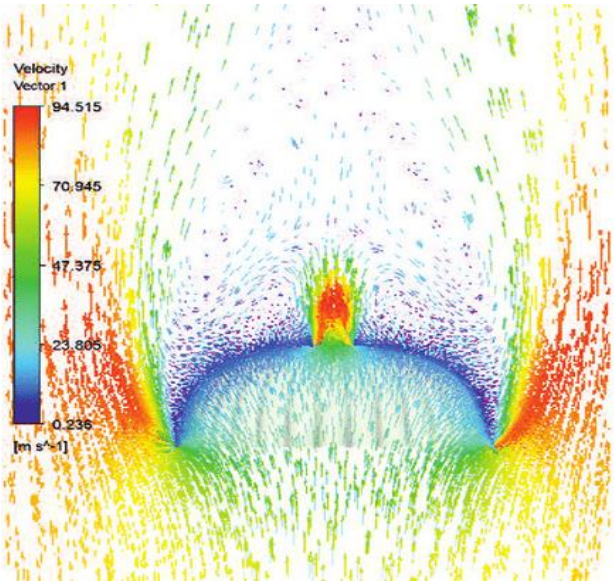
DEPARTMENT OF AEROSPACE ENGINEERING

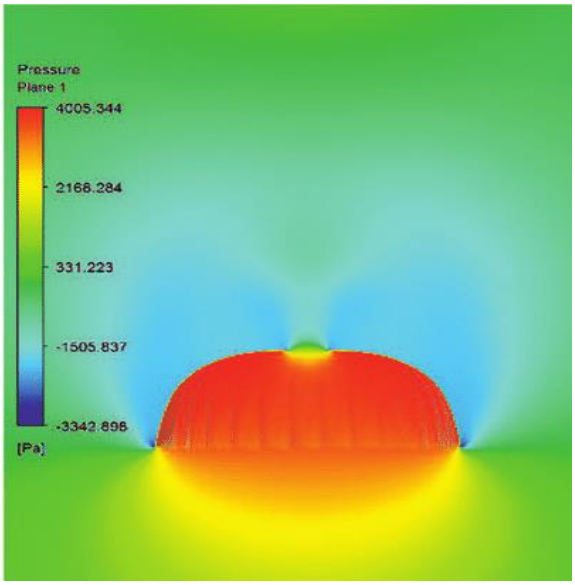
Faculty Name	:	**Dr.A.Arun Negemiya, ASP/ Aero**	Academic Year	:	**2024-2025 (Even)**
Year & Branch	:	**III AEROSPACE**	Semester	:	**VI**
Course	:	**19ASB304 - Computational Fluid Dynamics for Aerospace Application**			

UNIT V – FLOW FIELD ANALYSIS AND TURBULENCE MODELS

Overview of Flow Field Analysis in CFD

In Computational Fluid Dynamics (CFD), flow field analysis involves numerical methods to simulate and analyze fluid flow behavior, including properties like velocity, pressure, and temperature, to understand complex fluid dynamics problems.





- **Numerical Simulation:**

CFD uses computer simulations to model fluid flow, solving equations that govern fluid motion, such as the Navier-Stokes equations.

- **Analyzing Fluid Properties:**

The analysis focuses on understanding various fluid properties like velocity, pressure, density, temperature, and viscosity within the flow field.

- **Applications:**

CFD is used in diverse fields like aerospace (aerodynamics), automotive (engine cooling), and industrial processes (heat transfer).

- **Key Concepts:**
    - **Flow Field:** A multi-dimensional vector and scalar field that describes the physics of a fluid flow in space and time.
    - **Boundary Conditions:** These define the conditions at the edges of the simulation domain, such as inlet velocities, outlet pressures, or wall temperatures.
    - **Mesh Generation:** The computational domain is divided into a mesh of cells or elements, which are necessary for the numerical solution.
    - **Solvers:** These are the algorithms used to solve the governing equations and calculate the flow field.
    - **Post-processing:** After the simulation, results are visualized and analyzed to gain insights into the flow behavior.
- **CFD Analysis Process:**
    - **Problem Formulation:** Define the flow problem and its objectives.
    - **Geometry Modeling:** Create a 3D model of the flow domain.
    - **Mesh Generation:** Divide the domain into a mesh.
    - **Boundary Conditions:** Define the conditions at the boundaries.
    - **Simulation Setup:** Choose the simulation strategy and parameters.
    - **Simulation:** Perform the numerical calculations.
    - **Post-processing:** Analyze and visualize the results.

Benefits of Flow Field Analysis in CFD:

- **Predictive Capabilities:**

CFD allows engineers to predict fluid flow behavior and its impact on a system or product.

- **Optimization:**

CFD simulations can help optimize designs by identifying areas of poor flow or high pressure.

- **Cost and Time Savings:**

CFD simulations can reduce the need for expensive and time-consuming physical experiments.

- **Detailed Insights:**

CFD provides detailed information about the flow field, including velocity profiles, pressure distributions, and temperature gradients