

# PROJECT FLUID MECHANICS (THEORY)

## SIMULATION EXPERIMENT: NACA-23018

## ANALYSIS OF THE AERODYNAMICS AND STALL BEHAVIOUR USING ANSYS

# FLUID MECHANICS PROJECT THEORY



## GROUP MEMBERS & CONTRIBUTIONS:

- 1) Muhammad Ahmed Saeed (simulation and results)
- 2) Saad Ahmed (Domain building)
- 3) Ahsan Rehman Khan (Meshing)

**DEGREE: 43-ME**

**SYNDICATE: A**

**COURSE: Fluid Mechanics II**

**SUBMITTED TO: DR TARIQ TALHA**

# TABLE OF CONTENTS

ABSTRACT .....	4
INTRODUCTION .....	5
METHADODOLOGY & TUTORIAL.....	6
CREATING THE DOMAIN .....	6
SOLUTION AND SIMULATION .....	12
RESULTS AND DESCRIPTION.....	20
CONCLUSIONS AND RECOMMENDATONS .....	27

# ABSTRACT

This report presents a comprehensive analysis of the aerodynamic characteristics of the NACA-23018 aerofoil using Ansys simulation software. The NACA-23018 aerofoil is widely employed in various aerospace applications due to its favourable lift and drag properties. The objective of this study is to investigate the performance of the aerofoil under different operating conditions, such as varying angles of attack.

The simulation process involves the creation of a three-dimensional model of the NACA-23018 aerofoil in Ansys, followed by the implementation of a suitable turbulence model and boundary conditions. Computational Fluid Dynamics (CFD) techniques are utilized to solve the governing equations and obtain the aerodynamic coefficients.

The simulation results will provide valuable insights into the behaviour of the NACA-23018 aerofoil. The lift and drag coefficients are evaluated two different angles of attack, leading to the determination of the aerofoil's stall characteristics and maximum lift-to-drag ratio. Finally, the pressure distribution around the aerofoil surface is examined to understand the range of flow patterns and identify areas of low and high pressure.

The simulation results will be used for comparison with experimental data, providing a basis for validation for this model.

Overall, this report highlights the significance of Ansys simulation in assessing the aerodynamic behaviour of the NACA-23018 aerofoil. The comprehensive analysis of lift, drag, and pressure distribution characteristics aids in optimizing the aerofoil design and enhancing the overall efficiency of aerospace systems.

# INTRODUCTION

The analysis of an aerofoil plays a critical role in the optimization and the development of aerospace systems. Aerofoils are aerodynamic structures that generate lift and control the airflow in the system using Fluid Mechanics principles. The selection of such varying structures is essential in optimizing the aerodynamic performance of the airborne vehicle.

The NACA-23018, belonging to the 23XX series, is one of the widely used due to its favourable lift and drag ratio value against a range of angle of attacks. The use of simulation techniques such as Ansys (a powerful computational fluid dynamics (CFD) capabilities for conducting detailed airflow simulations around complex geometries) can provide a cost effective and an efficient solution to test the aerodynamic of this aerofoil type.

Our primary objective is to analyse the lift coefficient, study the stall behaviour the aerofoil shall exhibit and finally observe the pressure and velocity distribution of the aerofoil at two extreme angles of attack. The simulation results shall provide a general insight of the aerofoil's performance and help in further research and optimization of other aerodynamic components.

The sole equation that can be used in this entire study is the Bernoulli's equation for all pressure and velocity distributions.

(praxilabhtt)

$$P_1 + \frac{1}{2}\rho v_1^2 + \rho g h_1 = P_2 + \frac{1}{2}\rho v_2^2 + \rho g h_2$$

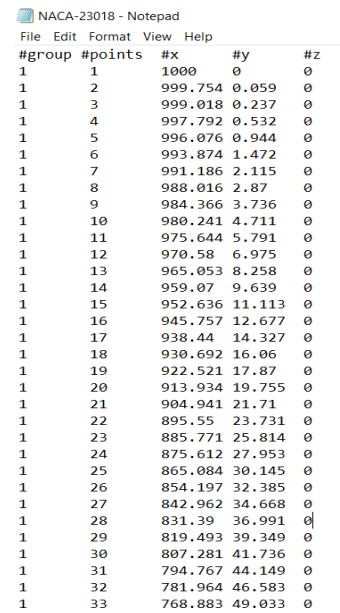
$$P_1 + \frac{1}{2}\rho v_1^2 + \rho g h_1 = \text{constant}$$

# METHADODOLOGY & TUTORIAL

## CREATING THE DOMAIN

### Importing the coordinates of the Aerofoil.<sup>1</sup>

- 1) Using the 5-digit aerofoil generator, increasing the points to 200 for more accurate results. Finally pasting it in the txt file and adding one more Z axis and initializing all z-axis rows to 0 to show compatibility in the Ansys application.
- 2) Launching Ansys and selecting Ansys fluent analysis system in the project schematic and load up the design modeller.
- 3) Importing the Aerofoil txt file from the 3D curve option in the Concept drop down menu (change the default settings to millimetres as our aerofoil has a chord length of 1 meter).
- 4) Generating the aerofoil.

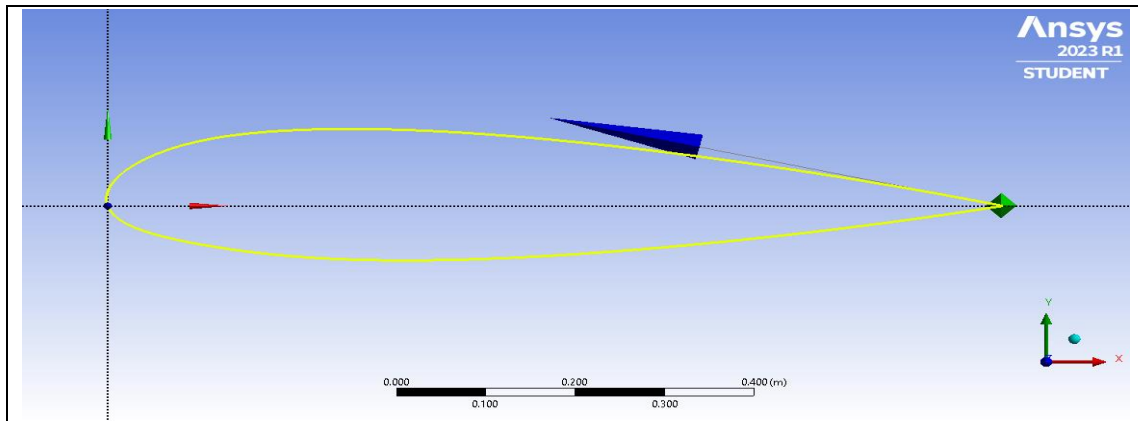


#group	#points	#x	#y	#z
1	1	1000	0	0
1	2	999.754	0.059	0
1	3	999.018	0.237	0
1	4	997.792	0.532	0
1	5	996.076	0.944	0
1	6	993.874	1.472	0
1	7	991.186	2.115	0
1	8	988.016	2.87	0
1	9	984.366	3.736	0
1	10	980.241	4.711	0
1	11	975.644	5.791	0
1	12	970.58	6.975	0
1	13	965.053	8.258	0
1	14	959.07	9.639	0
1	15	952.636	11.113	0
1	16	945.757	12.677	0
1	17	938.44	14.327	0
1	18	930.692	16.06	0
1	19	922.521	17.87	0
1	20	913.934	19.755	0
1	21	904.941	21.71	0
1	22	895.55	23.731	0
1	23	885.771	25.814	0
1	24	875.612	27.953	0
1	25	865.084	30.145	0
1	26	854.197	32.385	0
1	27	842.962	34.668	0
1	28	831.39	36.991	0
1	29	819.493	39.349	0
1	30	807.281	41.736	0
1	31	794.767	44.149	0
1	32	781.964	46.583	0
1	33	768.883	49.033	0

Figure 1: NACA-23018 coordinates

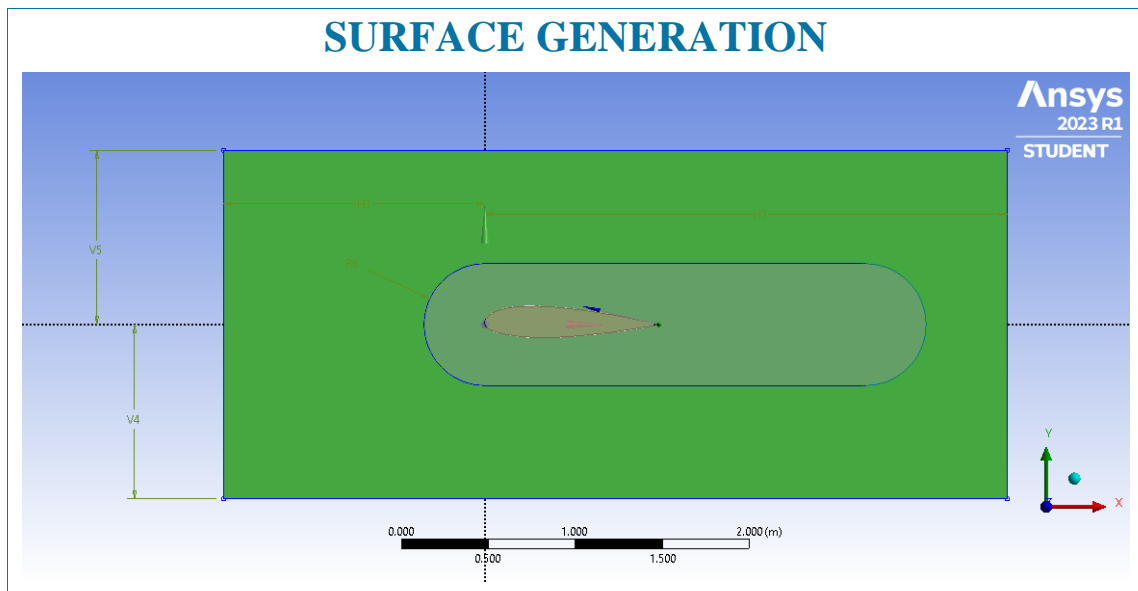
<sup>1</sup> <http://airfoiltools.com/airfoil/details?airfoil=naca23018-il>

## GENERATION OF AEROFOIL COORDINATES



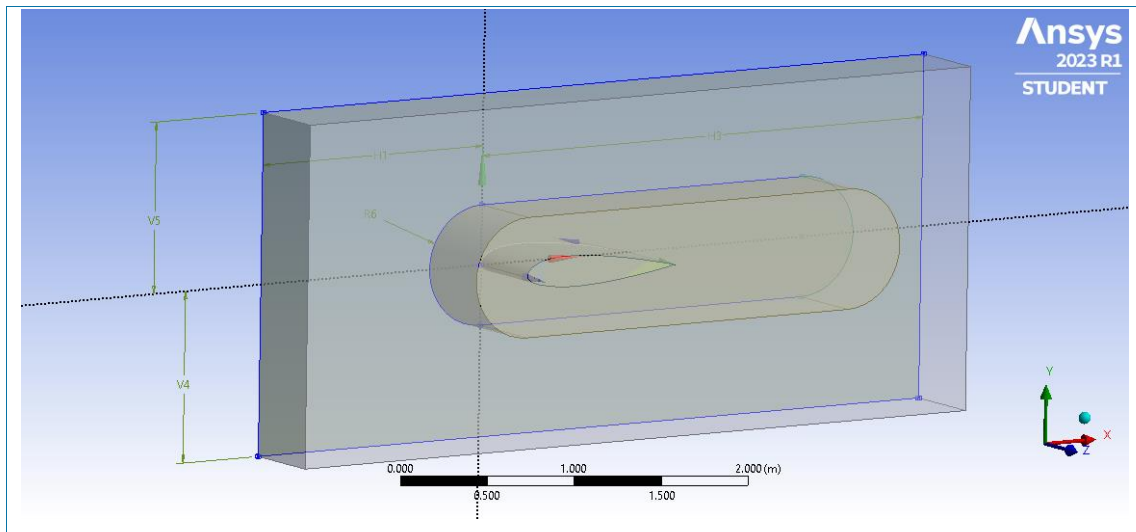
- 5) create another domain around the C section of rectangular shape of length 4.5 m and width of 2m
- 6) Finally, from the drop-down menu, use “surface from sketches” to create a surface for the aerofoil and select “surface from edges” to create surfaces for both domains and generate the surface.
- 7) Under the tree outline, we changed the properties of the surface to “frozen” for both the C and rectangular domain. This will ensure isolation from both domains required for proper meshing and simulation.

## SURFACE GENERATION



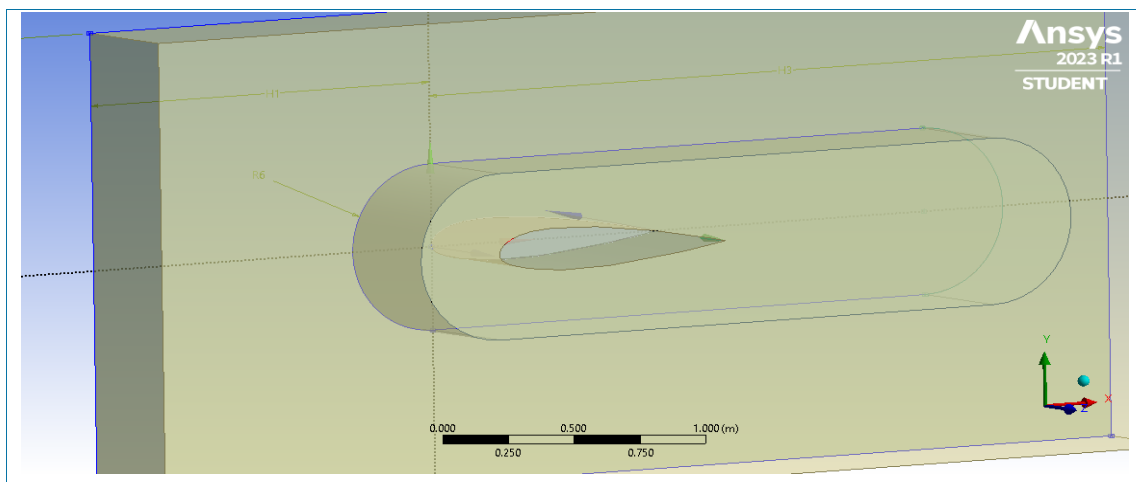
- 8) Finally select and extrude all three surfaces to a length of 0.5 m and selecting the direction vector as XY plane (use the same operation as “add frozen” to ensure all extruded surfaces are in isolation from one another)

## EXTRUSION OF ALL THREE SURFACES



- 9) At the end, we used the Boolean operation under the “create” drop down menu and subtracted the aerofoil surface from the rest of the domains.

## IMAGE DEMONSTRATION



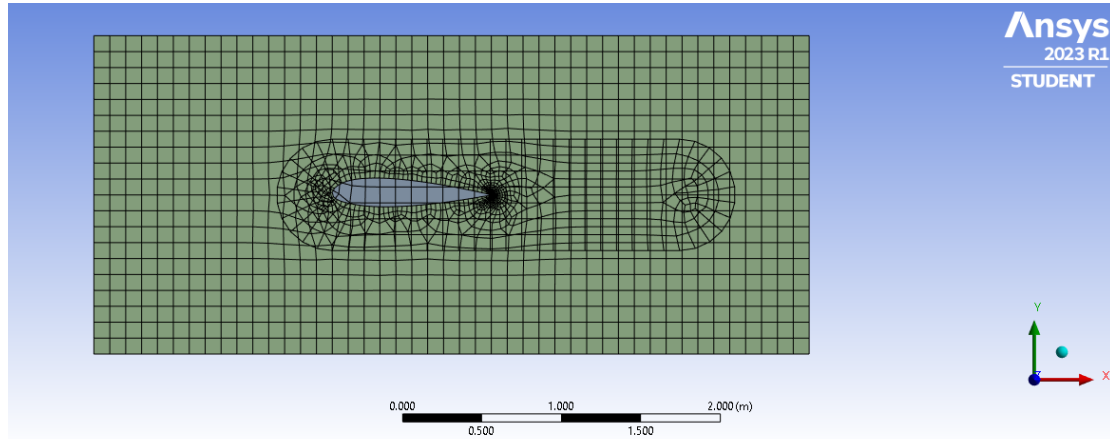
- 10) Saving the project from the file menu and moving on to Meshing.

## GENERATING THE MESH



- 1) Upon loading the meshing software. In the outline section in the left side, we generated the initial mesh from the start.

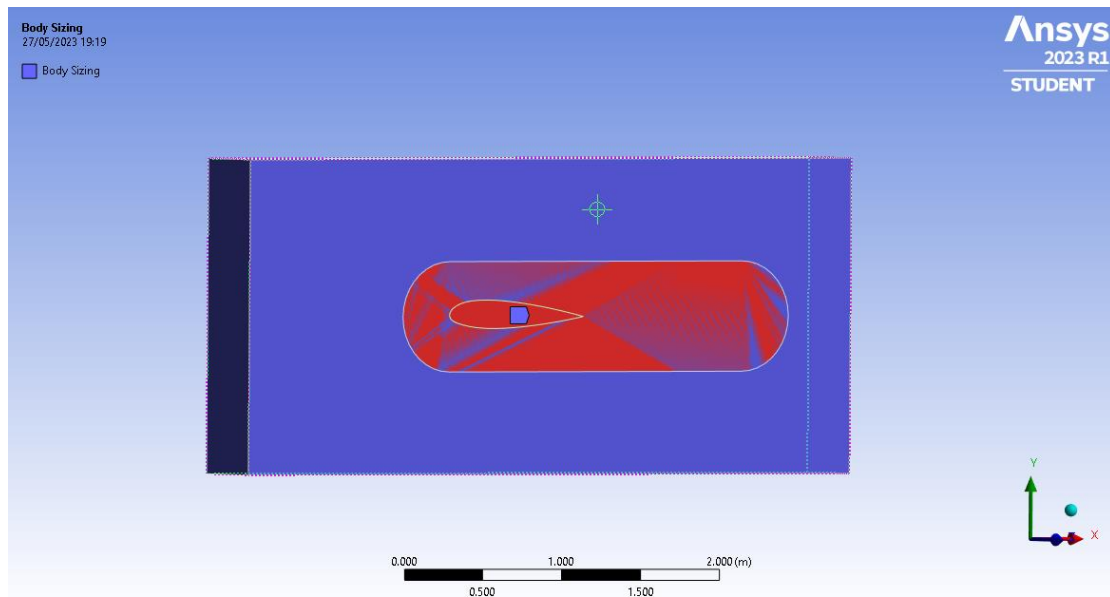
## IMAGE



Under “details of meshing,” we tweaked the settings to increase the number of nodes and decreasing cell size. Decreasing the element size to a factor of 0.1 m. under sizing, decrease the growth rate to 1.2 and the max size to 0.2 m.

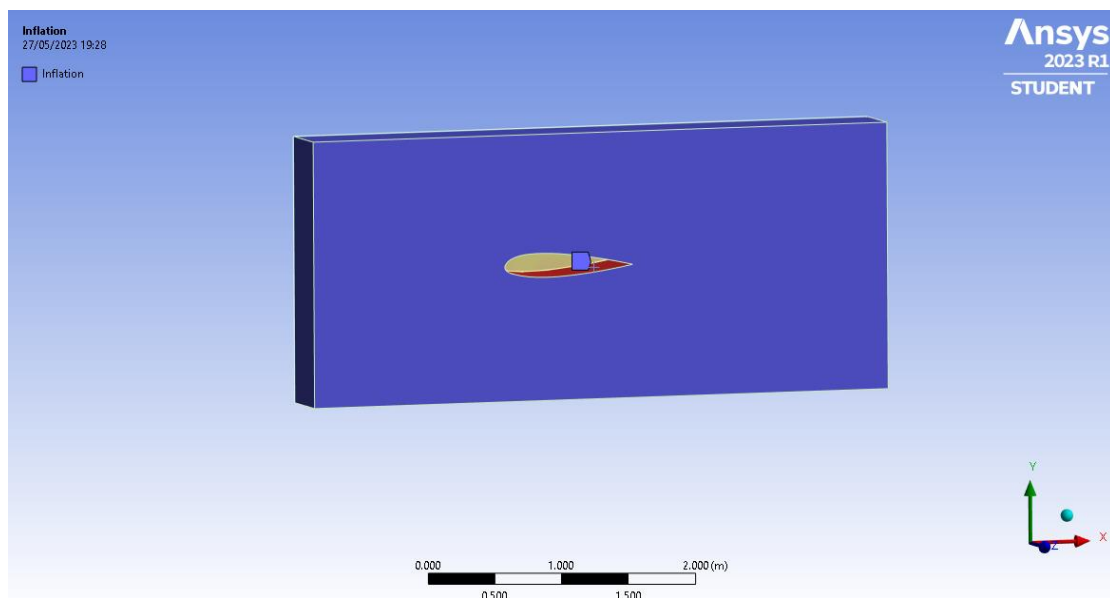
- 2) Left clicking meshing option and selecting “sizing” under the insert drop down menu.
- 3) By Selecting body of influence, we change how one domain effects the other. In this case we are going to select the rectangular domain as our body and the C section domain as our “body of influence.”

## IMAGE DEMONSTRATION



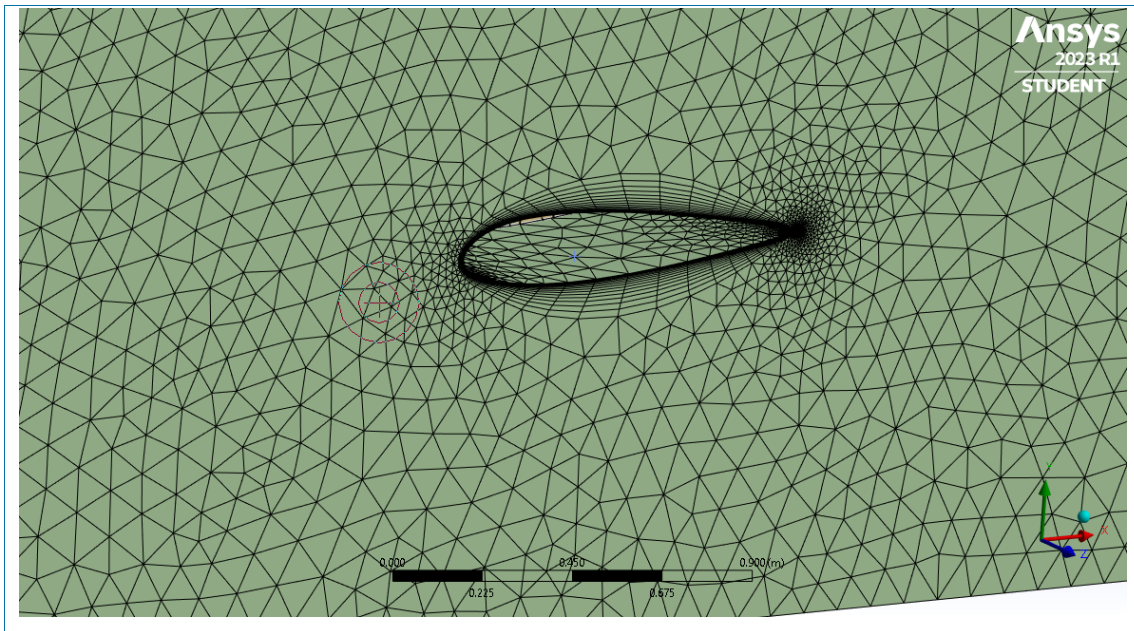
- 4) To create boundary layer condition on the airfoil surface, we will select inflation from the insert menu by right clicking on the mesh option. Select the airfoil face as your boundary geometry. Under the Inflation drop down menu, increase the number of layers to 15 and the transition ration to 0.27.

## IMAGE DEMONSTRATION



Then generate the mesh.

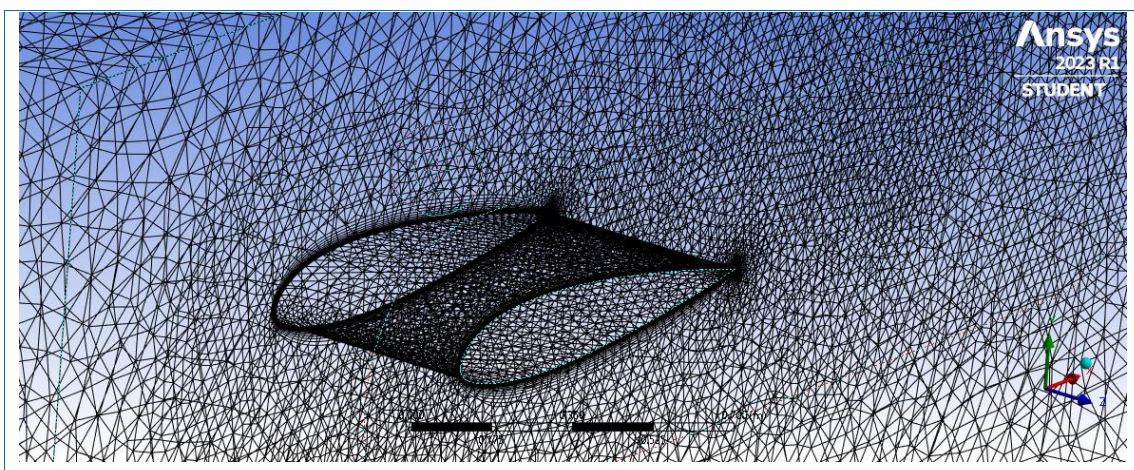
## GENERATION OF MESH



The boundary layer is now visible across the across surface. This ensures that the velocity on the surface of the aerofoil is 0 due to the no slip condition and increases with respect to the Y axis.

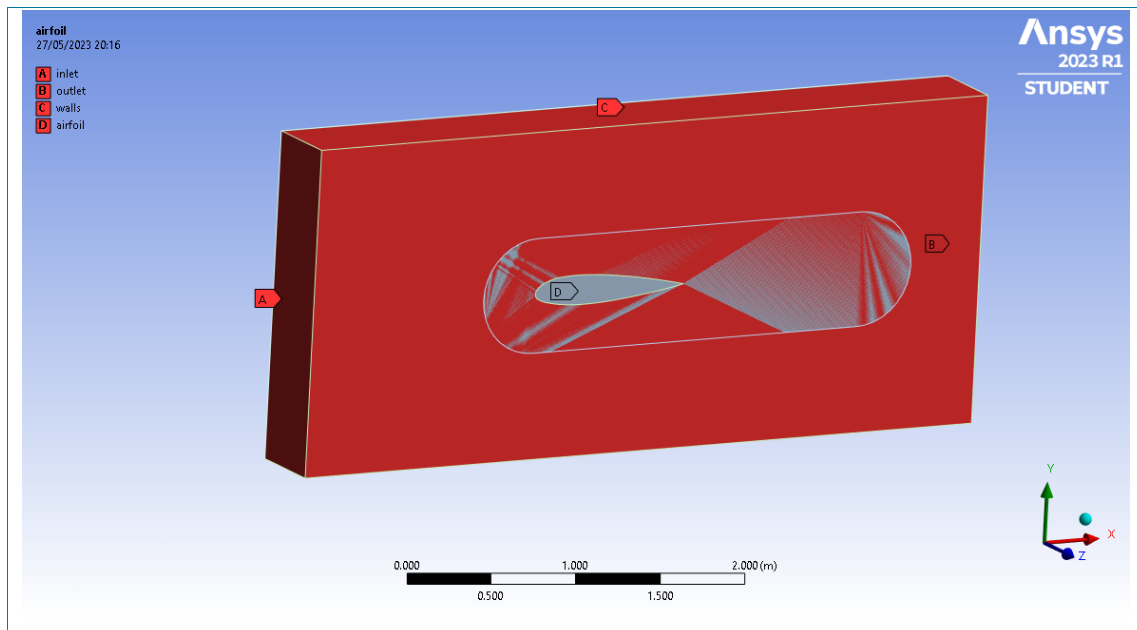
In Wireframe mode, the mesh is as follows.

## WIREFRAME MODE



- 5) Finally, we create the named sections, selection the front of the domain to be in the inlet condition. The top and bottom surfaces as well as the sides of the domain to be walls and the back surface to be the outlet.

## IMAGE DEMONSTRATION

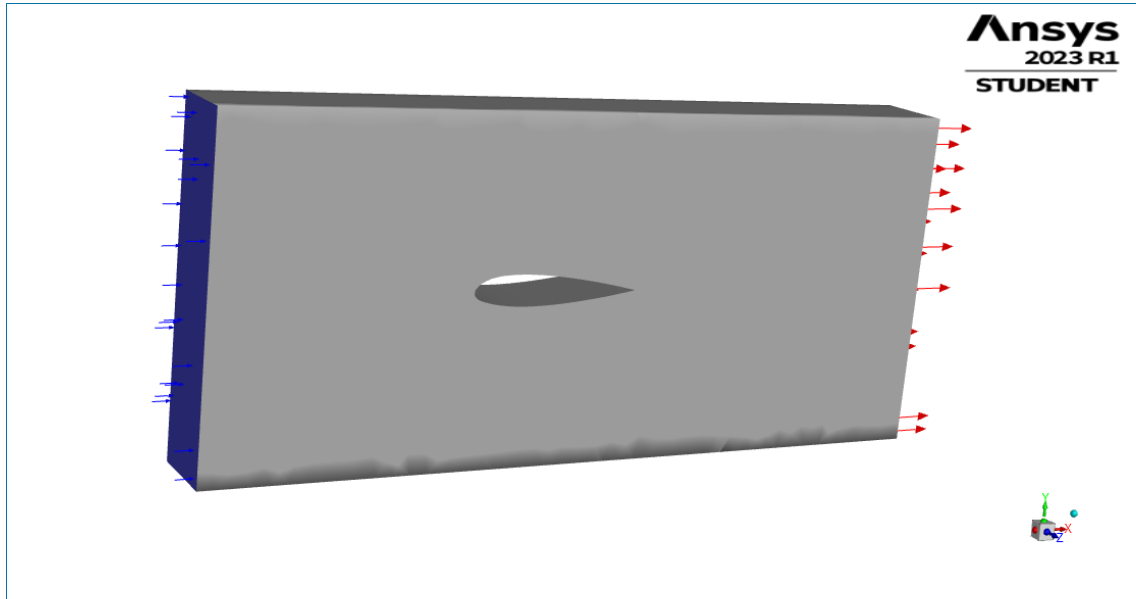


## END POINT OF MESH PREPARATION

## SOLUTION AND SIMULATION

- 6) We move to the final part of the simulation. Select “setup” in the project schematic. The number of cores that are to be selected are 4, since this is an Ansys student software. It also restricts the number of elements in the mesh to be no less than 500,000. We should be greeted with the following display of our model.

## DISPLAY IN SETUP



- 1) From this point forward, Use the following settings for an accurate simulation.
- 2) In the outline tree, double click on the setup and select Viscous “SST K – Omega” as a default model. This model is accurate for airflow simulations.
- 3) Under boundary conditions drop down menu, double click “inlet” and set the X – component velocity to the desired value and multiply it with  $\cos 15$  degrees. This will be our velocity for the 15-degree angle of attack. Keep Z velocity component to 0 and multiply the desired value of the Y – velocity component by  $\sin 15$  degrees. With this we can simulate the critical angle of attack that forces stall behaviour in the NACA – 23018.
- 4) Under solutions drop down menu, we can now set up the report definitions for the calculation of drag and lift coefficients. Left click on the report definition option and select add. Selecting the following options as below
- 5) The same setting applies to the lift coefficient as well.

## BOUNDARY CONDITONS



Velocity Inlet
×

Zone Name

Momentum
 Thermal
 Radiation
 Species
 DPM
 Multiphase
 Potential
 Structure
 UDS

Velocity Specification Method Components

Reference Frame Absolute

Supersonic/Initial Gauge Pressure [Pa] 0

Coordinate System Cartesian (X, Y, Z)

X-Velocity [m/s] 44.04

Y-Velocity [m/s] 11.802

Z-Velocity [m/s] 0

**Turbulence**

Specification Method Intensity and Viscosity Ratio

Turbulent Intensity [%] 5

Turbulent Viscosity Ratio 10

Apply

Close

Help

Pressure Outlet
×

Zone Name

Momentum
 Thermal
 Radiation
 Species
 DPM
 Multiphase
 Potential
 Structure
 UDS

Backflow Reference Frame Absolute

Gauge Pressure [Pa] 0

Pressure Profile Multiplier 1

Backflow Direction Specification Method Normal to Boundary

Backflow Pressure Specification Total Pressure

☐ Prevent Reverse Flow
   
☐ Radial Equilibrium Pressure Distribution
   
☐ Average Pressure Specification
   
☐ Target Mass Flow Rate

**Turbulence**

Specification Method Intensity and Viscosity Ratio

Backflow Turbulent Intensity [%] 5

Backflow Turbulent Viscosity Ratio 10

Apply

Close

Help

## REPORT DEFINITION FOR DRAG AND LIFT COEFFICIENT

**Drag Report Definition**

Name: drag-coefficient

**Options**

☐ Per Zone

Average Over(Iterations): 1

**Force Vector**

X: 0 Y: 0 Z: 0

**Report Files [1/2]**

- cd-rfile
- cl-rfile

**Report Plots [1/2]**

- cl-rplot
- cd-rplot

**Create**

☒ Report File

☒ Report Plot

Frequency: 1

☐ Print to Console

☐ Create Output Parameter

**Report Output Type**

☒ Drag Coefficient

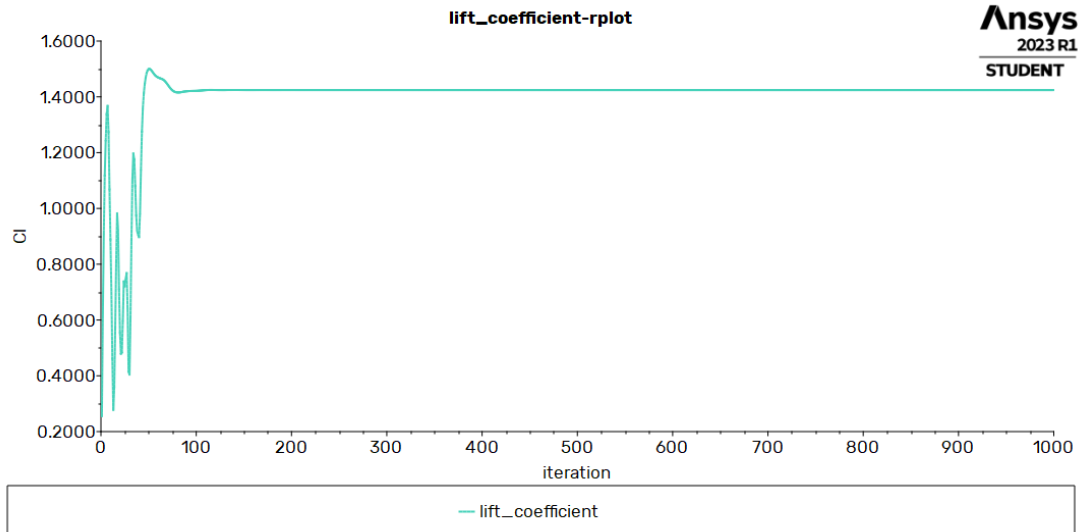
☐ Drag Force

**Zones** Filter Text

- airfoil
- walls

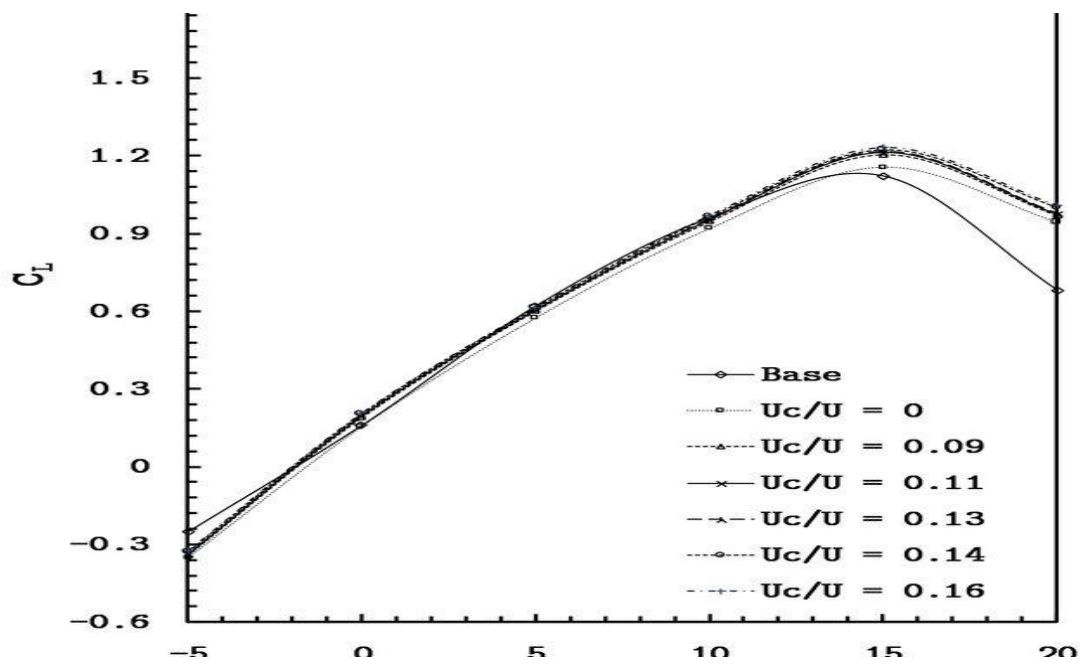
- 1) Then Run calculation in the “calculations” option and set the number of iterations to one thousand. The calculation should converge after a few hundred iterations due the lack of the complexity of the model. The following results confirm this statement.

## LIFT COEFFICIENT RESULTS



Let us compare this value to from an experimental source.

## LIFT COEFFICIENT DATA




- 2) The given chart confirms the value of the lift coefficient to be 1.3 at 15-degree angle of attack. Our calculation converges at 1.4. An approximate 7.69 percentage error. This can be due the lack of accuracy of the numbers



of elements in our mesh. For the error to reduce, the elements should be in the millions.

- 3) Finally, we create the velocity and the pressure contours. Since the aerofoil is in 3D, we must create a 2D plane to get a better view of the pressure and velocity distribution. From the top right corner, select create and then select planes. Change the method to “three points” and use the following settings.

## GENERATION OF PLANE SURFACE

 **Plane Surface** ×

**New Surface Name**

**Method**  

Three Points

**Point 1**

X [m]	Y [m]	Z [m]	Select with Mouse
<input type="text" value="0.8350364"/>	<input type="text" value="2.249987"/>	<input type="text" value="2.500368"/>	

**Point 2**

X [m]	Y [m]	Z [m]	Select with Mouse
<input type="text" value="0.8350364"/>	<input type="text" value="-2.250749"/>	<input type="text" value="2.500368"/>	

**Point 3**

X [m]	Y [m]	Z [m]	Select with Mouse
<input type="text" value="0.8350364"/>	<input type="text" value="-2.250749"/>	<input type="text" value="-2.000368"/>	

**Options**

☐ Bounded  
☐ Sample Points

**Sample Density**

Edge 1	<div>10</div>
Edge 2	<div>10</div>

**Surfaces**

1

Reset

Create


Close

Help

This shall create a plane in the middle of the domain.

- 4) Finally, under result drop down menu. Right click on “contours” and create both the velocity and the pressure contours.





## VELOCITY CONTOUR SETTINGS

 Contours ×



Contour Name  
velocity-contour

**Options**


- ☒ Filled
- ☒ Node Values
- ☒ Boundary Values
- ☐ Contour Lines
- ☒ Global Range
- ☒ Auto Range
- ☐ Clip to Range
- ☐ Draw Profiles
- ☐ Draw Mesh

**Contours of**  
Velocity...  
Velocity Magnitude  
Min [m/s] 0 Max [m/s] 0  
Surfaces       
airfoil  
inlet\_  
outlet  
plane-4  
walls

**Coloring**  
☐ Banded  
☒ Smooth  
Colormap Options...

**Display State**  
None    

## PRESSURE CONTOUR SETTINGS


Contours

Contours Name

pressure-contour

Options

☒ Filled  
☒ Node Values  
☒ Boundary Values  
☐ Contour Lines  
☒ Global Range  
☒ Auto Range  
☐ Clip to Range  
☐ Draw Profiles  
☐ Draw Mesh

Contours of

Pressure...

Static Pressure

Min [Pa]

0

Max [Pa]

0

Surfaces

Filter Text

☒
☐
☐
☐

airfoil  
 inlet\_  
 outlet  
 plane-4  
 walls

Coloring

☐ Banded  
☒ Smooth

Colormap Options...

Display State

None

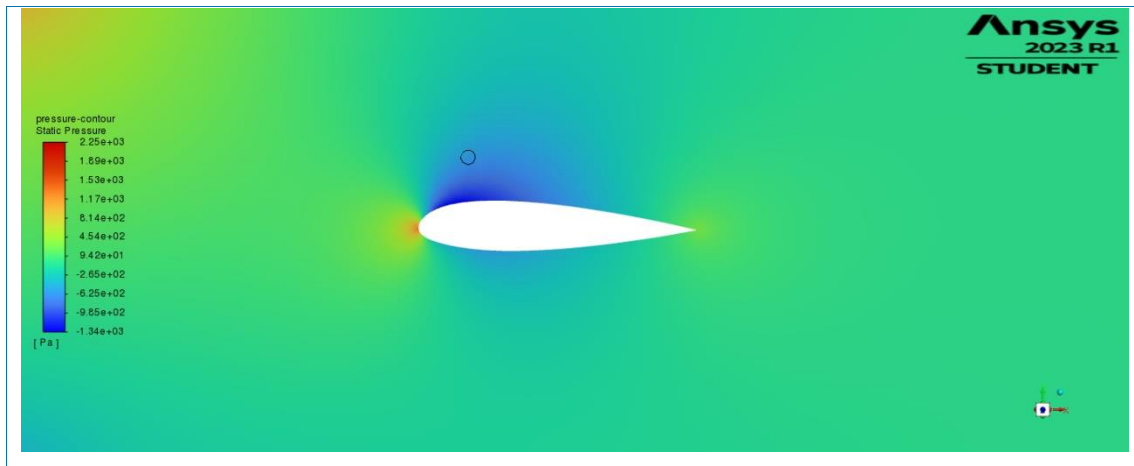
Use Active

New Surface

Finally, after entering save/display, we can now finally see our pressure and velocity contours,

# RESULTS AND DESCRIPTION

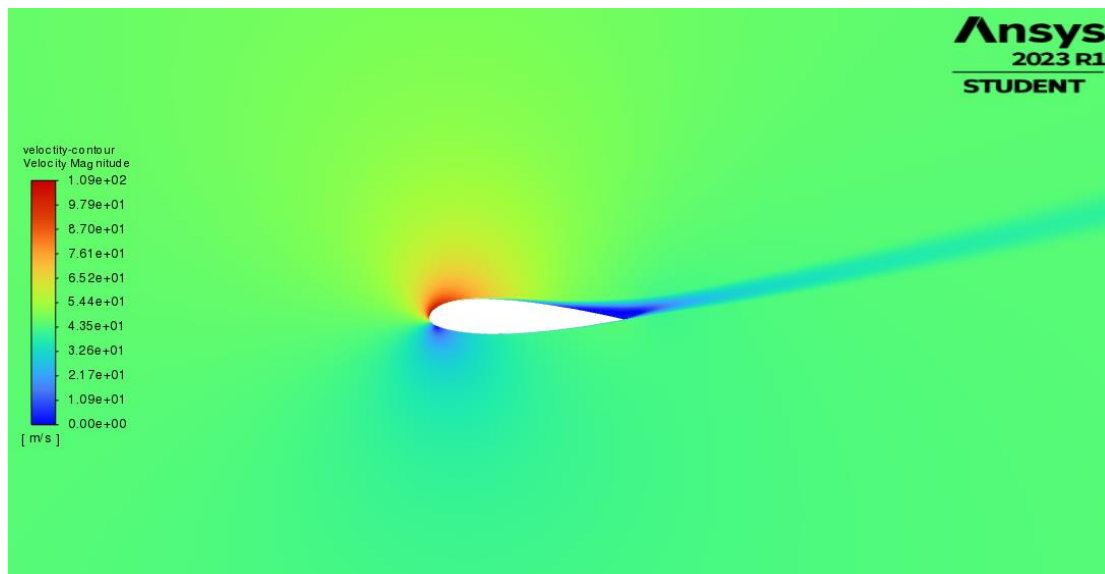
## PRESSURE CONTOUR (15-degrees AOA)



### Explanation:

- We can clearly demonstrate the stall location of the aerofoil; The yellow-orange region indicates an area of high pressure whereas the gradient from green to blue represents a decrease in the pressure. At an angle of 15 degree with respect to the wind axis, the flow separation is shifted much closer to the leading edge, creating a large number of standing vortices, hence increasing the pressure drag.

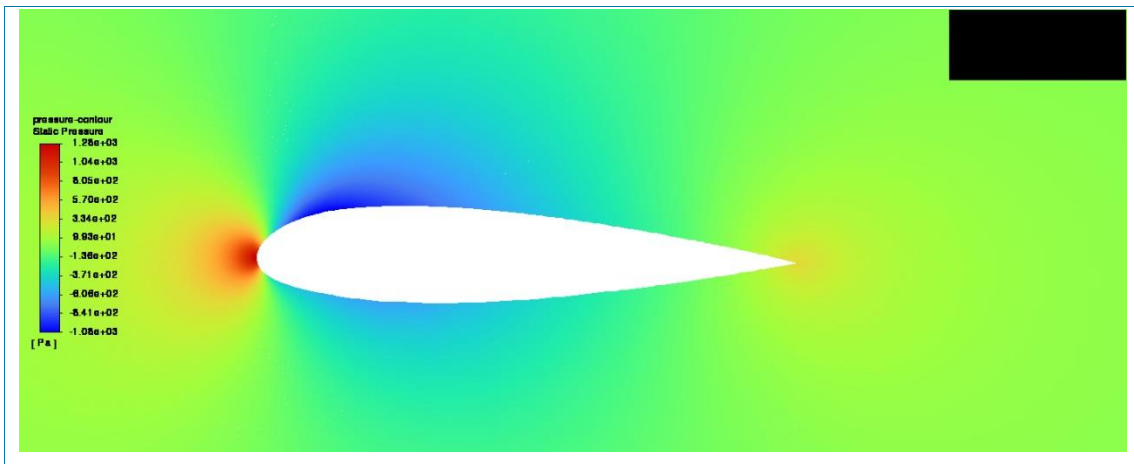
## VELOCITY CONTOUR (15-degrees AOA)



## Explanation:

- The above image demonstrates the velocity contour along different areas of the aerofoil. By using the Bernoulli's principle, the flow on the top surface is in the RED region which indicates high velocity according to the bar on the left. An area of high velocity would create a low-pressure region. Furthermore, the area below the aerofoil is of high-pressure region (of low velocity) as indicated by the BLUE colour region. This shows the effect of stall at high angle of attack that would create a downward force on the aerofoil and hence forcing the wing to stall to stay in equilibrium.
- This also shows how the flow separation has shifted backward, creating an influx of standing vortices (indicated by the trail of blue region from the trailing edge). The result is drag pressure that increases the pressure in this region.

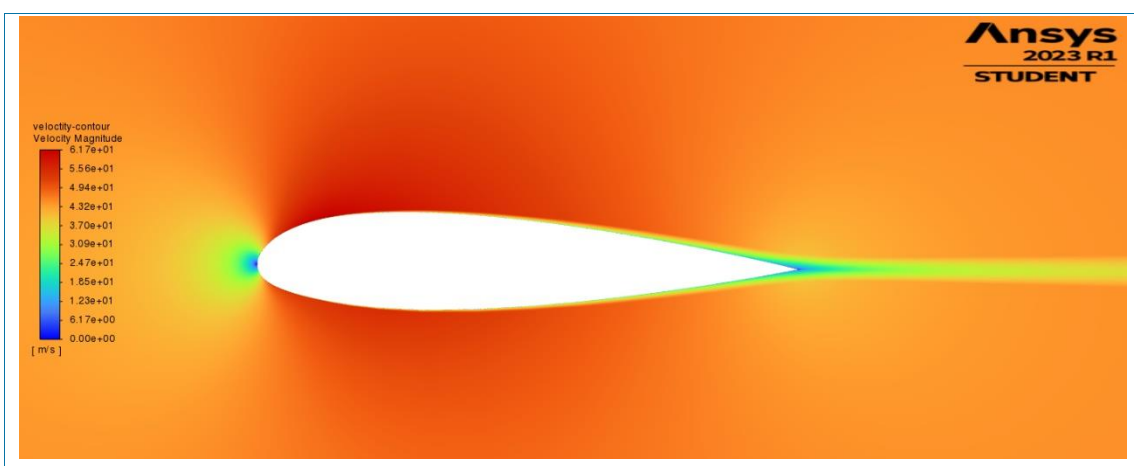
## PRESSURE CONTOUR (0-degree AOA)



### Explanation:

- The pressure distribution at the upper and lower surface differs slightly, creating insufficient lift. That is because the velocity distribution is the same in both surfaces. The highest pressure is located at the midpoint of the leading edge, indicating a maximum stagnation pressure.

## VELOCITY CONTOUR (0-degree AOA)

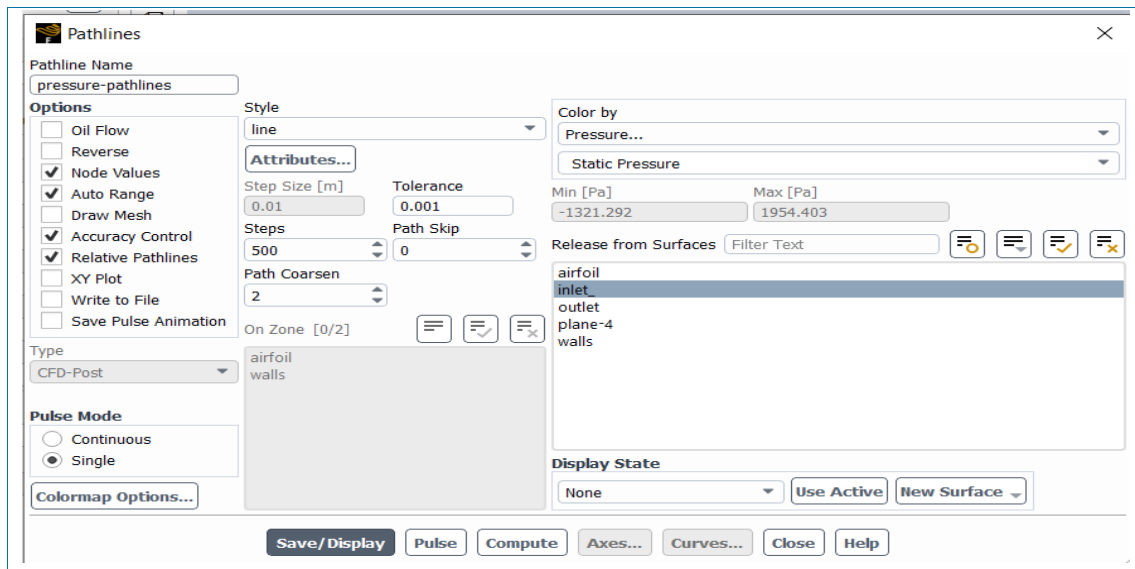


### Explanation:

- At 0-degree angle of attack, the flow separation occurs at the leading edge ( the stagnation point of the aerofoil where the velocity magnitude is 0. This is can verified with the above image, upon closer look, the blue region at the stagnation point indicates minimum velocity). The velocity distribution is almost the same in the upper and the lower surface of the aerofoil. A typical is that of an aircraft taking off in a runaway without changing the angle of attack, it does not create sufficient lift.

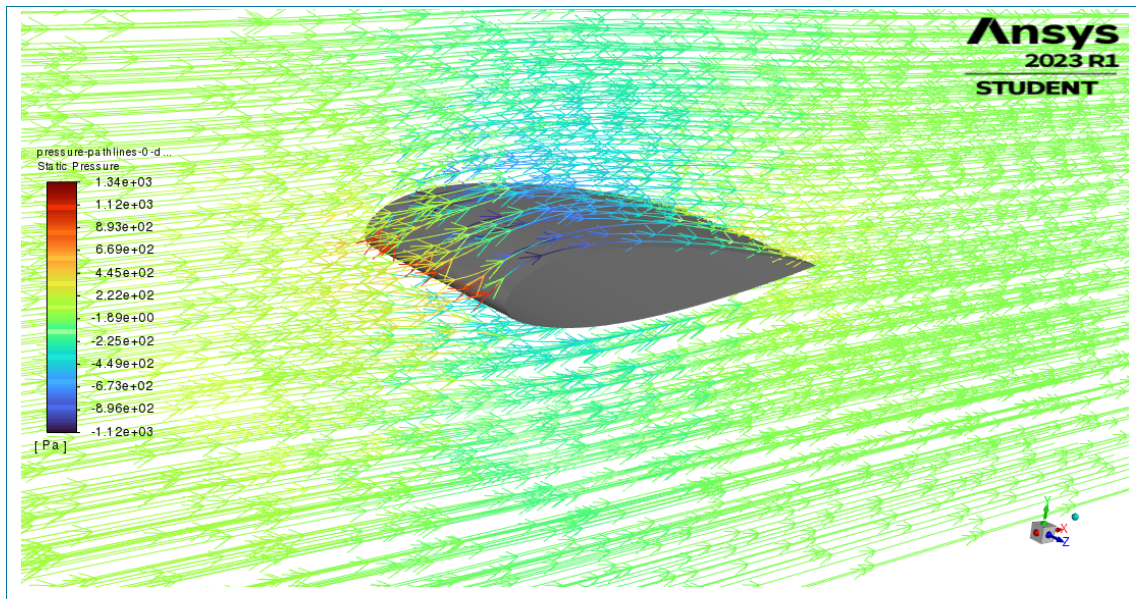
## PATHLINE SETTINGS

- To view in 3D, we can use the same settings, only this time leave the plane 4 and selecting all surfaces. We can also use the path lines options under the contour section. Use the following settings to see variation of path lines.

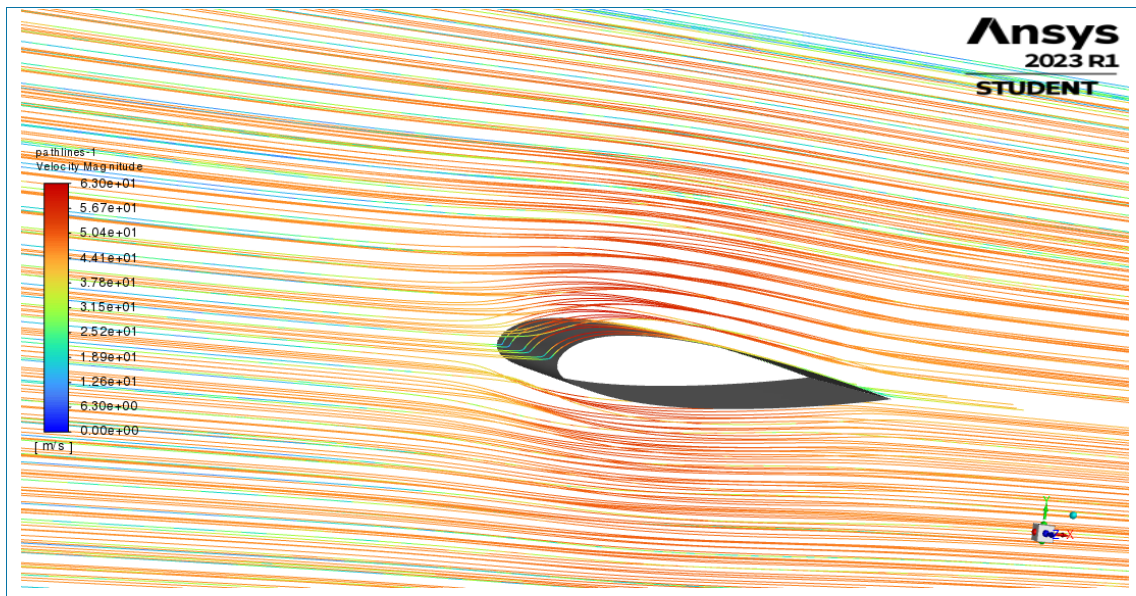




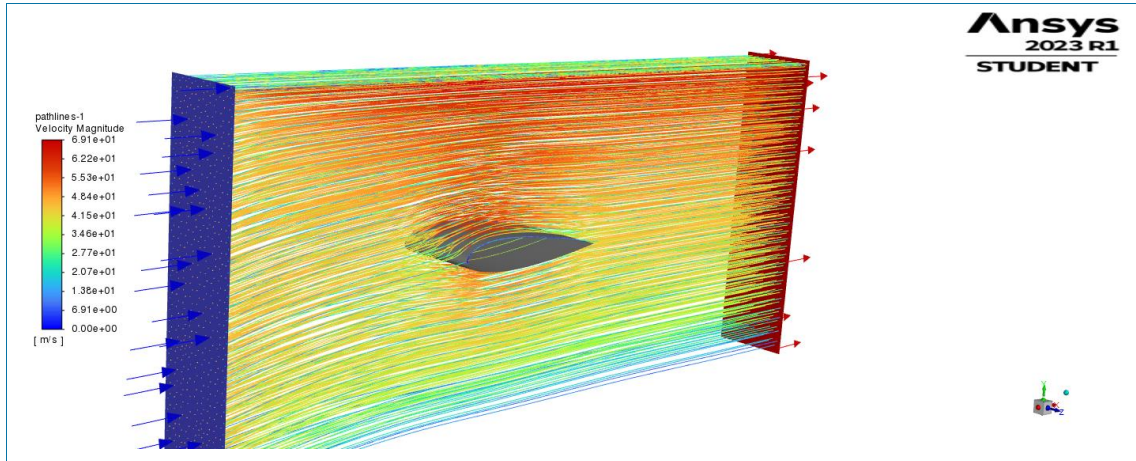
## PRESSURE CONTOUR PATHLINES (0-degrees)



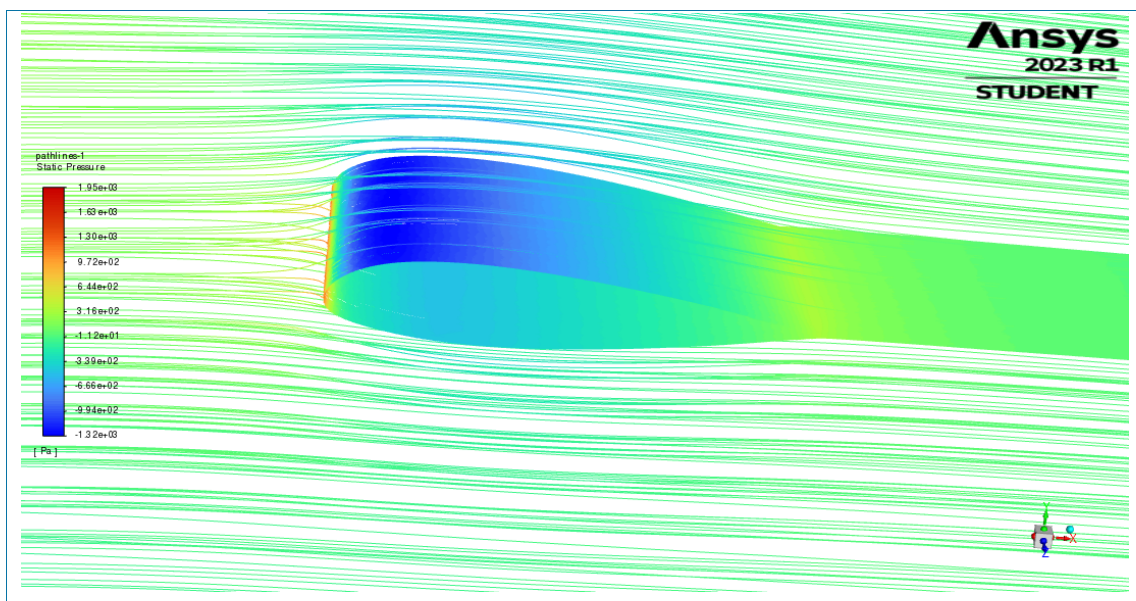
## VELOCITY CONTOUR PATHLINES (15-degrees )

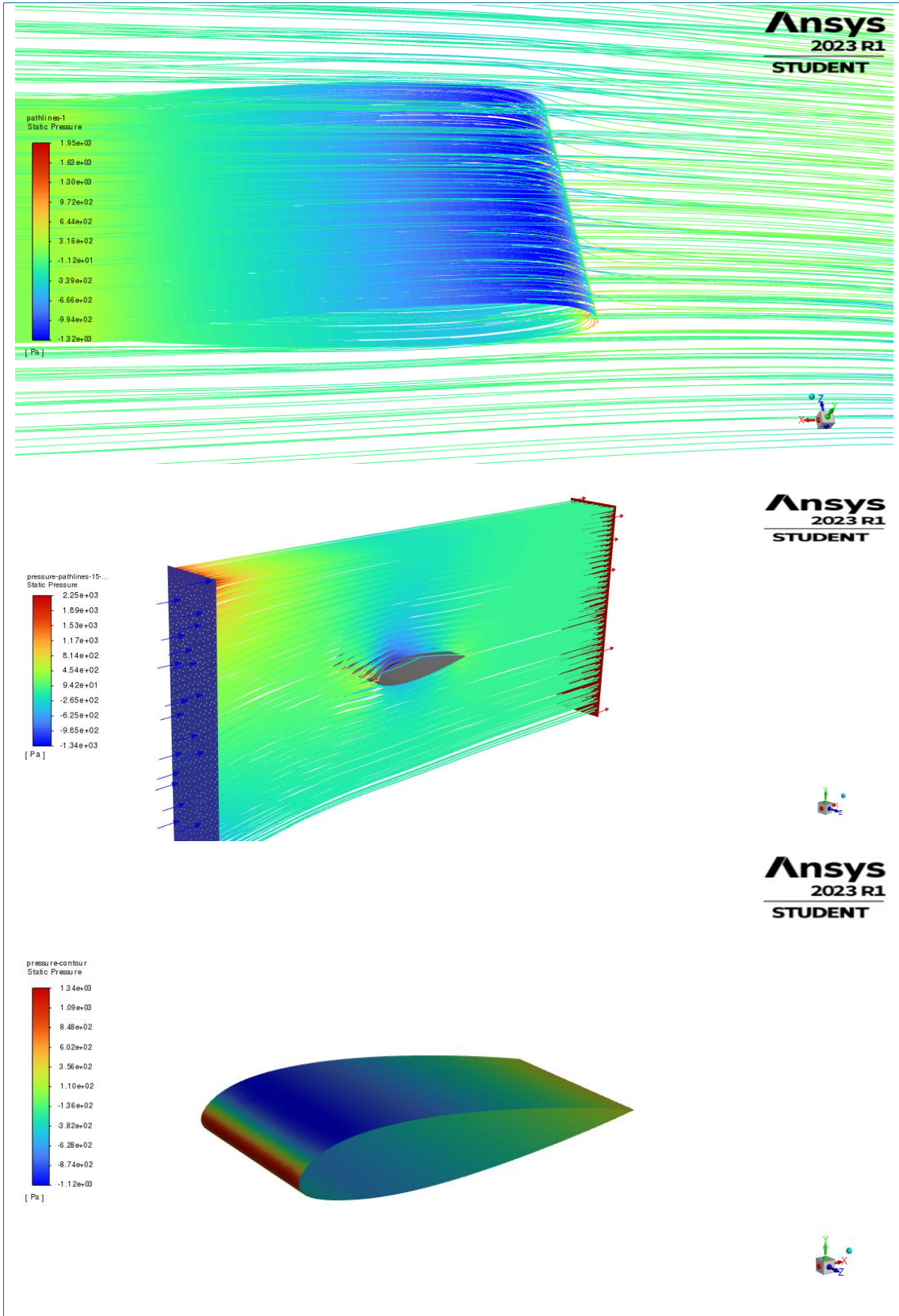






## PRESSURE CONTOUR PATHLINES (15 degrees)





## CONCLUSIONS AND RECOMMENDATIONS

- **Stall Prevention and Control:** The effect of stall can have disastrous effects on the aircraft, rendering the pilot unable to regain control until the lift is generated again. The NACA-23018 stalls at an angle of 15 degrees. It is crucial to keep the NACA-23018 under 15-degree angle of attack regardless of the incoming air velocity. Pilots should be aware of this limitation and ensure that the aerofoil remains within the recommended angle of attack range during operation.
- **Considering Flap Deployment:** It is crucial to remember situations in which the air velocity is insufficient, can also result in a stall. In such conditions (usually found during the landing of the airborne vehicle), Flaps mechanisms can be initiated to generate additional lift.
- **Investigate Morphing Wing Technology:** Ongoing research is focused on developing innovative mechanisms to delay the stall location and improve lift efficiency. One such technology is the concept of a morphing wing, inspired by the adaptability of wildlife birds. A morphing wing can change its shape during flight to optimize lift generation and control. Further exploration of this technology, its feasibility, and its integration into aircraft design

-----END-----