

Complex Engineering Problem

Submitted By: Huzaifa Khurshid

Roll NO: FA20-BME-071

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

CONTENTS :

1- INTRODUCTION

Problem description

2-GEOMETRY

3-MESHING

4-SETUP AND SOLUTION

5-RESULTS AND ANALYSIS

Lift and Drag

Wall Shear

Velocity

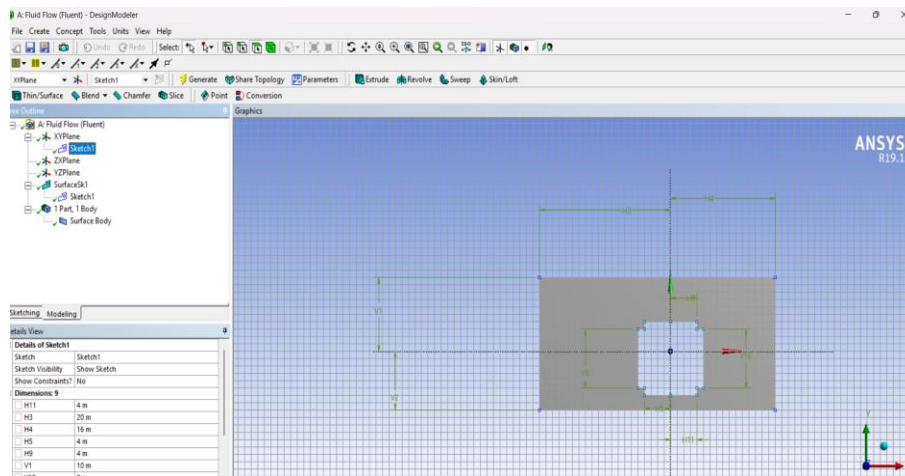
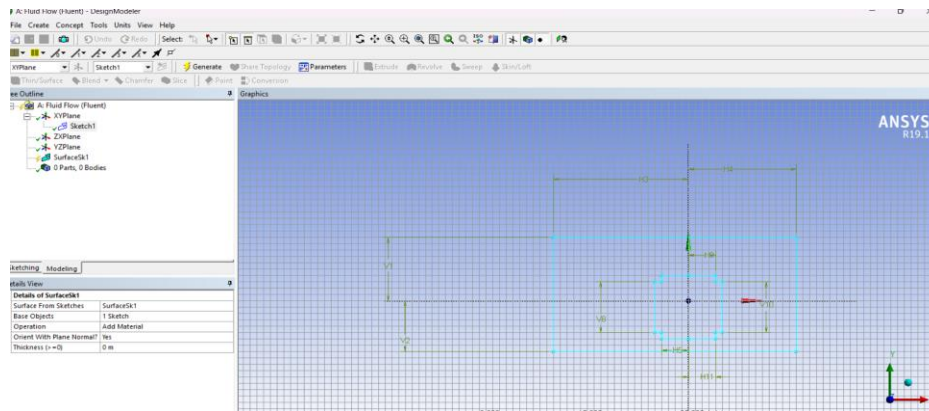
Pressure

1-Introduction:

This report focuses on the analysis of the flow around a modified corner square cylinder using Commercial 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.

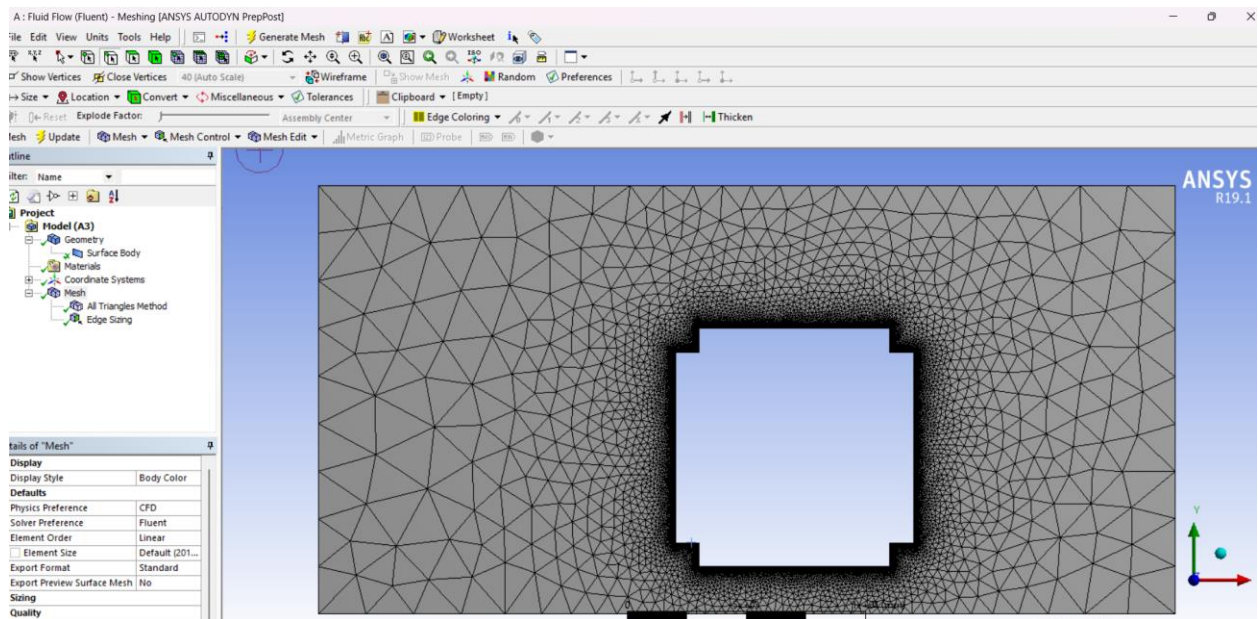


3-Meshing and Boundary Condition:

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”.

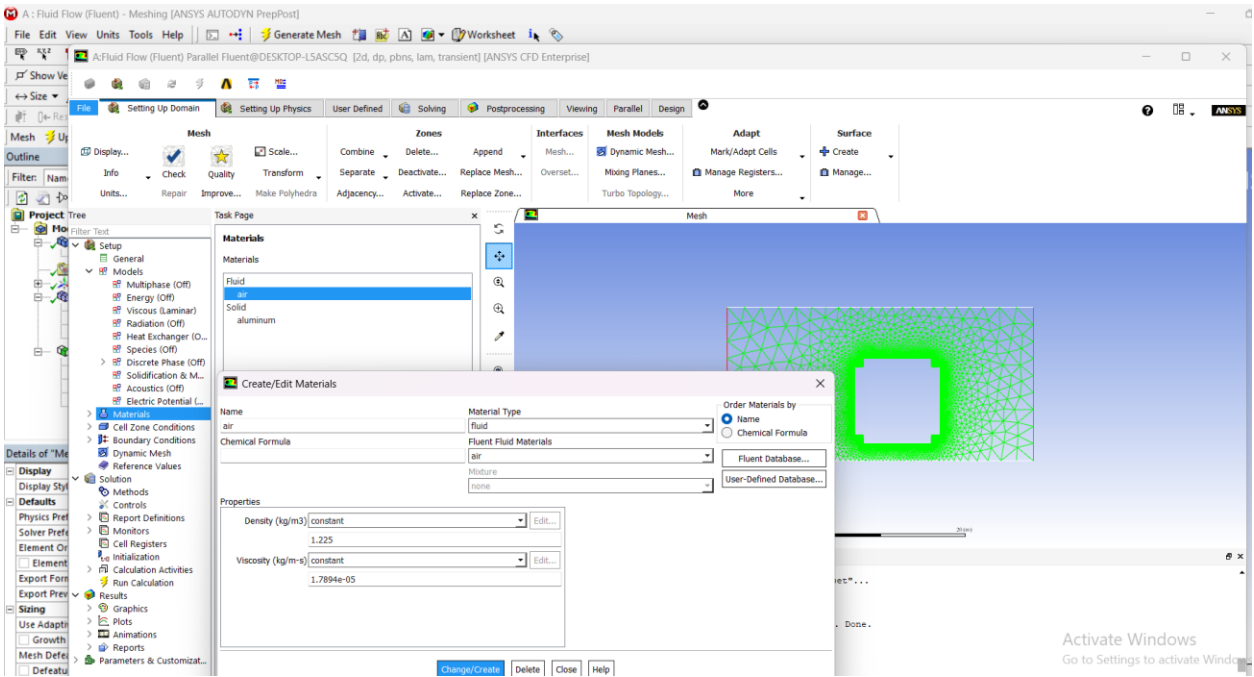
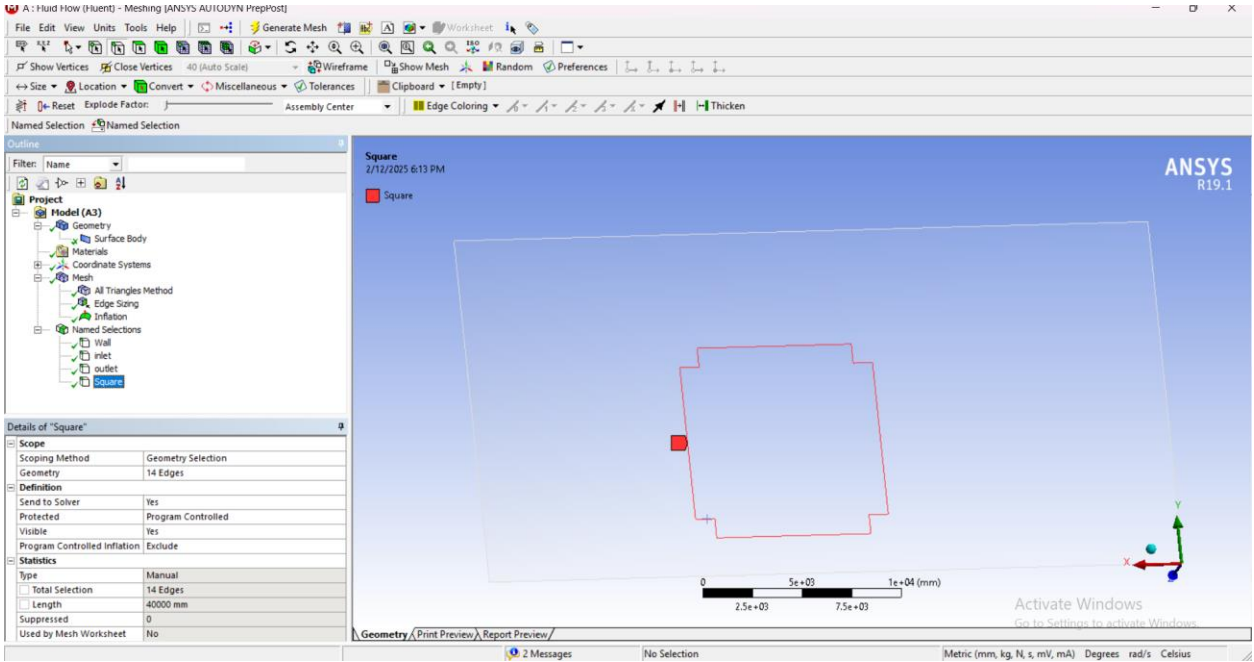
The next step in the meshing process is to determine the size and distribution of the mesh elements. Face sizing is applied to different faces of geometry model i.e vertical and horizontal lines and elliptical corners. Every mm is divided into 10 divisions for face sizing of body. Small to large type 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:



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

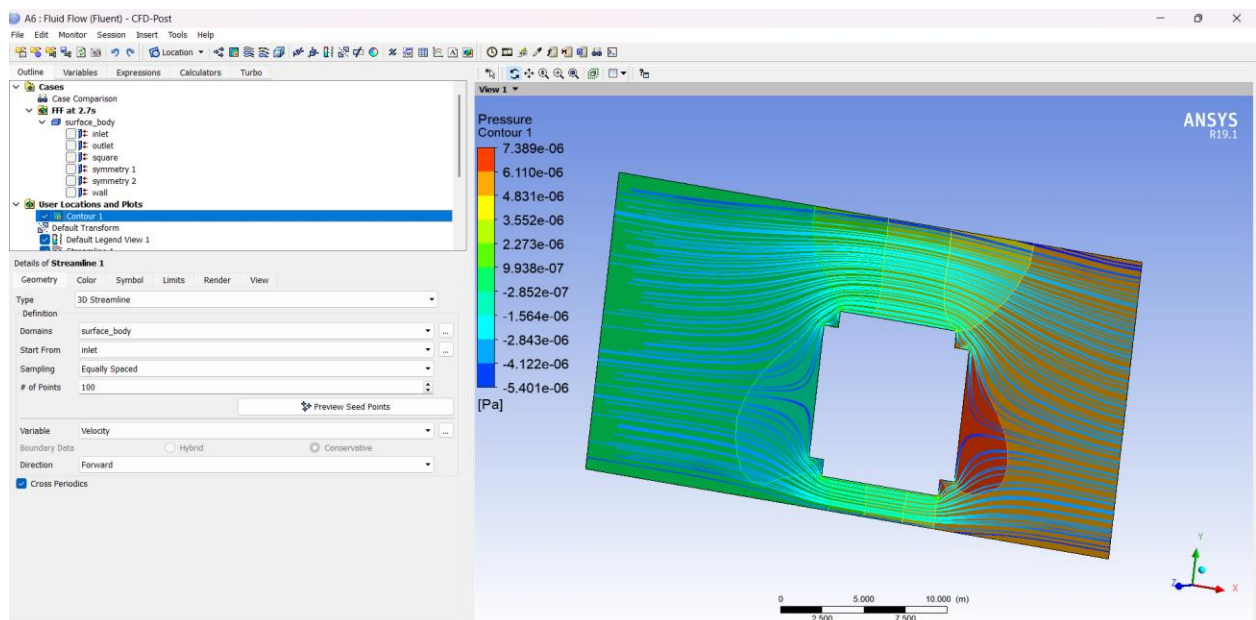
>Initialize

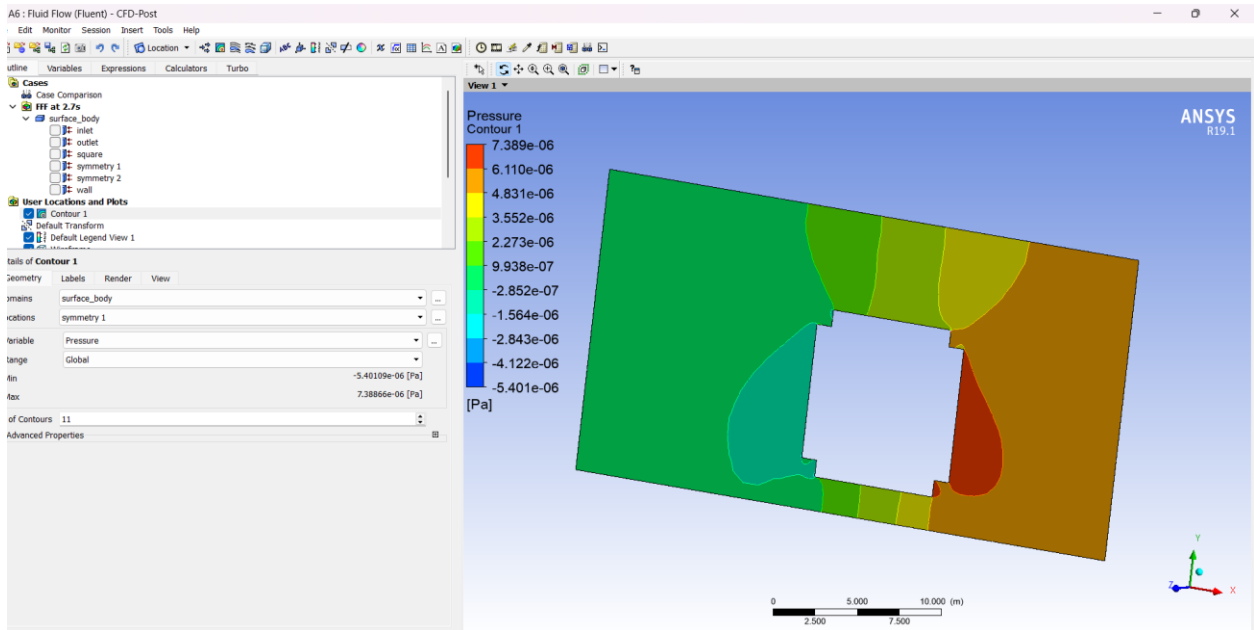
>Run Calculation

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

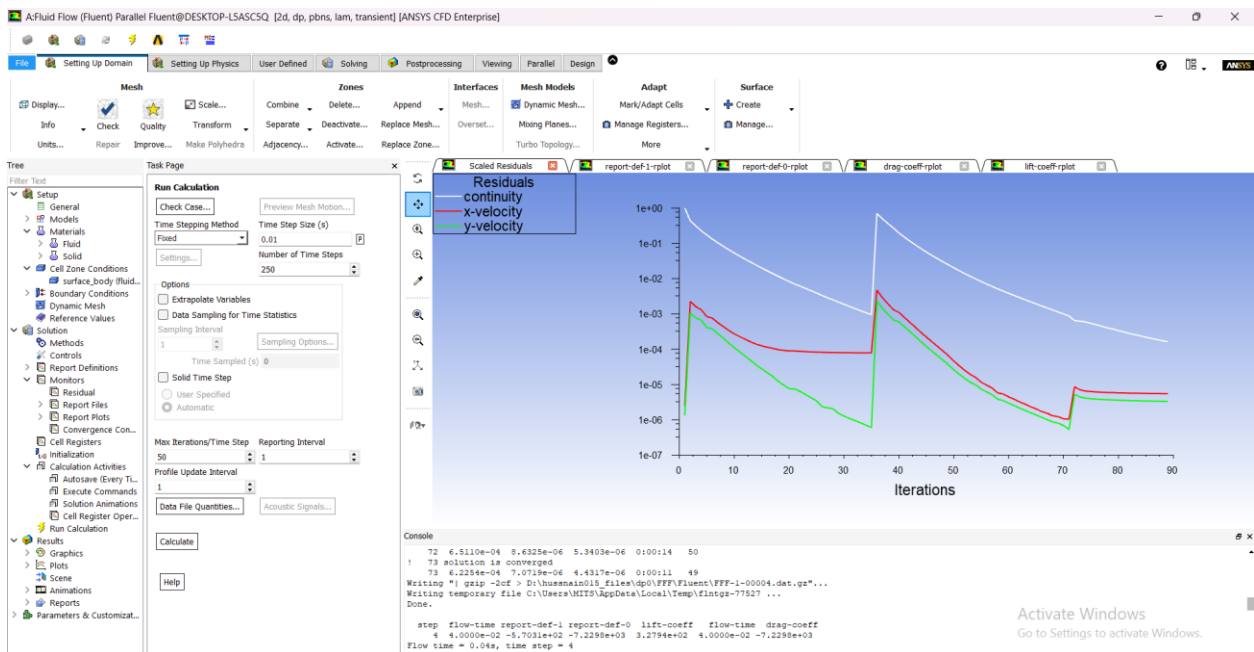
5-RESULTS AND ANALYSIS:

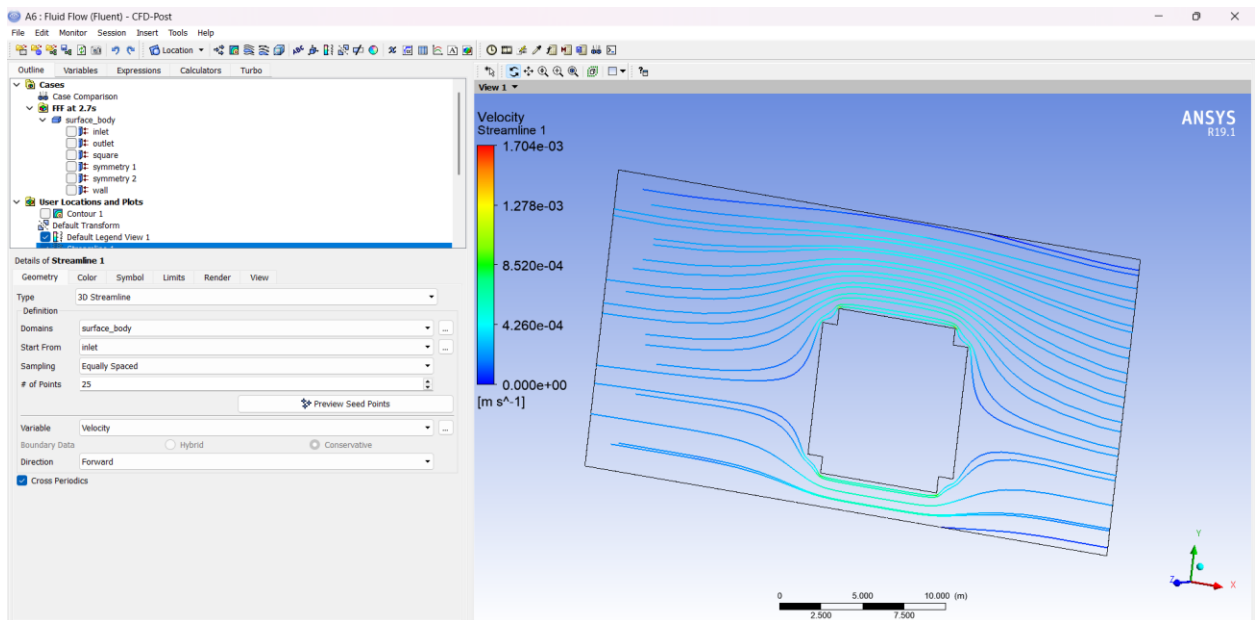
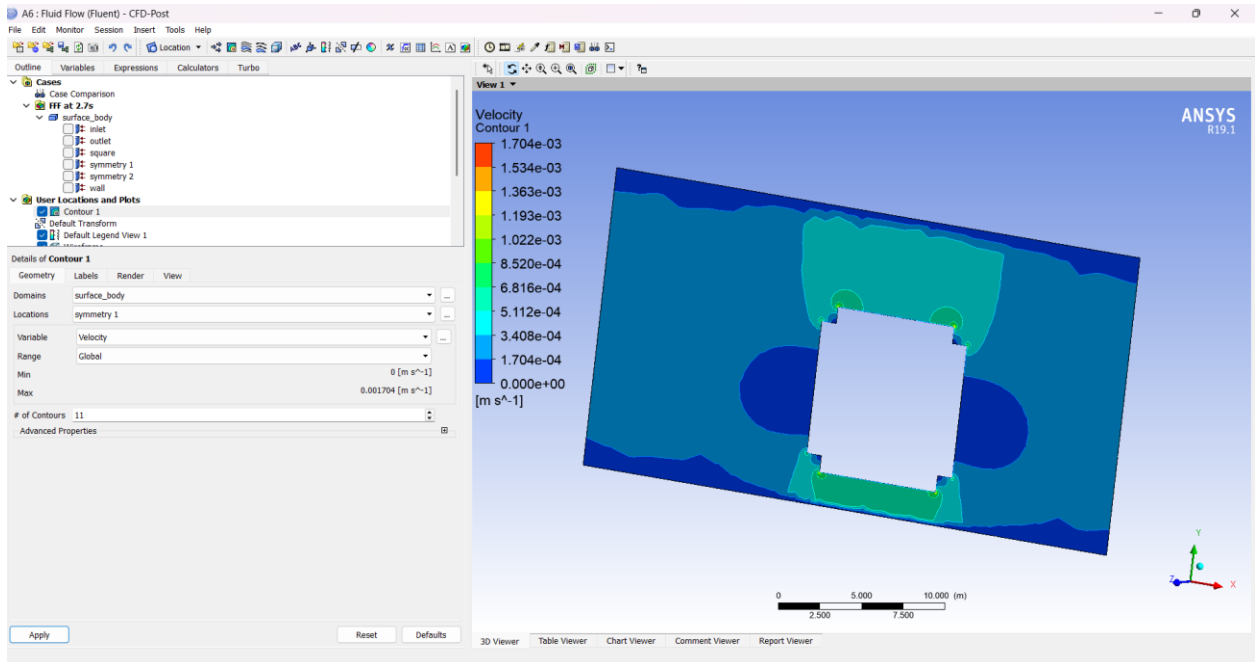
For results, we have plotted pressure and velocity contours as shown below: At left side, red color shows the high pressure which is reduced due to the Bernoulli's principle and the effects of flow separation. The Bernoulli's principle states that as the velocity of a fluid increases, the pressure decreases. 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 favourable 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.





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.

