

||JAI SHRI GURUDEV||



# S J C Institute of Technology, Chickballapur

DEPARTMENT OF AERONAUTICAL ENGINEERING

MINI PROJECT

ON

“Modelling of 2D Incompressible And Inviscid Flow Over A  
Cylinder Of 10cm Diameter And Plotting of Pressure Distribution  
And Velocity Vector For Subsonic Mach Number ”

Subject code:18AE72

Subject: COMPUTATIONAL FLUID DYNAMICS

## TEAM MEMBERS:

NAME: AMMAJAN B	USN:1SJ16AE003
NAME: NAGAMANI J V	USN:1SJ18AE038
NAME: SAI KEERTHAN	USN:1SJ18AE049
NAME: VAISHAK M	USN:1SJ18AE056
NAME: VARUN M	USN :1SJ18AE057
NAME: VEERESH	USN:1SJ18AE058

6/6

Mentor,  
Prof. Deepa M S  
Professor and HOD  
Department of Aeronautical Engineering  
S. J. C Institute of technology ,Chikkaballapur

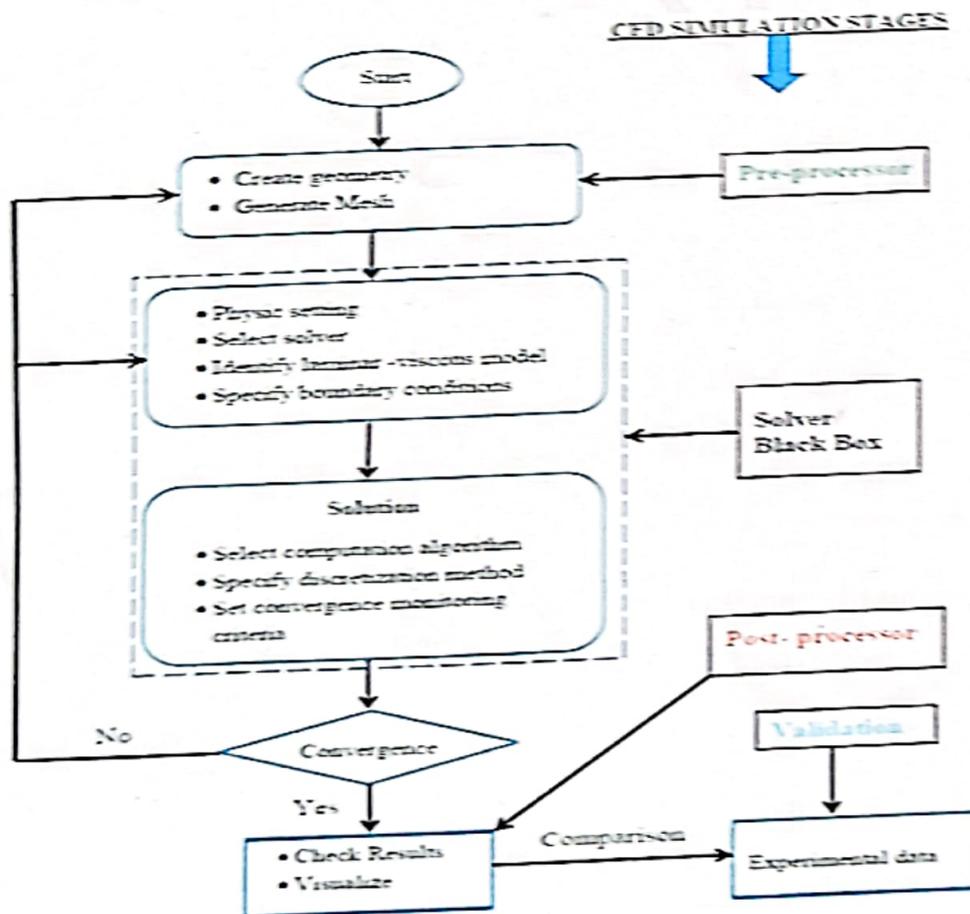
## INDEX

- INTRODUCTION
- METHODOLOGY
- PRE-PROCESSING
- SOLUTION
- POST-PROCESSING
- CONCLUSION

## INTRODUCTION:

- Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyse and solve problems that involve fluid flows.
- Computation fluid dynamics (CFD) is an engineering tool used to simulate the action of thermo-fluids in a system. It is used by many industries in their development work to analyse, optimise and verify the performance of designs before costly prototypes and physical tests.
- By applying CFD, even complex problems can be solved using a numerical iteration process. A commonly used method involves the Navier–Stokes equations which represent a system of equations for continuity, impulse and energy conservation of a fluid.
- Engineering fields where CFD analyses are frequently used are for example aerodynamics and hydrodynamics, where quantities such as lift and drag or field properties as pressures and velocities are obtained. Fluid dynamics is involved with physical laws in the form of partial differential equations.

## METHODOLOGY



**PRE-PROCESSING****Geometry:**

1. Modeling a Cylinder in ANSYS using Design-Modular.
2. Creating in the 2-Dimensional.
3. Give the dimensions for the Cylinder i.e, Diameter of the Cylinder is 10cm
4. Create a boundary for the Cylinder as shown below.

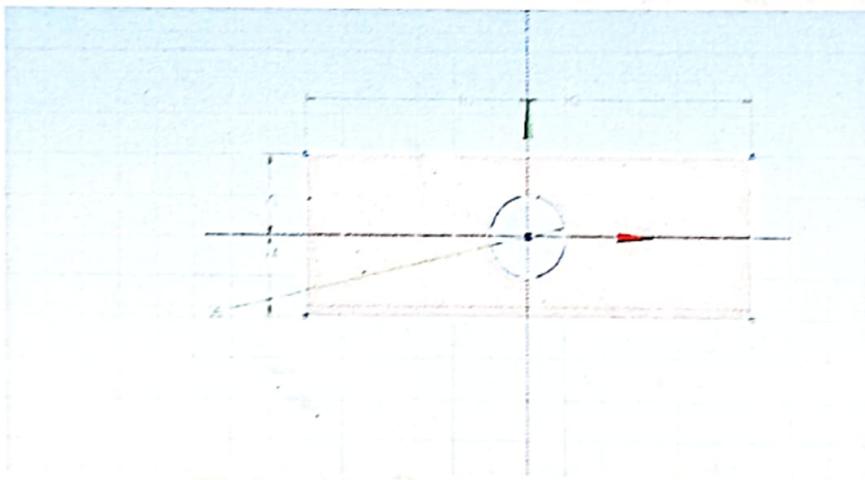


Fig 1. Geometry

Dimensions: 5

D5	10 cm
H1	30 cm
H2	30 cm
V3	10 cm
V4	10 cm

Fig .2 Dimensions

## MESHING :

1. Now Mesh the geometry which is been created.
2. By doing meshing we will get accurate result.
3. In this meshing is been done with Triangle shape.
4. And fine meshing is been given.

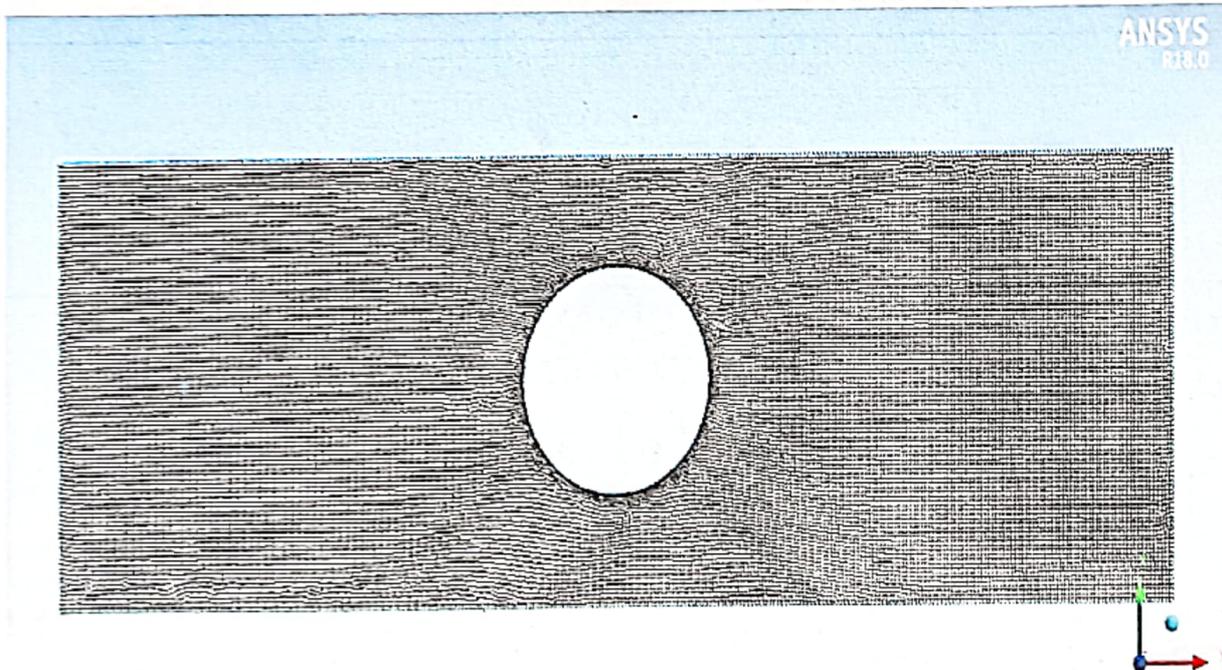
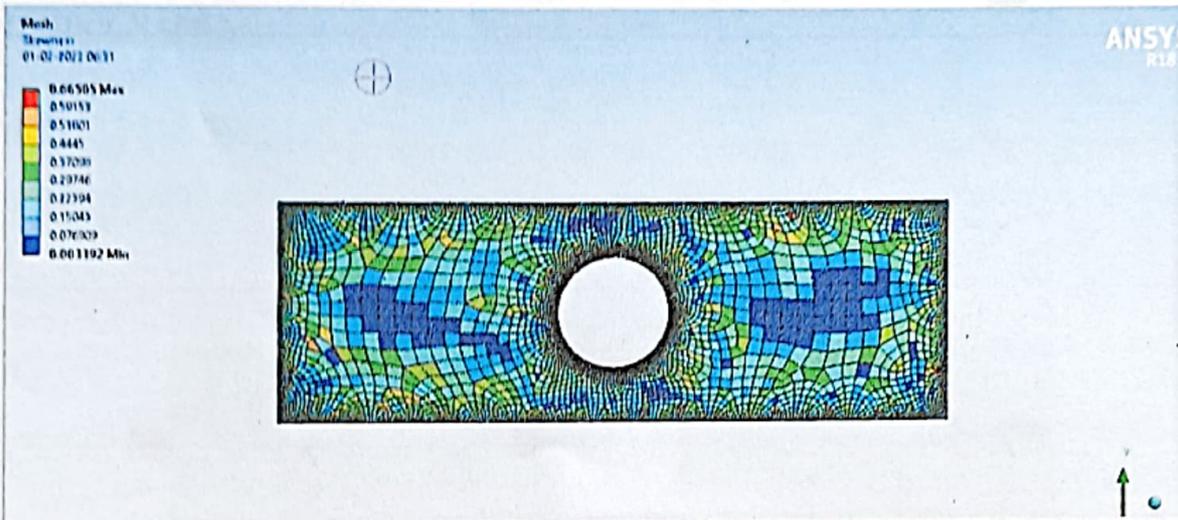


FIG.3 meshing

<b>Nodes</b>	<b>37556</b>
Elements	37011

SKEWNESS OF CYLINDER:



Skewness Ratio

0.003392-0.66050

ORTHOGONALITY QUALITY :

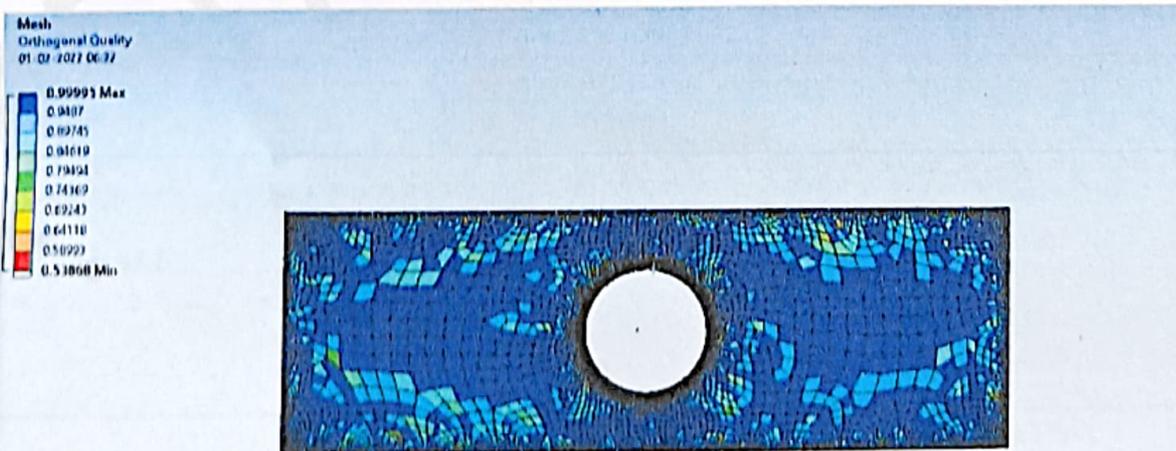


Fig .5

**Orthogonal Quality**

**0.538-0.9995**

**BOUNDARY NAMING:**

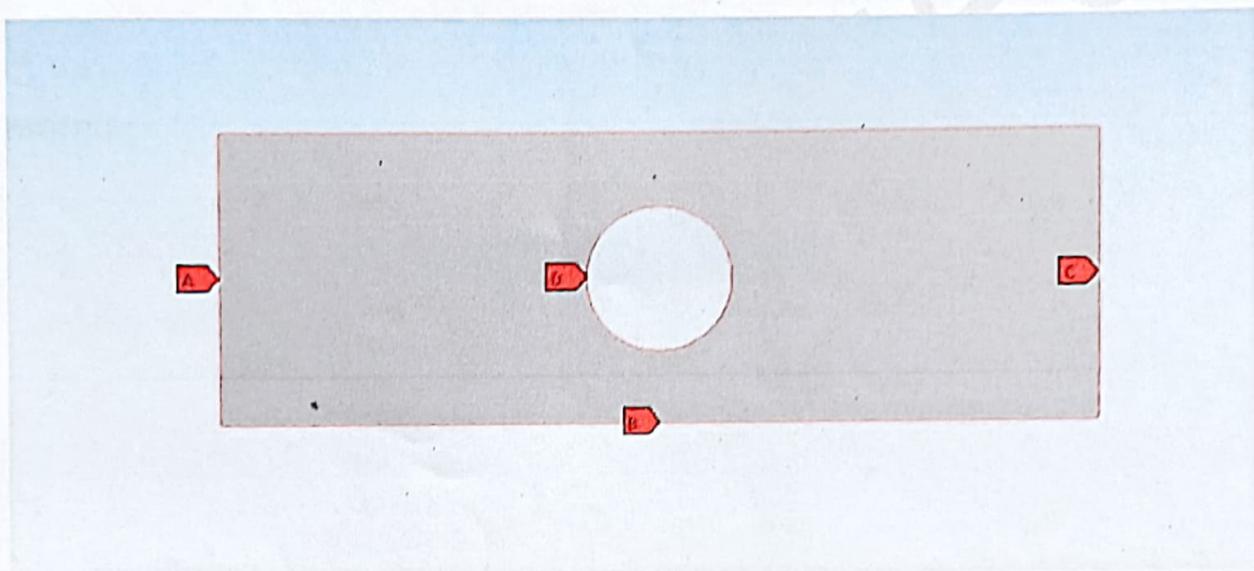


Fig.6

01-02-2022 06:27

<b>A</b>	inlet
<b>B</b>	wall
<b>C</b>	outlet
<b>D</b>	cylinder

**SOLVER:**

**GENERAL;**

General

Mesh

Scale...

Check

Report Quality

Display...

Solver

Type

- Pressure-Based
- Density-Based

Velocity Formulation

- Absolute
- Relative

Time

- Steady
- Transient

2D Space

- Planar
- Axisymmetric
- Axisymmetric Swirl

Gravity Units...

Help

MODEL:

Models

- Multiphase - Off
- Energy - On
- Viscous - Inviscid
- Radiation - Off
- Heat Exchanger - Off
- Species - Off
- Discrete Phase - Off
- Acoustics - Off
- Electric Potential - Off

MATERIAL TYPE:

**Create/Edit Materials**

Name:  Material Type:  Order Materials by:  Name  Chemical Formula

Chemical Formula:  Fluent Fluid Materials:  Fluent Database...  
User Defined Database...

Properties

Density (kg/m<sup>3</sup>): incompressible ideal-gas  Edit

Cp (Specific Heat) (J/kg-K): constant  Edit

Molecular Weight (kg/kmol): constant  Edit

**BOUNDARY CONDITIONS:**

**INLET:**

**Pressure Inlet**

Zone Name:

Momentum Thermal Radiation Species DPM Multiphase Potential UDS

Reference Frame: Absolute

Gauge Total Pressure (pascal):  constant

Supersonic/Initial Gauge Pressure (pascal):  constant

Direction Specification Method: Normal to Boundary

OK Cancel Help

**OUTLET:**

**Pressure Outlet**

Zone Name:

Momentum Thermal Radiation Species DPM Multiphase Potential UDS

Backflow Reference Frame: Absolute

Gauge Pressure (pascal):  constant

Backflow Direction Specification Method: Normal to Boundary

Backflow Pressure Specification: Total Pressure

Average Pressure Specification

Target Mass Flow Rate

OK Cancel Help

**SOLUTION INITIALIZATION :**

Solution Initialization

Initialization Methods

- Hybrid Initialization
- Standard Initialization

Compute from

Reference Frame

- Relative to Cell Zone
- Absolute

Initial Values

Gauge Pressure (pascal)

X Velocity (m/s)

Y Velocity (m/s)

Temperature (k)

Initialize   Reset   Patch...

Reset DPM Sources   Reset Statistics

**RUN CALUCULATION:**

Run Calculation

Check Case...

Update Dynamic Mesh...

Number of Iterations

2000

Reporting Interval

1000

Profile Update Interval

1

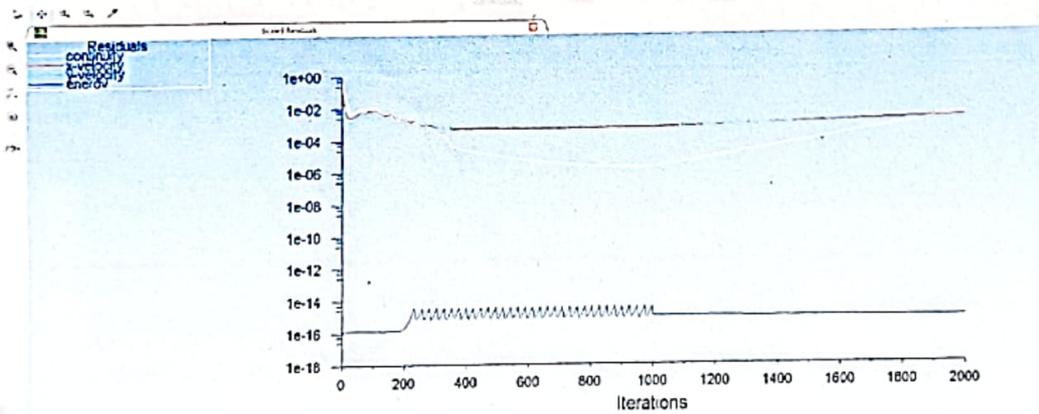
Data File Quantities...

Acoustic Signals...

Calculate

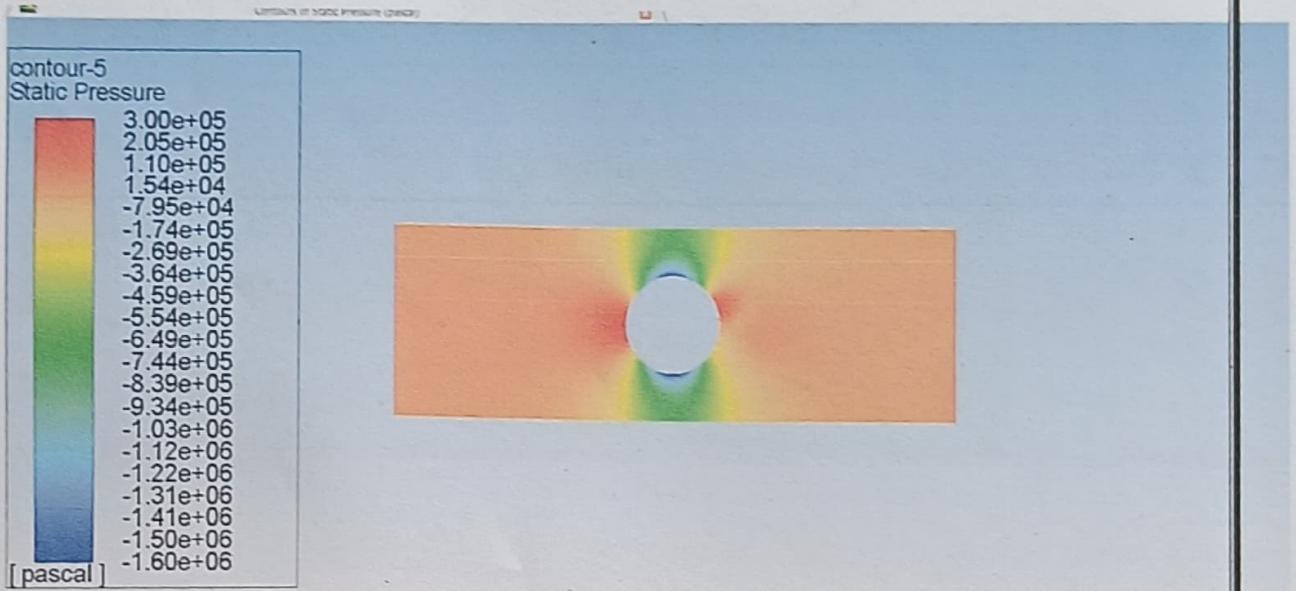
Help

CALCULATED GRAPH:

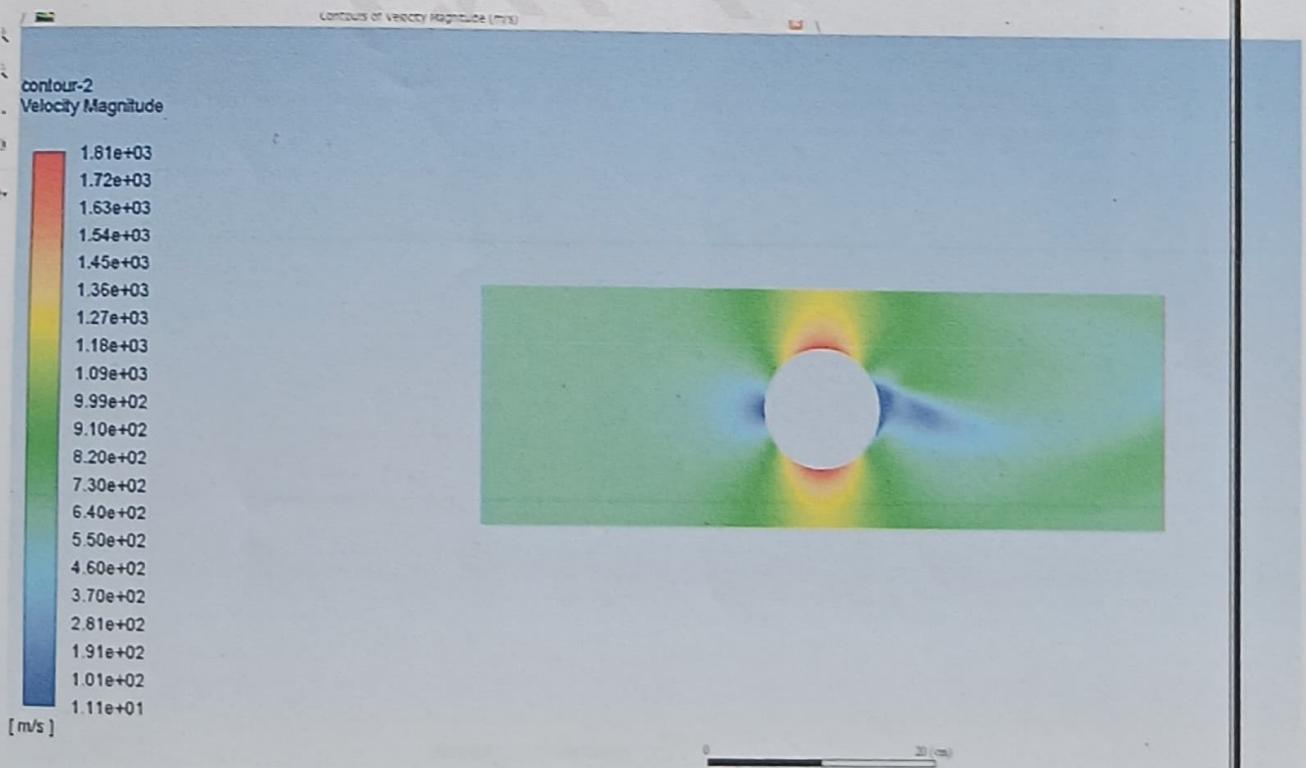


## POST PROCESSING:

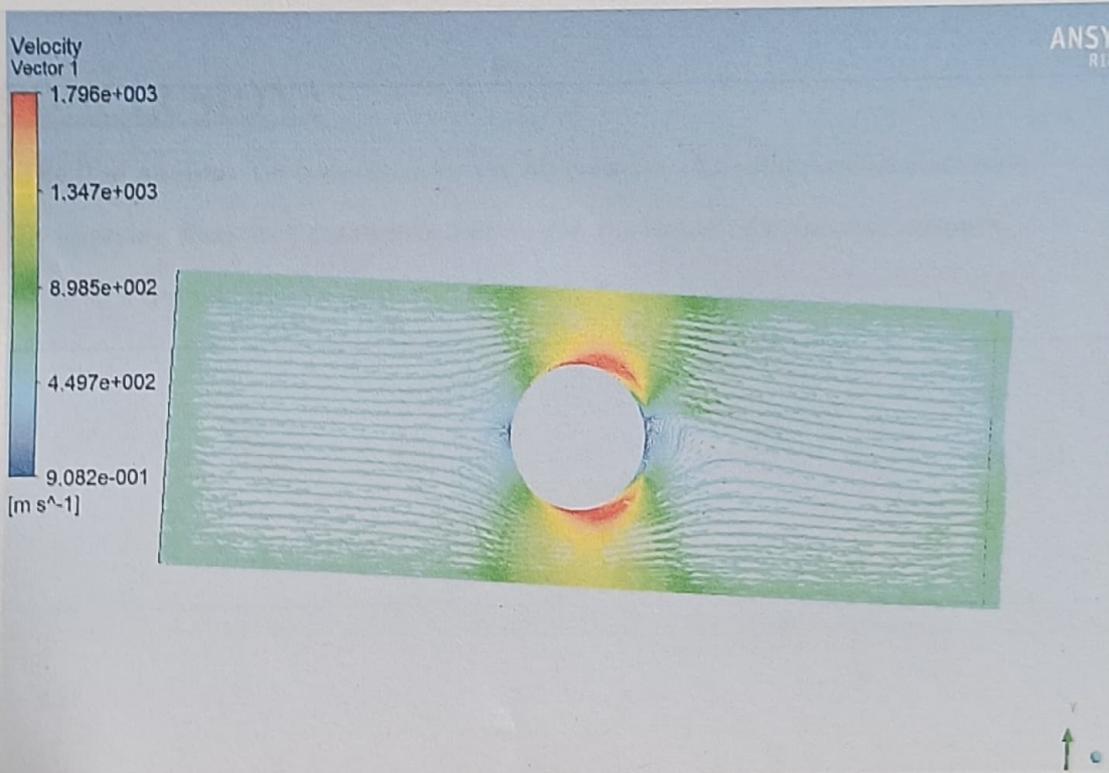
### STATIC PRESSURE COUNTER:



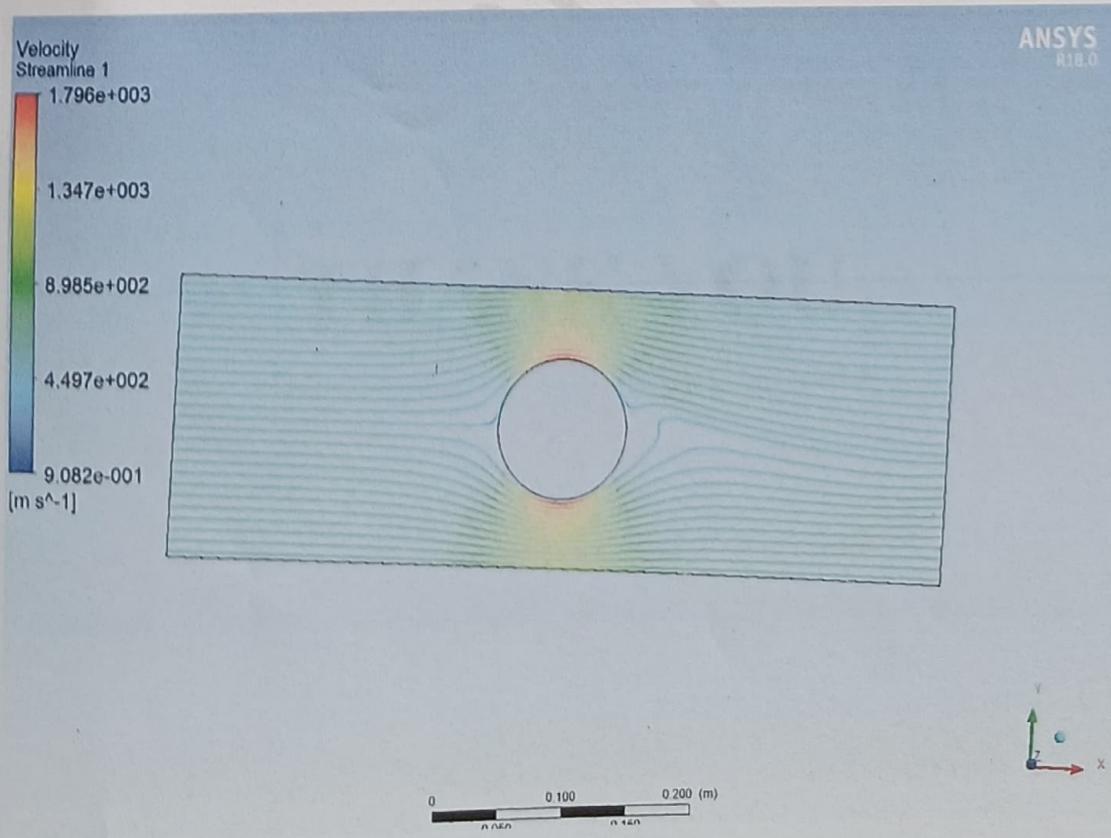
### VELOCITY MAGNITUDE:



VELOCITY VECTOR:



VELOCITY STREAMLINE :



## CONCLUSION :

The flow analysis has been done for the 2D cylinder of incompressible & inviscid by applying Boundary conditions and we got the Result of pressure Contours, velocity Contours and velocity vector.

**THANK YOU**