A J INSTITUTE OF ENGINEERING AND TECHNOLOGY
NH-66, Kottara Chowki, Mangaluru -575006, D. K District, Karnataka, INDIA.
A Unit of Laxmi Memorial Education Trust®
(Approved by AICTE, New Delhi. Affiliated to Visveswaraya Technological University, Belagavi)
Accredited by NAAC & NBA (BE: CV, CSE, ECE, ISE, ME)

Date:23/03/2026

Report on one day worshop on "Fluid Analysis using ANSYS"

Name of the Program:	Fluid Analysis using ANSYS	Program Dates & Timings:	20-03-2026 9.00 AM	
Name & Details of the Resource Person:	Dr. Sreejith B K			
Organized by (Clubs/ Dept.)	ARMS	In Association with (clubs)		
Number of Participants	Students	38	Faculty	-
Program Outcome (PO) Mapping	PO1, PO2, PO4, PO12			
Coordinators	Dr. Vighnesha Nayak, Dr. Nithesh K			

About the Program:

The Department of Mechanical Engineering, under the banner of the Association of Royal Mechanical Students (ARMS), organized a **one-day workshop on "Fluid Analysis using ANSYS"** at AJ Institute of Engineering and Technology, Mangaluru.

The primary objective of this workshop was to provide students with hands-on exposure to computational fluid dynamics (CFD) using ANSYS software and to bridge the gap between theoretical fluid mechanics and practical simulation-based analysis.

The session was conducted by **Dr. Sreejith B K, Associate Professor, Department of Mechanical Engineering, AJIET** who has extensive experience in the field of fluid mechanics and computational analysis. The workshop began with an introduction to the fundamentals of fluid flow analysis and the importance of CFD in modern engineering applications such as aerodynamics, heat transfer, and industrial flow systems.

Dr. Sreejith B K introduced students to the ANSYS Workbench environment and explained the complete workflow involved in fluid analysis.

Key topics covered during the workshop include:

- Introduction to CFD and its engineering applications
- Pre-processing: Geometry creation and mesh generation
- Setting up boundary conditions and solver parameters
- Simulation of fluid flow problems using ANSYS Fluent
- Post-processing and interpretation of results such as velocity, pressure, and contour plots

The workshop was highly interactive, with students actively participating in hands-on sessions. They performed basic simulations and gained practical knowledge of using ANSYS tools for analyzing fluid flow behavior.

The session emphasized the importance of simulation in design optimization, reducing experimental costs, and improving engineering efficiency. Students were also encouraged to explore advanced CFD applications and develop software skills relevant to industry requirements.

The program concluded with a vote of thanks, expressing gratitude to the Management, Principal, Head of the Department, faculty coordinators, and **Dr. Sreejith B K** for successfully conducting the workshop.

The program was concluded with a vote of thanks, expressing gratitude to the Management of AJIET, Principal, Head of the Department, faculty coordinators, and student organizers for their continuous support and efforts in making the event successful. Overall, the workshop was an informative and skill-oriented program that enhanced students' understanding of fluid analysis and provided valuable exposure to industry-relevant simulation tools.

Objectives: The program session enabled students to

1. To understand the fundamentals of computational fluid dynamics (CFD).
2. To provide hands-on training in ANSYS for fluid flow analysis.

3. To develop skills in pre-processing, solving, and post-processing of CFD problems.
4. To bridge the gap between theoretical fluid mechanics and simulation tools.
5. To enhance students' readiness for industry applications and research.

Outcomes: On successful completion of program, the student should able to:

1. Explain the basic principles of CFD and fluid flow analysis.
2. Perform basic fluid simulations using ANSYS software.
3. Interpret simulation results such as pressure, velocity, and flow patterns.
4. Apply CFD tools for solving engineering problems.
5. Recognize the importance of simulation in modern engineering design and analysis.

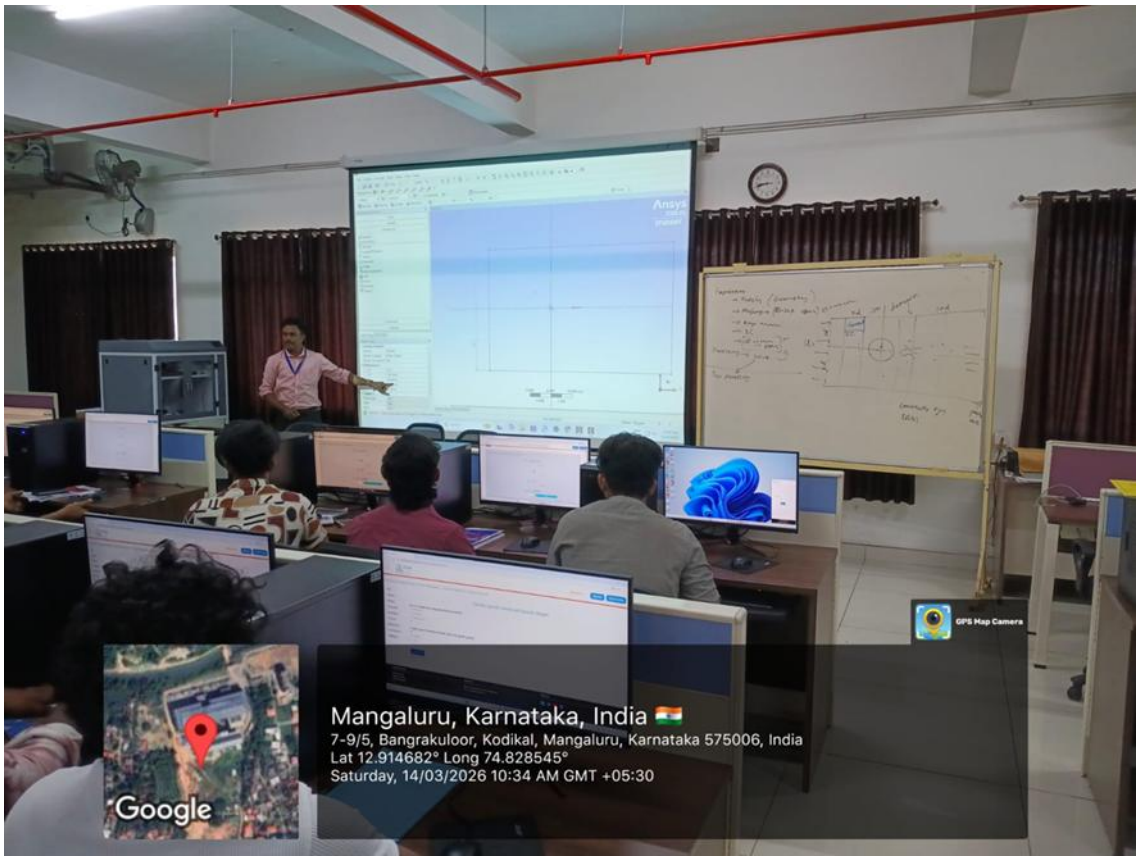
Articulation Matrix:

Course Outcomes	**Program Outcomes**											
	1	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**
1	2	2	-	-	-	-	-	-	1	-	-	-
2	2	2	-	-	2	-	-	-	1	-	-	-
3	-	-	1	1	2	-	-	-	1	-	-	-
4	2	-	-	-	2	-	-	-	1	-	-	-
5	2	-	-	-	2	-	-	-	1	-	-	-
Average	2	2	1	1	2	-	-	-	1	-	-	-

Photos:


Mangaluru, Karnataka, India
Google


Mangaluru, Karnataka, India
Google


Mangaluru, Karnataka, India
7-9/5, Bangrakuloor, Kodikal, Mangaluru, Karnataka 575006, India
Lat 12.914682° Long 74.828545°
Saturday, 14/03/2026 10:34 AM GMT +05:30
Google
GPS Map Camera

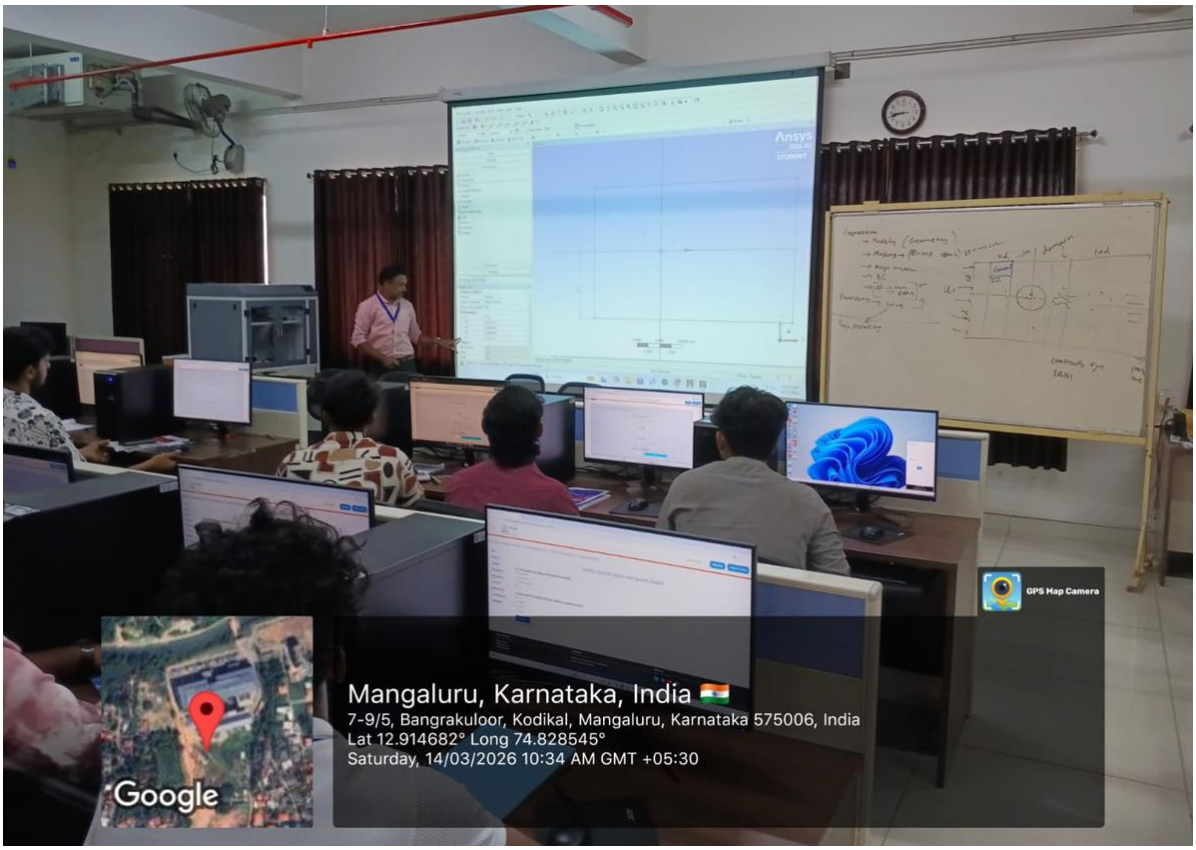
A J INSTITUTE OF ENGINEERING AND TECHNOLOGY

NH-66, Kottara Chowki, Mangaluru -575006, D. K District, Karnataka, INDIA.

A Unit of Laxmi Memorial Education Trust®

(Approved by AICTE, New Delhi. Affiliated to Visveswaraya Technological University, Belagavi)

Accredited by NAAC & NBA (BE: CV, CSE, ECE, ISE, ME)



Photos of workshop on Fluid Analysis using ANSYS

Event Coordinators **HOD**

Dean (Academics) **Principal**

For information: Vice president, LMET

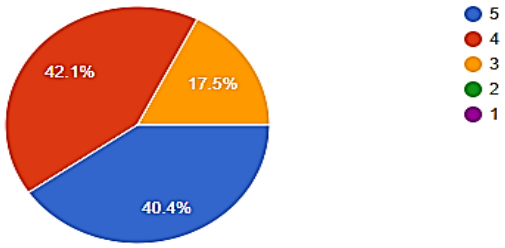
A J INSTITUTE OF ENGINEERING AND TECHNOLOGY
NH-66, Kottara Chowki, Mangaluru -575006, D. K District, Karnataka, INDIA.
A Unit of Laxmi Memorial Education Trust®
(Approved by AICTE, New Delhi. Affiliated to Visveswaraya Technological University, Belagavi)
Accredited by NAAC & NBA (BE: CV, CSE, ECE, ISE, ME)

Date: 23/03/2026

FEEDBACK ON 1-DAY WORSHOP ON "FLUID ANALYSIS USING ANSYS"

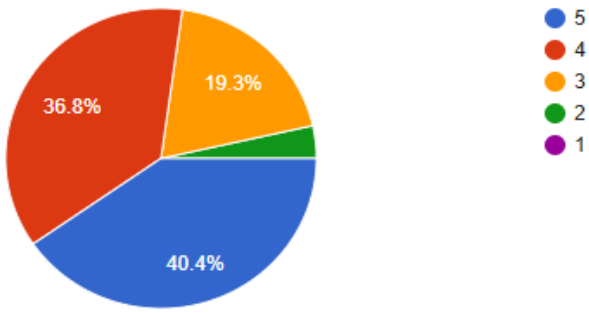
1. What is the confident level to perform basic fluid simulation in ANSYS?

38 responses



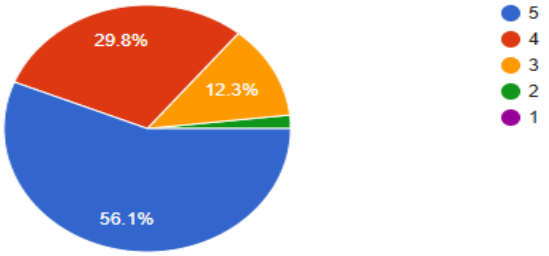
2. Rate the clarity of explanation by the resource person.

38 responses

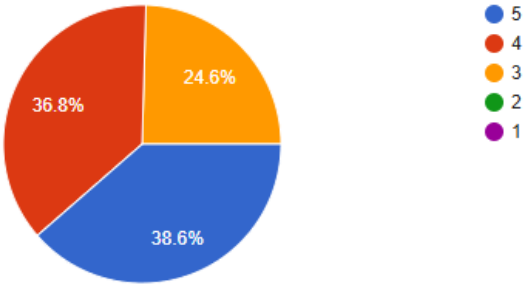


3. Rate the effectiveness of the ANSYS software demonstration and hands-on session.

38 responses

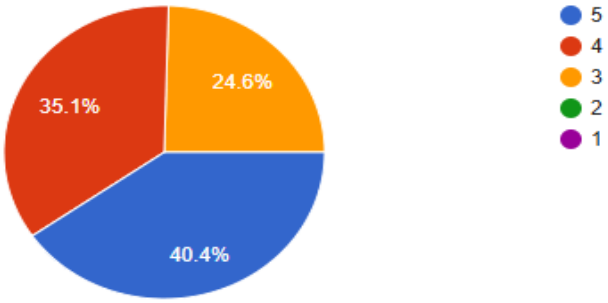


4. Rate the usefulness of the workshop for learning CFD and fluid analysis concepts.
38 responses



5. Rate the organization and coordination of the workshop.

38 responses



Event Coordinators **HOD**

Dean (Academics) **Principal**

For information: Vice president, LMET