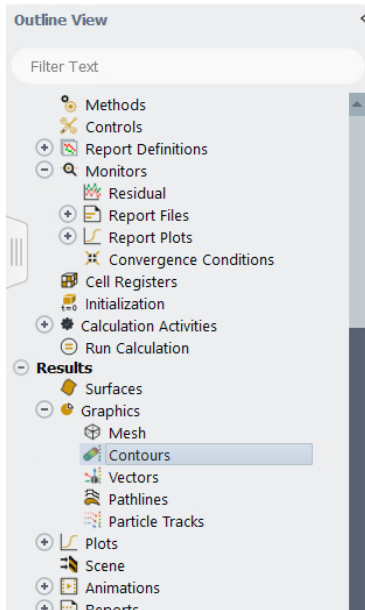
6.1.3. Click Save/Display. Then click Close.
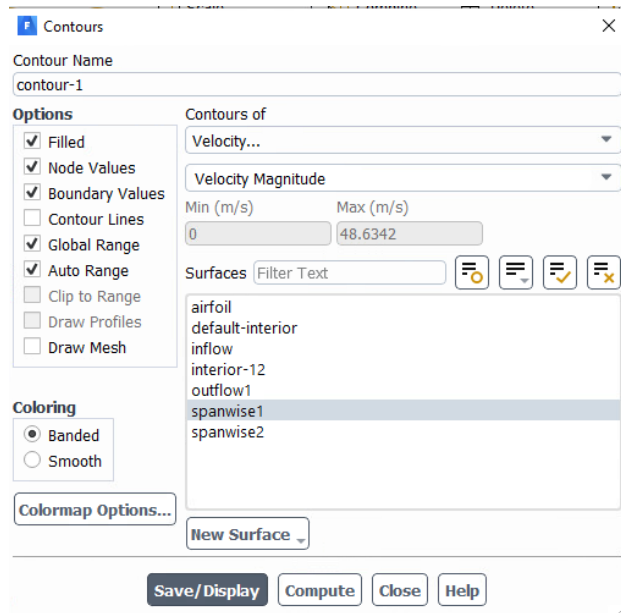

Figure 29. Selecting the "Contours" Graphic.

Figure 30. Velocity contour option window

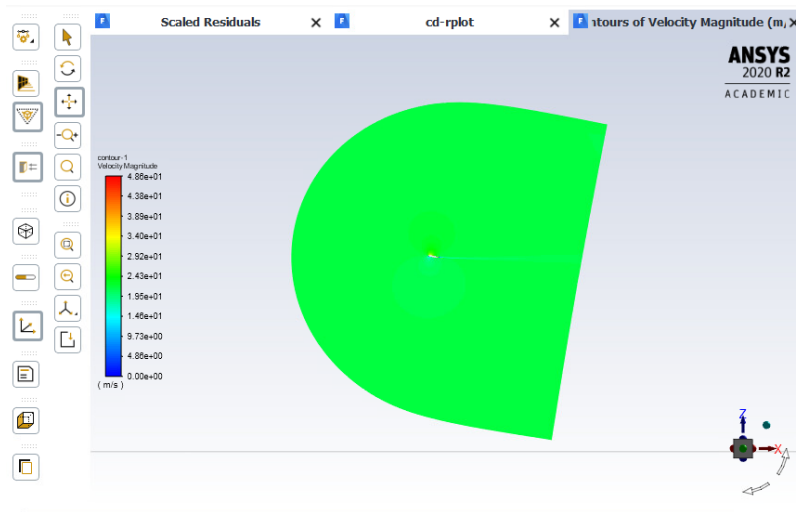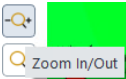6.1.4. First, make sure that the **Graphics** view is at **x-z plane** as shown in Figure 31. To get this view, click on the "y" axis of coordinate system shown.

Figure 31. Graphics view aligned with x-z plane

6.1.5. Then, use the "**Zoom In/Out**" [Zoom In/Out] option from the left menu, to draw a box around the airfoil and get a zoom-in view. Similar to Figure 32. Use the Pan option to drag the view if needed.
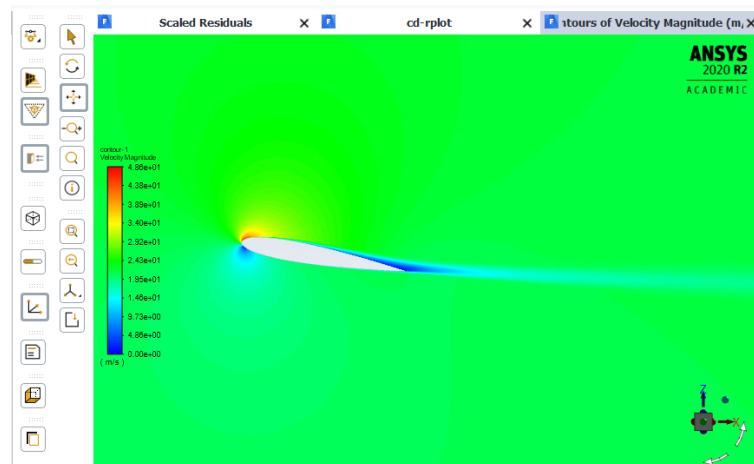

Figure 32. Zoom-in velocity contour plot around the airfoil

6.1.6. To save the plot: on the top menu bar, select **File > Save Picture**. In the "Save Picture" popup window, set the options as shown in Figure 33. Click [Save...]. And then, type in your file name in the "Select File" popup window.
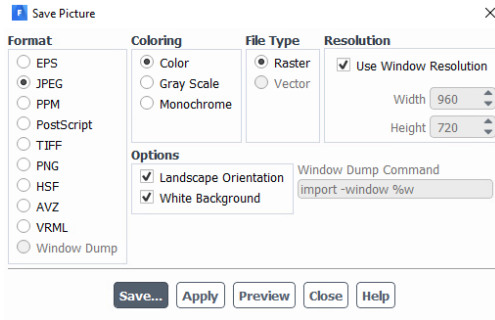
Figure 33. "Save Picture" window.

6.2. Generate a contour plot of the <u>pressure</u>.

6.2.1. In the **Outline View** pane, click **Results > Graphics > Contours** again.

6.2.2. In the "Contours" pop-up window, select the options as shown in Figure 34.

6.2.3. Click **Save/Display**. Then click **Close**.

6.2.4. You should see an image in the Graphics pane that looks like the one in Figure 35. Note, a pressure coefficient of 1 means the local static pressure is equal to the total pressure of the flow at the inlet (here, "total pressure" is the sum of the static and dynamic pressures). Whereas a pressure coefficient of zero means that the local static pressure is equal to the dynamic pressure of the flow at the inlet.
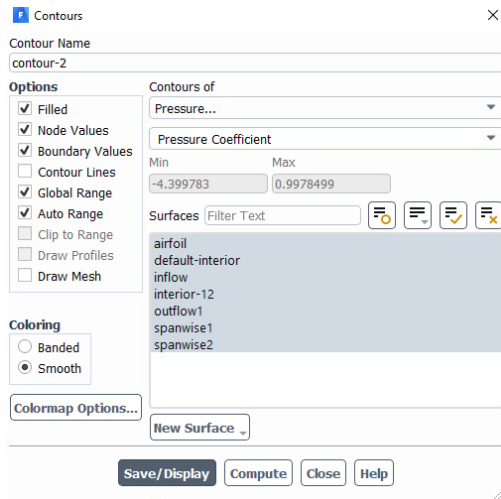

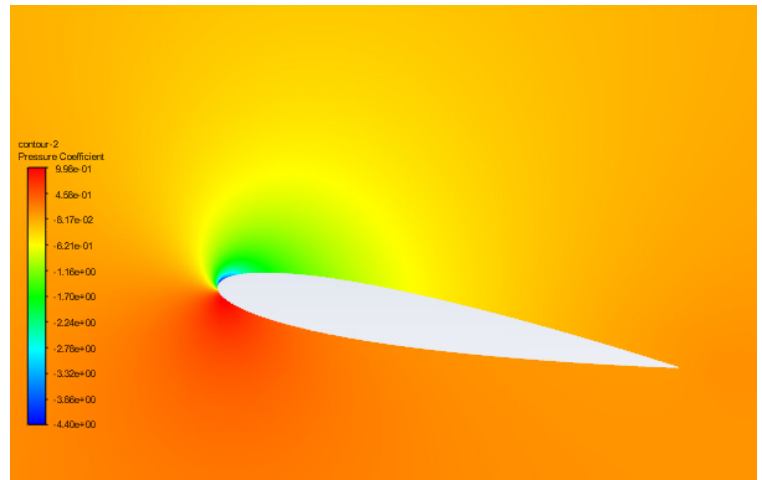Figure 34. Options for the pressure contours.

Figure 35. Pressure Coefficient contour in the vicinity of the airfoil.

21

6.3. Generate a contour of the Pathline.

6.3.1. In the **Outline View** pane, expand the headings **Results** > **Graphics**. Double click on **Pathlines**.

6.3.2. In the "Pathlines" popup window, click on New Surface>Line/Rake to define a line located at the front of airfoil, that the pathlines start from.

6.3.3. Set **Type** to **Rake**, **Number of Points** to **50**, and the following parameters for End Points:
x0 (m) 0   x1 (m) 0
y0 (m) 0   y1 (m) 0
z0 (m) -0.04   z1 (m) 0.04

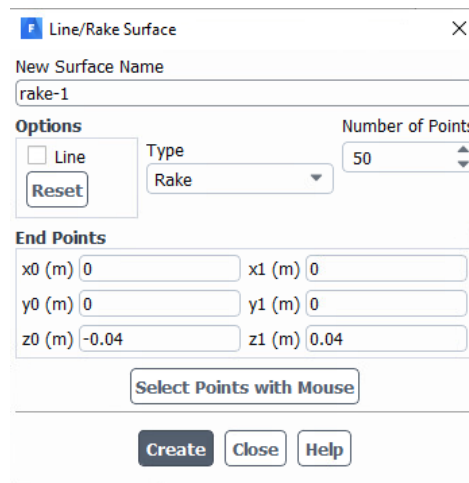6.3.4. The final settings should be similar to Figure 36. Click on **Create** and close.

Figure 36. Line/Rake Surface settings

6.3.5. Under the "Release from Surfaces", choose the rake that you just created. If needed, change the "Path Skip" to 1 or higher. This will coarsen the pathlines by ignoring some of the path lines. Keep the parameters as default, following Figure 37 (left image). The pathline should look like Figure 37 (right image). The pathlines can show wake in the flow and separation of the boundary layer from the airfoil.
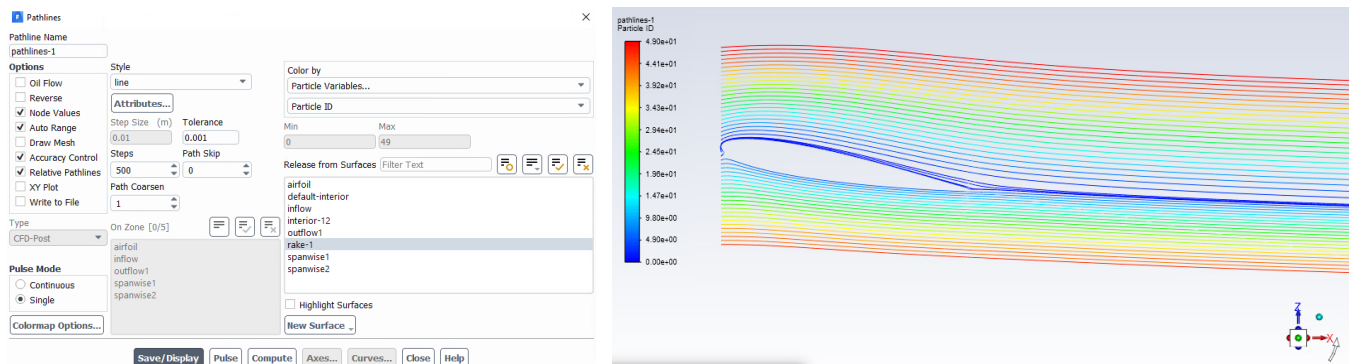

Figure 37. Pathline settings (left image), and Pathline contour (right image)