R.V. COLLEGE OF ENGINEERING, BENGALURU- 560 059

(Autonomous Institution Affiliated to VTU, Belagavi)

ANALYSIS OF SUPERSONIC FLOW OVER DOUBLE WEDGE USING ANSYS FLUENT

Project Report

Submitted by:

RITHWIK SHANKAR RAJ USN: 1RV19ME087

Under the Guidance
of

Prof. Bhaskar K.,
Assistant Professor,
Department of Aerospace Engineering,
RV COLLEGE OF ENGINEERING®, BENGALURU - 560059

in partial fulfilment for the award of degree of
Bachelor of Engineering
IN
Mechanical Engineering

\

1. OVERVIEW

- This project deals with the simulation of 2-D supersonic flow over a double wedge.
- Density based solver in ANSYS fluent is used.
- The contours of static pressure, velocity and Mach number are obtained from the results of the numerical computations.
- The results obtained are compared with theoretical values from shock-expansion theory.

2. DOUBLE WEDGE SUPERSONIC MODEL

2.1. MODEL AND DOMAIN DETAILS FOR THE NUMERICAL SIMULATION

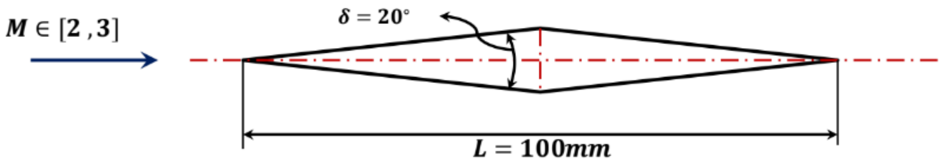


Fig. 1: Model details

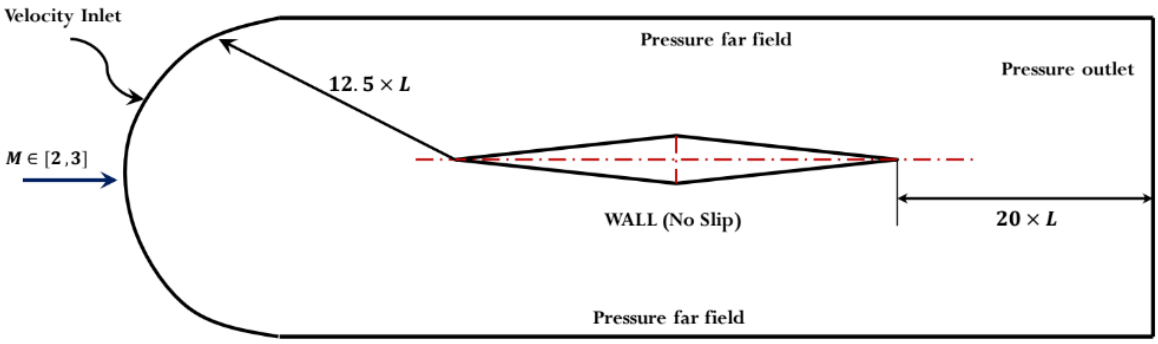


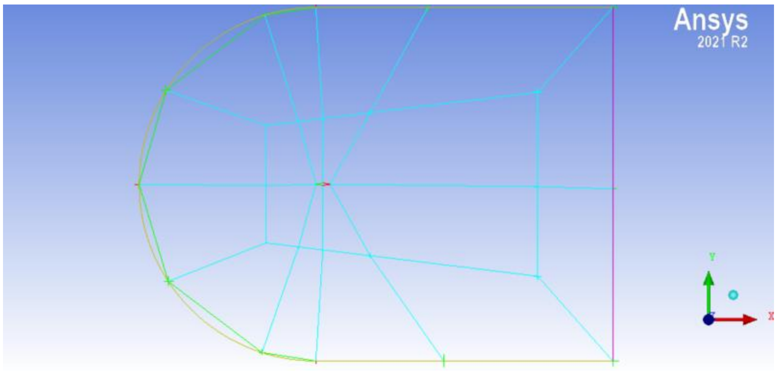
Fig. 2: Domain details for numerical simulation

2.2. MESHING

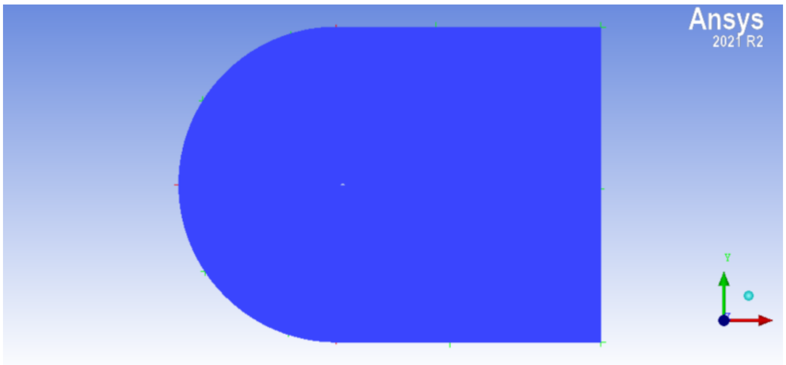
ICEM CFD software has been used to mesh the domain as specified above. The domain has been constructed within the ICEM CFD software and parts were created in ICEM CFD for the following regions:

- Fluid
- Far-field
- Outlet
- Air foil

Blocking procedure has been carried out to obtain the necessary configuration of mesh elements to accurately capture the shockwaves around the double wedge. The blocking strategy is shown below.

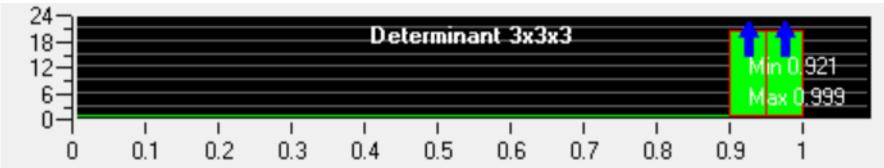


The final obtained mesh after specification of pre-mesh parameters is shown below.

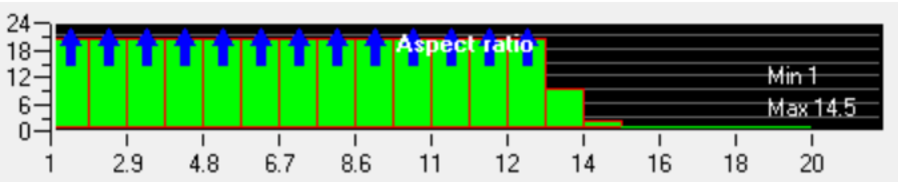


The mesh quality parameters are as follows:

- Mesh quality (3 x 3x 3 Determinant):



- Mesh aspect ratio:

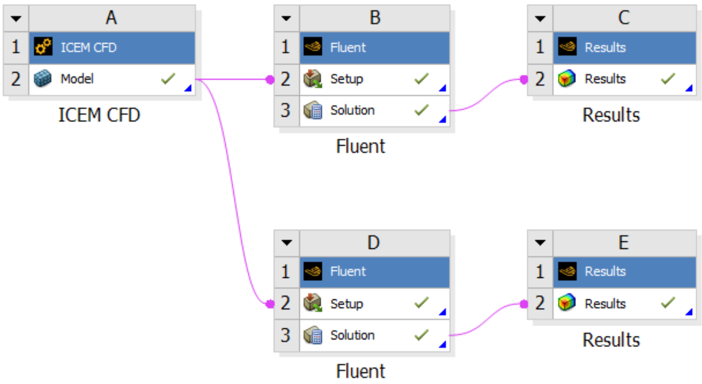


2.3. COMPONENT SETUP IN ANSYS WORKBENCH

The mesh data is translated to ANSYS FLUENT solver in ICEM CFD after specifying the appropriate boundary conditions of pressure far-field and velocity inlet at the corresponding boundaries of the computational domain.

The data from ICEM CFD is used as input to two FLUENT component systems for computation of results for the following two cases:

- Flow at M=2 at no angle of attack and
- Flow at M=2 with an angle of attack of 10 degrees.



3. SETUP AND BOUNDARY CONDITIONS

- Steady State simulation with Density Based Solver (DBNS) is selected.
- Energy Equation is turned on.
- Spallart Almaras(SA) 1 Equation Model is selected.
- The properties of the fluid (air) are changed to account for compressible flow conditions: density variation is according to ideal gas law and Sutherland model is chosen for viscosity.
- The boundary conditions for the two cases are as follows:

    - Case 1: For flow at Mach no 2 at no angle of attack:

    Pressure far-field:
    Mach number: 2
    X component of velocity: 1
    Y component of velocity: 0
    Airfoil: Wall boundary condition
    Outlet: Pressure outlet condition

    - Case 2: For flow at Mach no2 at an angle of attack of 10 degrees:

    Pressure far-field:
    Mach number: 2
    X component of velocity: 0.98480775
    Y component of velocity: 0.17364818
    Airfoil: Wall boundary condition
    Outlet: Pressure outlet condition

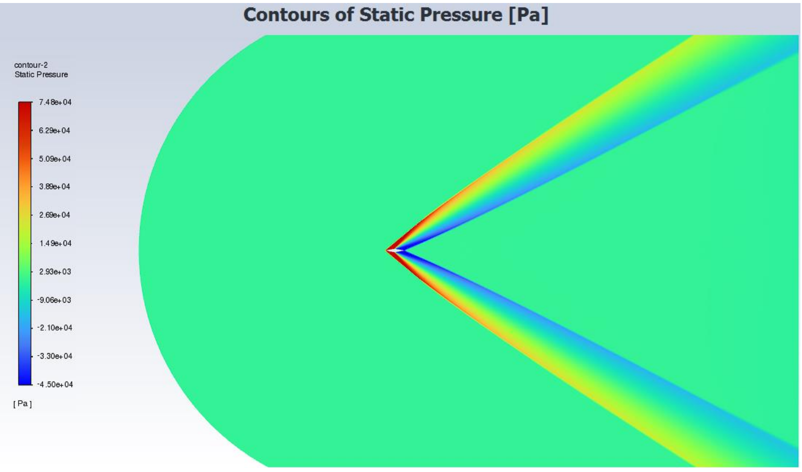
4. CALCULATION

- Reference values are calculated from the far-field in both cases.
- Both flow And Modified Turbulent Viscosity Methods are assigned Second Order Upwind.
- Report definitions are made for Lift and Drag coefficients plots (vs no of iterations) which also show convergence of the solution.
- Appropriate convergence conditions and residuals are set up.
- Standard initialisation with reference from far-field is selected.
- The number of iterations is chosen as 1000 and after checking the case the simulation is started.

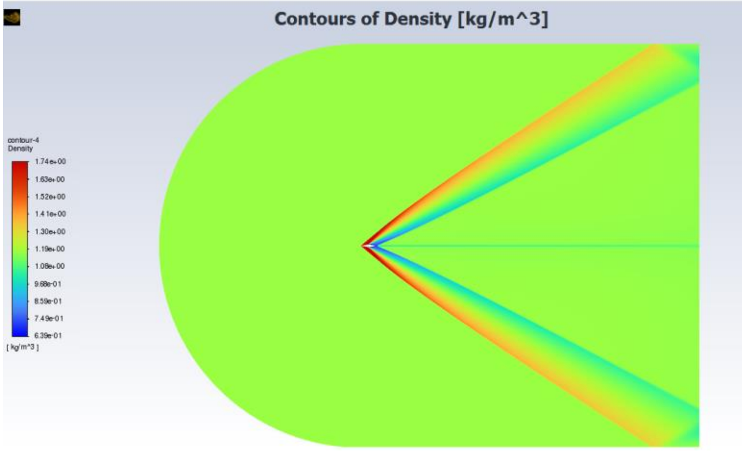
5. RESULTS

5.1. 0° ANGLE OF ATTACK

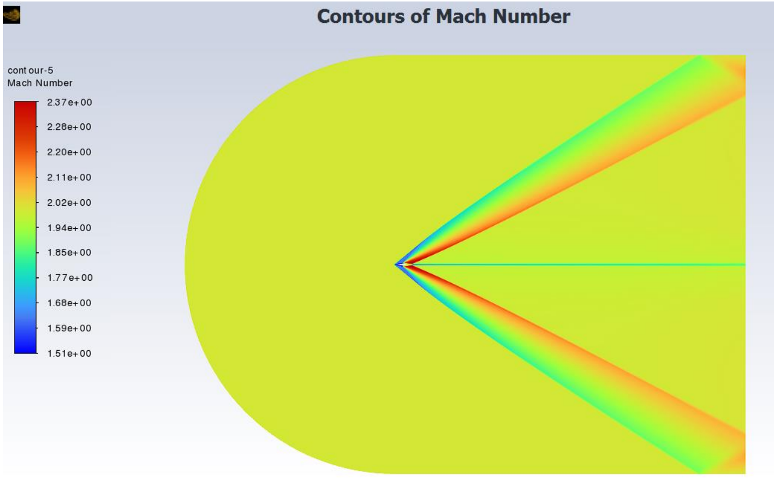
1. CONTOURS OF STATIC PRESSURE



2. CONTOURS OF DENSITY

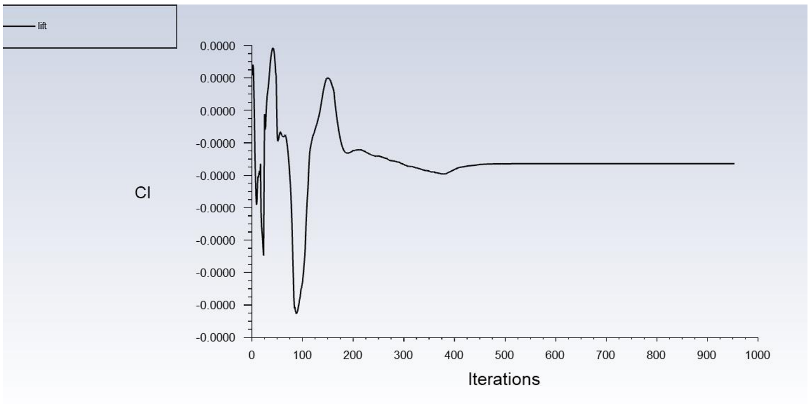


3. CONTOURS OF MACH NUMBER



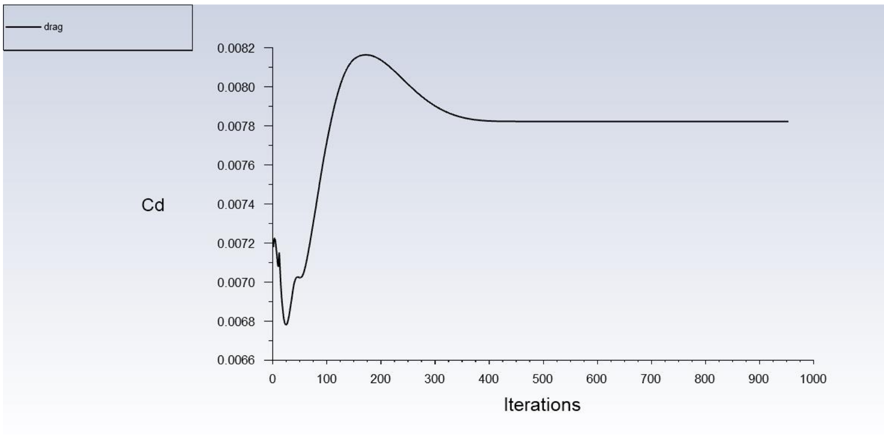
4. COEFFICIENT OF LIFT (Cl)

The value obtained for the lift coefficient is 0.0000. The convergence plot of the same is as shown:



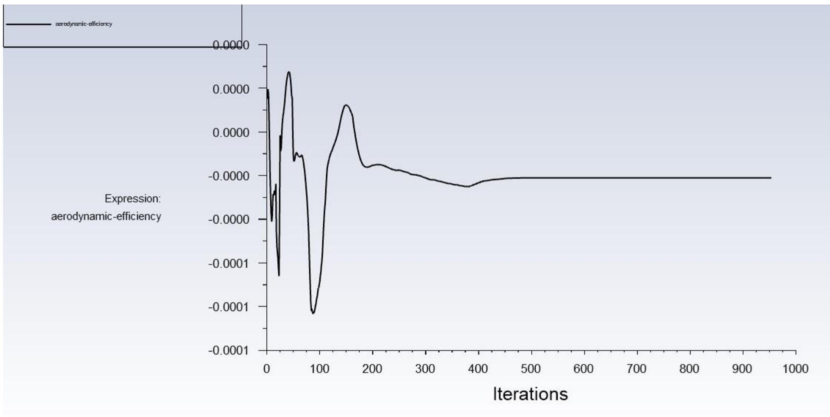
5. COEFFICIENT OF DRAG (Cd)

The value obtained for the drag coefficient is 0.0078219168. The convergence plot of the same is as shown:



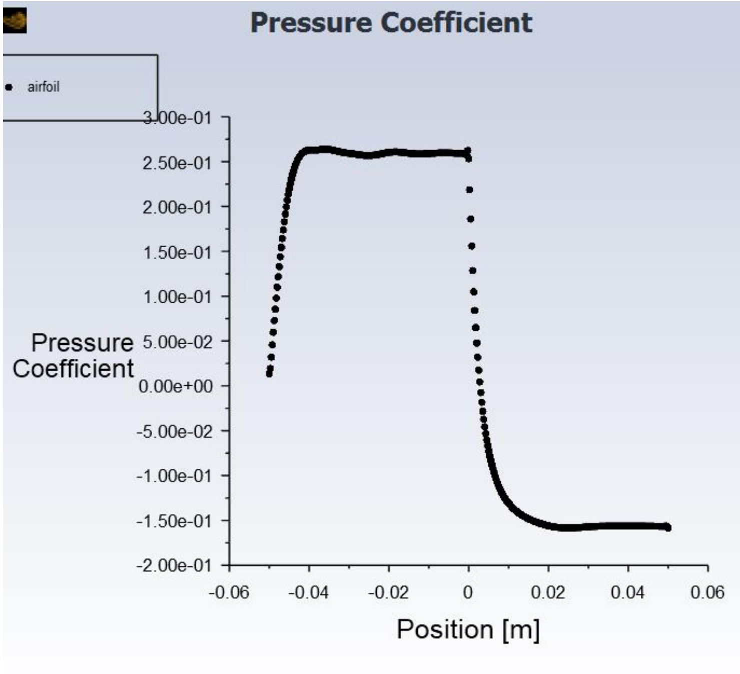
6. AERODYNAMIC EFFICIENCY

The aerodynamic efficiency obtained is 0 as there is no angle of attack. The convergence plot is as shown:



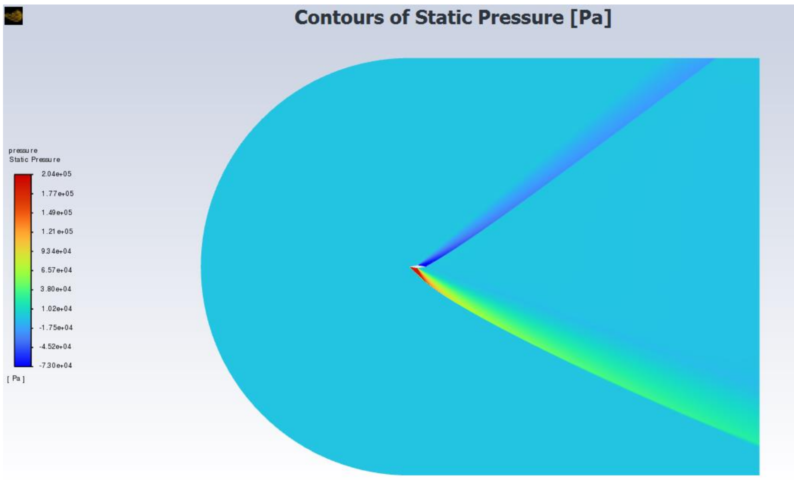
7. PRESSURE COEFFICIENT VS POSITION

The plot of pressure coefficient vs the position is as shown:

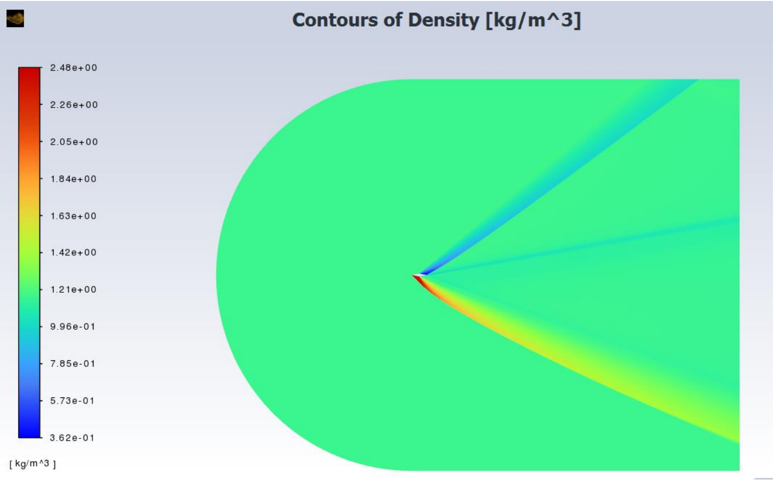


5.2. 10° ANGLE OF ATTACK

1. CONTOURS OF STATIC PRESSURE

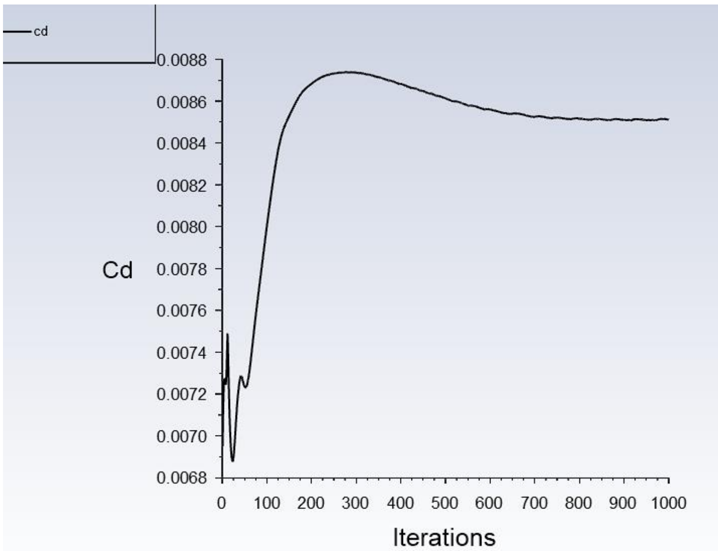


2. CONTOURS OF DENSITY



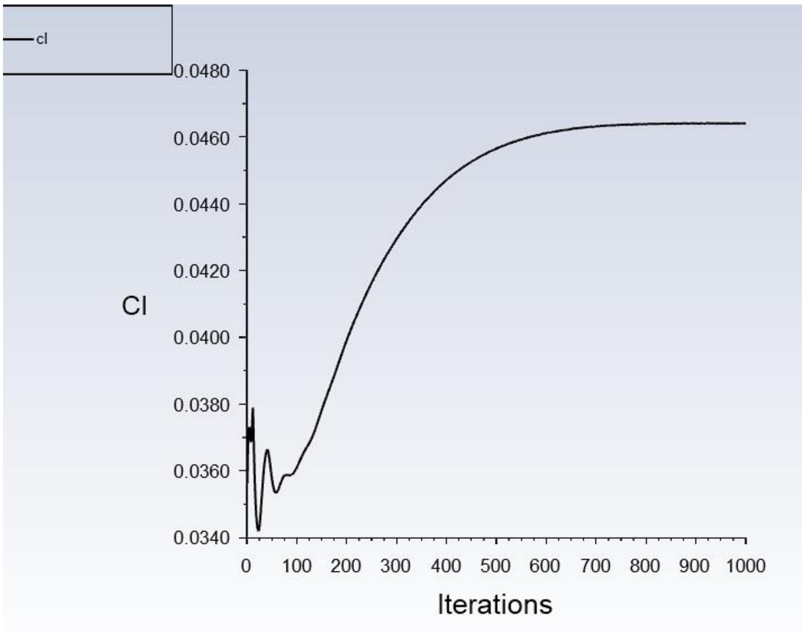
3. COEFFICIENT OF DISCHARGE (Cd)

The value obtained for the drag coefficient is 0.0085110719. The convergence plot of the same is as shown:



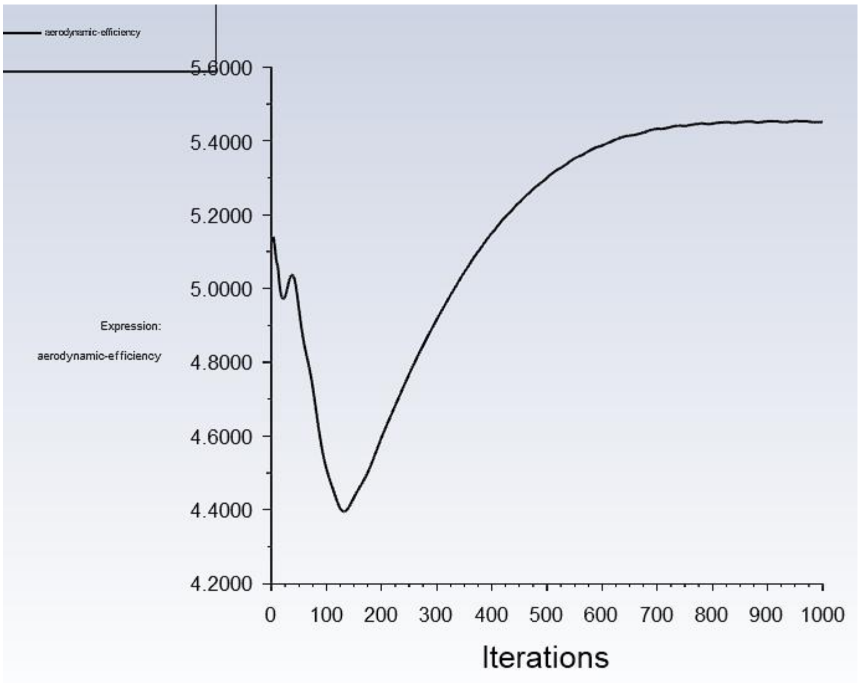
4. COEFFICIENT OF LIFT (Cl)

The value obtained for the lift coefficient is 0.046401218. The convergence plot of the same is as shown:



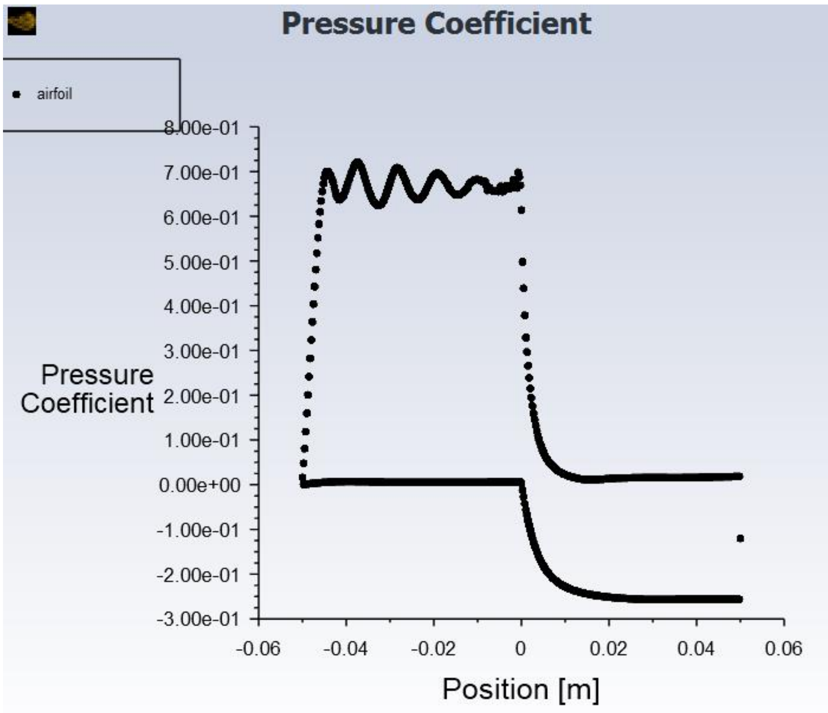
5. AERODYNAMIC EFFICIENCY (Cl/Cd)

The aerodynamic efficiency obtained is 5.4 as obtained from the report. The convergence plot is as shown:



6. PRESSURE COEFFICIENT VS POSITION

The plot of pressure coefficient vs the position is as shown:



6. COMPARISON WITH ANALYTICAL VALUES

SIMULATION	ANALYTICAL	PARAMETER
0.0078219168	0.0085	DRAG COEFFICIENT (Cd)
0	0	LIFT COEFFICIENT (Cl)

SIMULATION	ANALYTICAL	PARAMETER
0.0085110719	0.0078	DRAG COEFFICIENT (Cd)
0.046401218	0.04	LIFT COEFFICIENT (Cl)

It is seen that the results obtained from the analytical and simulation are both in agreement.

The difference in values obtained may be due to the approximations involved in obtaining the isentropic relations in the analytical method.

The model used for computation also may have had an effect as here Spalart Almaras model has been used.