LAKIREDDY BALI REDDY COLLEGE OF ENGINEERING
(AUTONOMOUS)
Accredited by NAAC & NBA (CSE, IT, ECE, EEE & ME)
Approved by AICTE, New Delhi and Affiliated to JNTUK, Kakinada
L.B.Reddy Nagar, Mylavaram-521230, Krishna Dist, Andhra Pradesh, India

DEPARTMENTS OF MECHANICAL ENGINEERING

- **REPORT ON EVENT : Student Certification Program on Finite Element Analysis using ANSYS from 14-06-2021 to 19-06-2021(Six day program)**

Event Type : SDP

Date / Duration : 14-06-2021 to 19-06-2021(Six day program)

Resource Person : Mr. K Harish Kumar - FEA Engineer EXXENTRIX Academy -Visakhapatnam

Name of Coordinator : B. CHAITANYA /A NAGESWARA RAO

Target Audience : IV-B.Tech STUDENTS

Total no of Participants: 44 Mechanical Engineering Students

Objective of the event:
This is a certification program on Finite element Analysis (FEA) using ANSYS and is specially meant for Engineering final students. The Finite Element Method (FEM) is a well-established technique for analysing the structural behaviour of mechanical components and systems. In recent years, the use of finite element analysis as a design tool has grown rapidly. ANSYS is a popular and well recognized general purpose finite element modelling package for numerically solving a large range of problems including static, dynamic mechanical, structural analysis (linear and nonlinear), heat transfer and fluid problems, as well as acoustic and electromagnetic problems. It is widely used in the mechanical, automobile, structural, chemical and aeronautical industries.

Outcome of event :
Introduce to the Finite Element Analysis (FEA) concepts and make familiar with the tools and techniques of the ANSYS software package. This SDP aims at providing **complete hands-on training** on FEA analysis. The SDP will help the participants to develop expertise on various aspects of ANSYS for FEA applications. The SDP serves the purpose of bringing together the engineers from various domains such as Structural, Thermal and Fluid Dynamics fields.

Feedback / Suggestions: Increase Laboratory Sessions, Real time case studies

Day to Day Report

14/06/2021

SESSION DETAILS:-

- Session started at 9.30 A.M.
- Registration of participants for the STUDENT CERTIFICATION PROGRAM ON FEA USING ANSYS
- Inauguration of program by Dr. S. Pichi Reddy-Head of the Department.
- Introduction to Finite Element Analysis theory and applications.
- Introduction to ANSYS software
- Demonstration on operating ANSYS environment.
- Steps in ANSYS solver
- Participants practiced and interacted with new ANSYS Platform software.

15/06/2021

SESSION DETAILS:-

- Session started at 10.00 A.M.
- Briefly explained ANSYS Design Modular Window, how to design model in ANSYS.
- Draw, modify too bars in Design Modular Window tool bars
- Participants performed operations draw tool bars like point, line, rectangular, polygon, and circle
- Edit tool options rotate, scale, mirror that was practiced

16/06/2021

SESSION DETAILS:-

- Session started at 10.00 A.M.
- Explained various commands types of meshing methods in ANSYS
- Demonstration on 2D and 3D model components are meshed
- Participants practiced 2D and 3D model components are meshed.
- Demonstrated how to consider and apply Boundary conditions for different models of problems
- Participants practiced mechanical components and aerospace components.

17/06/2021

MORNING SESSION:-

- Session started at 10.00 A.M.
- Explained Rack and gear problem with remote displacement/force options
- Mode imported in to ANSYS, performed meshing followed by boundary conditions and solved.
- Finally resultant momentum to drive the gear on pinion founded.
- Participants participated Rack and gear tutorial
- Explained different structural analysis problems like bars and trusses with procedures
- Participates participated structural analysis tutorials like bars and trusses

18/06/2021

SESSION DETAILS:-

- Session started at 10.00 A.M.
- Explained thermal analysis problems solved options
- Mode imported in to ANSYS, performed meshing followed by boundary conditions and solved.
- Finally heat flux and heat transfer analysis were studied.
- Participant's performed hands on session with thermal analysis of heat exchanger.

- Explained different thermal analysis problems like fins and heat pipes with procedures
- Participates participated structural analysis tutorials like bars and trusses

19/06/2021

SESSION DETAILS:-

- Session started at 10.00 A.M.
- Explained heat transfer analysis procedure in ANSYS
- Study state and transient temperature distribution steps explained
- Fin pin heat transfer analysis performed
- Participants practiced Fin pin heat transfer analysis using ANSYS tool
- Practice session was given to practice all the modules in related to FEA workbenches.
- 3.00PM valedictory started
- Dr. S. Pichi Reddy-Head of the Department addressed the gathering
- Certification of course completion was presented to participants.

Photographs:

Fig: Poster for ANSYS SDP

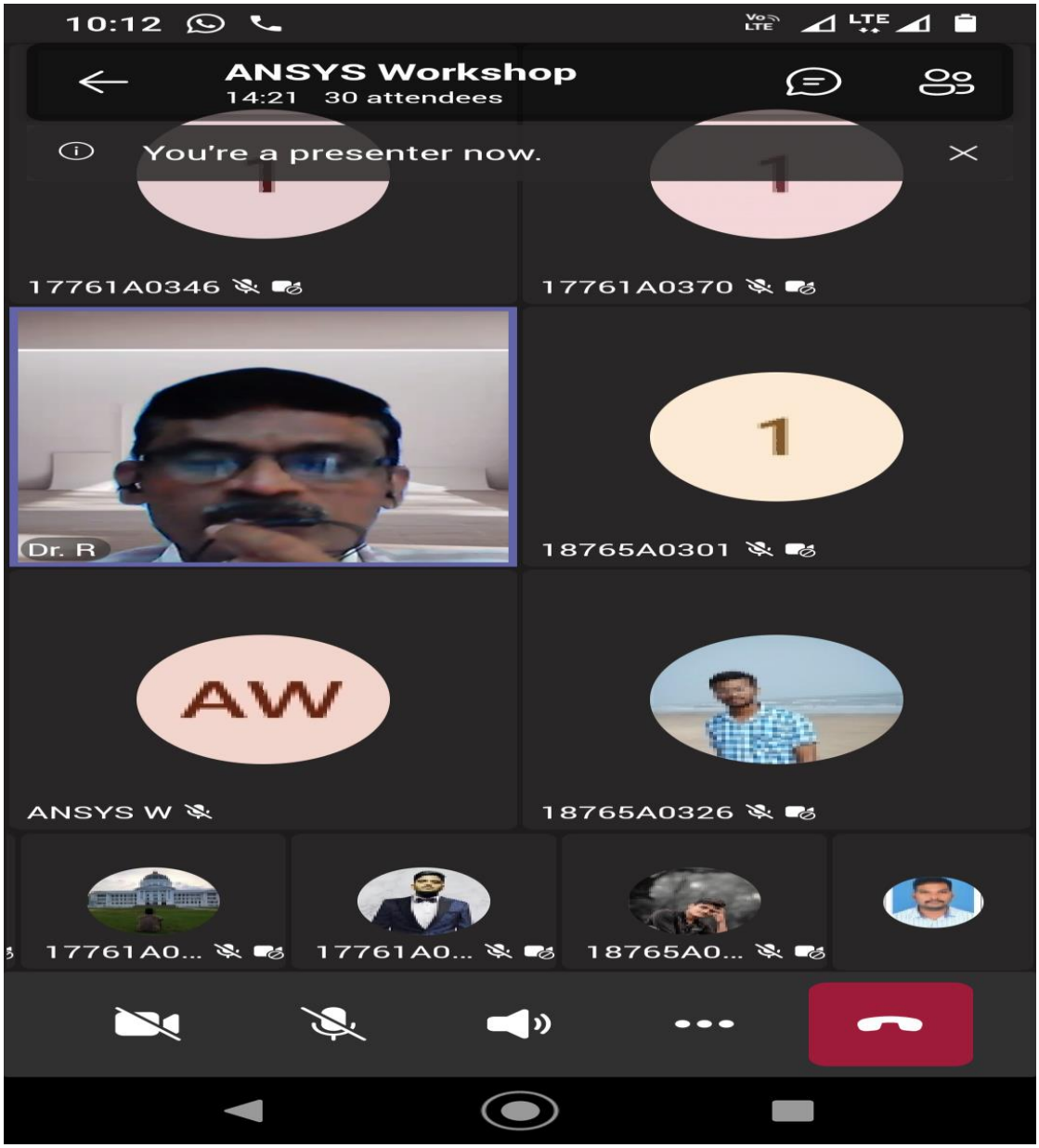
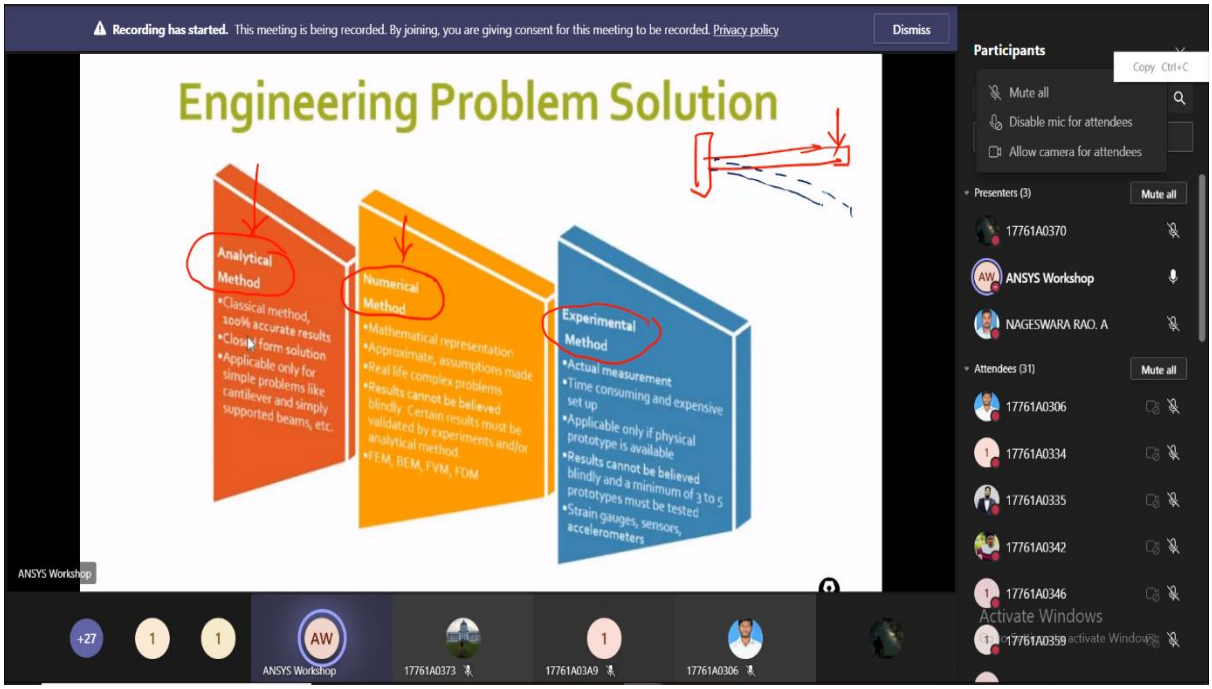
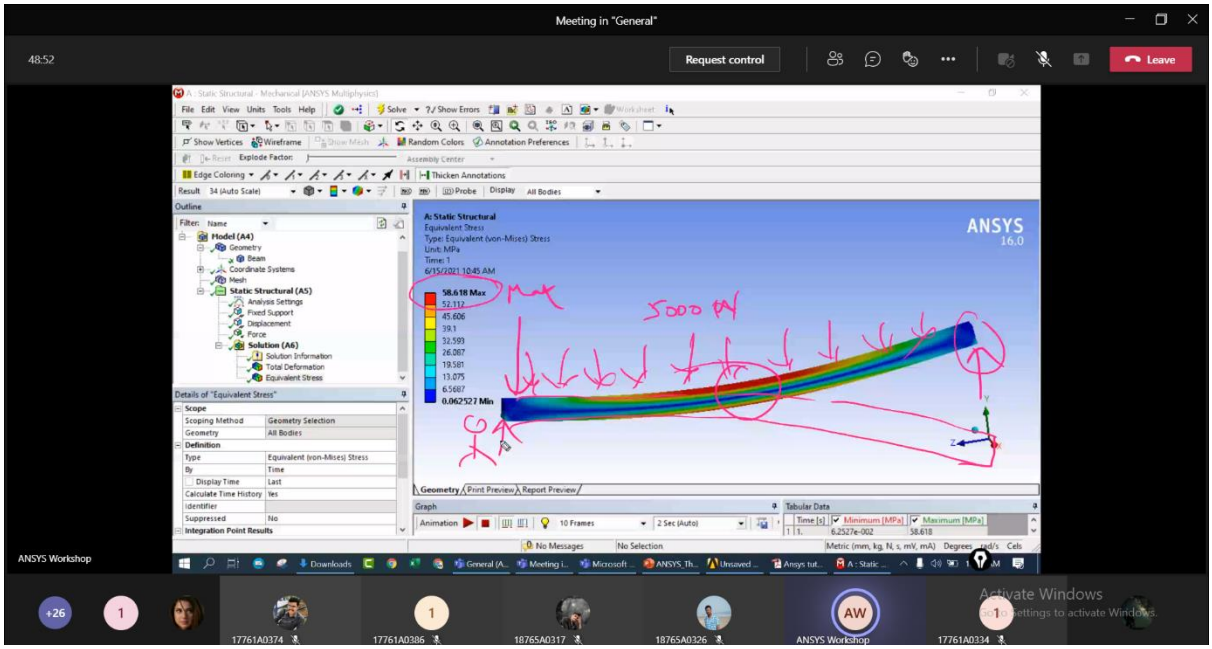


Fig: Inaugural speech by Dr. S. PICHI REDDY HoD-MED

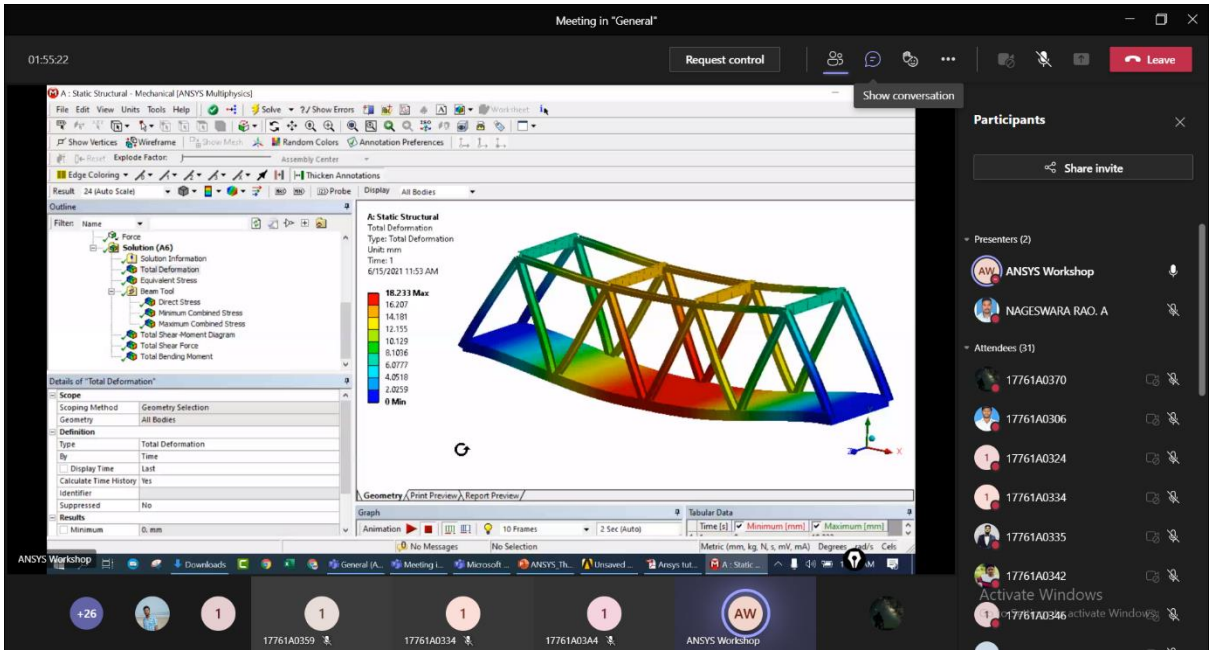
Fig: Inaugural speech by K Harish Kumar Resource person

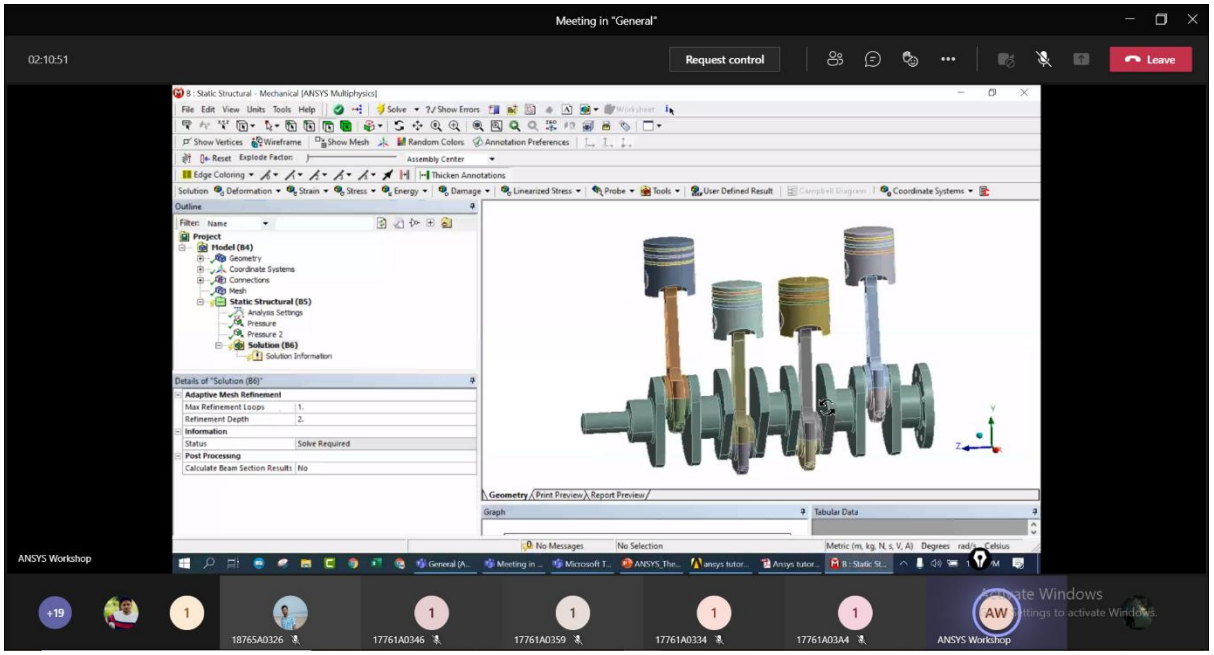
Day 1

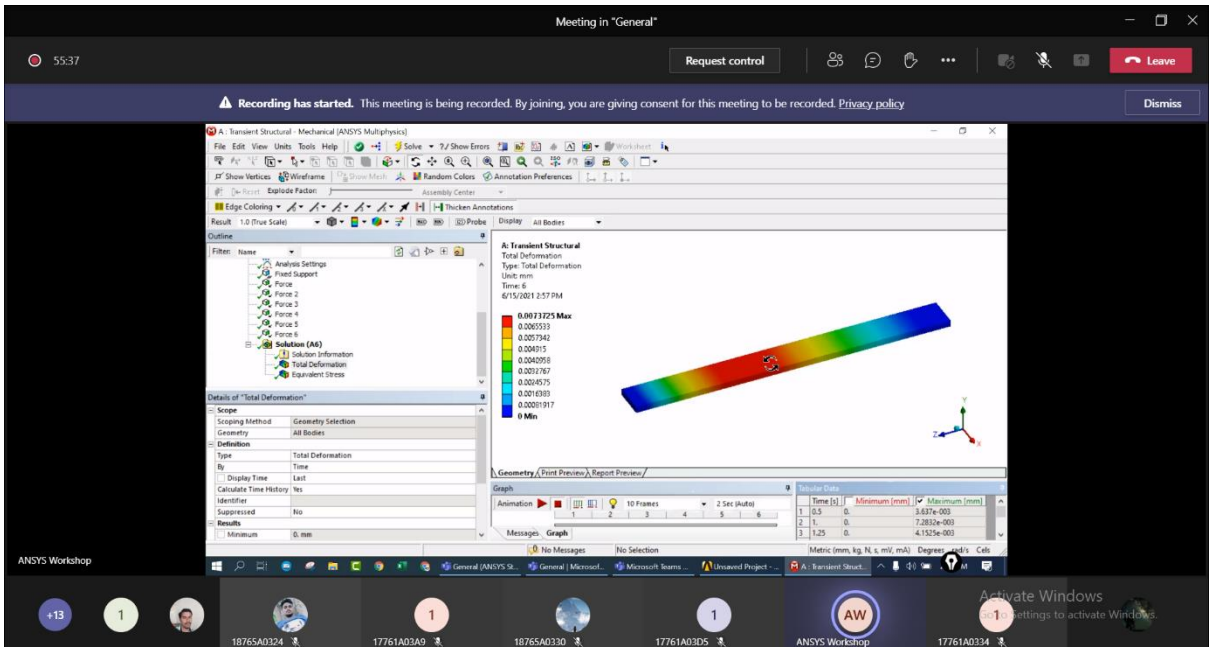



Meeting in "General"
48:52
Request control
Leave
ANSYS
16.0
A: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1
6/15/2021 10:45 AM
58.618 Max
52.112
45.606
39.1
32.593
26.087
19.581
13.075
6.5687
0.062527 Min
5000 kN
Max
ANSYS Workshop
17761A0374
17761A0386
18765A0317
18765A0326
ANSYS Workshop
17761A0334

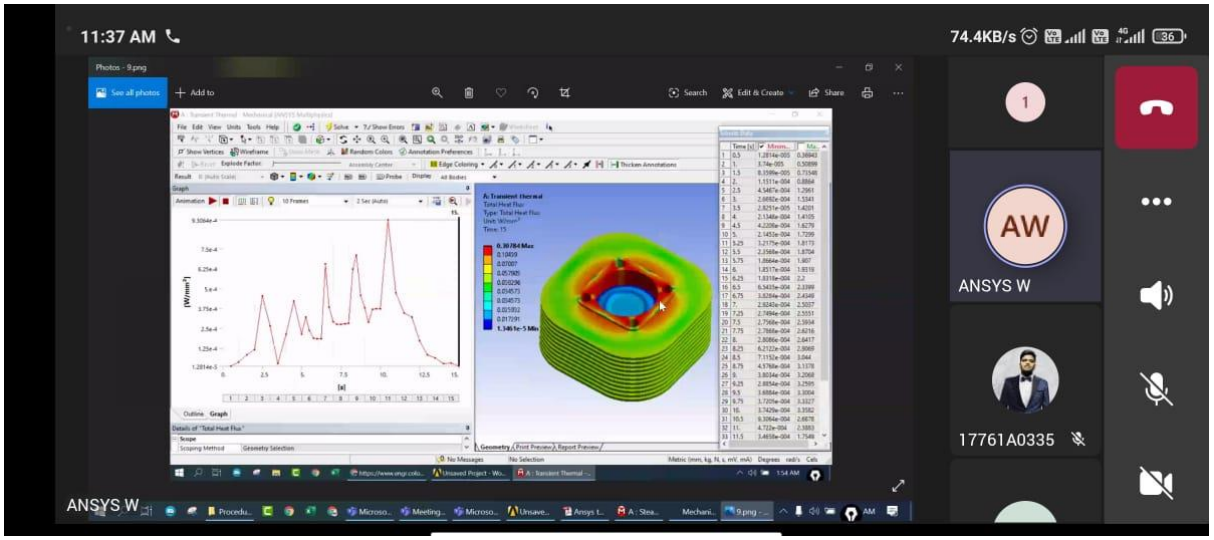
Day 2

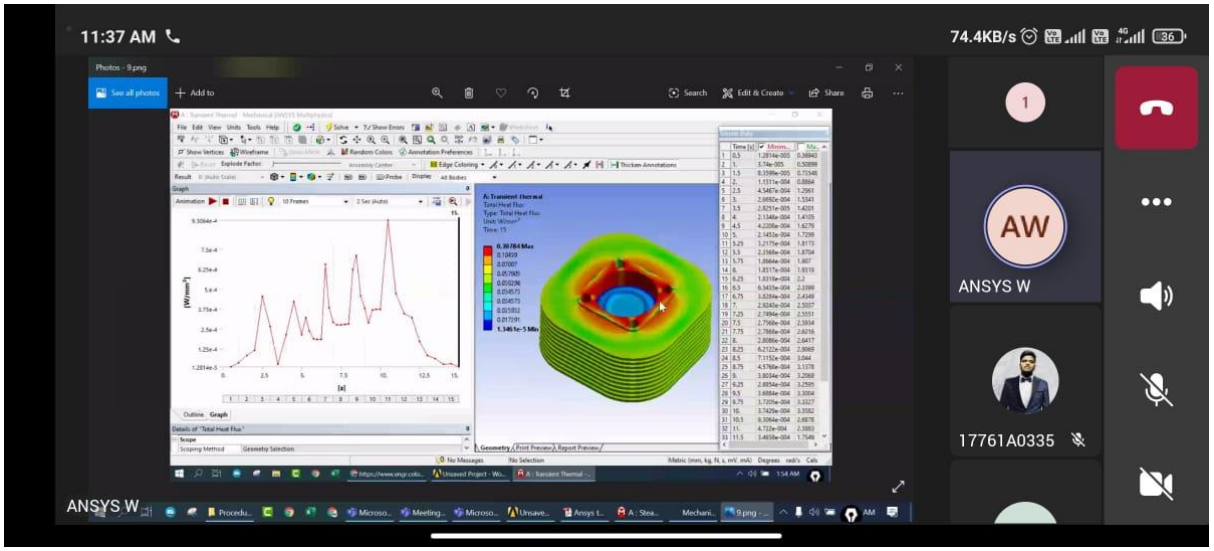

Meeting in "General"
01:55:22
Request control
Leave
Show conversation
Participants
Share invite
Presenters (2)
ANSYS Workshop
NAGESWARA RAO. A
Attendees (31)
17761A0370
17761A0306
17761A0324
17761A0334
17761A0335
17761A0342
17761A0346
A: Static Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1
6/15/2021 11:53 AM
18.233 Max
16.207
14.181
12.155
10.129
8.1036
6.0777
4.0518
2.0259
0 Min
ANSYS Workshop
17761A0359
17761A0334
17761A03A4
ANSYS Workshop

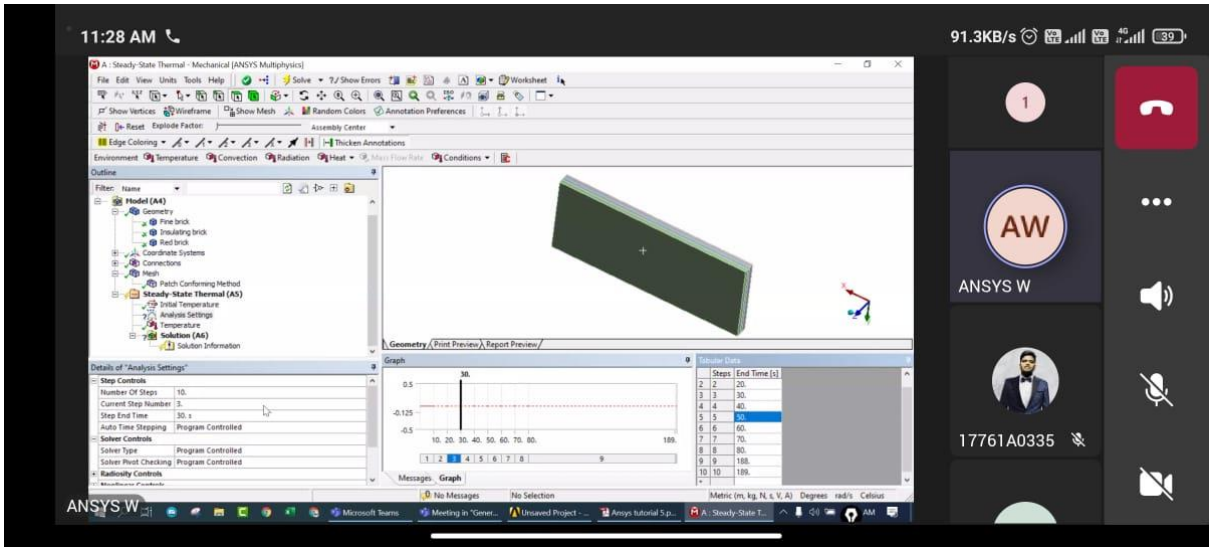

Meeting in "General"
02:10:51
Request control
Leave
B : Static Structural - Mechanical [ANSYS Multiphysics]
Project
Model (B4)
Static Structural (B5)
Solution (B6)
Details of "Solution (B6)"
Adaptive Mesh Refinement
Max Refinement Loops 1.
Refinement Depth 2.
Information
Status Solve Required
Post Processing
Calculate Beam Section Results No
Geometry
Graph
Tabular Data
No Messages
No Selection
ANSYS Workshop
18765A0326
17761A0346
17761A0359
17761A0334
17761A03A4
ANSYS Workshop


Meeting in "General"
55:37
Request control
Leave
Recording has started. This meeting is being recorded. By joining, you are giving consent for this meeting to be recorded. Privacy policy
Dismiss
A : Transient Structural - Mechanical [ANSYS Multiphysics]
A: Transient Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 6
6/15/2021 2:57 PM
0.0073725 Max
0.0065533
0.0057342
0.004915
0.0040958
0.0032767
0.0024575
0.0016383
0.00081917
0 Min
Solution (A6)
Solution Information
Total Deformation
Equivalent Stress
Details of "Total Deformation"
Scope
Scoping Method Geometry Selection
Geometry All Bodies
Definition
Type Total Deformation
By Time
Display Time Last
Calculate Time History Yes
Identifier
Suppressed No
Results
Minimum 0. mm
Geometry
Graph
Tabular Data
Time [s] Minimum [mm] Maximum [mm]
1 0.5 0. 3.637e-003
2 1. 0. 7.2832e-003
3 1.25 0. 4.1525e-003
No Messages
No Selection
ANSYS Workshop
18765A0324
17761A03A9
18765A0330
17761A03D5
ANSYS Workshop
17761A0334

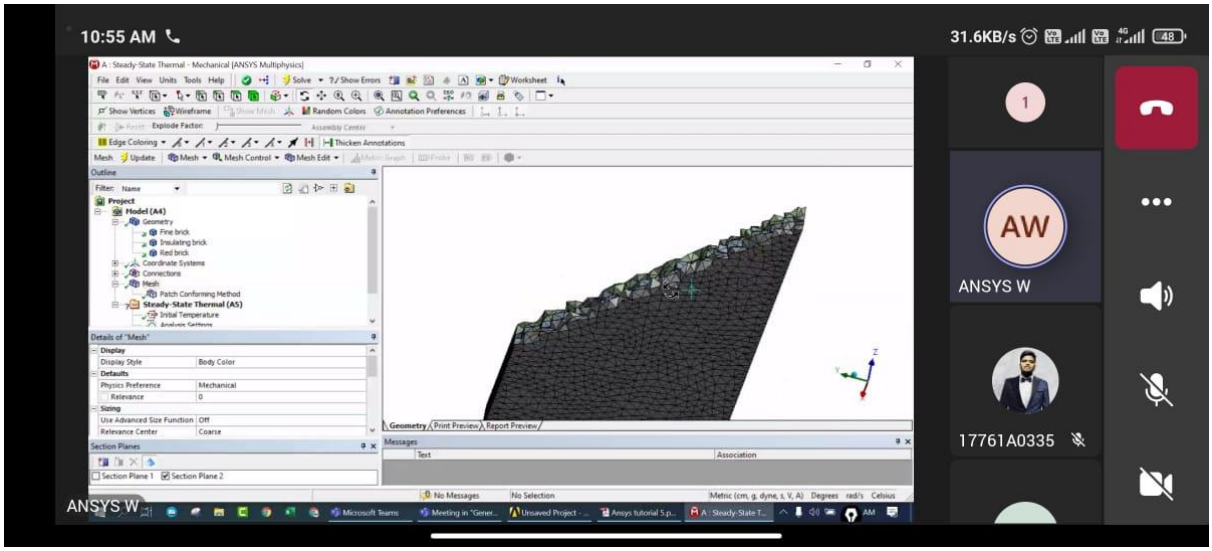
Day 3

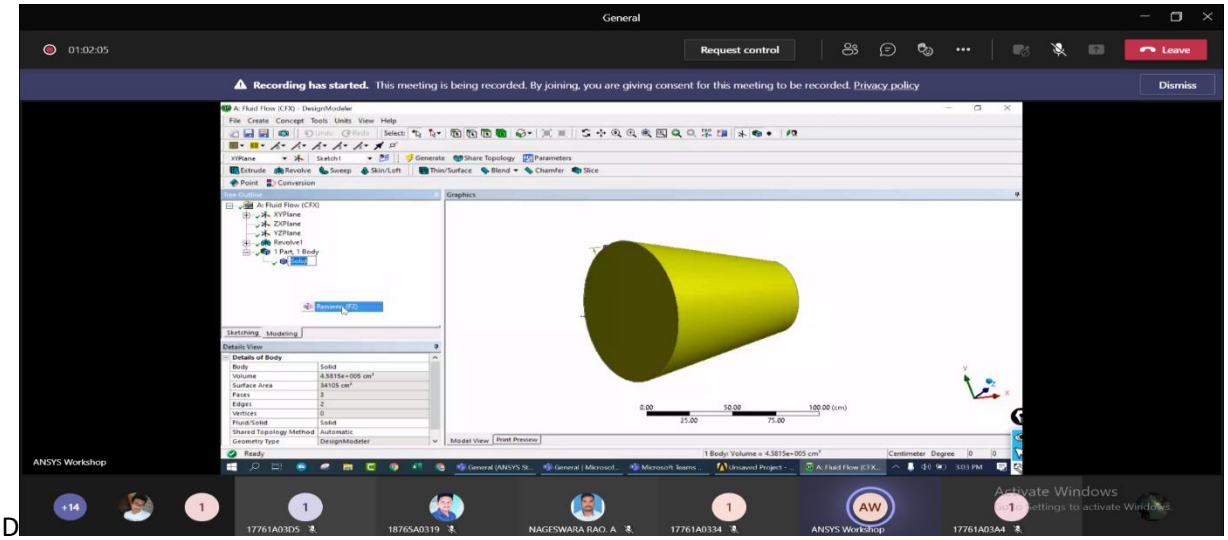

11:37 AM
74.4KB/s
AW
ANSYS W
17761A0335
ANSYS W


11:37 AM
74.4KB/s
AW
ANSYS W
17761A0335
ANSYS W

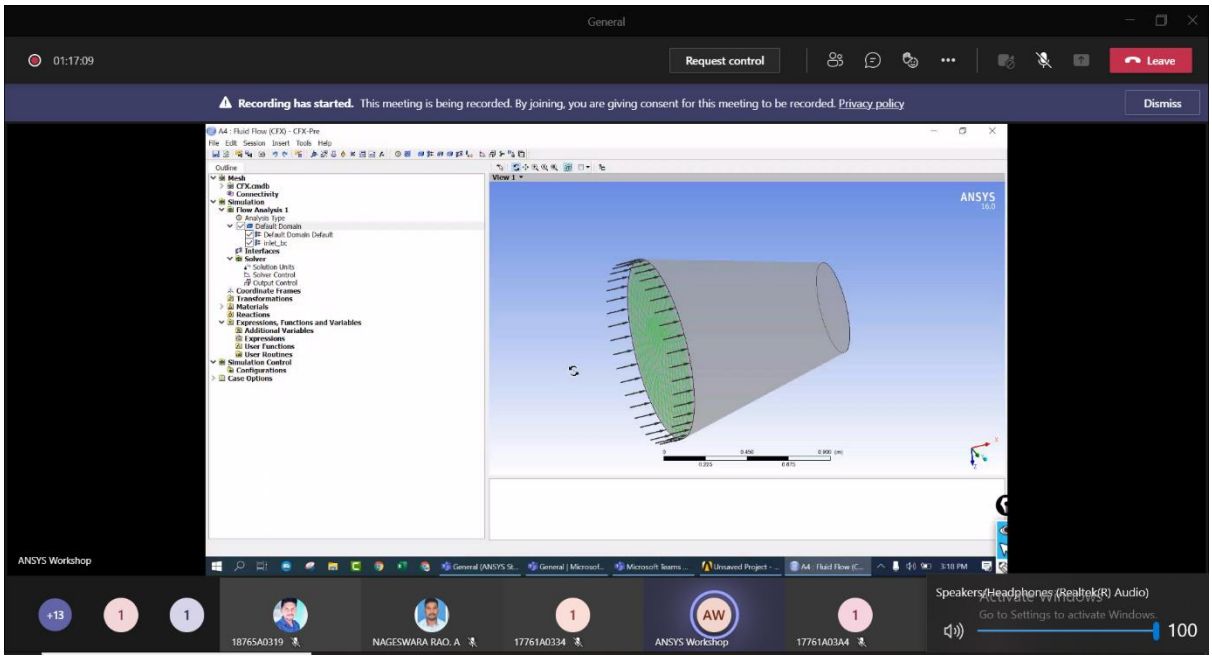

11:28 AM
91.3KB/s
AW
ANSYS W
17761A0335
ANSYS W

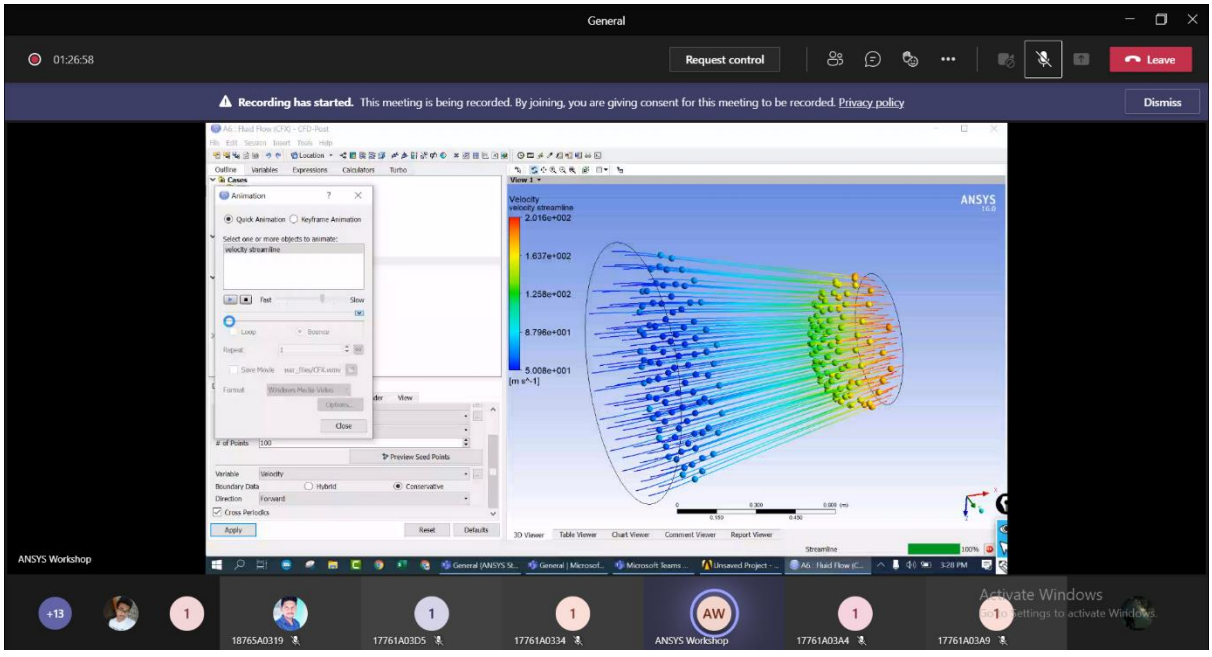
Day 4



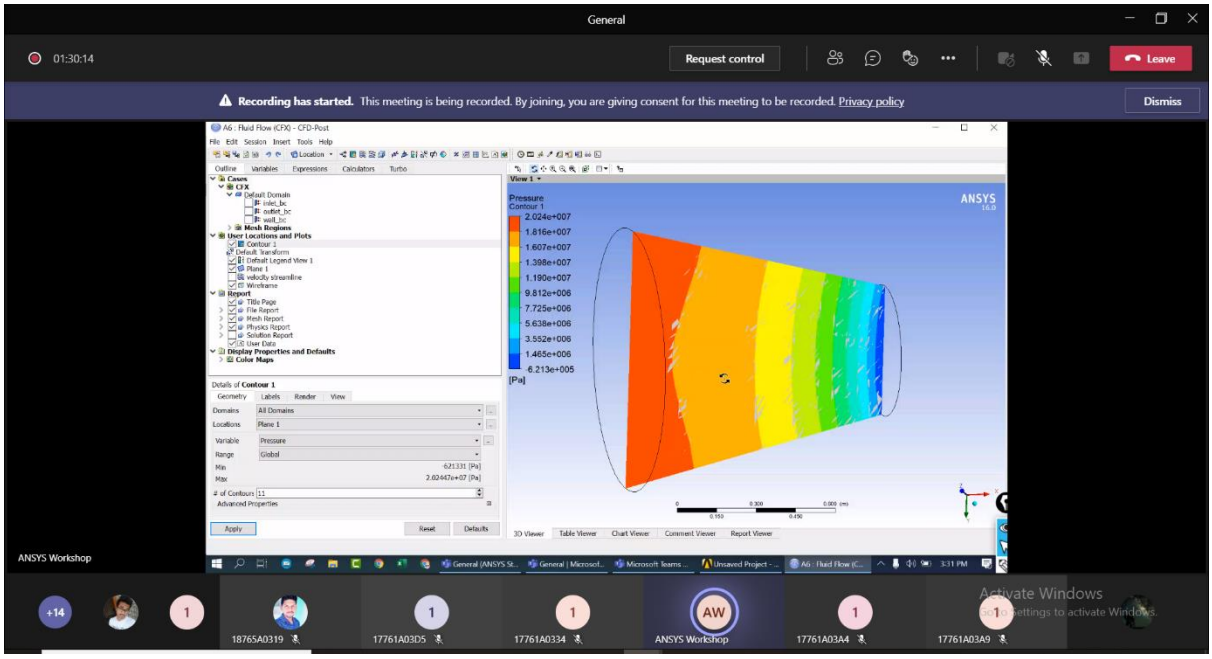

General
01:02:05
Request control
Leave
Recording has started. This meeting is being recorded. By joining, you are giving consent for this meeting to be recorded. Privacy policy
Dismiss
ANSYS Workshop
17761A03D5
18765A0319
NAGESWARA RAO. A
17761A0334
ANSYS Workshop
17761A03A4

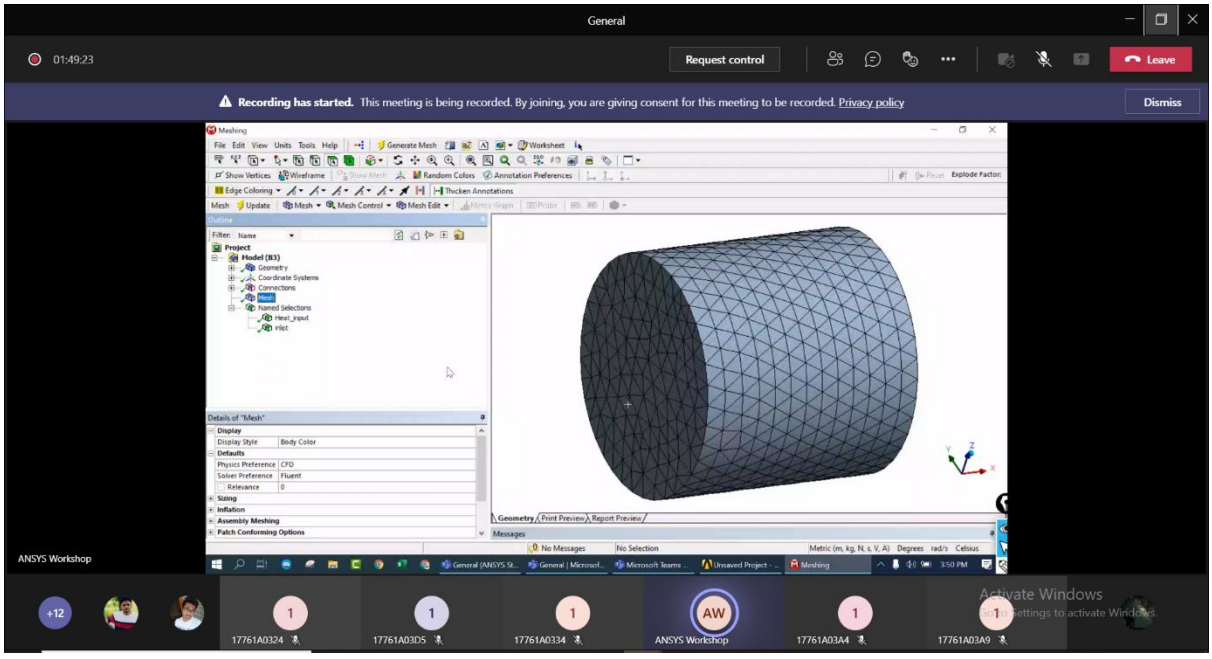
D

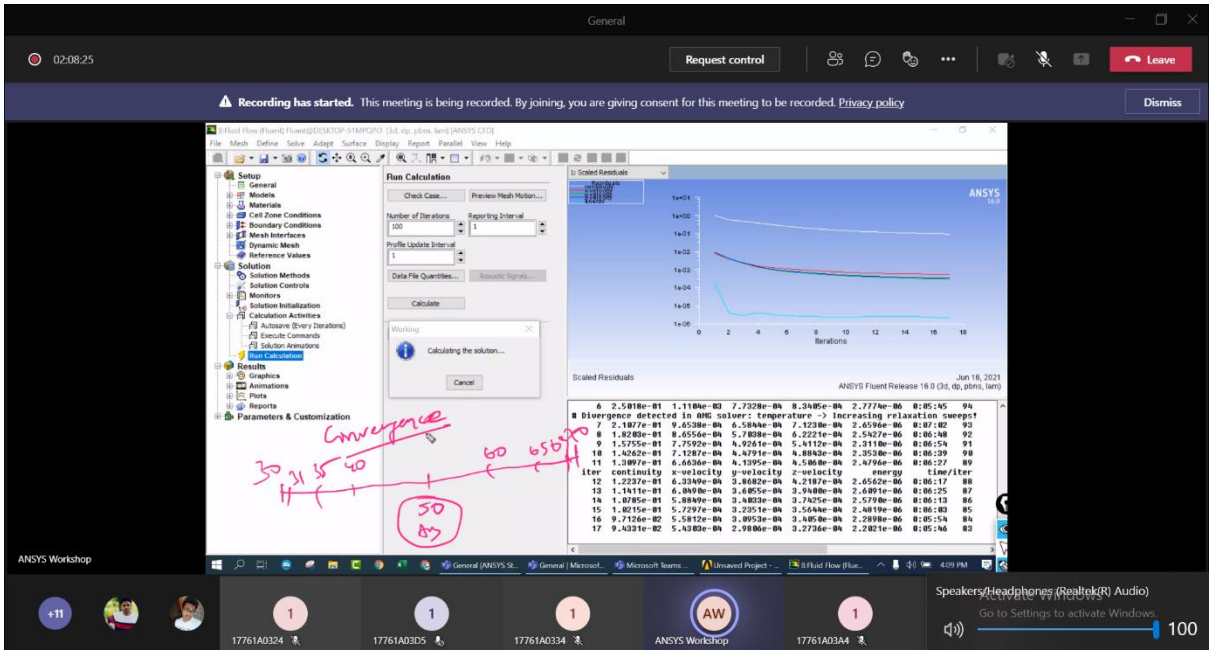

General
01:17:09
Request control
Leave
Recording has started. This meeting is being recorded. By joining, you are giving consent for this meeting to be recorded. Privacy policy
Dismiss
ANSYS Workshop
18765A0319
NAGESWARA RAO. A
17761A0334
ANSYS Workshop
17761A03A4
Speakers (Realtek(R) Audio)
100


General
01:26:58
Request control
Leave
Recording has started. This meeting is being recorded. By joining, you are giving consent for this meeting to be recorded. Privacy policy
Dismiss
ANSYS Workshop
18765A0319
17761A03D5
17761A0334
ANSYS Workshop
17761A03A4
17761A03A9

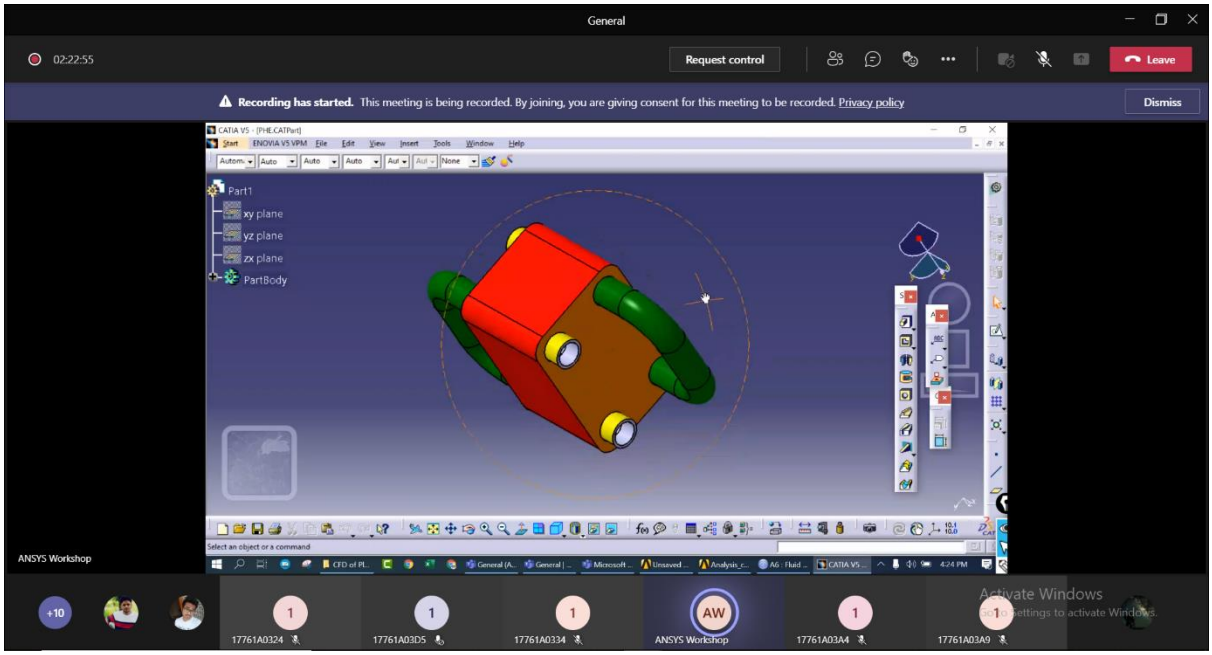
Day 5

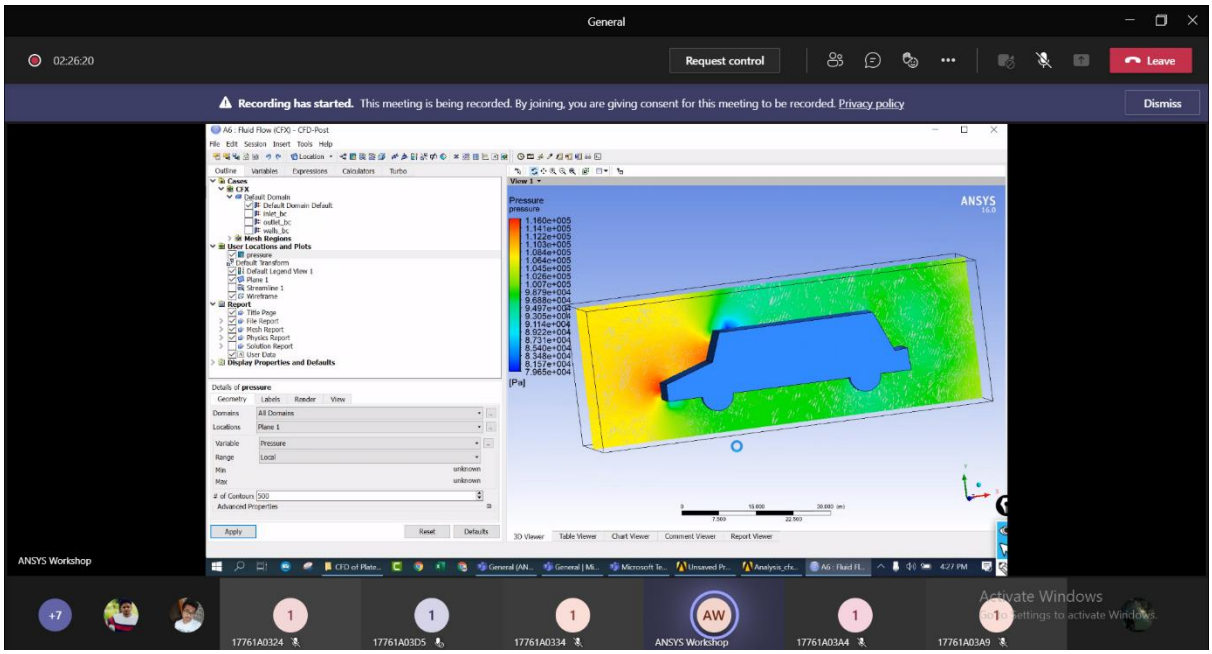

General
01:30:14
Request control
Leave
Recording has started. This meeting is being recorded. By joining, you are giving consent for this meeting to be recorded. Privacy policy
Dismiss
ANSYS Workshop
18765A0319
17761A03D5
17761A0334
ANSYS Workshop
17761A03A4
17761A03A9

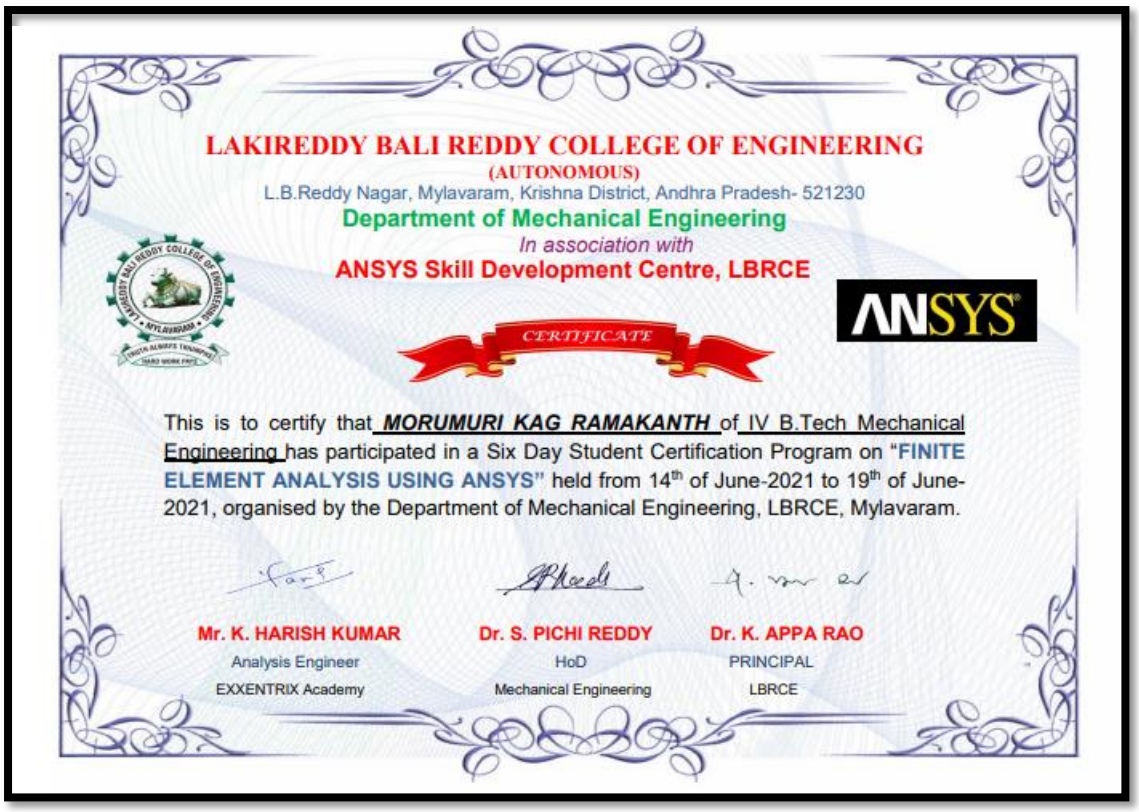

General
01:49:23
Request control
Leave
Recording has started. This meeting is being recorded. By joining, you are giving consent for this meeting to be recorded. Privacy policy
Dismiss
ANSYS Workshop
17761A0324
17761A03D5
17761A0334
ANSYS Workshop
17761A03A4
17761A03A9


General
02:08:25
Request control
Leave
Recording has started. This meeting is being recorded. By joining, you are giving consent for this meeting to be recorded. Privacy policy
Dismiss
Run Calculation
Check Case...
Preview Mesh Motion...
Number of Iterations
Reporting Interval
Profile Update Interval
Data File Quantities...
Calculate
Working
Calculating the solution...
Cancel
Convergence
Scaled Residuals
ANSYS Fluent Release 16.0 (3d, dp, pbns, lam)
Jun 16, 2021
ANSYS Workshop
Speakers (Realtek(R) Audio)
100
17761A0324
17761A03D5
17761A0334
ANSYS Workshop
17761A03A4

Day 6


General
02:22:55
Request control
Leave
Recording has started. This meeting is being recorded. By joining, you are giving consent for this meeting to be recorded. Privacy policy
Dismiss
CATIA V5 - [PHE.CATPart]
Part1
xy plane
yz plane
zx plane
PartBody
ANSYS Workshop
17761A0324
17761A03D5
17761A0334
ANSYS Workshop
17761A03A4
17761A03A9
Activate Windows


General
02:26:20
Request control
Leave
Recording has started. This meeting is being recorded. By joining, you are giving consent for this meeting to be recorded. Privacy policy
Dismiss
A6 : Fluid Flow (CFX) - CFD-Post
Pressure
ANSYS
[Pa]
ANSYS Workshop
17761A0324
17761A03D5
17761A0334
ANSYS Workshop
17761A03A4
17761A03A9
Activate Windows



LAKIREDDY BALI REDDY COLLEGE OF ENGINEERING

(AUTONOMOUS)

L.B.Reddy Nagar, Mylavaram, Krishna District, Andhra Pradesh- 521230

Department of Mechanical Engineering

In association with

ANSYS Skill Development Centre, LBRCE

CERTIFICATE

This is to certify that ***MORUMURI KAG RAMAKANTH*** of IV B.Tech Mechanical Engineering has participated in a Six Day Student Certification Program on **"FINITE ELEMENT ANALYSIS USING ANSYS"** held from 14th of June-2021 to 19th of June-2021, organised by the Department of Mechanical Engineering, LBRCE, Mylavaram.

Mr. K. HARISH KUMAR	Dr. S. PICHI REDDY	Dr. K. APPA RAO
Analysis Engineer	HoD	PRINCIPAL
EXXENTRIX Academy	Mechanical Engineering	LBRCE

Fig: certificate for ANSYS SDP participants