INSTITUTE OF AERONAUTICAL ENGINEERING
(Autonomous)
Dundigal, Hyderabad - 500043, Telangana

AEROSPACE ENGINEERING

ATTAINMENT OF COURSE OUTCOME - ACTION TAKEN REPORT

Name of the faculty:	**Dr. ATHOTA RATHAN**	Department:	**Aerospace Engineering**
Regulation:	**IARE - MT23**	Batch:	**2023-2025**
Course Name:	**Advanced Aerodynamics Laboratory**	Course Code:	**BAED11**
Semester:	**I**	Target Value:	**60% (1.8)**

Attainment of COs:

	Course Outcome	Direct Attainment	Indirect Attainment	Overall Attainment	Observation
CO1	Apply the philosophy behind the computational fluid dynamics for recognizing flow properties in solving fluids and heat transfer problems.	1.20	0.00	1.2	Not Attained
CO2	Select the structured, unstructured mesh and multi-blocking strategy in basic, complex geometries and flow domains for computing aerodynamic characteristics.	1.20	0.00	1.2	Not Attained
CO3	Identify the appropriate physical boundary conditions for attaining the precise results of fluid flow over a body.	1.20	0.00	1.2	Not Attained
CO4	Choose the suitable numerical modeling and schemes for computational simulations of aerodynamics and thermo-fluid problems using ANSYS.	1.20	0.00	1.2	Not Attained
CO5	Analyze the numerical solution of fluid flow problems using flow visualization Software's and wind tunnel for recognizing the flow physics in and around the supersonic intake and free jet.	1.20	0.00	1.2	Not Attained
CO6	Make use of the Wind Tunnel for predicting the profile drag using boundary layer and wake momentum theory.	1.20	0.00	1.2	Not Attained

Action Taken Report: (To be filled by the concerned faculty / course coordinator)

CO1: applied the principles of Computational Fluid Dynamics (CFD) to analyze and interpret flow properties for solving fluid flow and heat transfer problems.

CO2: Apply appropriate mesh generation strategies—structured, unstructured, and multi-blocking—for solving flow and heat transfer problems in simple and complex geometries.

CO3: To equip students with the ability to select and apply appropriate physical boundary conditions in CFD simulations to achieve accurate and realistic solutions f

CO4: enabled students to select appropriate numerical models, in ANSYS Fluent for accurate computational simulations of aerodynamic and thermo-fluid problems.

CO5: Developed an understanding of compressible flow behavior ithrough CFD simulations and wind tunnel testing, and to validate numerical results using flow visualization tools.

CO6: Enabled students to experimentallyy using **boundary layer measurements** and **wake momentum theory** in a subsonic wind tunnel.

Course Coordinator **Mentor** **Head of the Department**

Head of the Department
Aeronautical Engineering
INSTITUTE OF AERONAUTICAL ENGINEERING
Dundigal, Hyderabad - 500 043