

TWO DAYS SOFTWARE WORKSHOP ON ANSYS WORKBENCH

Event No	ME001
Organizing Department	Mechanical Engineering
Associate Dept. NSC	Mechanical Engineering
Date	22/03/2024 to 23/03/2024 (2 Days)
Time	09:30 AM to 04:10 AM
Event Type	Workshop
Event Level	Dept. Level
Venue	HPC Lab
Total Participants	52
Faculty - External	3
Students - External	49

Related SDG



Resource Persons

Sl	Type	Name	Designation	Company	Email	Phone
1	Resource Person	Ms L Kavya	NVH Simulation Engineer	Volvo Group India Private Limited	kvyaloganathang@gmail.com	xxxxxxxxxx

Involved Staffs

Sl	Name	Role
1	Sathish S	Coordinator
2	Dharani Kumar S	Coordinator

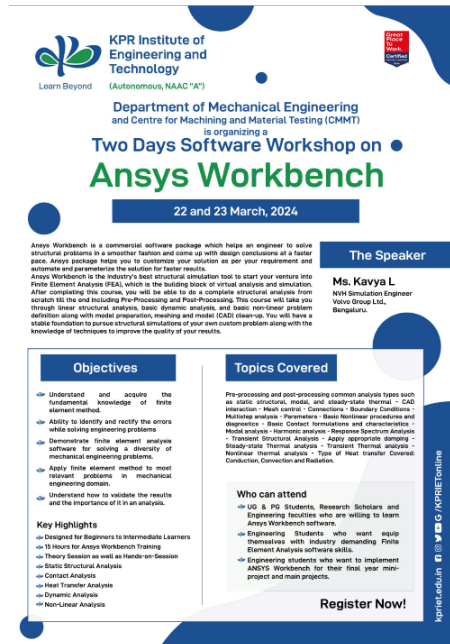
Outcome

External participants were able to analysis static structural problems. External participants were able to analysis heat transfer problems. External participants were able to analysis dynamic problems. External participants were able to analysis non-linear problems.

Event Summary

Day 1: The event was inaugurated by Ms. Saravana Gowri, a student of III Mechanical Engineering, who delivered the welcome address and introduced the esteemed guest speakers. The workshop aimed to shed light on various aspects of Computer-Aided Engineering (CAE) applications in the automobile industry. **Session 1:** Role of CAE in Automobile Industries Ms. L. Kavya, a seasoned professional in the field of CAE, kick-started the first session. She eloquently discussed the pivotal role of CAE in the realm of automobile industries. Her presentation highlighted how CAE techniques are instrumental in designing, analyzing, and optimizing automotive components and systems, leading to improved performance, safety, and cost-effectiveness. **Session 2:** Static Structural Analysis The second session was led by Dr. S. Dharani Kumar, an expert in structural analysis. The topics covered included importing and exporting material properties, basic meshing techniques, and defining boundary conditions. The participants also received hands-on training by solving two structural problems, which provided them with practical insights into the software's capabilities. **Session 3:** Contact Analysis Continuing the momentum, Dr. S. Dharani Kumar conducted the third session, focusing on contact analysis. Participants gained understanding about bonded and frictionless contact problems. **Day 2: Session 4:** Joints and Linkage Problems of IC Engine The second day of the workshop commenced with an insightful session on joints and linkage problems of internal combustion engines. This session, facilitated by domain experts, explored the challenges associated with engine components' connections and provided insights into effective solutions. **Session 5:** Thermal Problems. S. Dharani Kumar returned with a session on thermal problems. He explained steady-state thermal problems, encompassing conduction, convection, and radiation. The participants were exposed to practical problem-solving techniques using ANSYS Workbench, enhancing their comprehension of heat transfer phenomena. **Session 6:** Vibration Problems. The final technical session of the workshop focused on vibration problems. Dr. Dharani kumar elucidated modal analysis of an aircraft wing, harmonic analysis of a beam, and transient analysis of a beam. These sessions gave participants exposure to dynamic analysis techniques and their significance in engineering design and evaluation. Feedback Session and Certificate Distribution: Following the technical sessions, a feedback session was conducted, allowing

participants to express their opinions and suggestions for future improvements. The attendees lauded the workshop's organization, content, and the expertise of the speakers. The workshop concluded with the distribution of certificates to all participants, acknowledging their active engagement and successful completion of the ANSYS Workbench workshop.



KPR Institute of Engineering and Technology
 (Autonomous, NAAC "A")

Department of Mechanical Engineering
 and Centre for Machining and Material Testing (CMMT)

is organizing a

Two Days Software Workshop on Ansys Workbench

22 and 23 March, 2024

ANSYS Workbench is a commercial software package which helps an engineer to solve structural problems in a smoother fashion and come up with design conclusions at a faster pace. Ansys package helps you to convert your solution as per requirement and automate and parameterize the solution for faster results. Ansys Workbench is the industry's best structural simulation tool to start your venture into Finite Element Analysis (FEA), which is the building block of virtual analysis and simulation. After completing this course, you will be able to do a complete structural analysis from scratch till the end including Pre-Processing and Post-Processing. This course will take you through linear structural analysis, basic dynamic analysis, and basic non-linear problem definition along with model preparation, meshing and model (CAD) cleanup. You will have a stable foundation to pursue structural simulations of your own custom problem along with the knowledge of techniques to improve the quality of your results.

The Speaker
Ms. Kavaya L.
 NPL Simulation Engineer
 Volvo Group Ltd.,
 Bengaluru.

Objectives	Topics Covered
<ul style="list-style-type: none"> Understand and acquire the fundamental knowledge of finite element method. Ability to identify and rectify the errors while solving engineering problems. Demonstrate finite element analysis software for solving a diversity of mechanical engineering problems. Apply finite element method to most relevant problems in mechanical engineering domain. Understand how to validate the results and the importance of it in an analysis. 	<p>Pre-processing and post-processing common analysis types such as static structures, modal, and steady-state thermal - CAD Interaction - Mesh control - Connections - Boundary Conditions - Multiscale analysis - Parameters - Basic nonlinear procedures and equations - Basic Contact Formulations and characterizations - Model analysis - Harmonic analysis - Response Spectrum Analysis - Transient Structural Analysis - Linear static analysis - Steady-state Thermal analysis - Transient Thermal analysis - Nonlinear Thermal analysis - Type of Heat transfer Convection, Conduction and Radiation.</p>

Key Highlights

- Designed for Beginners to Intermediate Learners
- 10 Hours for Ansys Workbench Training
- Theory Session as well as Hands-on-Session
- Static Structural Analysis
- Contact Analysis
- Heat Transfer Analysis
- Dynamic Analysis
- Non-Linear Analysis

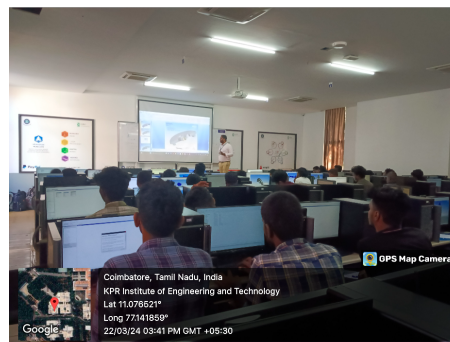
Who can attend

- UG & PG Students, Research Scholars and Engineering faculties who are willing to learn Ansys Workbench software.
- Engineering Students who want equip themselves with industry demanding Finite Element Analysis software skills.
- Engineering students who want to implement ANSYS Workbench for their final year mini-project and main projects.

Register Now!

kprietr.edu.in | KPRIETonline

[Click to View](#)



[Click to View](#)



[Click to View](#)

*** END ***