

Ansys Software Tutorial

Introduction to Ansys Fluent #4: Aerodynamics Analysis Part 1

Mitchell Boots

University of Newcastle

Edited by the Ansys Academic Development Team

education@ansys.com

Summary

Ansys Fluent is a comprehensive computational fluid dynamics (CFD) software that allows you to model fluid domains.

In this set of tutorials, we will introduce basic functionalities of Ansys Fluent through the Ansys Workbench interface. Ansys Workbench is the integration and workflow platform that connects Ansys products.

This tutorial will cover some modeling techniques to determine lift and drag on given objects. We'll cover how to import geometry, set up symmetry boundary conditions, and sweeps.

This tutorial is #4 of a seven-part tutorial series that serves as an introduction to Ansys Fluent. Details of the topics covered and the order can be found in the table below. These tutorials build on one another, so it is recommended that they are followed in order. Other tutorials can be found on the [Ansys Education Resources site](#).

Tutorial Order	Tutorial Topic
1	Introduction to Ansys Fluent
2	Mesh Sensitivity
3	Steady State vs. Transient
4	Aerodynamic Analysis Part 1
5	Aerodynamic Analysis Part 2
6	Heat Transfer with Ansys Fluent
7	Compressible Flows

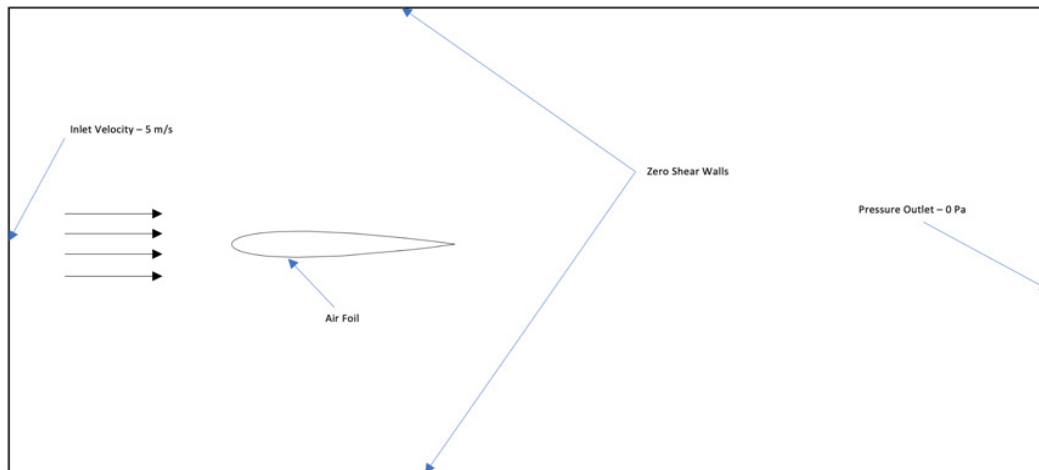
*This tutorial was created using the 2023R1 Student Version of Ansys Workbench. Some screens may look different, depending on your version. Check the [Ansys Learning Forum](#) if you have any questions.

Table of Contents

Exercise 1: 2D Turbulent Flow over an Air foil – Steady State:	3
Exercise 2: 3D Turbulent Flow over an infinite Air foil:	11

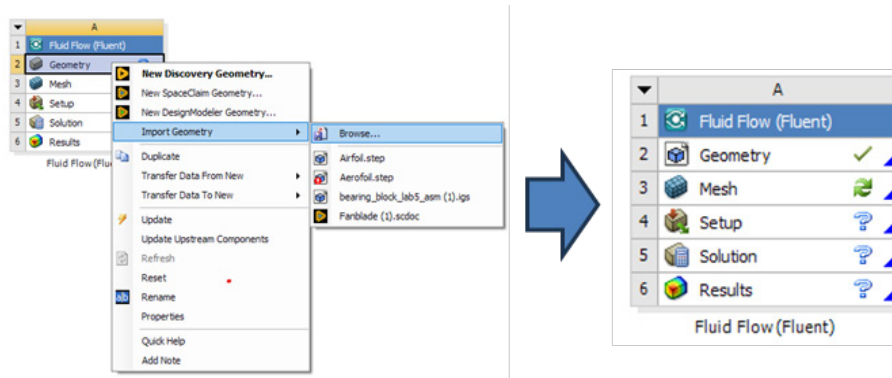
In this tutorial, we will be running through some modeling techniques to determine lift and drag on given objects. We'll cover how to import geometry, set up symmetry boundary conditions, and sweeps. You will need to use two CAD files named '2D-Airfoil.STEP' and '3D-Airfoil.STEP'. Please download both files and place them in a suitable folder.

The following image depicts a diagram of the setup we will be testing in today's lab.



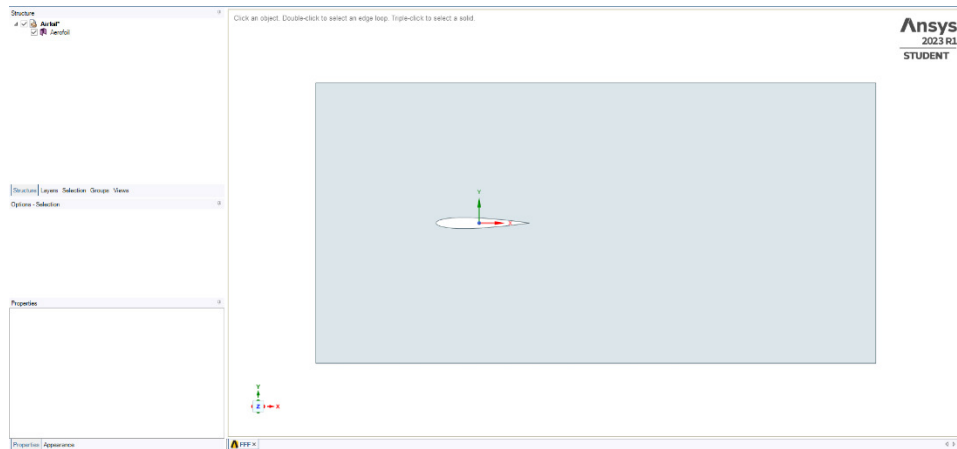
Exercise 1: 2D Turbulent Flow over an Air foil – Steady State:

Start a new Ansys Workbench project and load in Fluid Flow (Fluent). Load in the geometry file given by right-clicking on the dropdown arrow and selecting 'Import Geometry'. Then, choose the '2D-Airfoil.STEP' file provided.

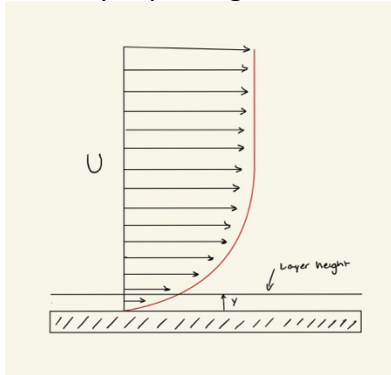


Once the geometry has loaded in, open the geometry in SpaceClaim¹. You should see the following surface loaded.

¹ Ansys Discovery is our go forward CAD tool, providing an improved user experience, it can be used to handle CAD instead of SpaceClaim



When you've confirmed that the geometry has loaded correctly, proceed to the 'Mesh' section. In our lab during week two, 'yplus' was introduced. Since we are working on our first turbulence model, let's examine 'yplus' to ensure we are accurately capturing the viscous layer gradient at the boundary.



$$y^+ = \frac{\rho u y}{\mu}$$

Where:

- y^+ is the dimensionless distance from the wall.
- ρ is the fluid density.
- u is the velocity of the fluid at the cell face adjacent to the wall.
- y is the distance from the cell center to the wall.
- μ is the dynamic viscosity of the fluid.

If we want to model the sharp gradient around the surface of a wall, we need to use a metric to determine how small our first couple of boundary layers need to be to accurately capture this steep gradient. Typically, a y^+ value of 5 or below is considered good, although it may vary depending on your specific simulation.

Example:

$$U = 5 \text{ m/s}$$

$$\rho = 998 \text{ kg/m}^3$$

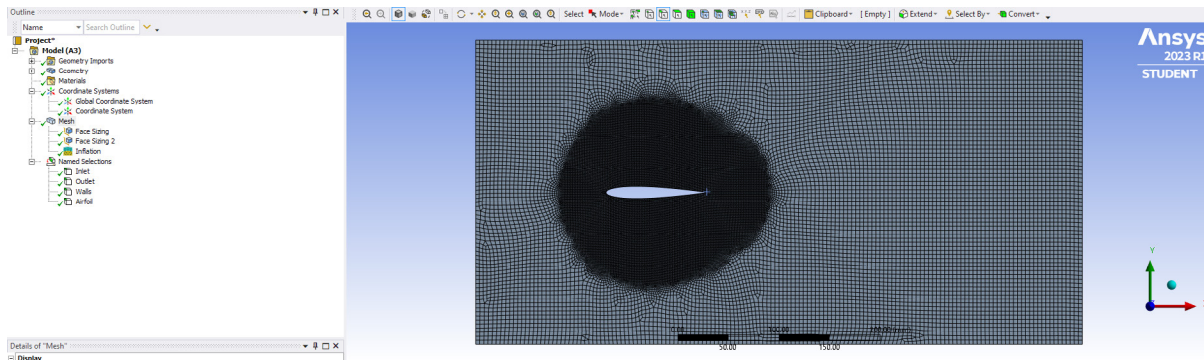
$$\mu = 0.001003$$

Using the Yplus formula to select our first layer thickness. $Y^+ < 5$

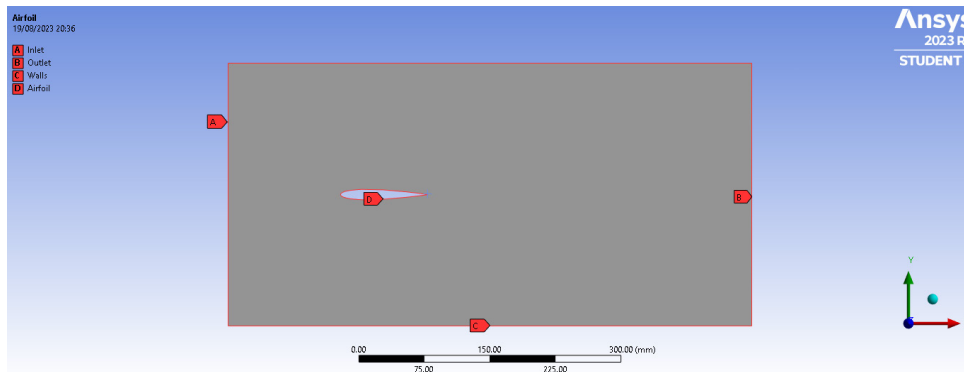
$$y = \frac{Y^+ \mu}{\rho u} = \frac{5 * 0.001003}{998 * 5} \approx 1e-6 \text{ m}$$

Thus, we would need a first layer height of $1e-6 \text{ m}$. Now calculate y based on the conditions of exercise 1.

Generate a mesh with appropriate settings to accurately model the wall boundary. Below is an example of the mesh configuration.



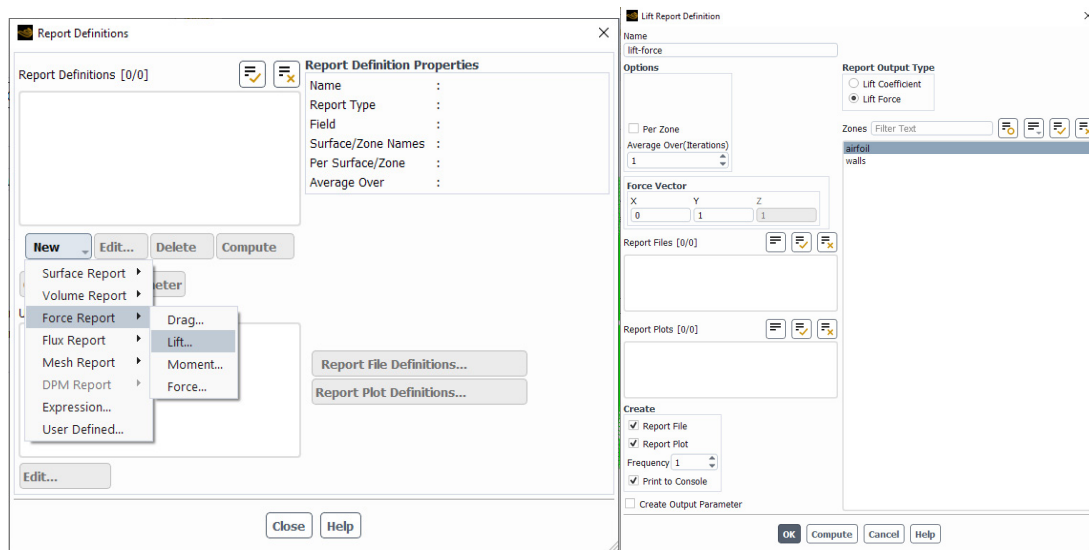
Before exiting the meshing section, make sure you've finished setting up the named selections. Assign names to the inlet, outlet, walls, and air foil.



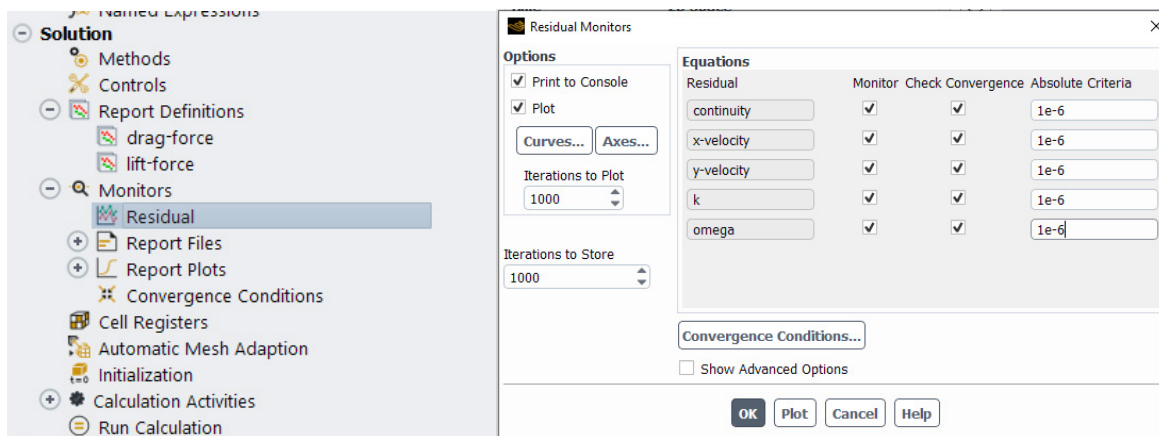
Once you have finished with the mesh tab move over to setup/solution and set the model and boundary conditions as follows:

1. Set the model to k omega SST.
2. Set the cell zone to 'air'.
3. Set the Inlet velocity to 5 m/s.
4. Set the walls to zero shear.

As done in the second tutorial, create 2 report definitions one for lift force and the other for drag, print both to the console and both to have plots.

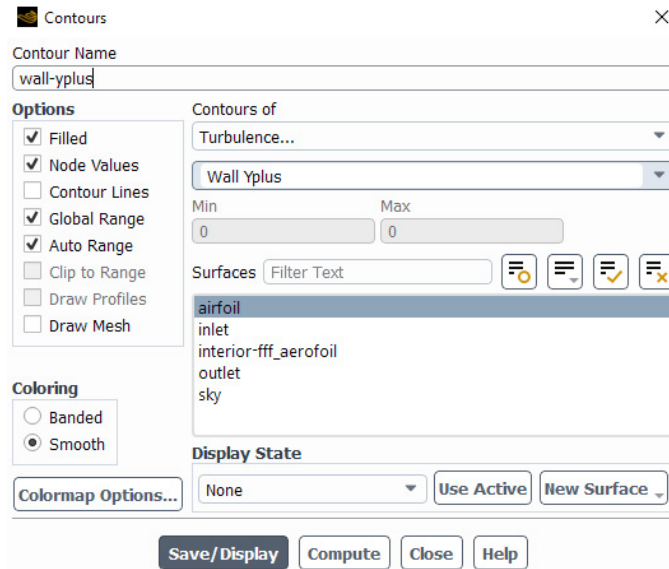


Before we continue, we will modify the residual convergence criteria to achieve greater convergence. Click on the monitor dropdown arrow and select residuals. Set all the residuals to $1e-6$.

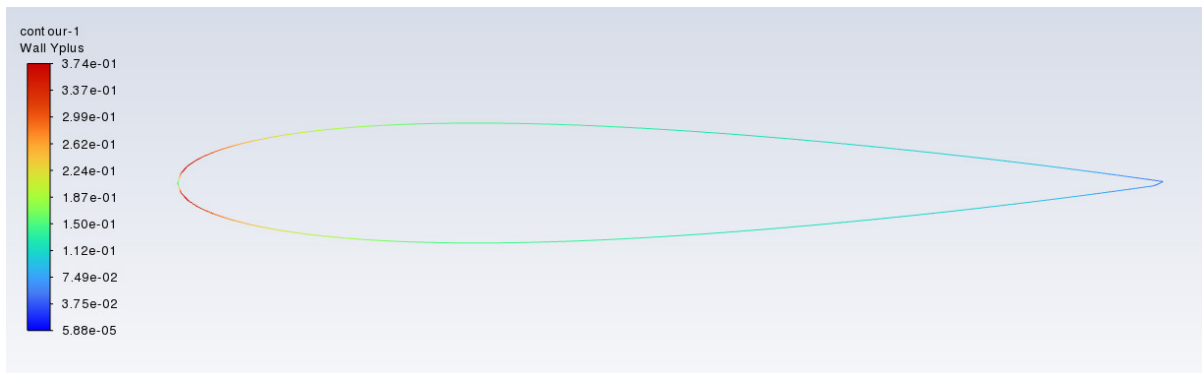


Initialize the simulation and run the calculation for 250 iterations.

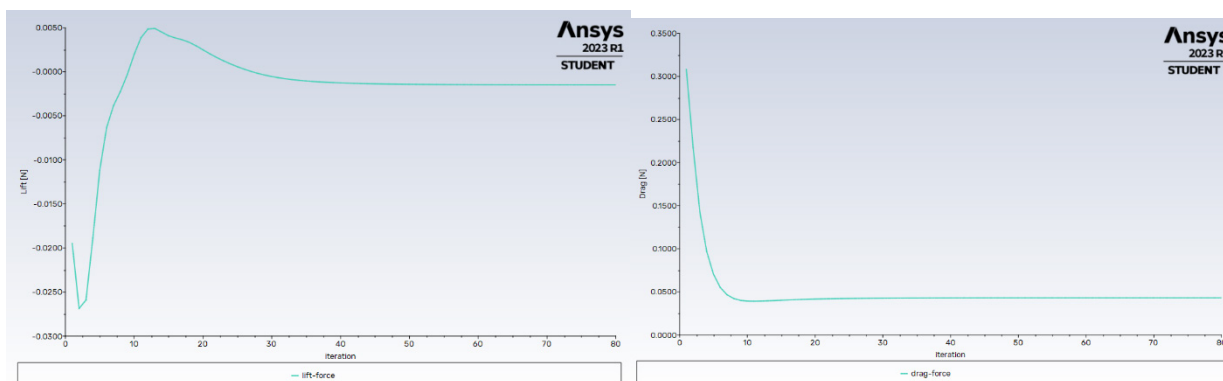
After the simulation has converged, this is the stage where you can assess the yplus values to determine if any mesh adjustments are necessary. This can be achieved by inserting a contour plot along the airfoil's edge for turbulence wall yplus.



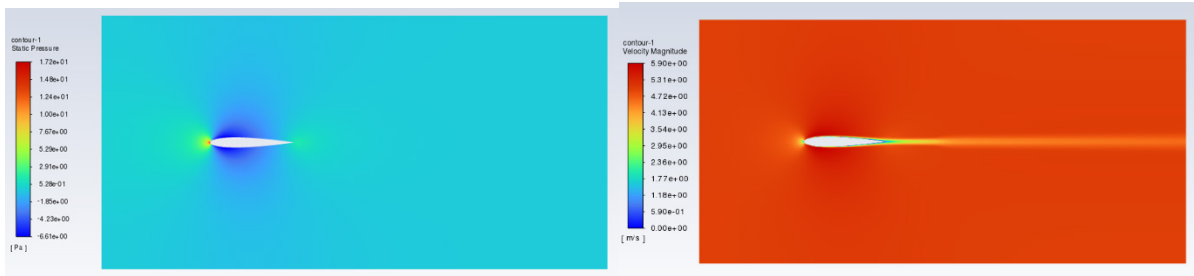
Once Inserted, analyzing the yplus around the air foil boundary helps identify potential areas requiring adjustments. While simpler geometries like a basic air foil may pose fewer issues, as geometries become more complex, thorough yplus analysis becomes crucial.



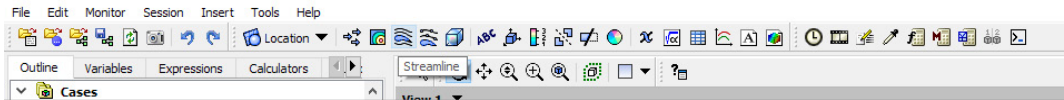
After you are satisfied with your yplus values, analyze the lift and drag force plots. Do the results seem reasonable, and can you explain why the forces exhibit fluctuations?



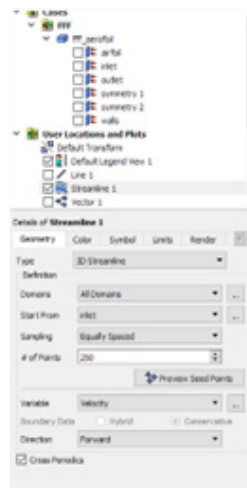
Generate contour plots for pressure and velocity distribution, identify stagnation points, and examine the wake behind the body.



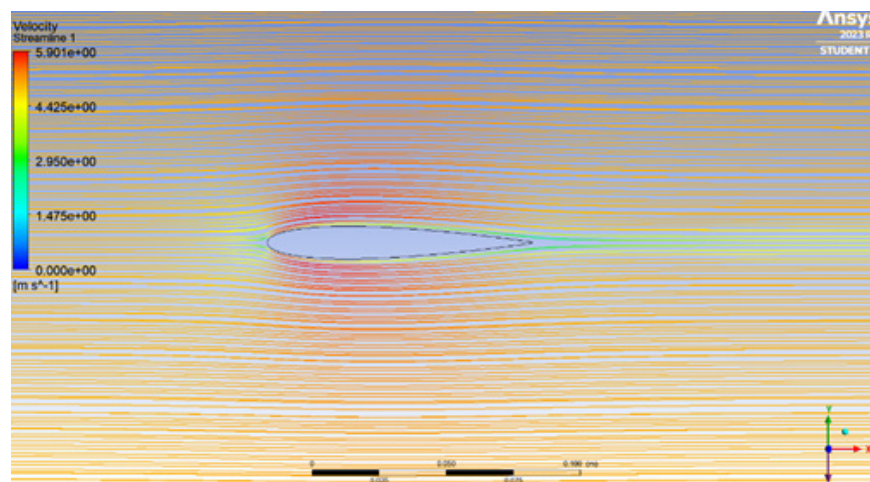
Now, let's generate a streamline visualization. Move to CFD Post. After loading the interface, click on the streamline icon.



After clicking on the streamline icon, set the streamlines to originate from the inlet. Adjust the number of points to 250.

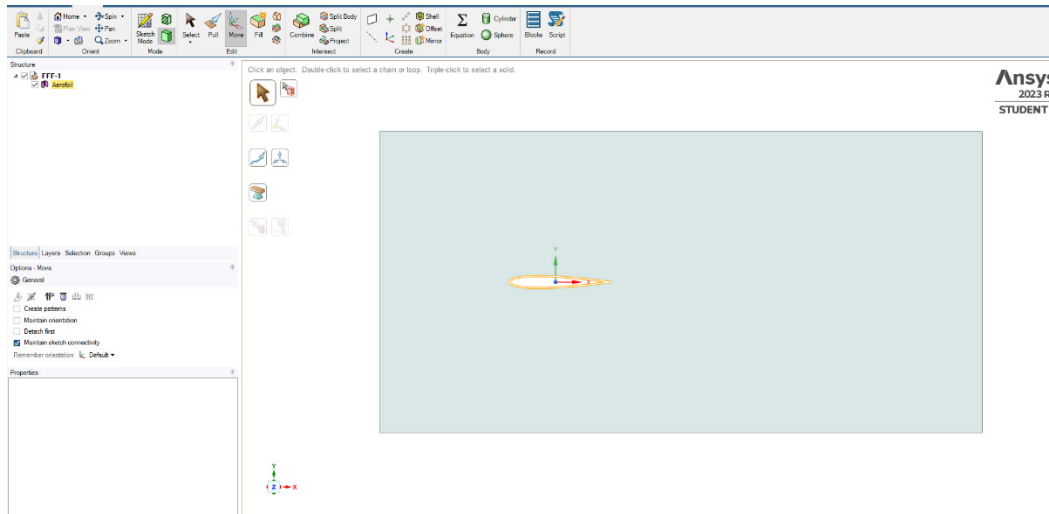


Observe the flow arrows around the air foil. Zoom in closely to examine the streamlines.

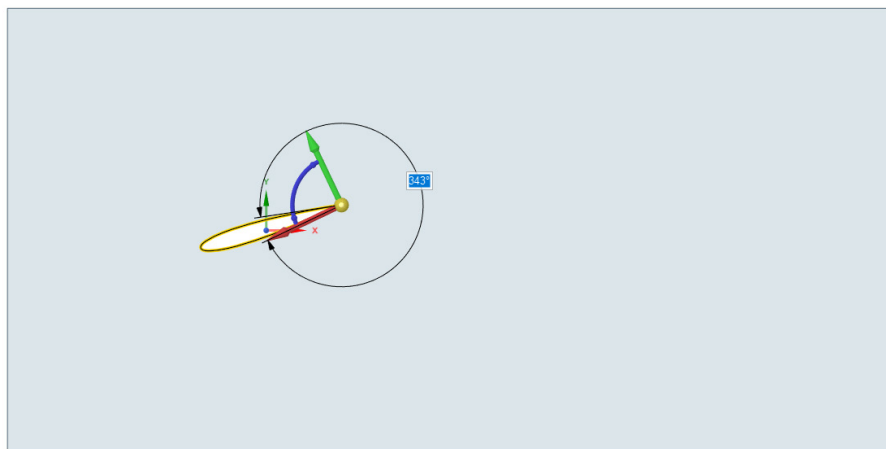


Now, let's adjust the Angle of Attack of the air foil to change the lift-to-drag ratio.

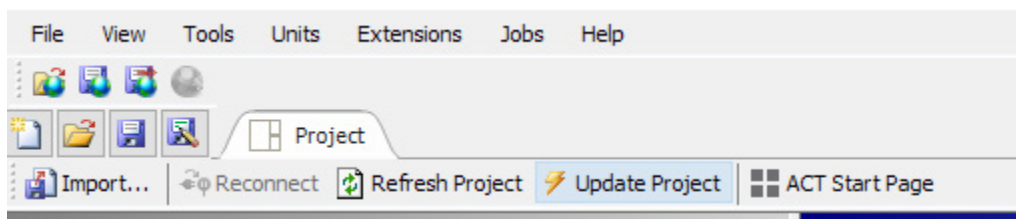
Duplicate your simulation in the Workbench and reopen SpaceClaim. After the geometry has loaded, employ the move tool and select the edge of the air foil.



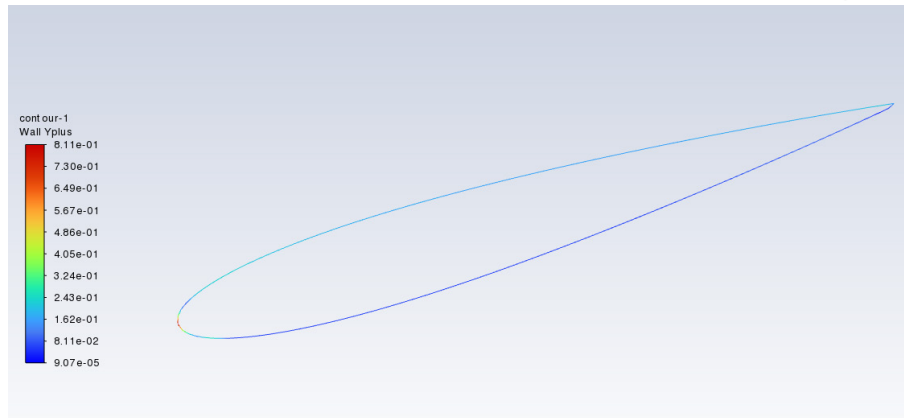
Change the angle by approximately 17 degrees, ensuring that the rotation is centered around the origin, as depicted in the following image.



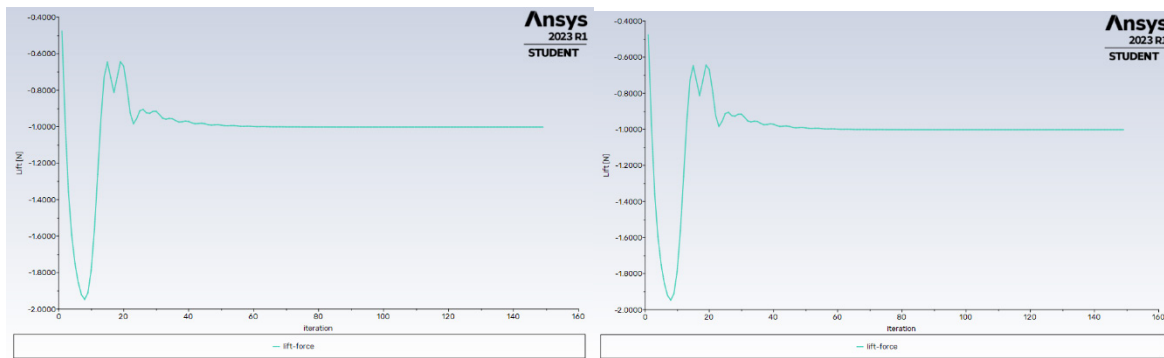
Close SpaceClaim. At the top of the 'Project Schematic,' click the 'Update Project' button. This action will re-run the simulation according to the adjusted parameters.



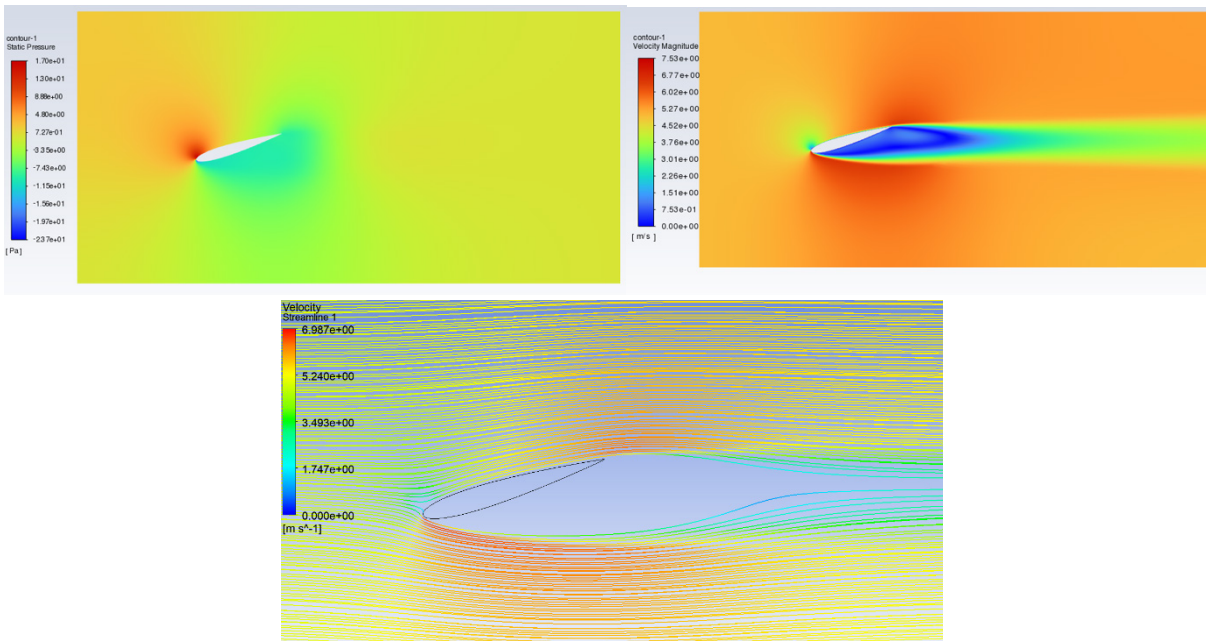
Double-check that your yplus is suitable.



Open Fluent once more and examine the updated lift and drag values.



Inspect the contour plots and generate another streamline visualization.

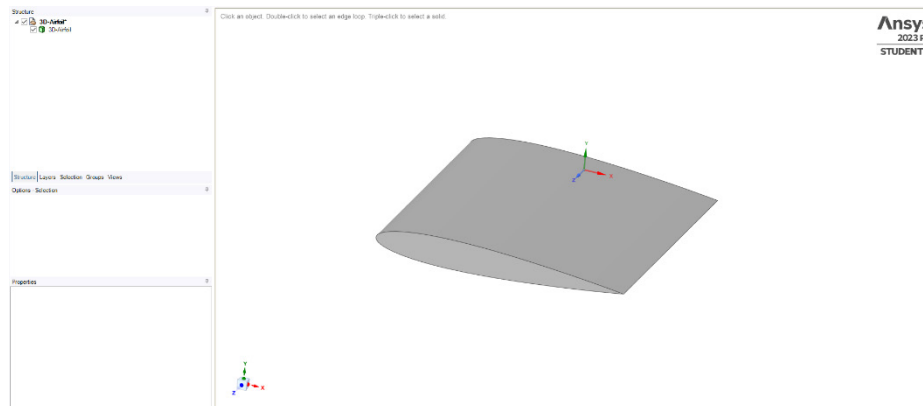


Is the flow noticeably different? How have the lift and drag forces changed? What observations can you make about the wake?

Exercise 2: 3D Turbulent Flow over an infinite Air foil:

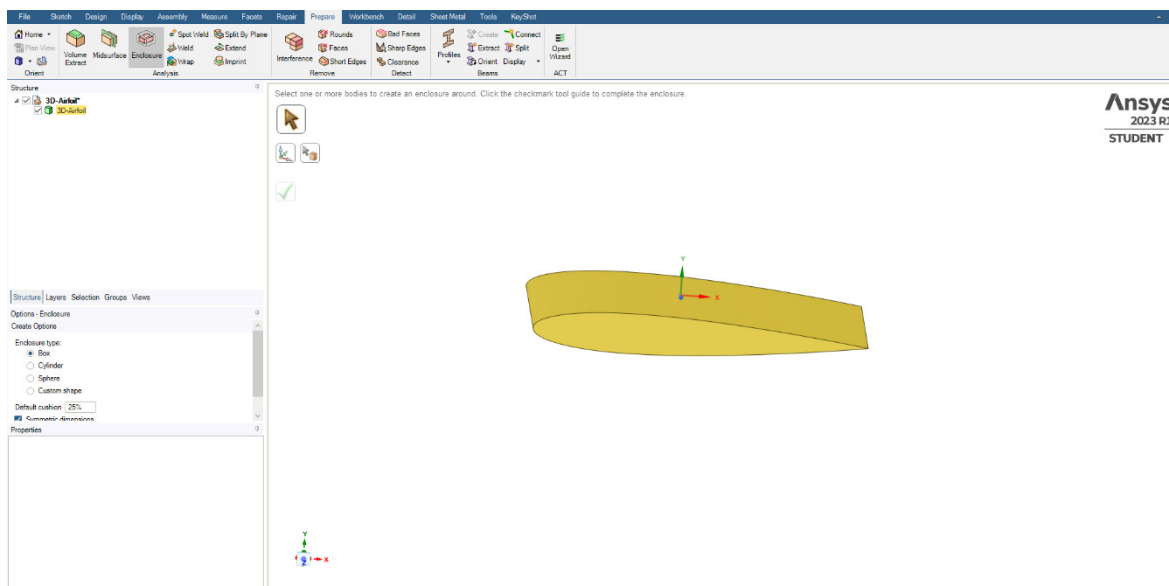
Add a new Fluid Flow (Fluent) analysis to the Workbench and import the other provided CAD document, '3D-Airfoil.STEP'. We are now going to conduct a 3D analysis.

Once it's loaded in the 'Geometry' tab, open SpaceClaim. You should see the following geometry loaded.

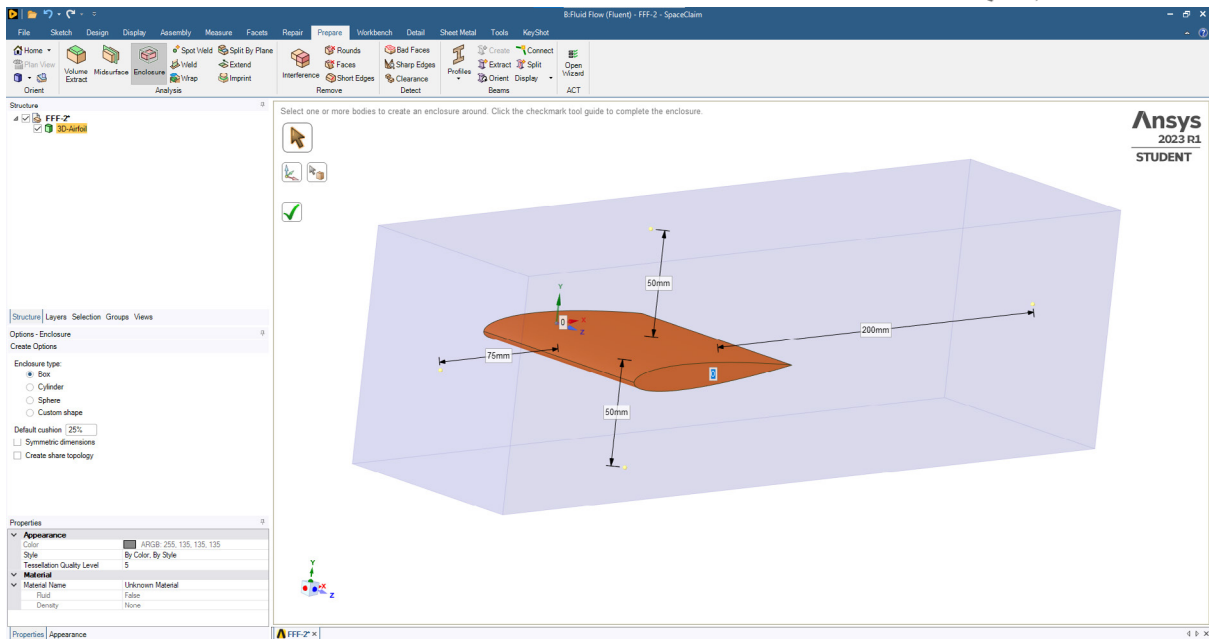


Usually, it's a good practice to conduct a quick check on the quality of the provided geometry. In an upcoming lab, we will cover techniques for repairing and fixing geometry.

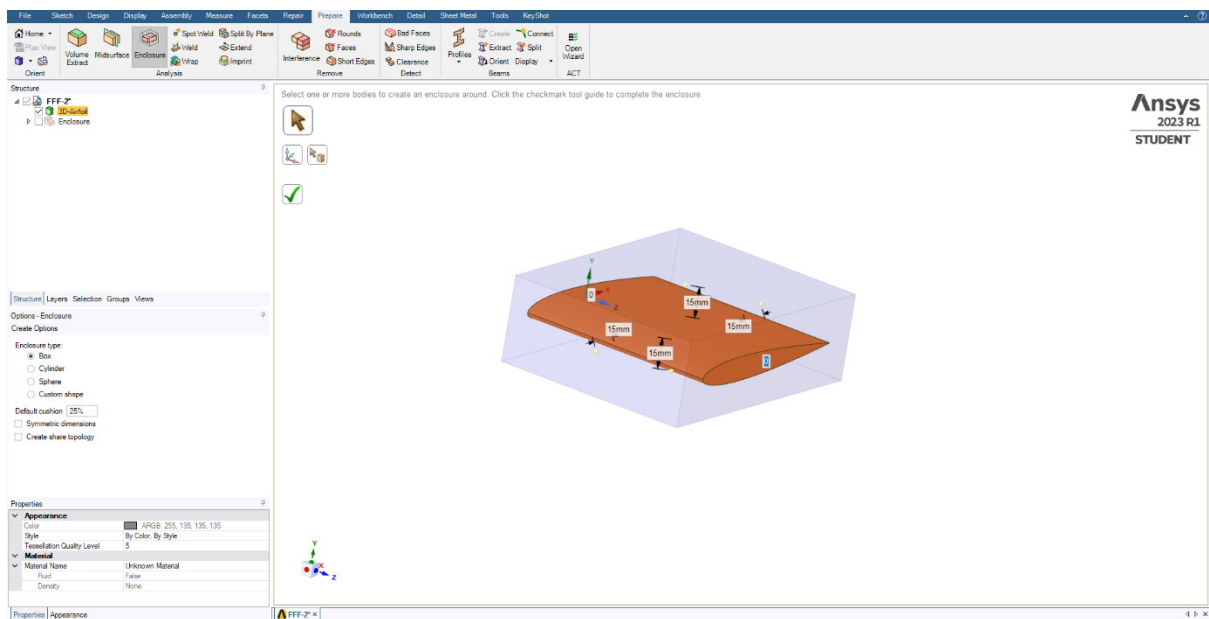
For now, let's create an enclosure around the air foil profile. Navigate to Prepare -> Enclosure and select the air foil surface. After selecting it, click the green checkmark to confirm.



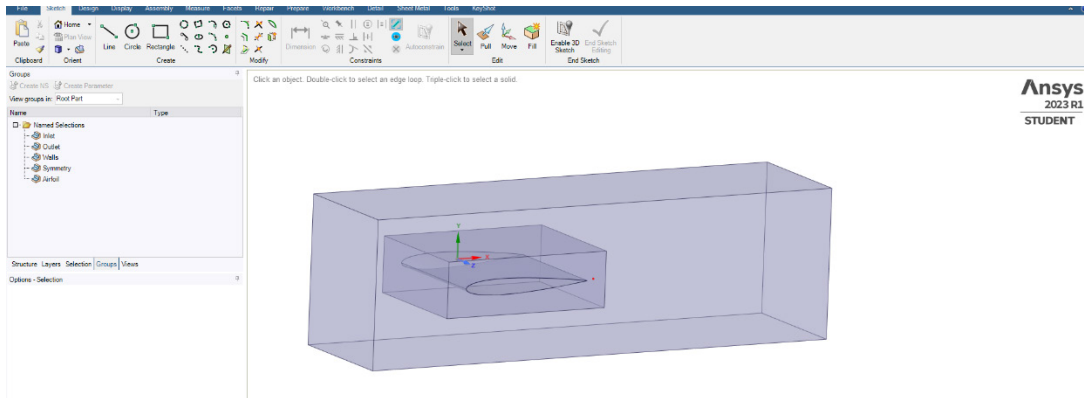
On the left side, you should see several options. Disable 'Symmetric Dimensions' and set the dimensions as follows:



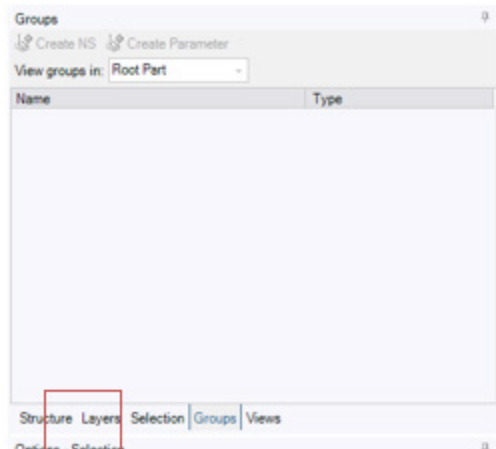
Click the green checkmark to confirm, and then hide 'Enclosure' by clicking on the checkmark in the model tree. Now, let's create another smaller enclosure for a 'body of influence'. Create this new enclosure with the following dimensions.



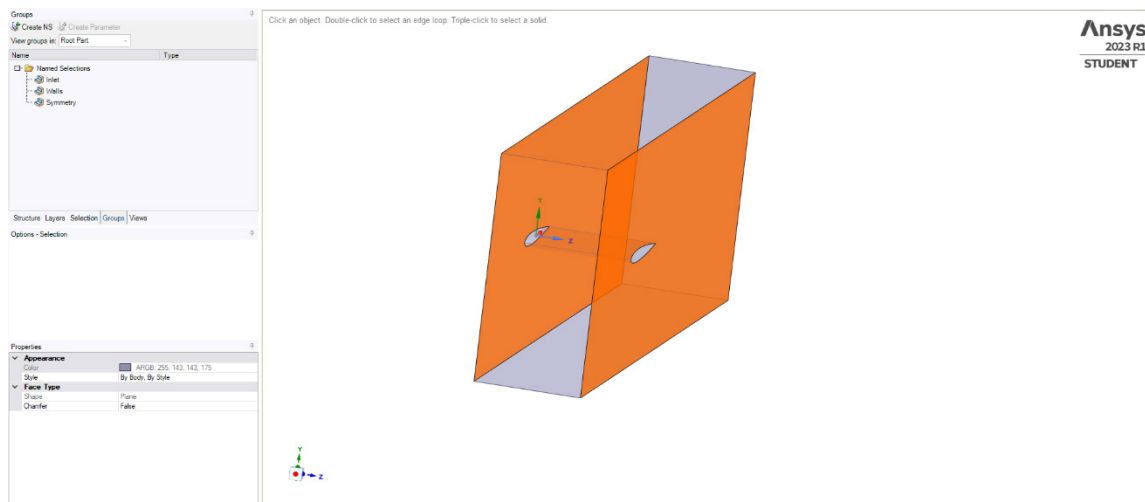
With the two enclosures in place, let's hide and suppress the airfoil from the physics analysis. After completing these steps, your setup should resemble the following image.



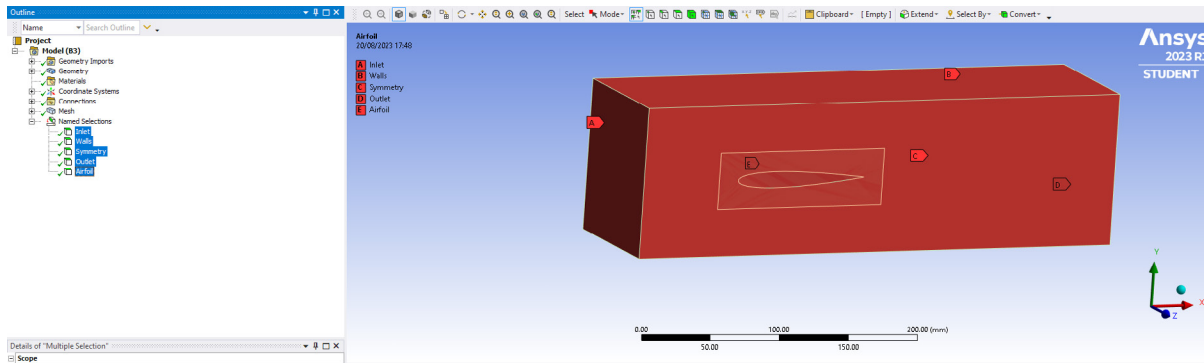
Next, we will create our named selections. Click on 'Groups' below the model tree.



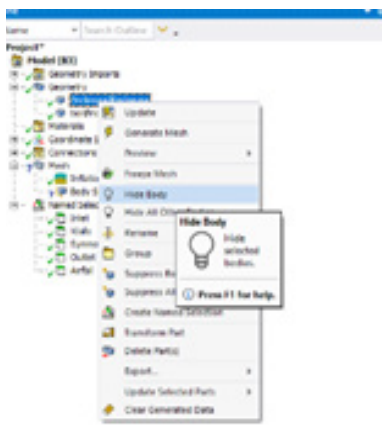
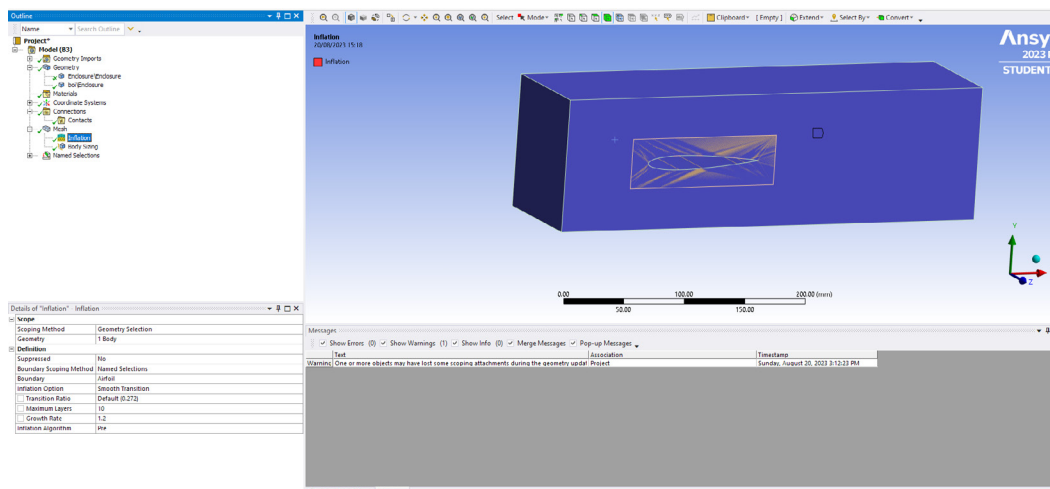
To establish a named selection, hold the Ctrl key and click on the surfaces you want to assign a name to. After selecting the desired surfaces, click on 'Create NS'.



Name each of the boundary conditions as follows: wall, airfoil, inlet, outlet, and symmetry. To simulate an infinite airfoil, set the faces at the end of the airfoil to a symmetry condition, as shown in the image above. After naming all the sections, move to the meshing section and ensure they have loaded in correctly.



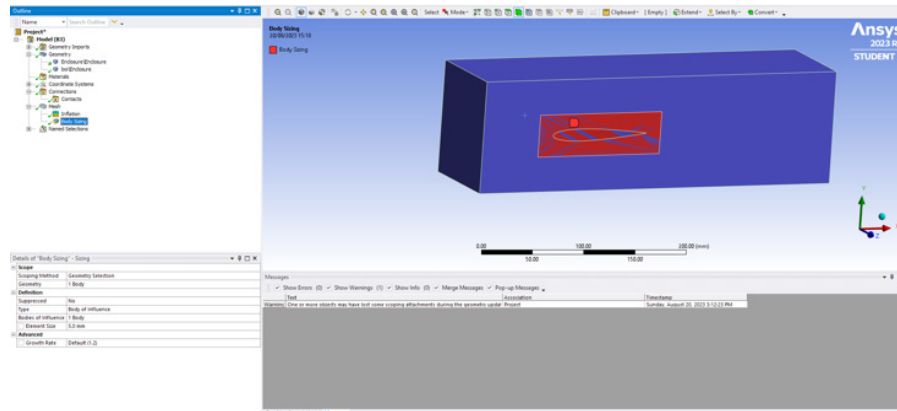
Set the global element size to 10mm and proceed to create an inflation layer. Select the geometry as the main body, and boundary to the named selection 'airfoil'. Keep the inflation method as 'smooth transition' and set the number of layers to 10.



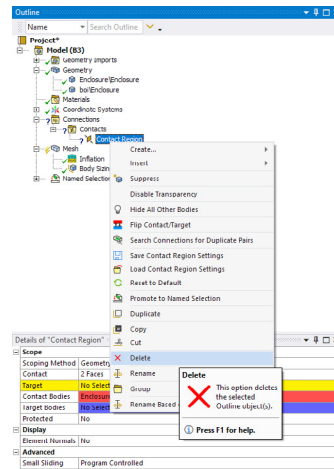
Now, let's create a body of influence. Insert a 'Sizing' into the mesh tree. Choose the largest body as the scoped geometry. Under 'Type', select 'Body of Influence'. Here, we will utilize the smaller box we created in SpaceClaim.

To proceed, you'll need to hide the larger body. Click on the 'Geometry' dropdown menu, then right-click on the larger body and select 'Hide'.

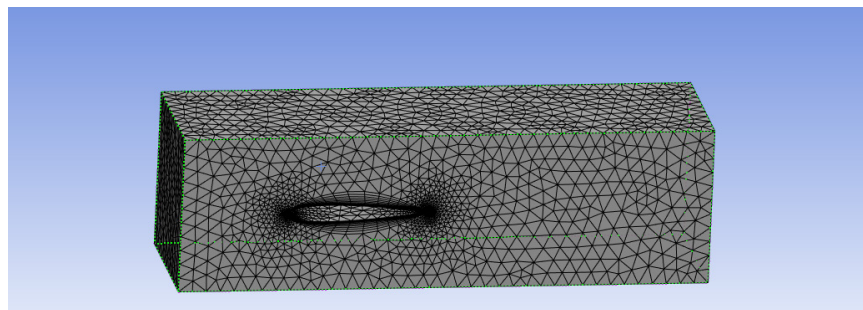
Next, return to 'Body Sizing' and choose the smaller body as the body of influence. Set the element size to 5mm. This informs Ansys that we want a 5mm element size within the box. Configure it according to the image provided. Don't forget to display the larger enclosure again.



When performing this step, if a bonded contact is generated, you'll need to delete it. Navigate to 'Connections' -> 'Contacts', then right-click and delete the corresponding region. If no contact has been generated, you can simply proceed to the next step.

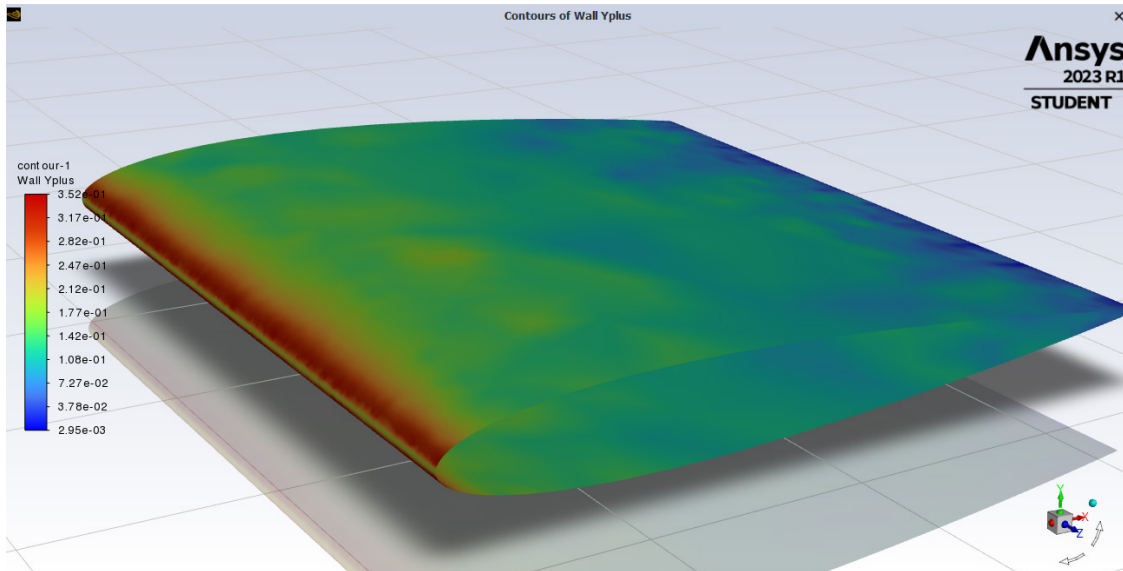


Generate the mesh. It should resemble the following image. While the mesh is relatively coarse and could be improved, we will proceed with this mesh due to computational limitations and cell count constraints.

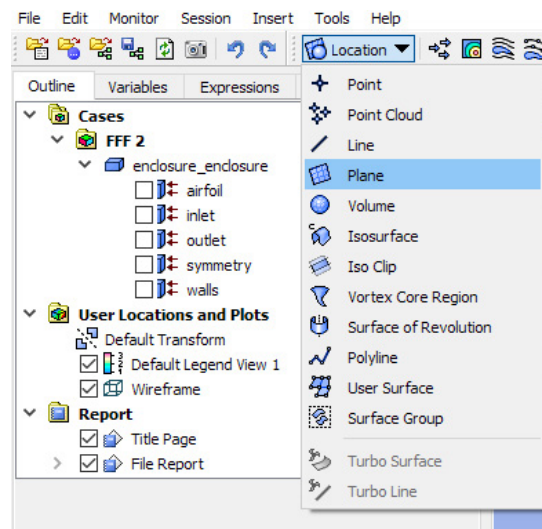


Once the mesh has generated, this will take some time. Update mesh and move over to fluent. The symmetry boundary condition will be automatically loaded if named correctly, Set the rest of the solution up with the same conditions as before.

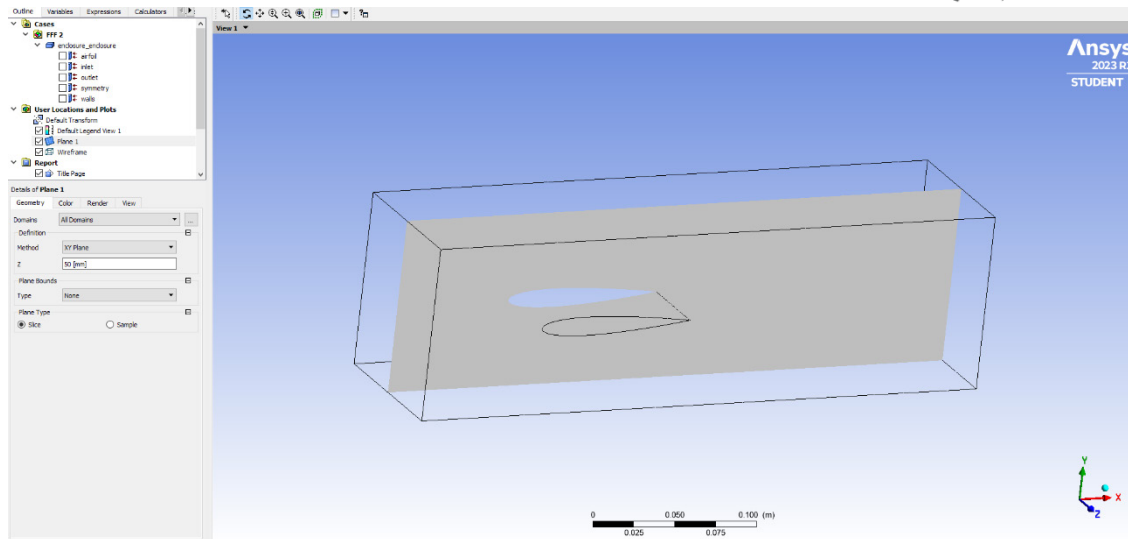
1. Set the model to k omega SST.
2. Set the cell zone to 'air'.
3. Set the Inlet velocity to 5 m/s.
4. Set the walls to zero shear.
5. Initialize the simulation and run over 250 iterations. This will take significantly more time than our previous 2D simulations.
6. Once completed verify that you have an adequate yplus.



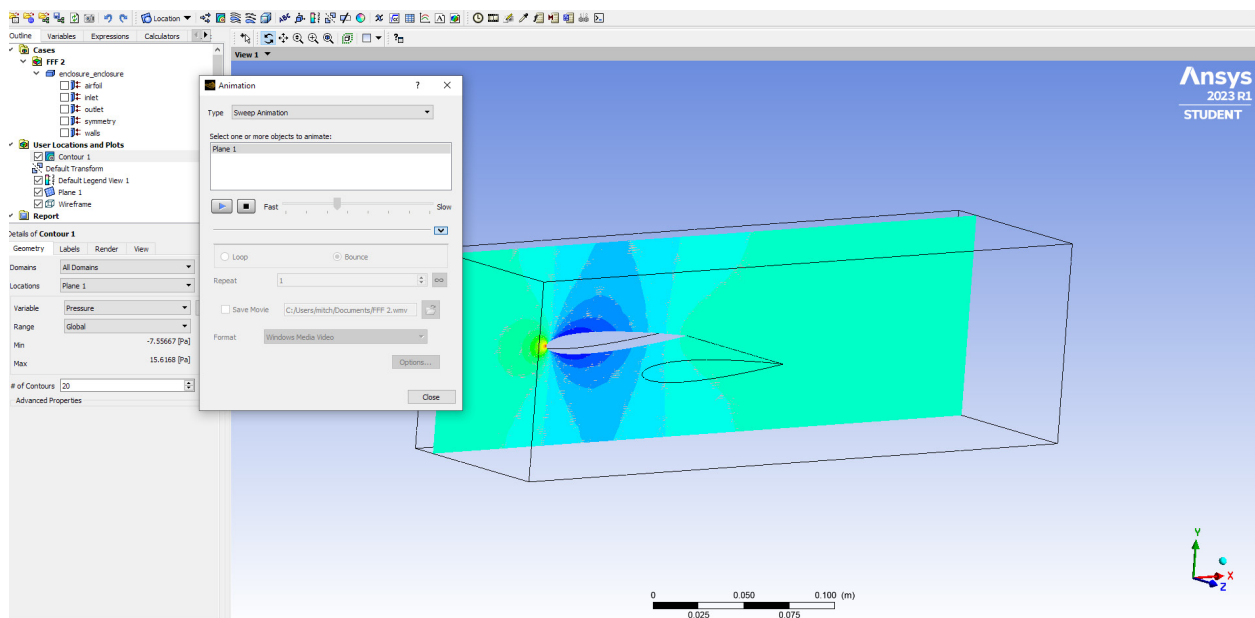
Once completed move over to CFD post. When CFD post has loaded create a plane for us to put a contour on. Location -> Plane.



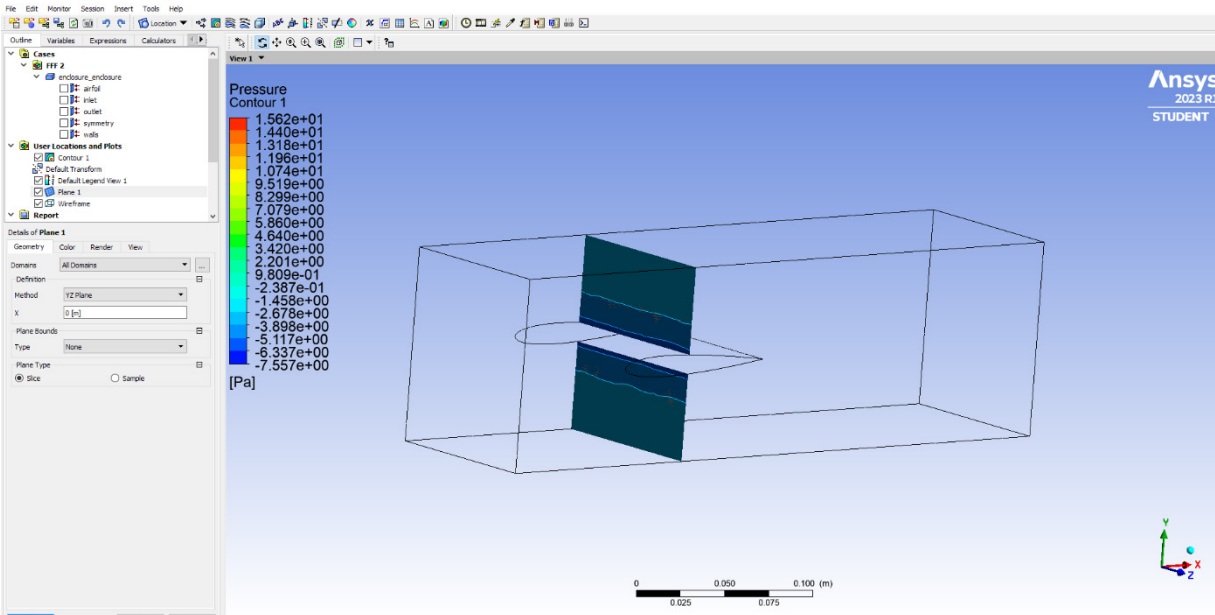
Select 'Plane' and place a plane on the XY-plane. Position the plane at Z = 50mm, positioned halfway across the airfoil.



Now, generate a contour plot on the plane we just created. When setting up the contour, choose the animation button that we used to record our transient simulation last week. Instead of creating a film, we'll make a sweep. When you click on this option, it should resemble the following image.



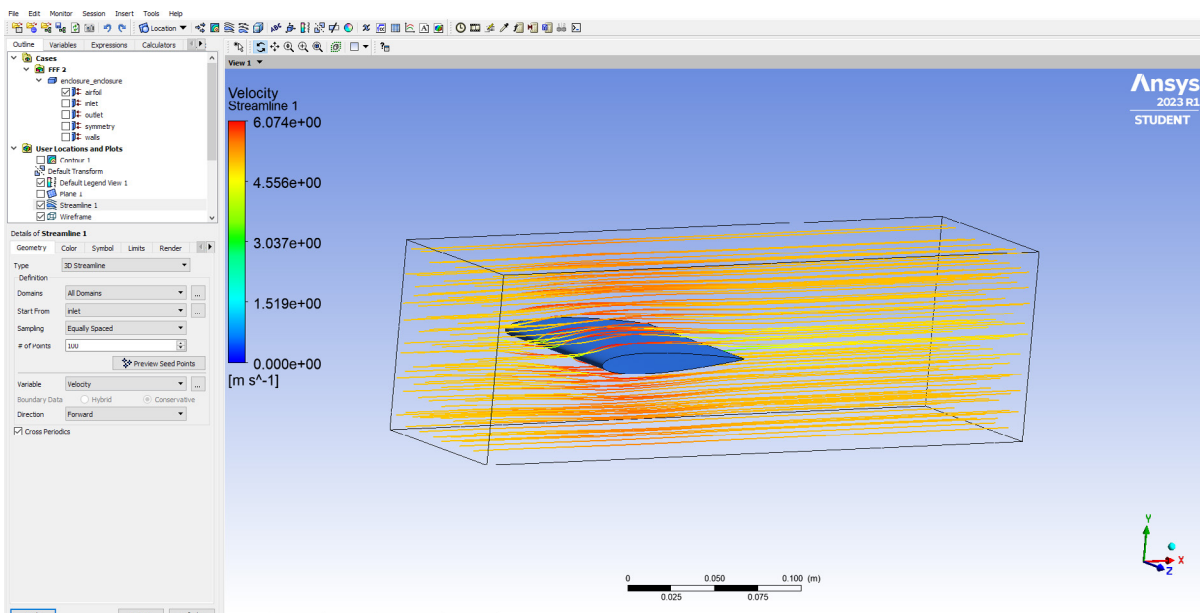
Click the play button, and you should observe the plane moving across the Z-axis. Now, let's repeat the same process on the XZ plane. Right-click and edit the first plane you created, then change it to an XZ plane.



Sweeps are an effective method for visualizing contours along a specific axis.

Set up a streamline starting from the inlet and configure it with 100 points. This should result in a plot resembling the one shown in the image.

Additionally, you can visualize the airfoil by clicking the checkbox next to the airfoil named selection, as seen in the image below.



Is the pressure and velocity distribution the same as in 2D? How can we optimize this?

© 2024 ANSYS, Inc. All rights reserved.

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information

This case study is part of a set of teaching resources to help introduce students to topics focused on structures and structural simulations.

Ansyes Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, MicroProjects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

If you notice any errors in this resource or need to get in contact with the authors, please email us at education@ansys.com.

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit www.ansys.com for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2024 ANSYS, Inc. All Rights Reserved.