MADANAPALLE INSTITUTE OF TECHNOLOGY & SCIENCE
(Deemed to be University)
Affiliated to JNTUA, Ananthapuramu & Approved by AICTE, New Delhi
NAAC Accredited with A+ Grade, NIRF India Rankings 2025 - Band: 201-300 (Engg.)
NBA Accredited - B.Tech. (CIVIL, CSE, ECE, EEE, MECH, CST), MBA & MCA

A Report on One week training program on "ANSYS & Space Claim" Organized by Department of Mechanical Engineering In association with IEI Student Chapter from 13.10.2025 to 18.10.2025

Report Submitted by: Mr. Manoj Kumar S Assistant Professor, Department of Mechanical Engineering.
Resource Person Details: Mr. GM Raja Mahendra, Mayinkrish Pvt Ltd, Bangalore
Total Participants: 50 students from the Department of ME
Venue: Siemens Computer Lab (EB019)
Mode of Conduct: Offline

The Department of Mechanical Engineering, Madanapalle Institute of Technology and Science, Andhra Pradesh, Madanapalle, in association with the IEI Student Chapter, MITS, Madanapalle, organized a one-week Training program on "**ANSYS & Space Claim**" from Oct 13, 2025, to Oct 18, 2025. A total of 50 students participated in this program, making the event a success.

A summary of the skill development program is as follows:

Dr. Dipankar Roy, Dean School of Engineering **& Dr. S. Baskaran**, Associate Professor and head of the Department of Mechanical Engineering, participated in the inaugural function. **Mr. Aravindhan D**, Asst.Prof./ME delivered the Welcome Address, followed by **Dr. S. Baskaran**, who welcomed the resource person and emphasized the importance of the training program. **Mr. Manoj Kumar**, Asst.Prof./ME, introduced the resource person and handed over the session to them.

Objectives of the program.

- To introduce the fundamentals of ANSYS Workbench and Space Claim interface.
- To train students in pre-processing, meshing, and boundary condition applications.
- To perform static, thermal, and modal analysis on real-time engineering components.
- To develop design and analysis skills relevant to industrial applications.
- To encourage students to use simulation tools for academic and project-based research.

Day-Wise Schedule and Topics Covered:

Day 1 – 13.10.2025 (Monday)
Topic: *Introduction to ANSYS Workbench*

- Covered Analytical and Experimental Methods and explained their industrial significance.
- Introduced the Importance of Numerical Methods in engineering problem-solving.
- Provided an overview of CAD and CAE concepts and demonstrated their integration in design and analysis.

Day 2 – 14.10.2025 (Tuesday)

Topic: *Solid Modeling and Assembly.*

- Continued with Rigid Dynamics and completed case studies on Simple Pendulum and Oldham Coupling with detailed step-by-step explanations.
- Introduced Explicit Dynamics with a small Cricket Ball Impact Analysis problem to demonstrate real-time applications.

Day 3 – 15.10.2025 (Wednesday)

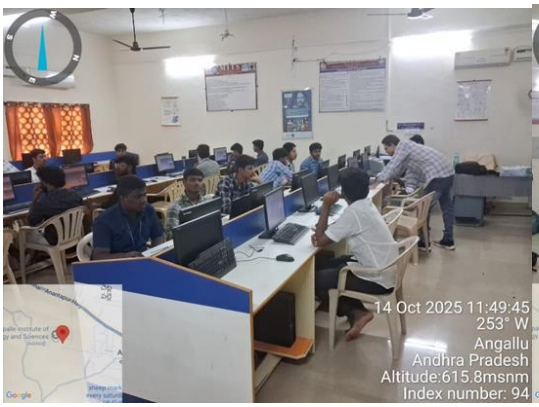
Topic: *Static Structural Analysis using ANSYS*

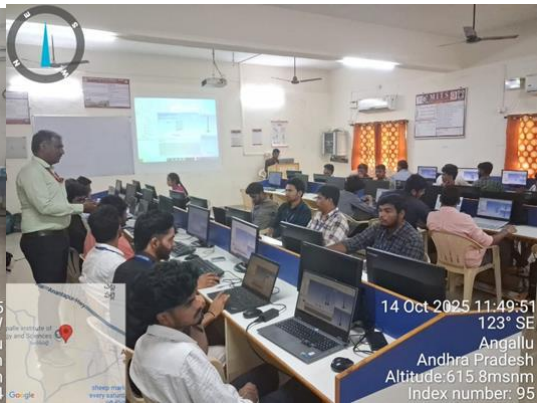
- Importing models from Space Claim to ANSYS Workbench.
- Applying boundary conditions and constraints.
- Meshing techniques and mesh quality evaluation.
- Solving static structural problems.
  Model Practiced: Structural analysis of a Cantilever Beam and a Bracket under load.

Day 4 – 16.10.2025 (Thursday)

Topic: Bucking analysis and some design models using space claim

- **Eigenvalue (linear) buckling** — finds theoretical critical load multipliers and mode shapes quickly. Good for a first check and for ideal, linear-elastic designs, but it *overestimates* capacity because it ignores imperfections and nonlinear effects. Use as first-pass.
- **Nonlinear (geometric + material) buckling / post-buckling** — includes initial imperfections, large-deformation (geometric) nonlinearity and plasticity. Use when you need realistic collapse loads or post-buckling behaviour. You typically feed an eigenmode as a perturbation and run a nonlinear static to capture realistic response.





Day 5 – 17.10.2025 (Friday)

Topic: *Coupled Field Analysis and Result Interpretation*

- Performing coupled thermal-structural analysis.
- Understanding solver convergence and result validation.
- Report generation and result visualization in contour plots.
  Model Practiced: Thermal-Structural Coupled analysis of a Brake Disc.

Day 6 – 18.10.2025 (Saturday)

Topic: *Project Demonstration*

- Students presented mini-projects based on analysis tasks.
- Evaluation of simulation accuracy and modeling efficiency.
- Discussion on industrial applications of ANSYS and future learning paths.

Models Presented by Students:

1. Static analysis of a Bike Frame.
2. Thermal stress analysis of a Heat Exchanger Plate.
3. Modal analysis of an Automotive Suspension Arm.

Outcomes of the Program:

Students gained **hands-on experience** in using ANSYS Workbench and Space Claim for modelling and simulation. Improved understanding of **structural, thermal, and modal analysis** concepts. Participants successfully completed **mini-projects** integrating theory and practical knowledge. Enhanced employability skills relevant to **design and simulation industries**.

Conclusion:

The one-week training program on "ANSYS & SpaceClaim" was highly beneficial for the participants. It provided an in-depth understanding of CAD-CAE integration and its applications in mechanical design and analysis. The initiative by the **Department of Mechanical Engineering** and the **IEI Student Chapter** was appreciated by students and faculty alike

I, Mr. Manoj Kumar S, take this opportunity to express my deep sense of gratitude to all those who have supported and contributed to the successful organization of this program. First and foremost, I extend my sincere thanks to the Management of our institution for their unwavering encouragement and continuous support in promoting academic and professional development initiatives. I am especially grateful **to Dr. P. Ramanathan**, Principal, for his kind permission, motivation, and financial support, which were instrumental in planning and executing this program effectively.