Flow Over an Airfoil: Ansys Solution Outline

Pre-analysis

- See video

Geometry

We start by defining the domain of the boundary value problem. This is the fluid region around the airfoil.

- Download the NACA0012.txt file to a useful location
- Re-name Fluent Analysis: “NACA 0012 Flow”
- In Workbench, Right-Click Geometry > Properties > Analysis Type > 2D
- Double-Click Geometry to open Discovery
- Click the Menu icon in the top left > Units and Display Precision > Length: m, Grid Spacing: .1 m
- Click the Menu icon again > Insert Geometry > Navigate to the NACA0012.txt file and select it > Open

This uses the airfoil coordinates to create a curve in Discovery. We can now define the boundaries for the fluid region around the airfoil

- Select 3-Point Arc Tool > Click the Y-Axis above the airfoil > Click the Y-Axis below the airfoil > Drag cursor so the center of the arc is on the Y-Axis > Left Click
- Select Constraints in the top bar > Coincident > Click the point at the center of the arc > Click the Origin to adjust the arc
- Click the Line Tool > Click on the top point of the arc > Move cursor to the right create a horizontal line (Right-angle marker should appear at the left end) > Left Click to create a line
- Move cursor downward so it is aligned with the bottom point in the arc (Right-angle marker should appear at the top end, and marker should appear at the bottom end to show you are aligned with the arc) > Left Click to create another line
- Move cursor to the bottom point on the arc > Left Click to create another line
- Select the Dimension Tool > Click the Arc > Left Click to Create the Dimension
- Click the Top Horizontal Line > Left Click to Create the Dimension
- Click on the arc dimension > Change to 12.5 m
- Click on the horizontal line dimension > Change to 12.5
- Enter 3D mode to create the surface
- Select the Project Tool > Select the airfoil curve > Select the Receiving Surface Option > Click on the surface > Click check mark
- Right Click on the surface inside the airfoil curve > Click Delete

- In the tree Right-Click Sketching Plane > Exclude from Simulation > Right-Click it again > Hide
- Click on the Named Selection Tool on the Bottom Right > Create the following Named Selections:
    - Arc and Both Horizontal Edges: inlet
    - Rightmost Vertical Edge: outlet
    - Airfoil Curve: airfoil

We have defined the fluid domain over which we want to solve the governing equations.

- Close Discovery
- Save project in .wbpj project

Mesh

The domain needs to be discretized for the solver to obtain an approximate numerical solution to the BVP and calculate the velocity components and pressure at the cell centers.

- In Workbench, Double Click on Mesh Step
- Select Mesh > Generate
- Make this mesh less coarse by Selecting Mesh in the Tree > Element Size > .5 m
- Select Mesh > Update

The mesh can be refined by decreasing the element size close to the airfoil.

- Right-Click Coordinate Systems in the Tree > Insert > Coordinate System
    - Name: Airfoil Center
    - Define By: Global Coordinates
    - Origin X: .5 m
    - Origin Y: 0 m
- Select Mesh in the Tree > Select Sizing in the Top Bar
    - Select the Surface > Click Apply under Geometry in the Details Pane
    - Type: Sphere of Influence
    - Sphere Center: Airfoil Center
    - Sphere Radius: 3 m
    - Element Size: .05 m
- Select Mesh > Update

The mesh can be further refined along the edge of the airfoil.

- Select Mesh in the Tree > Sizing
    - Select Edge Selection Filter > Select Airfoil Curve > Click Apply under Geometry in the Details Pane
    - Type: Number of Divisions
    - Number of Divisions: 250

  - Capture Curvature: No
  - Behavior: Hard
- Select Mesh > Update

Finally, the mesh can be refined in the direction normal to the airfoil boundary.

- Select Mesh in the Tree > Inflation
  - Select Face Selection Filter > Select Flow Domain > Click Apply under Geometry in the Details Pane
  - Select the Edge Selection Filter > Select Airfoil Edges > Click Apply under Boundary in the Details Pane
  - Inflation Option: Total Thickness
  - Number of Layers: 10
  - Growth Rate: 1.2
  - Maximum Thickness: .01 m

- Select Mesh > Update

This creates a much finer mesh with smaller elements close to the airfoil. These elements represent control volumes; the primary unknowns will be determined directly at the centers of the control volumes.

- Exit Mesher
- Save Project

Physics Setup

In this step, we define the governing equations and boundary conditions.

- Double Click Setup in Workbench
- When the Fluent Launcher opens, select Double Precision, and change Solver Processes to match the number of CPU cores on your computer > Click Start to start Fluent
- Perform Mesh Check
- Under General, change Solver Type to Density-Based
- Double-Click Viscous > Select Inviscid > Click Ok

This specifies which equations need to be solved. Now, material properties and boundary conditions must be provided.

- Under Materials > Fluid > Double Click on air > Change Fluid Properties
  - Density: 1 kg/m^3
  - Click Change/Create

- Double Click Boundary Conditions and set the following Types:
    - inlet: velocity inlet > Change Velocity Specification Method to Components > Enter values to produce an angle of attack of six degrees
        - X-Velocity: .9945 m/s
        - Y-Velocity: .1045 m/s
    - outlet: pressure-outlet > Gauge Pressure: 0 Pa
    - airfoil: wall
- Select Operating Conditions > Ensure Operating Pressure is 101,325 Pa

Solution

Let's get the Fluent solver to determine the primary unknowns at the cell centers. The algebraic equations generated by the solver are nonlinear, so they must be solved iteratively to get the cell center values of the primary unknowns.

- Add the drag coefficient to be reported by Double Clicking Report Definitions > New > Force Report > Drag
    - Name: cd
    - Force Vector: X = .9945, Y = .1045
    - Report Output Type: Drag Coefficient
    - Zones: airfoil
    - Select Report File, Report Plot, and Print to Console
- Add the drag coefficient to be reported by Clicking New > Force Report > Lift
    - Name: cl
    - Force Vector: X = -.1045, Y = .9945
    - Report Output Type: Drag Coefficient
    - Zones: airfoil
    - Select Report File, Report Plot, and Print to Console
- Double Click Reference Values > Compute from > Inlet
- Reduce the criteria for convergence my expanding Monitors > Double Click Residual > Reduce Absolute Criteria to 1E-6 for all fields
- Provide initial cell-center guess values by Double Clicking Initialization > Standard Initialization > Compute from: inlet > Initialize
- Start the iterations by Double Clicking Run Calculation > Number of Iterations: 4000 > Calculate
- Monitor the Residual Plot to ensure each data series falls below the 1E-6 tolerance, and ensure that the Lift and Drag Coefficient Plots have flattened out and converged.

Numerical Results (Fluent)

Here we will post-process the results; all results are calculated from the cell center values of the primary unknowns.

Velocity Vectors

- From the Results Tab > Vectors > New
  - Name: velocity_vectors
  - Vectors of: Velocity
  - Color by: Velocity > Velocity Magnitude
  - Scale: .2
  - Skip: 1
  - Check Draw Mesh
    - Select airfoil from Surface List > Click Display
  - Click Save/Display

Velocity Magnitude Contours

- From the Results Tab > Contours > New
  - Name: velocity_contours
  - Contours of: Velocity > Velocity Magnitude
  - Surfaces: Ensure none are selected
  - Click Save/Display

Pressure Contours

- From the Results Tab > Contours > New
  - Name: pressure_contours
  - Contours of: Pressure > Static Pressure
  - Surfaces: Ensure none are selected
  - Click Save/Display

Streamlines

Streamlines can be created quickly using contours of the stream function:

- From the Results Tab > Contours > New
  - Name: streamfunction_contours
  - Contours of: Velocity > Stream Function
  - Uncheck Filled
  - Uncheck Auto Range
  - Check Draw Mesh
    - Select airfoil from Surface List > Click Display
  - Min: 13.11 kg/s, Max: 14.16 kg/s
  - Click Save/Display to view

A more detailed visualization of the streamlines can be created using pathlines:

- From the Results Tab > Pathlines > New
  - Name: streamlines
  - Color by: Velocity > Velocity Magnitude
  - Release from Surfaces: inlet
  - Check Draw Mesh
    - Select airfoil from Surface List > Click Display
- To get better resolution near the airfoil, Create > Line/Rake
  - New Surface Name: seedline
  - Type: Rake
    - This forms a line using a series of points which can be used as seed points for the streamlines
  - End Points: (-3, -1), (-3, 1)
  - Number of Points: 101
- Use this to create streamlines by expanding Pathlines > Right Click streamlines > Click Edit
  - Release from Surface: Add seedline, remove inlet
  - Click Save/Display to view

Coefficient of Pressure Plot

- From the Results Tab > XY Plot > New
  - Name: cp_plot
  - Y-Axis Function: Pressure > Pressure Coefficient
  - X-Axis Function: Direction Vector
  - Surfaces: airfoil
  - Click Plot