

DR. S. & S. S. Ghandhy Government Engineering College,
Surat

ANSYS Training Report

(Training under RUSA Initiative)

Training Duration : 24/02/2020 to 07/03/2020

Training Start Date : 24/02/2020

Training End Date : 07/03/2020

Training Location : Indo German Tool Room (IGTR), Ahmedabad

Training Instructor : Rupesh Sir

Trainee Details:

Sr. No.	Name	Enrolment No.
1	Karan Champaneria	160230119001
2	Muaaviya Kaduji	160230119018
3	Nauman Momin	160230119025
4	Daxal Naheriwala	160230119026
5	Daulatsingh Parmar	160230119029
6	Yogesh Prajapati	160230119045

On the very first day, we met with training in-charge of IGTR. He gave us forms for enrolment. We filled the forms and then he showed us our class and laboratory. He introduced us to our instructor Rupesh sir. Our instructor gave us a brief overview of the course. He gave us details about timing of classes and solved our doubts for the same.

The timing for our lectures was from 10:00AM to 5:00PM. In lecture, our instructor used to demonstrate on ANSYS and during practice in laboratory, we tried to do it ourselves. The detail break-up of timing is given below:

Time	Work
10:00 to 11:00	Practice in lab
11:00 to 12:00	Lecture in class
12:00 to 12:30	Lunch Break
12:30 to 14:00	Practice in lab
14:00 to 15:00	Lecture in class
15:00 to 15:15	Tea Break
15:15 to 17:00	Practice in Lab

The modules of ANSYS included in our training were,

1. Static Stress Analysis
2. Steady State Thermal Analysis

First seven days, we were taught Static stress analysis. Next three to four days, we were taught Steady state thermal analysis and in remaining days we were taught how to perform static stress and steady state thermal analysis combined. After completion of the training, exam was taken for certification.



Lecture Hours



Practical Hours

Different practical that we performed during training are listed below,

Sr.No	Practical
1	Bending of solid and hollow shaft
2	Twisting of solid and hollow shaft
3	Bending of beams of different cross-section
4	Determining thickness of the pressure vessel
5	Applying Hydrodynamic pressure
6	Determining pipe thickness for given centrifugal pump
7	Applying Remote Force
8	Conduction through a Rod under free convection
9	Conduction through a Rod under forced convection
10	Radiation on surface of a plate
11	Combined Static stress and steady state thermal analysis

Training Outcome:

After completing this training, we are now able to,

- Perform Static stress analysis in ANSYS
- Perform Steady state analysis in ANSYS
- Understand the results obtained from analysis
- Define type of analysis to be performed
- Perform combination of static stress analysis and steady stress thermal analysis in ANSYS

At the end, we would like to thank all the individuals and organization involved for successfully arranging such a nice training. We learnt a lot of new things in the training. We would like to thank our college for sponsoring and giving their consent for training. We would also like to thank Indo German Tool Room and our instructor.