



# 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 tool 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
- Model 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
- Participants 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
- Model 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:**

**LAKIREDDY BALI REDDY COLLEGE OF ENGINEERING**  
(AUTONOMOUS)  
Accredited by NBA,NAAC, ISO 9001:20105 certified Institution  
Approved by AICTE,New Delhi and Affiliated to JNTUK, Kakinada  
L.B.Reddy Nagar, Mylavaram-521230, Krishna Dist, Andhra Pradesh, India

**DEPARTMENT OF MECHANICAL ENGINEERING**

**ANSYS**  
ANSYS Skill Development Center

**Dr. K. APPA RAO**  
PRINCIPAL

**CONVENER**  
**Dr. S. PICHI REDDY**  
PROFESSOR&HOD  
MECHANICAL ENGINEERING

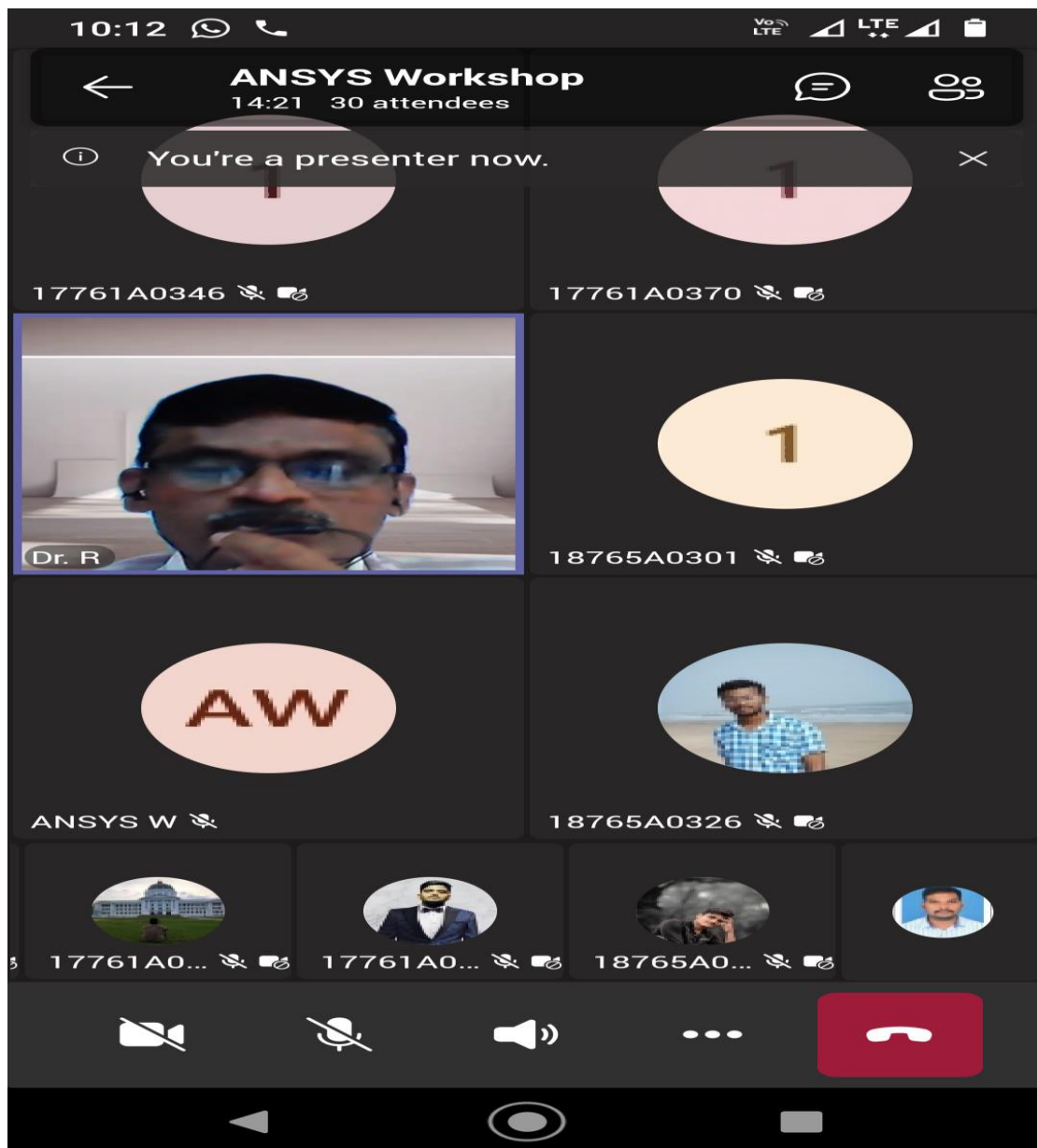
**STUDENT CERTIFICATION PROGRAM**  
**ON FEA USING ANSYS**  
**FROM 14/06/2021 TO 19/06/2021**  
**REGISTRATION FEE : RS 500**

**RESOURCE PERSON**  
**Mr.K.HARISH KUMAR**  
Analysis Engineer,  
EXXENRIX Academy,  
Visakhapatnam

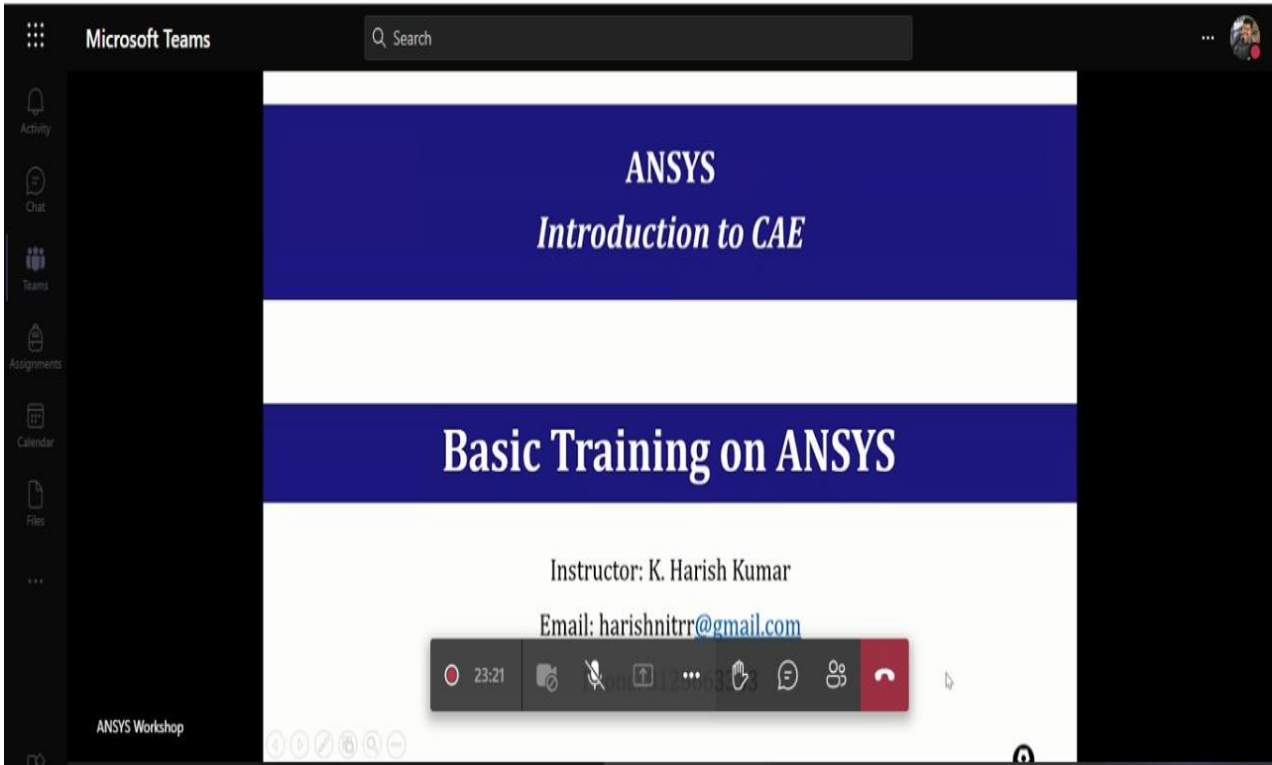
**COURSE CONTENT**

TOOL	TOPICS	WORKSHOP
ANSYS Meshing	Introduction to ANSYS Meshing	Tee Junction Meshing
	Meshing Methods	Exhaust System
	Global Mesh Controls	Workshop on Global Mesh Controls
	Local Mesh Controls	Home Assignment "Exhaust Manifold Meshing"
ANSYS Mechanical	Mechanical Basics	Basics using Control Box cover
	General Pre-processing	2D Gears
	Meshing in Mechanical	Object Generator
	Modeling Connections	Mesh creation & Control
ANSYS Mechanical	Modeling Connections	Contact Offset control
	Remote Boundary Conditions	Joints
	Static structure analysis	Screw jack
	Thermal Analysis	Constraints Equation
	Introduction to Heat Transfer	Beam Connection
	Heat Transfer Fundamentals	Linear Structural Analysis
	Pre-processing	Steady State Thermal Analysis
	Boundary Conditions	Thermal Bar
ANSYS Mechanical	Steady State Heat Transfer	Heating Coil
	Nonlinear Thermal Analysis	Thermal Contact
	Transfer Analysis	Radiating System
		Solenoid & Fin/Tube Heat Transfer
<b>Coordinators</b>		
1) Mr.B. Chaitanya, Associate Professor		
2) Mr.A.Nageswara Rao, Assistant Professor		

**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

Recording has started. This meeting is being recorded. By joining, you are giving consent for this meeting to be recorded. [Privacy policy](#) Dismiss

## Engineering Problem Solution

- Analytical Method**
  - Classical method, 100% accurate results
  - Closed form solution
  - Applicable only for simple problems like cantilever and simply supported beams, etc.
- Numerical Method**
  - Mathematical representation
  - Approximate, assumptions made
  - Real life complex problems
  - Results cannot be believed blindly. Certain results must be validated by experiments and/or analytical method.
  - FEM, BEM, FVM, FDM
- Experimental Method**
  - Actual measurement
  - Time consuming and expensive set up
  - Applicable only if physical prototype is available
  - Results cannot be believed blindly and a minimum of 3 to 5 prototypes must be tested
  - Strain gauges, sensors, accelerometers

Participants

Mute all  
Disable mic for attendees  
Allow camera for attendees

Presenters (3)

- 17761A0370
- ANSYS Workshop
- NAGESWARA RAO. A

Attendees (31)

- 17761A0306
- 17761A0334
- 17761A0335
- 17761A0342
- 17761A0346
- Activate Windows
- 17761A0359 activate Windows

ANSYS Workshop

17761A0373

17761A03A9

17761A0306



Meeting in "General"

48:52

Request control

Leave

A: Static Structural - Mechanical [ANSYS Multiphysics]

File Edit View Units Tools Help

Solve 7/ Show Errors

Work sheet

Show Vertices Wireframe Show Mesh Random Colors Annotation Preferences

Edge Coloring

Result 34 (Auto Scale)

Thicken Annotations

Outline

Filter: Name

Model (A4)

- Geometry
- Beam
- Coordinate Systems
- Static Structural (AS)
- Analysis Settings
- Pinned Support
- Displacement
- Force
- Solution (A6)
- Solution Information
- Total Deformation
- Equivalent Stress

Details of "Equivalent Stress"

Scope

- Scoping Method: Geometry Selection
- Geometry: All Bodies

Definition

- Type: Equivalent (von-Mises) Stress
- By: Time
- Display Time: Last
- Calculate Time History: Yes
- Identifier: No
- Suppressed: No
- Integration Point Results: No

A: Static Structural

Equivalent Stress

Type: Equivalent (von-Mises) Stress

Unit: MPa

Time: 1

6/15/2021 10:45 AM

58.618 Max

45.606

39.3

31.959

26.087

19.385

13.075

6.9807

0.062327 Min

5000 N

Graph

Geometry (Print Preview) Report Preview

Animation

10 Frames

2 Sec (Auto)

Tabular Data

Minimum [MPa]

Maximum [MPa]

Metric (mm, kg, N, s, mV, mA) Degrees rad/s Cells

ANSYS Workshop

17761A0374

17761A0386

18765A0317

18765A0326

ANSYS Workshop

17761A0334

Activate Windows

Settings to activate Windows

## Day 2

Meeting in "General"

01:55:22

Request control

Show conversation

Leave

A: Static Structural - Mechanical [ANSYS Multiphysics]

File Edit View Units Tools Help

Solve 7/ Show Errors

Work sheet

Show Vertices Wireframe Show Mesh Random Colors Annotation Preferences

Edge Coloring

Result 24 (Auto Scale)

Thicken Annotations

Outline

Filter: Name

Force

Solution (A6)

- Solution Information
- Total Deformation
- Equivalent Stress
- Beam Tool
- Direct Stress
- Minimum Combined Stress
- Maximum Combined Stress
- Total Shear Moment Diagram
- Total Shear Force
- Total Bending Moment

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: No
- Suppressed: No
- Results: Minimum 0. mm

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

10.119

8.1036

6.0777

4.0518

2.0259

0 Min

Graph

Geometry (Print Preview) Report Preview

Animation

10 Frames

2 Sec (Auto)

Tabular Data

Minimum [mm]

Maximum [mm]

Metric (mm, kg, N, s, mV, mA) Degrees rad/s Cells

ANSYS Workshop

17761A0359

17761A0334

17761A0344

ANSYS Workshop

Participants

Share invite

Presenters (2)

- ANSYS Workshop
- NAGESWARA RAO, A

Attendees (81)

- 17761A0370
- 17761A0306
- 17761A0324
- 17761A0334
- 17761A0335
- 17761A0342
- 17761A0346 activate Windows

Activate Windows

Settings to activate Windows

Meeting in "General"

02:10:51

Request control

Leave

8 - Static Structural - Mechanical [ANSYS Multiphysics]

File Edit View Units Tools Help

Solve 7/ Show Errors

Show Vertices Wireframe Show Mesh Random Colors Annotation Preferences

Reset Explode Factor Assembly Center

Edge Coloring Thicken Annotations

Solution Deformation Strain Stress Energy Damage Linearized Stress Probe User Defined Result Campbell Diagram Coordinate Systems

Outline

Filter: Name

- Project
  - Flowed (84)
    - Geometry
    - Coordinate Systems
    - Connections
    - Mesh
    - Static Structural (85)
      - Analysis Settings
      - Pressure
      - Pressure 2
      - Solution (86)
      - Solution Information

Details of "Solution (86)"

Adaptive Mesh Refinement

Max Refinement Loops: 1

Refinement Depth: 2

Information

Status: Solve Required

Post Processing

Calculate Beam Section Results: No

Geometry / Print Preview / Report Preview

Graph Tabular Data

No Messages No Selection Metric (m, kg, N, s, V, A) Degrees rad/ Celsius

ANSYS Workshop

+19

18765A0326

17761A0346

17761A0359

17761A0334

17761A0344

Activate Windows

Settings to activate Windows

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

8 - Transient Structural - Mechanical [ANSYS Multiphysics]

File Edit View Units Tools Help

Solve 7/ Show Errors

Show Vertices Wireframe Show Mesh Random Colors Annotation Preferences

Reset Explode Factor Assembly Center

Edge Coloring Thicken Annotations

Result: 1.0 (True Scale) Display: All Bodies

Outline

Filter: Name

- Project
  - Analysis Settings
  - Fixed Support
  - Force
  - Force 2
  - Force 3
  - Force 4
  - Force 5
  - Force 6
  - Solution (A6)
    - Total Deformation
    - Equivalent Stress
  - Solution Information

Details of "Total Deformation"

Scope

Scoping Method: Geometry Selection

Geometry: All Bodies

Definition

Type: Total Deformation

By: Time

Display Time: Last

Calculate Time History: No

Identifier

Suppressed: No

Results

Minimum: 0 mm

A: Transient Structural

Total Deformation

Type: Total Deformation

Units: mm

Time: 6

6/15/0811 2:57 PM

0.0013225 Max

0.0002059

0.0007342

0.0004915

0.0004056

0.001767

0.0014575

0.0016383

0.0001917

0 Min

Geometry / Print Preview / Report Preview

Graph Tabular Data

No Messages No Selection Metric (mm, kg, N, s, mV, mA) Degrees rad/ Cel

ANSYS Workshop

+13

18765A0324

17761A03A9

18765A0330

17761A03D5

ANSYS Workshop

17761A0334

Activate Windows

Settings to activate Windows

ANSYS Workshop

# Day 3

11:37 AM 74.4KB/s

ANSYS W

17761A0335

11:37 AM 74.4KB/s

ANSYS W

17761A0335

11:28 AM 91.3KB/s

ANSYS W

17761A0335



Day 4

10:55 AM 31.6KB/s

A: Steady-State Thermal - Mechanical [ANSYS Multiphysics]

File Edit View Units Tools Help Solve 7/ Show Errors Worksheet

Show Vertices Wireframe Show Mesh Random Colors Annotation Preferences

Edge Coloring Mesh Update Mesh Mesh Control Mesh Edit Thicken Annotations

Mesh Update Mesh Mesh Control Mesh Edit Thicken Annotations

Outline

- Project
  - Model (A4)
    - Geometry
      - Brick
      - Insulating brick
      - Red brick
    - Coordinate Systems
    - Connectors
    - Mesh
      - Patch Conforming Method
    - Steady-State Thermal (A5)
      - Initial Temperature
      - Boundary Conditions

Details of "Mesh"

Display

- Display Style Body Color

Defaults

- Physics Preference Mechanical
- Reference Center 0

Sizing

- Use Advanced Size Function Off
- Reference Center Coarse

Section Planes

- Section Plane 1
- Section Plane 2

Messages

- Text
- Association

ANSYS W

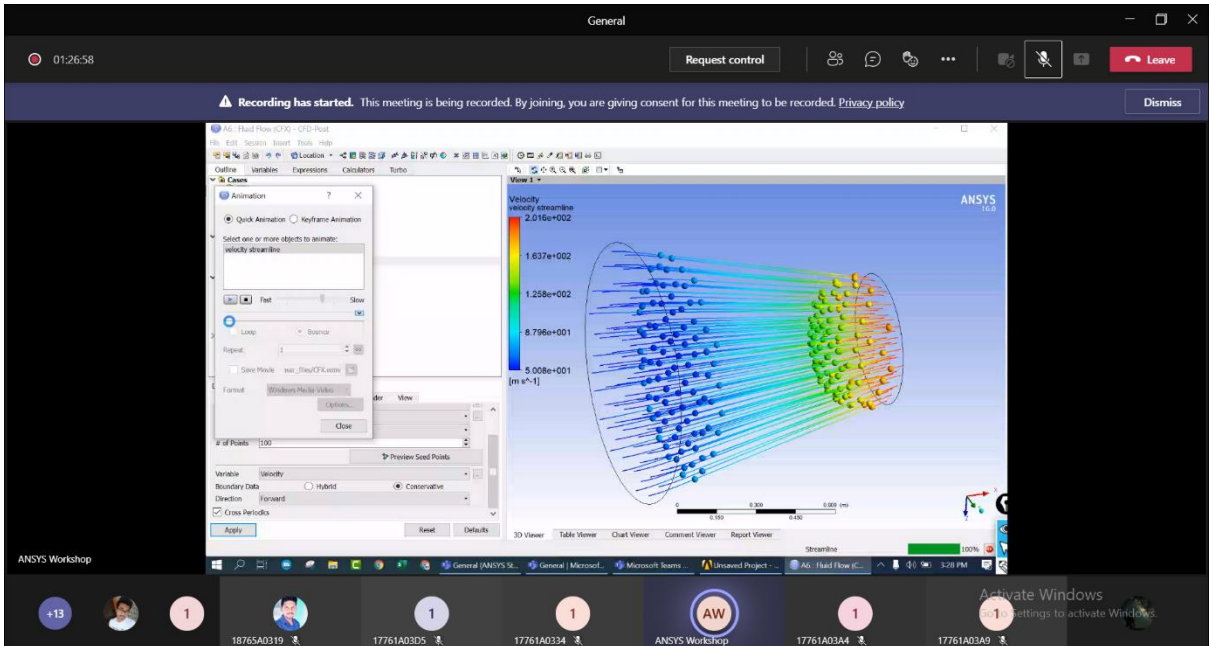
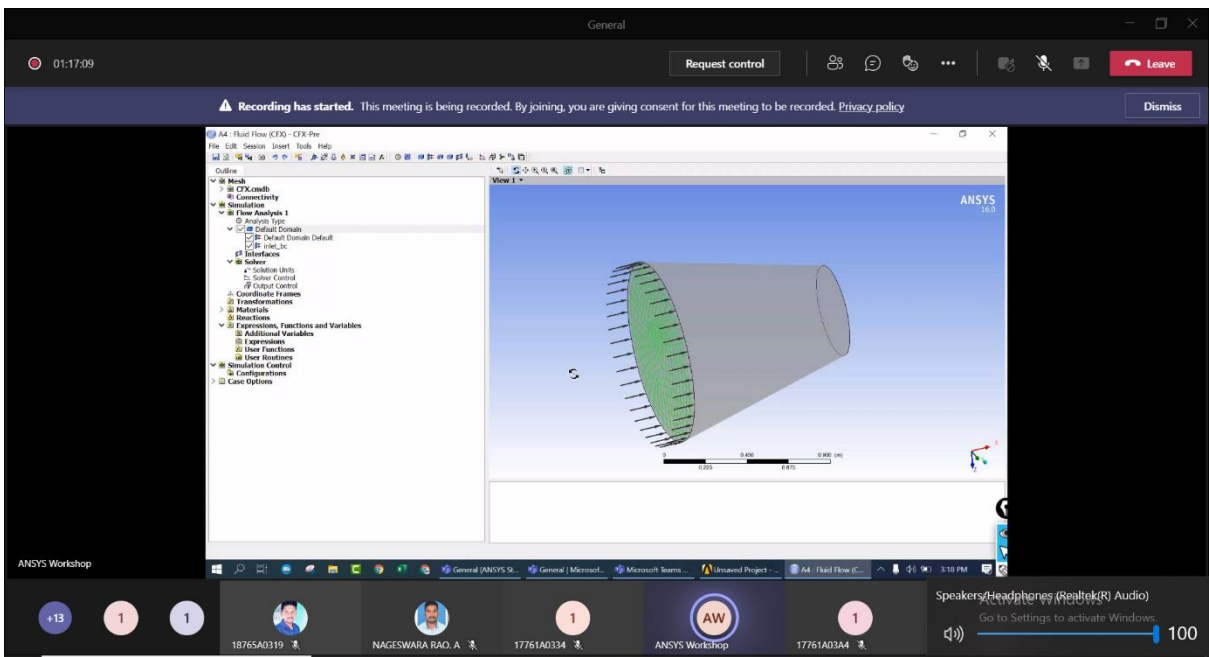
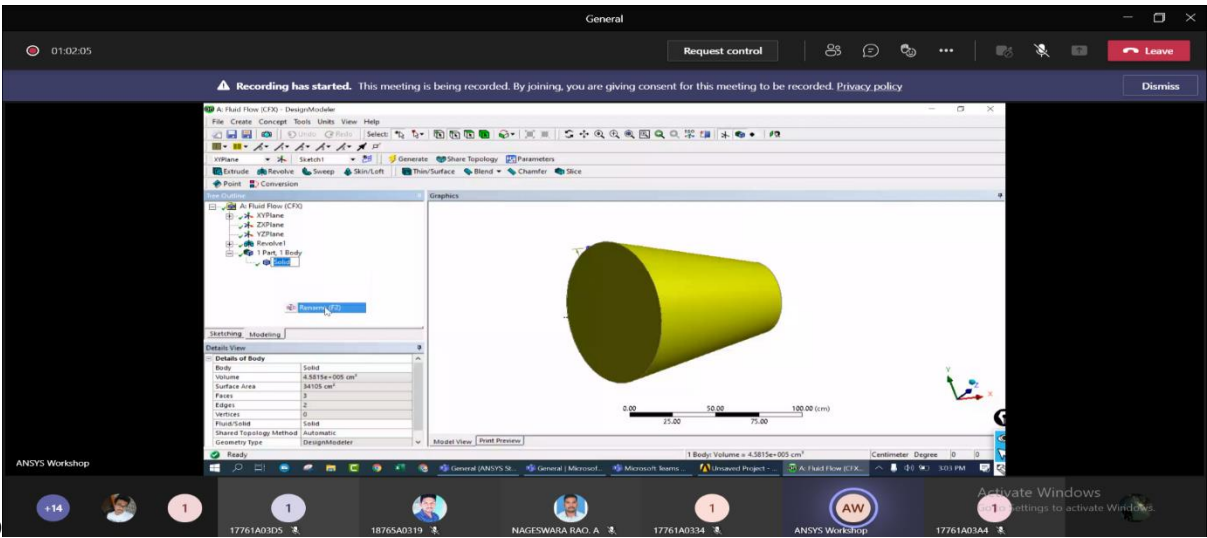
No Messages No Selection Metric (cm, g, dyne, s, V, A) Degree rad/s Celsius

1

AW

ANSYS W

17761A0335



# Day 5

General

01:30:14

Request control

Recording has started. This meeting is being recorded. By joining, you are giving consent for this meeting to be recorded. [Privacy policy](#)

Dismiss

ANSYS 16.0

ANSYS Workshop

Pressure Contour 1

2.024e+007

1.816e+007

1.607e+007

1.398e+007

1.190e+007

9.812e+006

7.725e+006

5.638e+006

3.552e+006

1.465e+006

6.213e+005 [Pa]

0 0.150 0.300 0.450 m

3D Viewer Table Viewer Chart Viewer Content Viewer Report Viewer

ANSYS 16.0

ANSYS Workshop

18765A0319

17761A03D5

17761A0334

ANSYS Workshop

17761A03M4

17761A03A9

Activate Windows

Settings to activate Windows

General

01:49:23

Request control

Recording has started. This meeting is being recorded. By joining, you are giving consent for this meeting to be recorded. [Privacy policy](#)

Dismiss

ANSYS 16.0

ANSYS Workshop

Meshing

Project

Head (R3)

Geometry

Coordinate Systems

Connectors

Mesh

Named Selectors

Mesh\_inport

inlet

Details of "Mesh"

Display

Display Style

Body Color

Defaults

Physics Preference

CFD

Solver Preference

Fluent

Reference

0

Scaling

Inflection

Assembly Meshing

Patch Conforming Options

Messages

No Messages

No Selection

Metric (m, kg, N, & V, A) Degrees rad/s Celsius

ANSYS Workshop

17761A0324

17761A03D5

17761A0334

ANSYS Workshop

17761A03M4

17761A03A9

Activate Windows

Settings to activate Windows

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**

Number of Iterations: 100  
Reporting Interval: 1  
Profile Update Interval: 1  
Data File Quantities: Automatic Reports

Working: Calculating the solution...

**Scaled Residuals**

ANSYS Fluent Release 16.0 (3d, 64-bit, serial)

iter	continuity	x-velocity	y-velocity	z-velocity	energy	time/iter
4	2.5810e-01	1.1100e-03	7.7520e-04	8.2085e-04	2.7774e-04	8:35:45 94
7	2.1877e-01	9.6538e-04	6.5848e-04	7.1230e-04	2.6596e-04	8:37:42 93
8	1.6240e-01	8.4556e-04	5.7830e-04	6.2221e-04	2.5427e-04	8:38:48 92
9	1.5755e-01	7.7592e-04	4.9261e-04	5.4112e-04	2.3110e-04	8:38:54 91
10	1.4260e-01	7.1287e-04	4.4797e-04	4.8843e-04	2.2520e-04	8:38:59 90
11	1.3097e-01	6.6636e-04	4.1395e-04	4.5808e-04	2.1796e-04	8:38:27 89
12	1.2227e-01	6.2389e-04	3.8682e-04	4.2187e-04	2.1552e-04	8:38:17 88
13	1.1411e-01	6.0090e-04	3.6855e-04	3.9400e-04	2.1699e-04	8:38:25 87
14	1.0785e-01	5.8049e-04	3.4853e-04	3.7457e-04	2.1579e-04	8:38:13 86
15	1.0215e-01	5.7297e-04	3.2351e-04	3.5644e-04	2.14819e-04	8:38:03 85
16	9.7126e-02	5.5812e-04	3.0952e-04	3.4058e-04	2.2899e-04	8:38:54 84
17	9.4231e-02	5.4382e-04	2.9180e-04	3.2736e-04	2.2802e-04	8:38:46 83

Handwritten red annotations: 'Convergence' with arrows pointing to iterations 30-40, and a circled '50' with '87' below it.

ANSYS Workshop

Speakers/Headphones (Realtek(R) Audio) 100

Day 6



General

02:22:55

Request control

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

Start ENOVA V5 VPM File Edit View Insert Tools Window Help

Autom Auto Auto Auto Aut All None

Part1

- xy plane
- yz plane
- xz plane
- PartBody

ANSYS Workshop

+10

17761A0324

17761A03D5

17761A0334

ANSYS Workshop

17761A03A4

17761A03A9

Activate Windows  
Go to Settings to activate Windows.

General

02:26:20

Request control

Recording has started. This meeting is being recorded. By joining, you are giving consent for this meeting to be recorded. [Privacy policy](#)

Dismiss

ANSYS FLUENT - CD-Post

File Edit Session Insert Tools Help

Outline Variables Expressions Calculators Turbo

CFX

- Default Domain: Default
- Mesh Regions: pressure
- Use Locations and Plots: pressure
- Report: Title Page, File Report, Mesh Report, Physics Report, Solution Report, User Data
- Display Properties and Defaults

Details of pressure

Geometry: Labels Reader View

Domains: All Domains

Locations: Plane 1

Variable: Pressure

Range: Local

Min: unknown

Max: unknown

# of Contours: 500

Advanced Properties

Apply Reset Defaults

Pressure

pressure

1	1.60E+005
1	1.14E+005
1	1.22E+005
1	1.10E+005
1	1.084E+005
1	1.054E+005
1	1.045E+005
1	1.025E+005
1	1.007E+005
1	9.87E+004
1	9.688E+004
1	9.482E+004
1	9.305E+004
1	9.114E+004
1	8.925E+004
1	8.724E+004
1	8.549E+004
1	8.348E+004
1	8.157E+004
1	7.965E+004

[Pa]

ANSYS 16.0

3D Viewer Table Viewer Chart Viewer Command Viewer Report Viewer

ANSYS Workshop

+7

17761A0324

17761A03D5

17761A0334

ANSYS Workshop

17761A03A4

17761A03A9

Activate Windows  
Go to Settings to activate Windows.



**Fig:** certificate for ANSYS SDP participants