Complex Engineering Problem

Submitted By: Hussnain Muhgal

Roll NO: FA20-BME-015

Study the flow around a modified corner square cylinder using Commerical CFD codes having Reynolds Numbers (Re) of 150

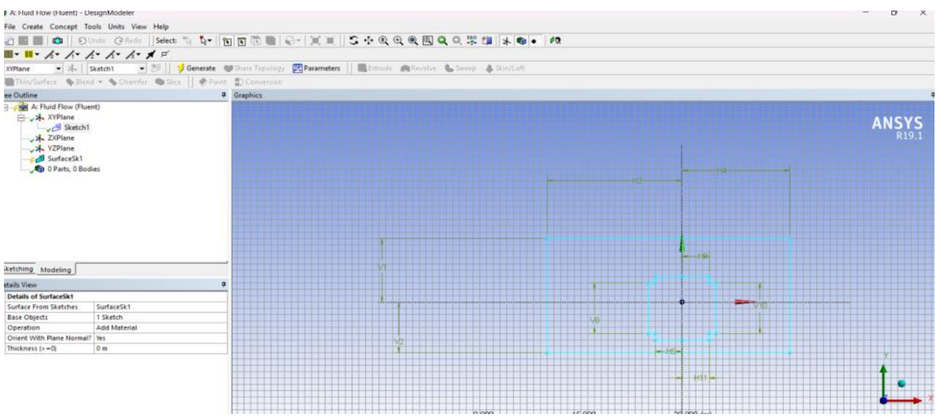
CONTENTS :

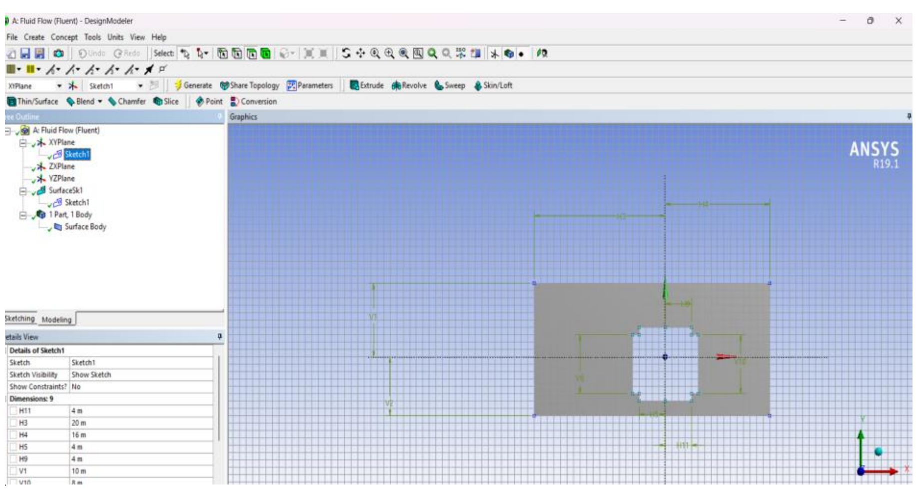
1-Introduction:

This report focuses on the analysis of the flow around a modified corner square cylinder using Commerical CFD codes having Reynolds Numbers (Re) of 150, using the CFD software ANSYS Fluent. By using this programme, we will examine different parameters for flow around a modified corner square cylinder which includes velocity and pressure distributions along with lift and drag coefficients for body and the effects of different flow parameters on the characteristics of the flow field. We also provide insights into the characteristics of the flow field, including the regions of high and low pressure, the wake structure, and the vortical patterns that form.

2-Geometry:

We have created 2D (10 x 10 mm) square with four equal elliptical corners. It has two vertical sides of same length and two horizontal sides of same length.



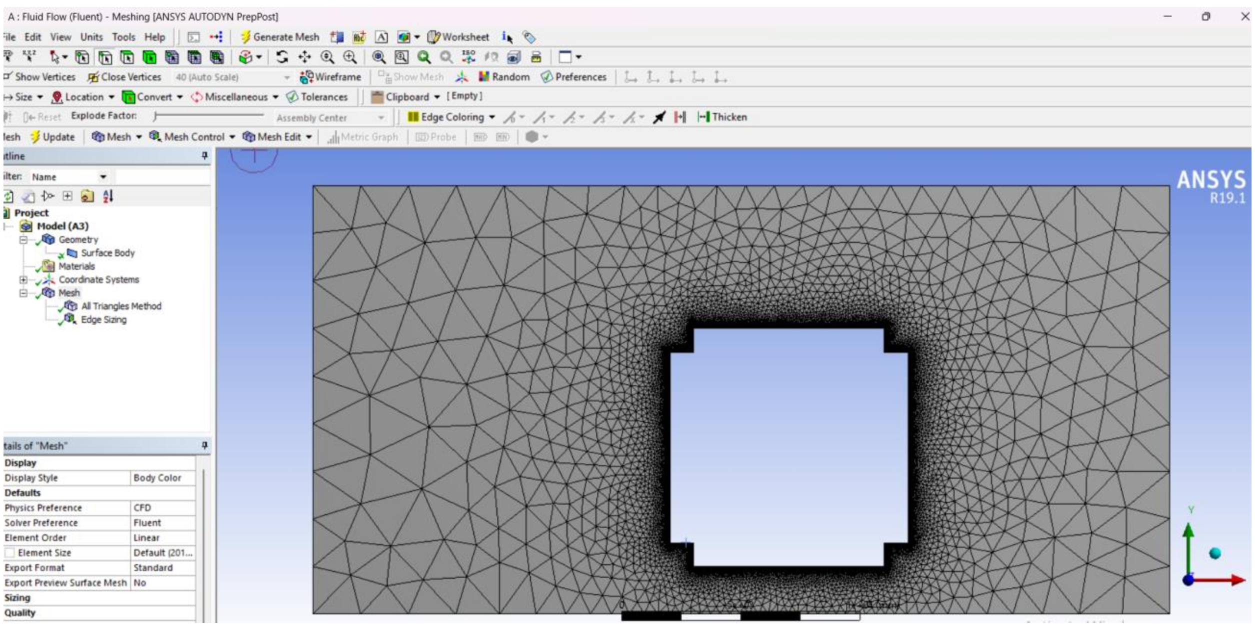


3-Meshing and Boundary Condition:

The meshing process in ANSYS involves dividing the geometry into small, finite-sized elements. The meshing process involves several steps, including geometry preparation, mesh sizing, and mesh generation. First step of geometry preparation is detailed in above step of “Geometry’’.

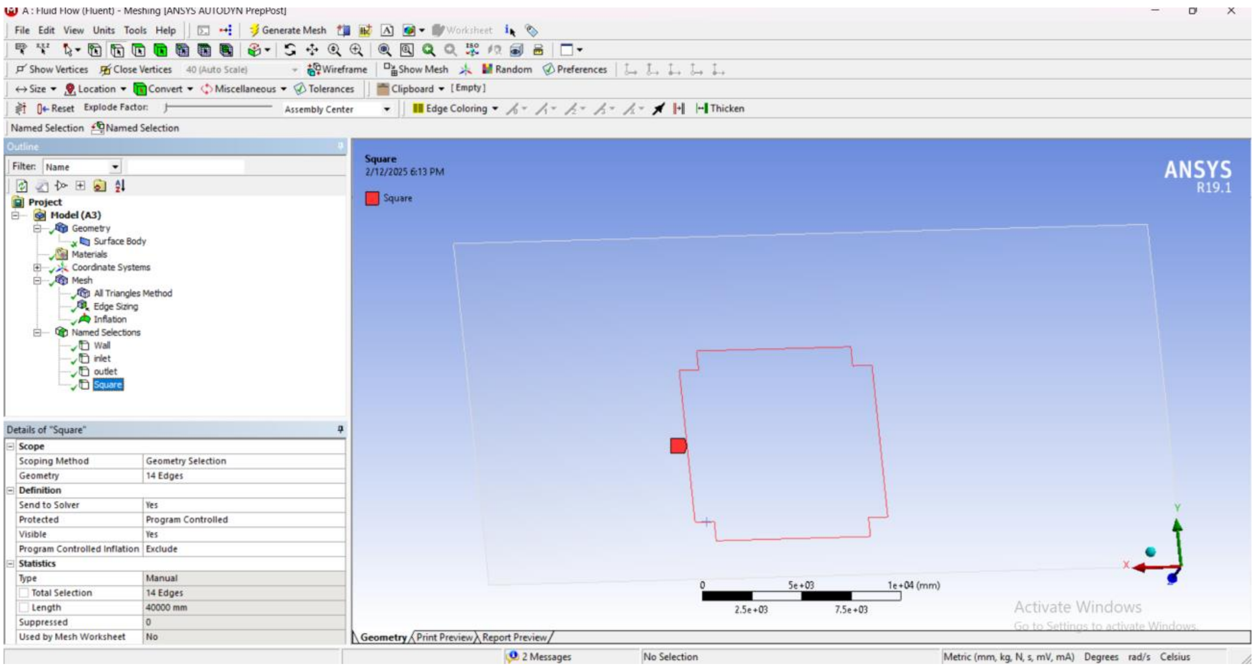
The next step in the meshing process is to determine the size and distribution of the mesh elements. Face sizing is applied to different faces of geometry model i.e vertical and horizontal lines and elliptical corners. Every mm is divided into 10 divisions for face sizing of body. Small to large type biasing is selected so that meshing near the surface of the body is more dense as compare to other areas.

Meshing is shown in figure below:

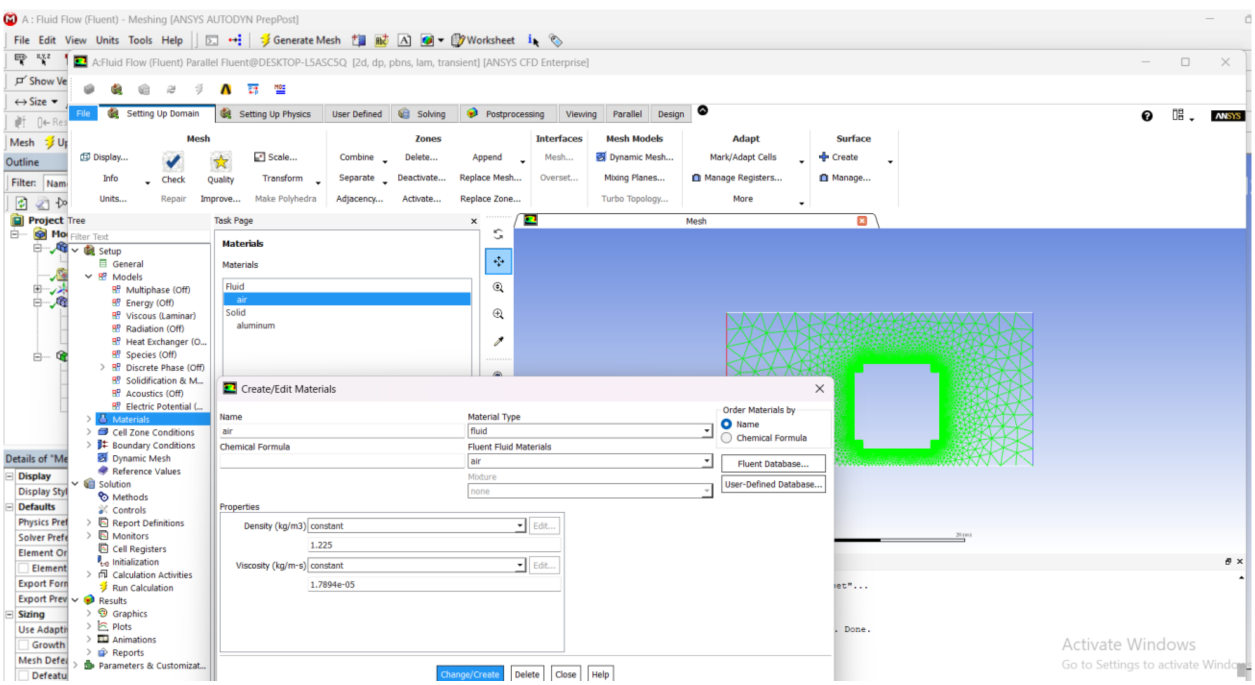


The left side of the geometry is considered as flow inlet and right side of the geometry is considered as flow outlet

Condition:



A : Fluid Flow (Fluent) - Meshing [ANSYS AUTODYN PrepPost]
Generate Mesh
Outline
Project
Model (A3)
Geometry
Surface Body
Materials
Coordinate Systems
Mesh
All Triangles Method
Edge Sizing
Inflation
Named Selections
Wall
inlet
outlet
Square
Square
2/12/2025 6:13 PM
ANSYS
R19.1
Details of "Square"
Scope
Scoping Method
Geometry Selection
Geometry
14 Edges
Definition
Send to Solver
Yes
Protected
Program Controlled
Visible
Yes
Program Controlled Inflation
Exclude
Statistics
Type
Manual
Total Selection
14 Edges
Length
40000 mm
Suppressed
0
Used by Mesh Worksheet
No
0
5e+03
1e+04 (mm)
2.5e+03
7.5e+03
Geometry
Print Preview
Report Preview
2 Messages
No Selection
Metric (mm, kg, N, s, mV, mA)
Degrees
rad/s
Celsius



A:Fluid Flow (Fluent) Parallel Fluent@DESKTOP-L5ASC5Q [2d, dp, pbns, lam, transient] [ANSYS CFD Enterprise]
Setting Up Domain
Setting Up Physics
User Defined
Solving
Postprocessing
Viewing
Parallel
Design
Setup
General
Models
Multiphase (Off)
Energy (Off)
Viscous (Laminar)
Radiation (Off)
Species (Off)
Discrete Phase (Off)
Acoustics (Off)
Materials
Cell Zone Conditions
Boundary Conditions
Dynamic Mesh
Reference Values
Solution
Methods
Controls
Report Definitions
Monitors
Cell Registers
Initialization
Calculation Activities
Run Calculation
Results
Graphics
Plots
Animations
Reports
Task Page
Materials
Fluid
air
Solid
aluminum
Mesh
Create/Edit Materials
Name
air
Material Type
fluid
Chemical Formula
Fluent Fluid Materials
air
Mixture
none
Order Materials by
Name
Chemical Formula
Fluent Database...
User-Defined Database...
Properties
Density (kg/m3)
constant
1.225
Viscosity (kg/m-s)
constant
1.7894e-05
Change/Create
Delete
Close
Help

4-SETUP AND SOLUTION:

Upon opening ANSYS Fluent, certain parameters need to be determined based on the type of analysis being performed.

>Setup

>General>Time>Transient

>Models>Energy>Check Energy Equation

>Models>Viscous Model>Check Laminar

>Materials>Fluid>air

>Materials>Solid>aluminum

>Boundary Conditions

>Reference Values

Area 0.01 m^2 , Density 1.225 kg/m^3 ,Depth 1 m, Enthalpy 0 J/kg. Length 0.01 m, Pressure 0 Pa, Velocity 0.0009271 m/s

>Solutions

>Methods

>Solution Methods>Pressure-Velocity Coupling>Scheme>Fractional Step

>Transient Formulation>Second Order Implicit

>Check Non-Iterative Time Advancement

>Controls> Non-Iterative Solver Relaxation Factors

>Report Definitions>New

>Surface Report>Area-Weighted Average>Field Variable>Wall Fluxes>Surface Nusselt Number>Surface>Select cylinder

>Force Report>Drag> Select cylinder

>Force Report>Lift> Select cylinder

>Initializing

>Initializing Methods>Check Standard Initializing

>Compute from>inlet

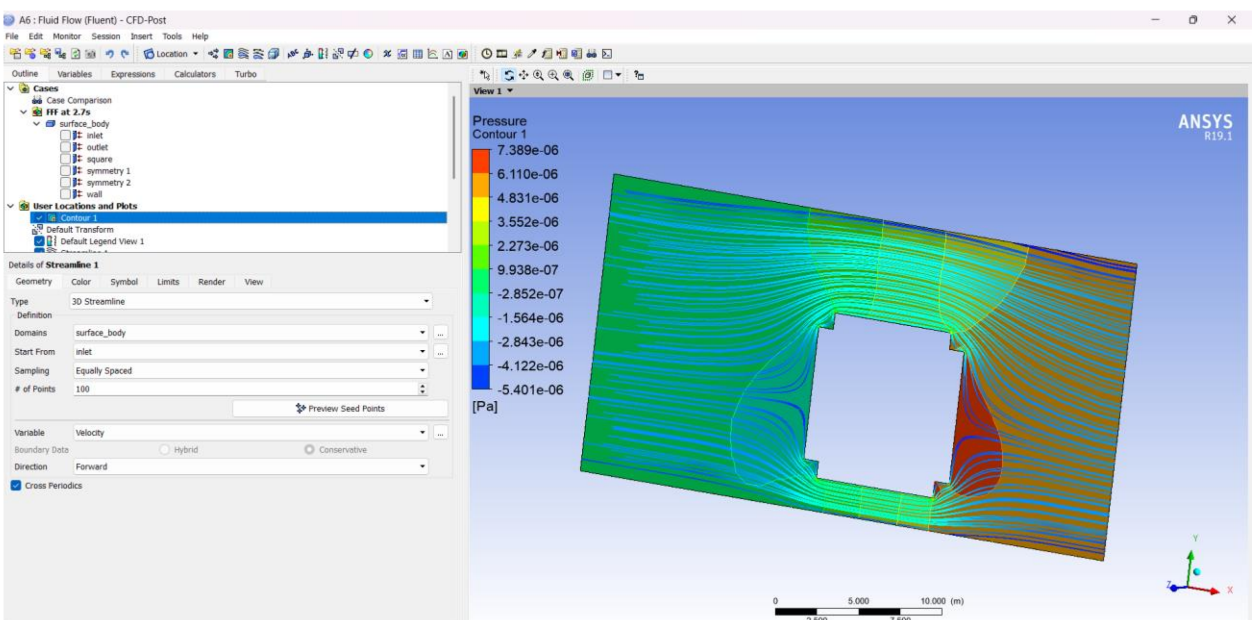
>Initialize

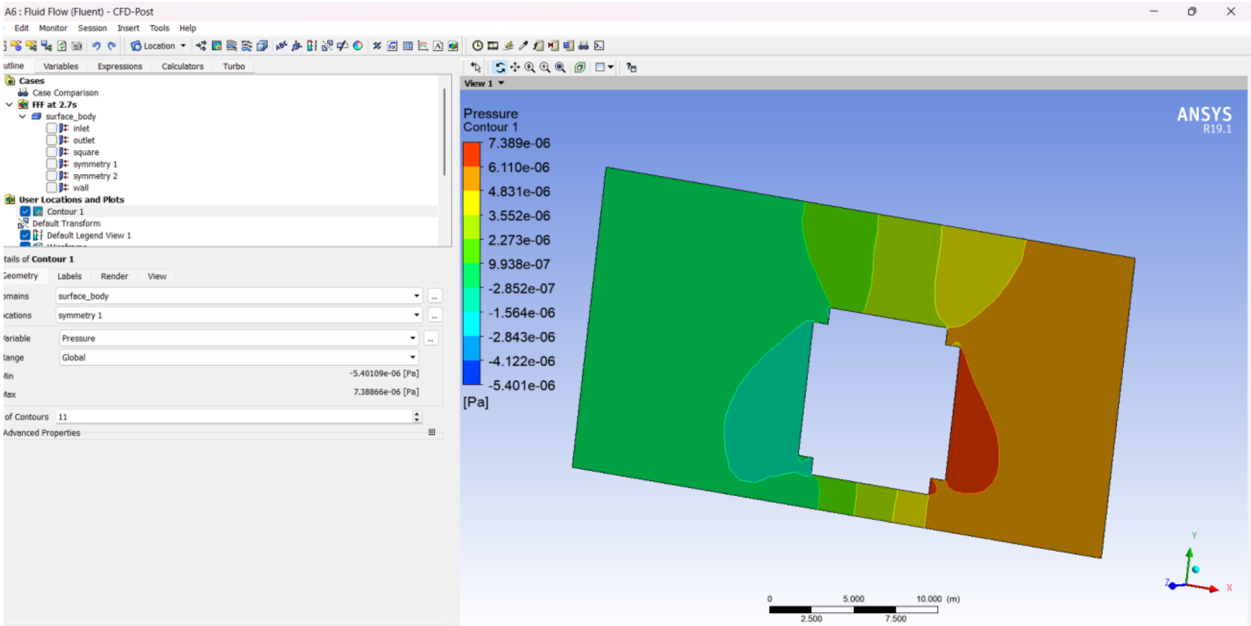
>Run Calculation

Unsteady Calculation Parameters	
Number of Time Steps	1000
Time Step Size [s]	0.001

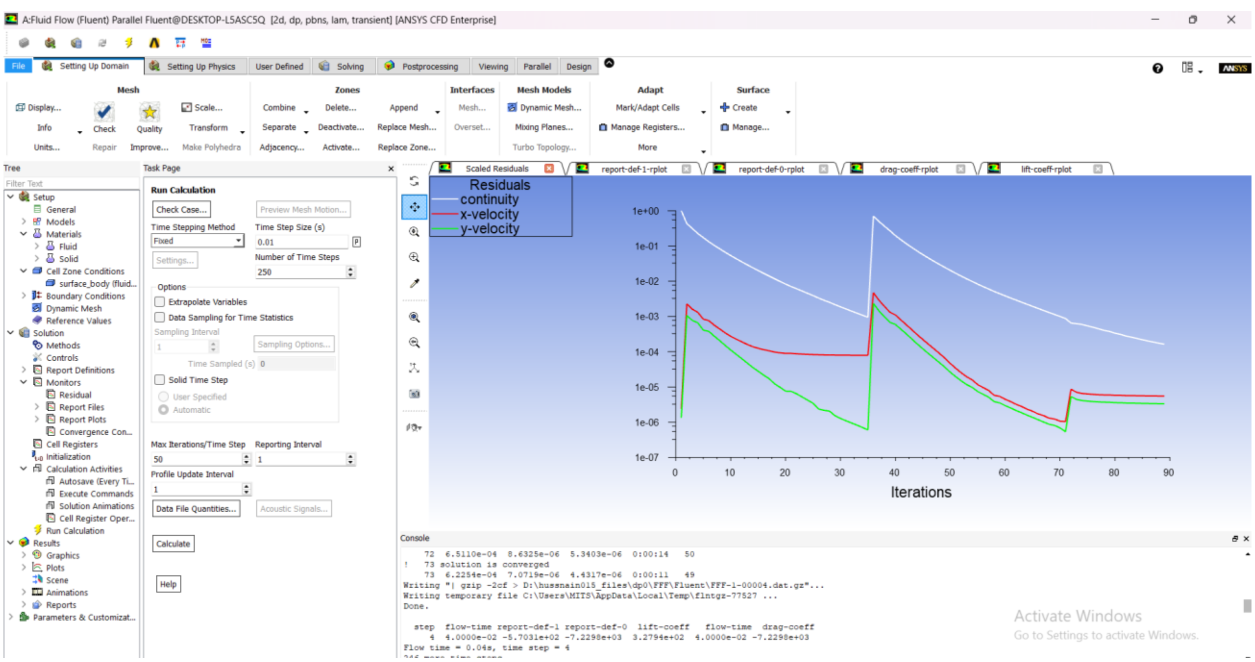
5-RESULTS AND ANALYSIS:

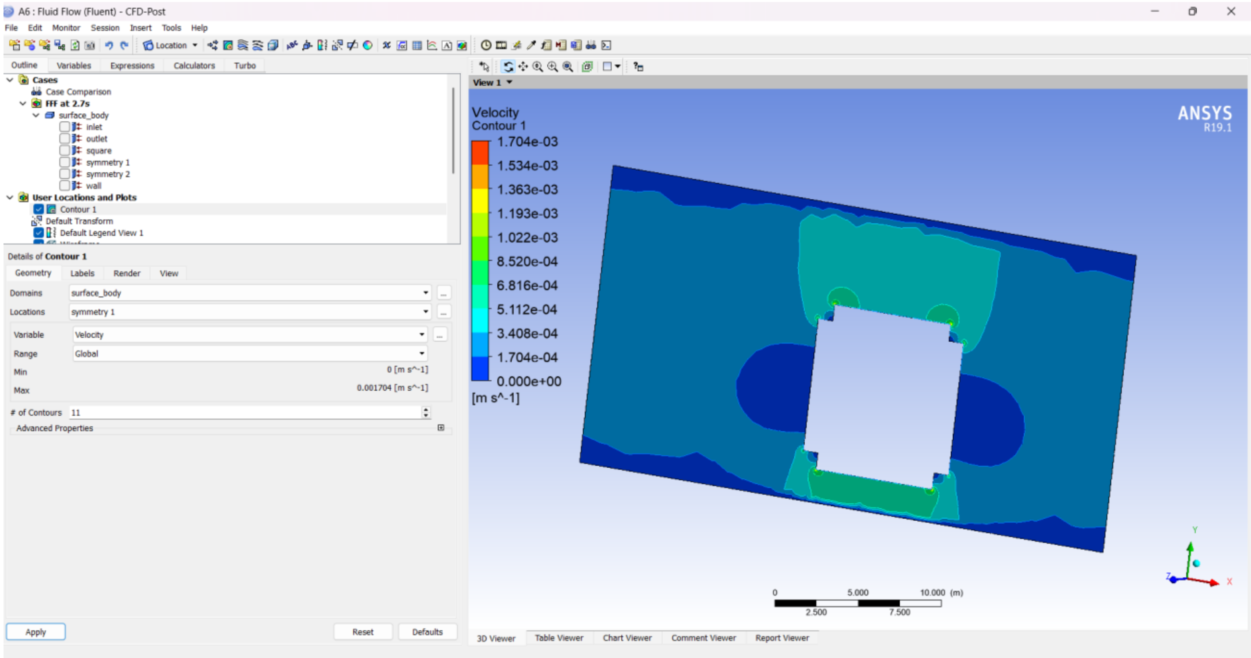
For results, we have plotted pressure and velocity contours as shown below: At left side, red color shows the high pressure which is reduced due to the Bernoulli's principle and the effects of flow separation. The Bernoulli's principle states that as the velocity of a fluid increases, the pressure decreases. As the fluid flows around an elliptical blunt body, the flow accelerates in certain regions, particularly around the leading edges and upper surface of the body. This acceleration results in a decrease in pressure in those regions, which is reflected in the pressure contours. Additionally, as the flow passes over the blunt trailing edge of the body, it separates from the body and creates a region of lowpressure wake behind it. This wake can further decrease the pressure.

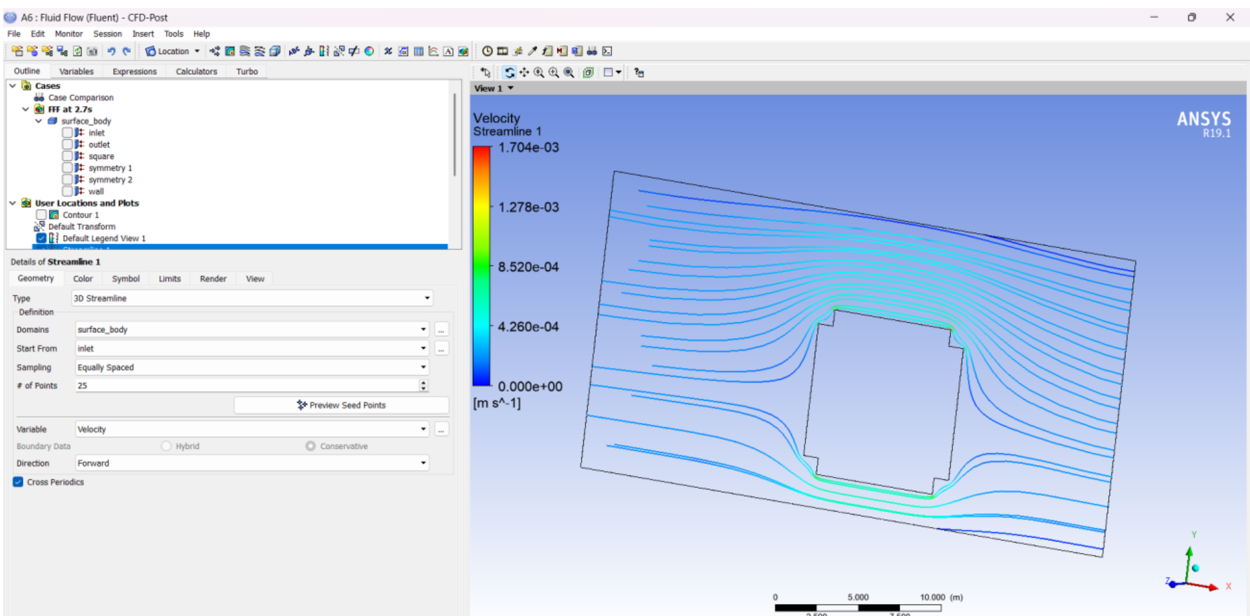




Following figure shows velocity contour at Reynold number of 150.We can see in the figure below that on the very left side of our surface body stagnation point exists where maximum kinetic energy of the fluid has converted into pressure energy and velocity became zero and after that decreasing pressure gradient provides favourible conditions to the flow and velocity enhances. Due to formation of the boundary layer, just along the surface of the body the velocity is small as compared to the incoming velocity.

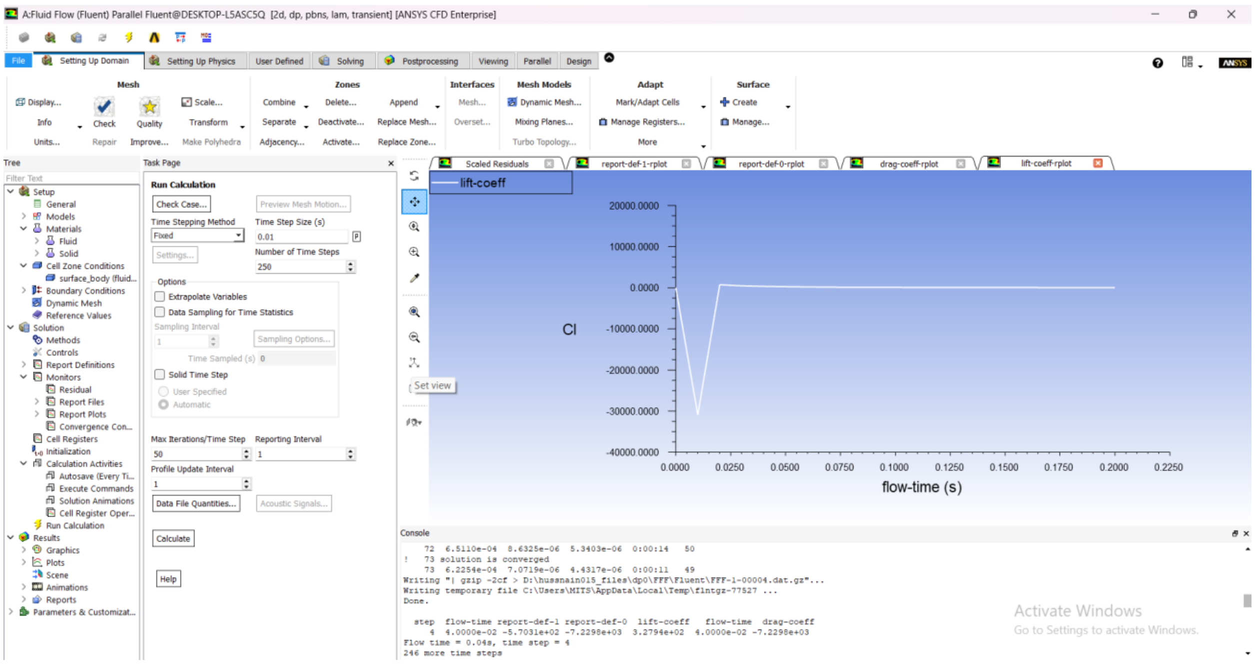


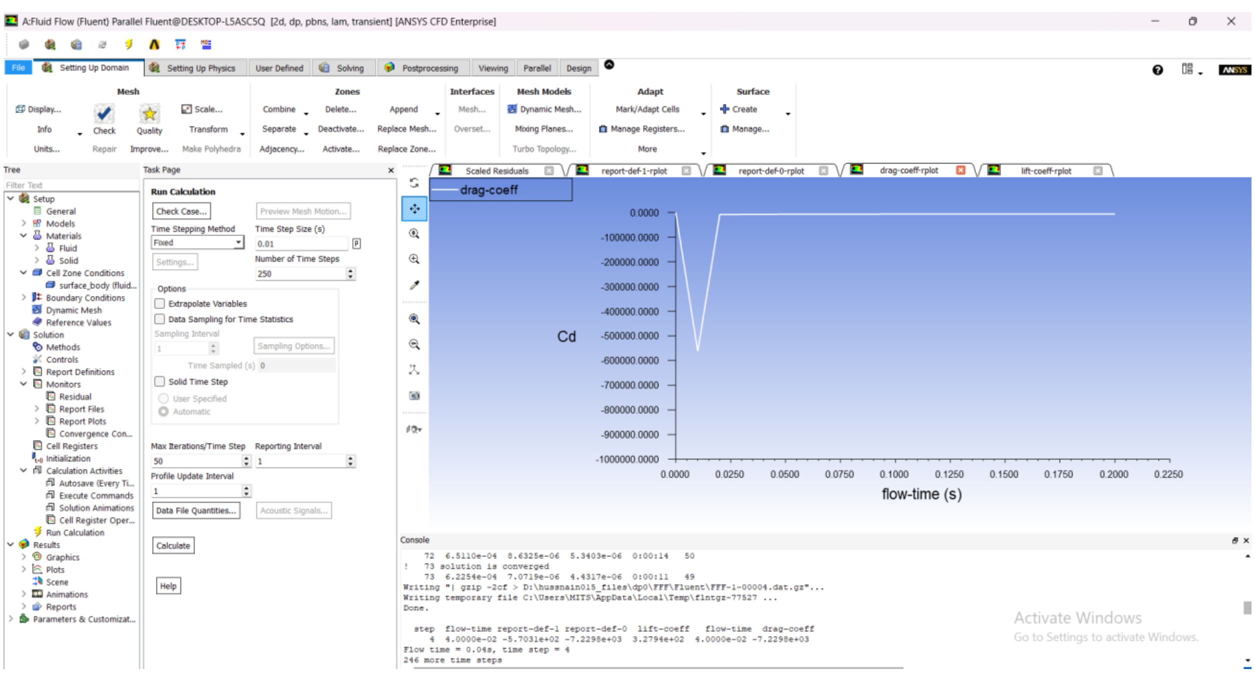

A6 : Fluid Flow (Fluent) - CFD-Post
Velocity
Contour 1
1.704e-03
1.534e-03
1.363e-03
1.193e-03
1.022e-03
8.520e-04
6.816e-04
5.112e-04
3.408e-04
1.704e-04
0.000e+00
[m s^-1]
ANSYS
R19.1
Details of Contour 1
Domains surface_body
Locations symmetry 1
Variable Velocity
Range Global
Min 0 [m s^-1]
Max 0.001704 [m s^-1]
# of Contours 11

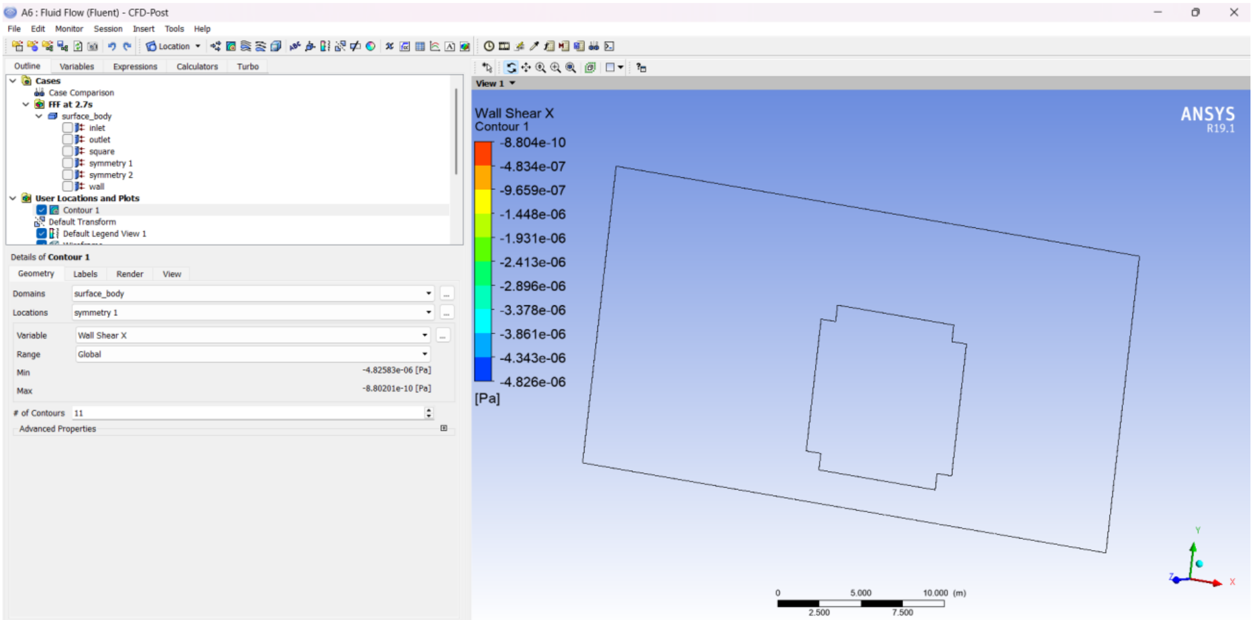

A6 : Fluid Flow (Fluent) - CFD-Post
Velocity
Streamline 1
1.704e-03
1.278e-03
8.520e-04
4.260e-04
0.000e+00
[m s^-1]
ANSYS
R19.1
Details of Streamline 1
Type 3D Streamline
Domains surface_body
Start From inlet
Sampling Equally Spaced
# of Points 25
Variable Velocity
Direction Forward

LIFT AND DRAG PLOTS:

We have explained in the pressure contours that flow separation and Bernoulli effect causes pressure variations. These pressure variations causes pressure drag.







A6 : Fluid Flow (Fluent) - CFD-Post
Wall Shear X
Contour 1
-8.804e-10
-4.834e-07
-9.659e-07
-1.448e-06
-1.931e-06
-2.413e-06
-2.896e-06
-3.378e-06
-3.861e-06
-4.343e-06
-4.826e-06
[Pa]
ANSYS
R19.1
Details of Contour 1
Domains surface_body
Locations symmetry 1
Variable Wall Shear X
Range Global
Min -4.82583e-06 [Pa]
Max -8.80201e-10 [Pa]
# of Contours 11