



Department of Mechanical Engineering
Academic Year 2021 – 2022 (Even Semester)

Degree, Semester & Branch: VI Semester B.E. Mechanical Engineering A

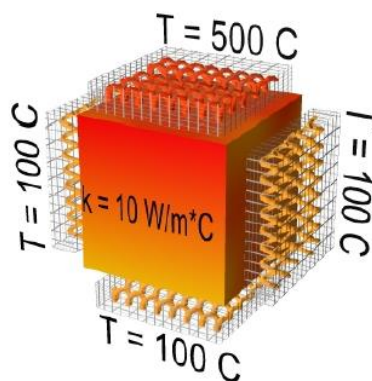
Course Code & Title: ME8692 Finite Element Analysis

Name of the Faculty member (s): Dr. J. Jabinth, AP/Mechanical

Innovative Practice Description

- **Unit / Topic:** Unit 2 / Problems in Heat Transfer
- **Course Outcome:** CO 2, Perform finite element formulations to solve one dimensional problems
- **Topic Learning Outcome:** TLO 5
- **Activity Chosen:** Thermal Analysis using ANSYS software (Theory to Practical)
- **Justification:**
As per stakeholder, M/s. Eleation's suggestion a hands on training in simulation software is taught to students.
- **Time Allotted for the Activity:** 40 minutes
- **Details of the Implementation:**

Students are taught to design a plane stress thermal problem using creo 7.0 software and the component is export in IGES format. The exported format is imported to ANSYS workbench. Thermal analysis module is selected from the type of analysis. The IGES component and the material properties were imported. The necessary boundary conditions are imported as shown in figure below to the designed model. The thermal distribution in a beam is determined.



- **CO – PO / PSO mapping:**

CO	PO1	PO2	PO5	PO12	PSO1	PSO4
CO2	3	2	3	1	1	2

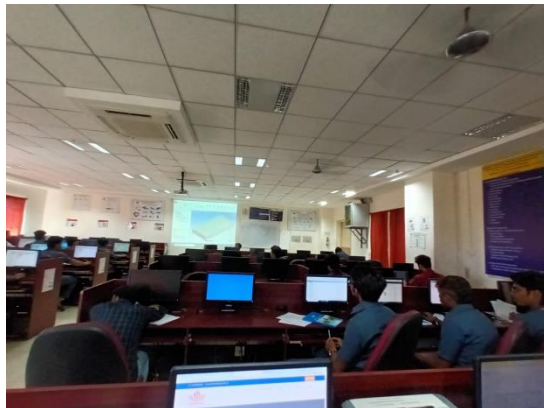
(1 – Low 2 – Moderate 3 – High)



- **PO / PSO mapped:**

Innovative practice	PO1	PO2	PO5	PO12	PSO1	PSO4
	3	2	3	1	1	2
Justification for correlation	Basic engineering knowledge is required	The type of problem need to be studied and analyzed	Modern tools like creo and Ansys is used	More different types of thermal problems can be solved using this method	Component development using Creo	Thermal simulation done using ANSYS software

- **Images / Screenshot of the practice:**



- **Reflective Critique:**

- ❖ *Feedback of practice from students and other stakeholders:*

Feedback from students attached separately

- ❖ *Benefit of the practice:* (E.g.: Outcome attainment would have increased due to innovative practice over conventional practice)

In future, Students would be able to do simulation analysis of thermal problems using ANSYS.

- ❖ *Challenges faced in implementation:*

Software must be available to every student. As it is a new software for students, it took time for them to grasp options.



RAMCO INSTITUTE OF TECHNOLOGY

Approved by AICTE, New Delhi & Affiliated to Anna University

Accredited by NAAC & An ISO 9001:2015 Certified Institution

NBA Accredited UG Programs: CSE, EEE, ECE and MECH

References:

- ❖ Finite Element Analysis using ANSYS 11.0 by Paleti Srinivas
- ❖ S. Md. Jalaludeen., “Finite Element Analysis in Engineering”, Anuradha Publications

Signature of Faculty Member

HOD

Student feedback on Innovative Practice (Thermal analysis in ANSYS)

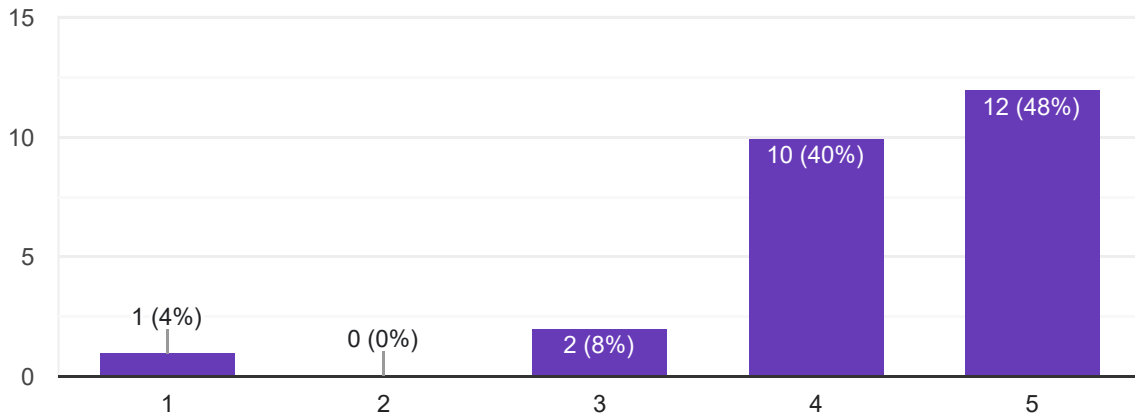
25 responses

[Publish analytics](#)

Instructor clarifies difficult aspects of this innovative activity.

 Copy

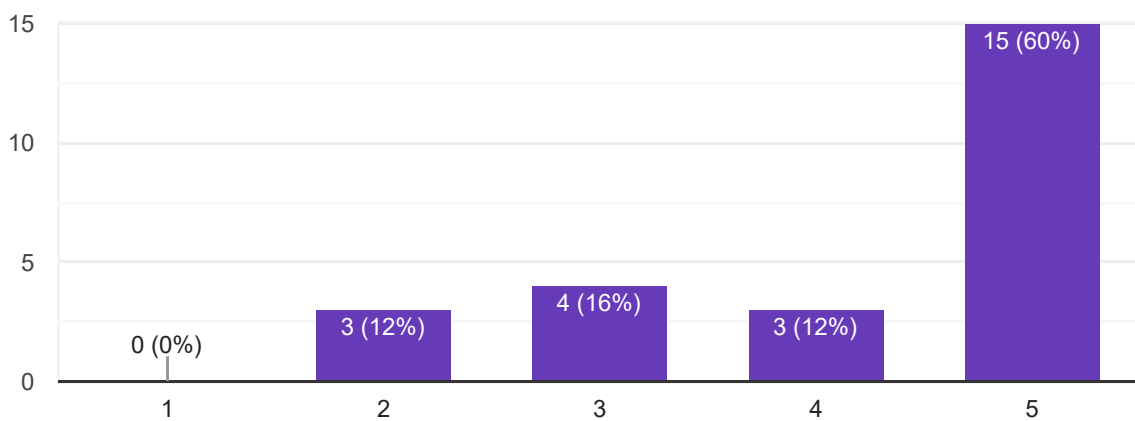
25 responses



This innovative activity improves my opinion about the content of the subject

 Copy

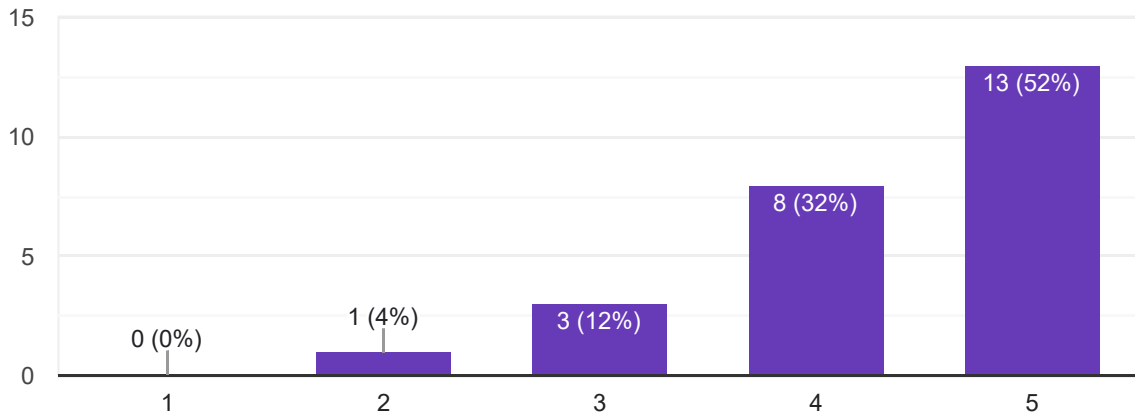
25 responses



I find new information about the topics and subjects using new technologies.



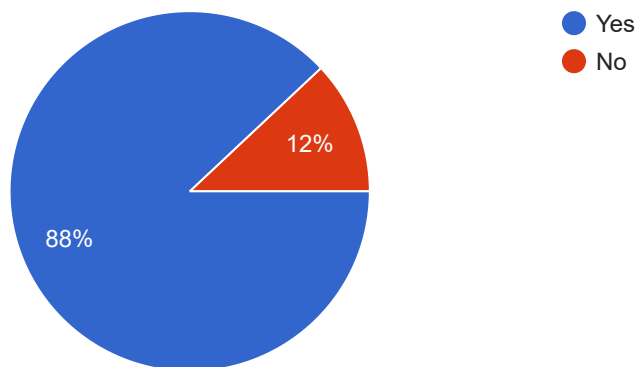
25 responses



I suggest this innovative practice to teach the topic for forthcoming students



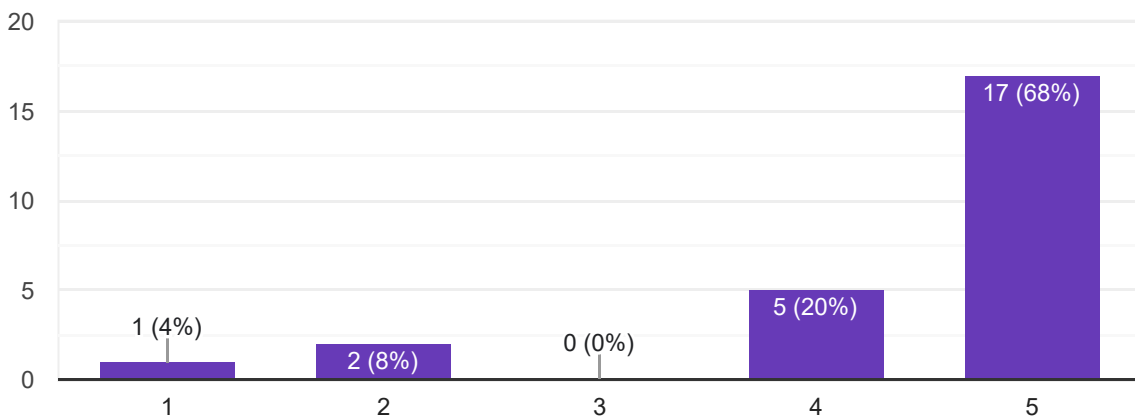
25 responses



This innovative activity builds any self-confidence to understand the content of the delivery.



25 responses



The most useful thing/skill I learned from this activity was...

25 responses

.

Nil

NA

I really understand FEA Problem analysis in software

Edgecam

Thermal analysis in ANSYS

Discussion

Numerical solving a wide variety of mechanical problems

Faster time to market and lower development costs

Learnt to do the temperature apply in surface

Nothing

ANSYS

N

.

Matrix

This content is neither created nor endorsed by Google. [Report Abuse](#) - [Terms of Service](#) - [Privacy Policy](#).

Google Forms

