Report on One Week Training Programme

On

Advanced Structural Non-Linear Analysis using ANSYS

From 16th Dec to 20th Dec 2019

Organized By Mentor Institute

Sardar Patel College of Engineering, Andheri, Mumbai

Under Twinning Activity of TEQIP – III for Mentee Institute

Govt. Engineering College, Rewa, M.P.

Conducted By

ShirshTechnosolutions, Mumbai

ANSYS Channel Partner

Venue: Room No.213,
SardarPatel College of Engineering, Andheri

Organiser	Coordinator
Dr. Santosh B. Rane,	Dr. Sudhakar U. Umale,
Dean Academics, SPCE	Associate Professor, MED, SPCE

Report of One Week Training Programme on
Advanced Structural Non-Linear Analysis using ANSYS

One Week training programme was organised by Mentor Institute Sardar Patel College of Engineering, Andheri, Mumbai under Twinning Activity of TEQIP – III for Mentee Institute Govt. Engineering College, Rewa, M.P. on Advanced Structural Non-Linear Analysis using ANSYS from 16th December to 20th December 2019 for students and faculty.

This program was explicitly structured so as to give a detailed knowledge regarding non-linear stress analysis using ANSYS. The programme included presentations on fundamentals of Finite Element Analysis (FEA), informative case studies in industries regarding failures, Hands-On experience on ANSYS software for nonlinear structural analysis problems delivered by expert speaker Mr. AshishVajir, CTO at ShirshTechnosolutions, Mumbai.

Numerical simulation plays an indispensable role in manufacturing process by improving product design time, quality and performance. Simulation of collapse of a shell, interaction of multiple parts, behavior of a rubber steel, post yield strength of metals, manufacturing process and so on using linear approximation gives poor accuracy hence they have become less acceptable in industry. ANSYS, Inc. is a pioneer in the non linear analysis. The ANSYS mechanical program's non linear capabilities have evolved according to emerging analysis needs, maturity of analysis methods and increasing computing power.

The structured training programme and valuable inputs from the speaker brought a clear idea about the Nonlinear Structural problems. The details for the session for the respective days are as follows:

DAY-1
The first day of the program commenced with the greeting of programme instructor Mr.AshishVajir alumni of SPCE by Dr.Santosh Rane, Dean Academics. Dr.Sudhakar Umale, Associate Professor, MED gave the introduction of guest and felicitated him with sapling as a welcome gift.
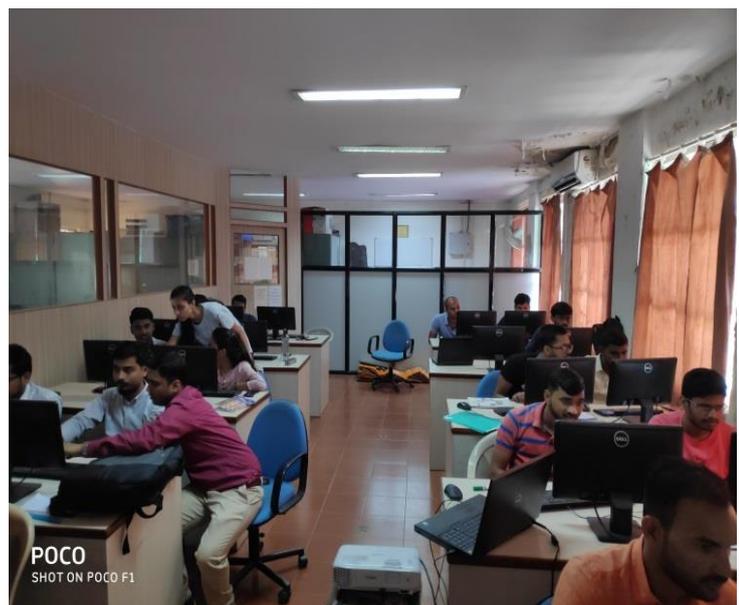
Then, Dr. Santosh Rane gave an overall idea of a training programme which included opportunities in the field of simulation, the importance of simulation software, objectives of the programme, expected outcomes, schedule of the programme, modules covered in the programme. He informed that

participants (2) faculty members from GEC,Rewa, MP and 20 students from SPCE, Andheri has participated in the workshop. He elaborated that this program was specifically designed to provide a practical hands-on learning experience of non-linear analysis using ANSYS software.
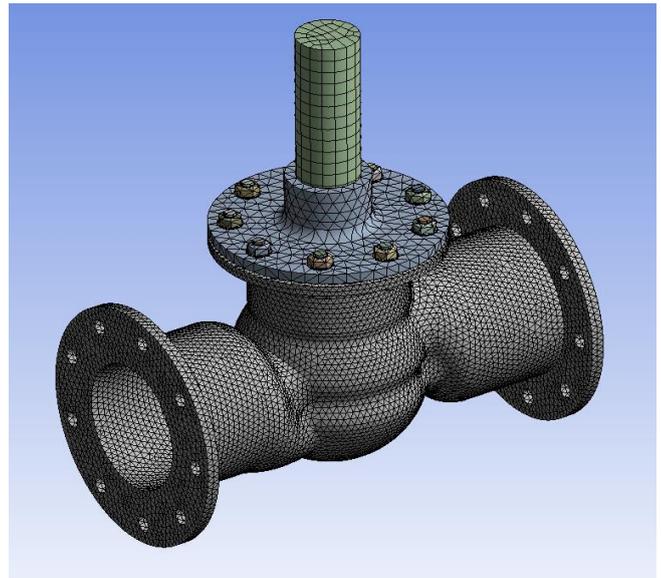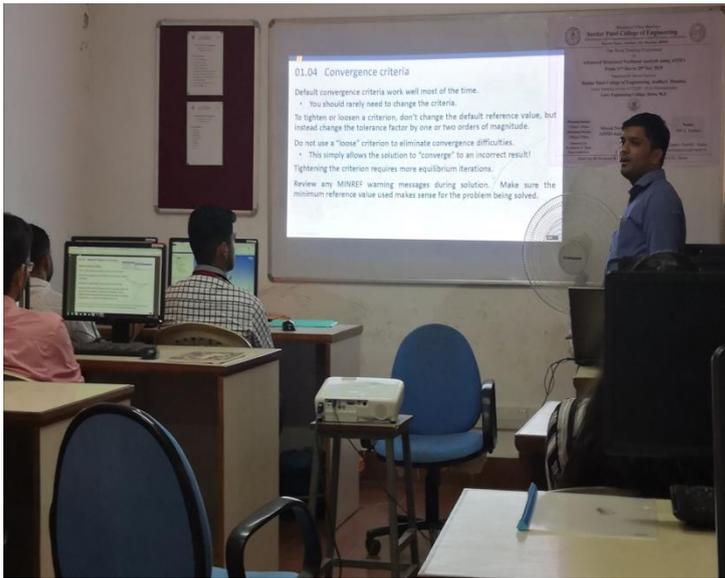

Program instructor Mr.AshishVajir took the charge of the session and addressed the participants by valuable information on the basics of FEA, Linear analysis, a key application of software and career opportunities and challenges in the stress analysis field. He explained the features of the software and stress analysis for various loading criteria and boundary conditions.

After a lunch break, using presentation, program instructor discussed problem of globe valve model for the involvement of participants in a training programme and complete understanding of linear structural problems. Later he discussed different ways to approach analysis problems. He demonstrated the computation of stress and deformation values for different loading condition. Important concepts of modelling approach, geometry, material selection, coordinate system, connection were explained to the students. He also explained the procedure of mesh generation, mesh control, force and constraint application to the component. Lab systems with ANSYS version 19.0 were used for conducting the analysis for theworkshop.

DAY-2

Second day of workshop started with detail information on factors influeincing the selection of elements viz. structural behavior (bulk or bending deformation) and material behavior (nearly incompressible to fully incompressible). Further analysis started with improved modeling approach and enhanced mesh techniques of using multizone meshing on globe valve assembly model. Realistic conditions for application of load and constraints were considered with bolt pre load and contact friction for simulation. Results were obtained for total deformation and stress developed. Program instructor explained different ways of solving the stress analysis problem usingANSYS.
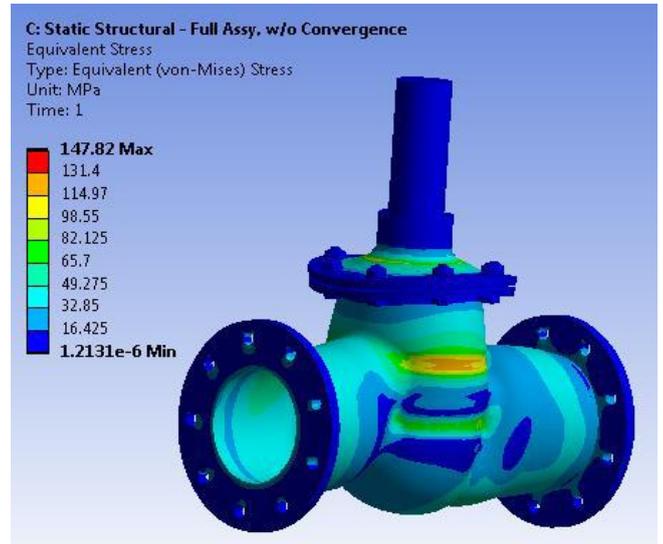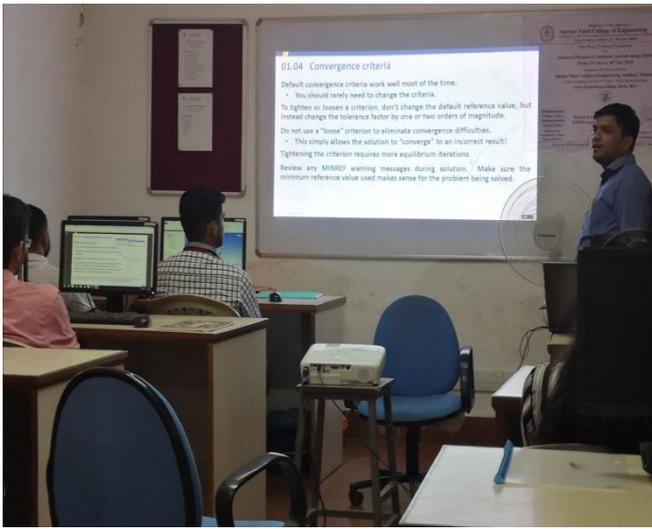
DAY-3

Program instructor gave introduction on concept of rate independent plasticity, metal plasticity and hardening, yield criteria, flow rule and hardening rules. He also explained nonlinear behavior and types of non linearities such as geometric nonlinearities, marerial nonlinearities and contact nonlinearities. Shell disk model is used to validate non linearities therby different material properties were varied to evaluate the change in stress and strain on application of loads. After a lunch break, again the hands-on session on Structural Non-Linear Analysis started with detailed training on making a cross-section, result with different types of meshing and loading at different location of the components.
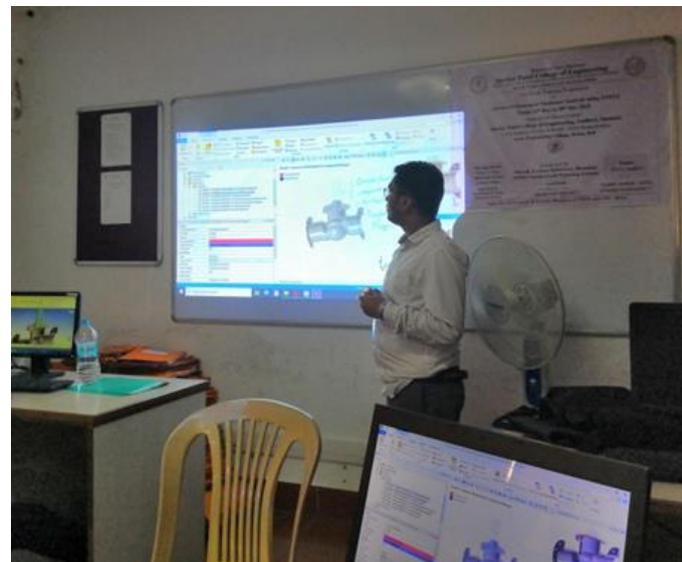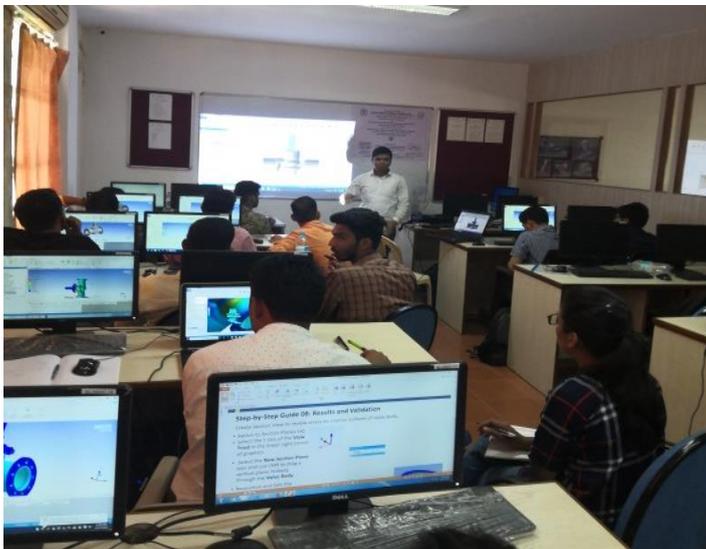

DAY-4

Detailed explanation was given on contact stiffness study with different types such as bonded, no separation, frictionless, rough and frictional. Various models like shell inside shell, thin spring plate were taken for practice to understand different behavior between solid face to solid face, surface body to surface body and surface body edge to surface body face in body types contact. Participants performed analysis on cantilever beam like structure to understand mechanical support relationships with rigid bodies for contact between two rigid bodies and contact between one rigid and one flexible body.
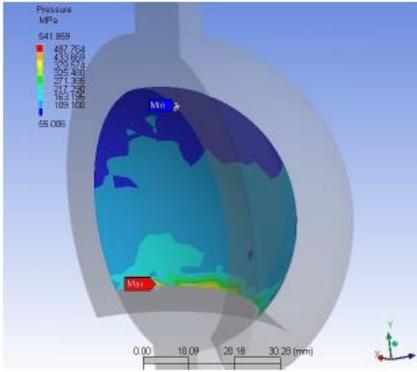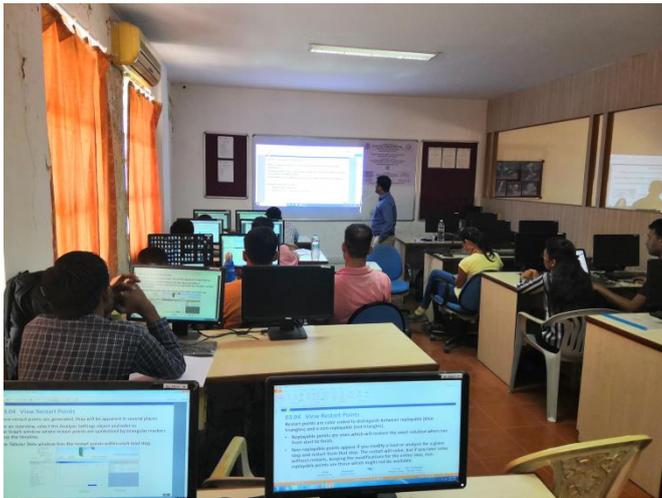
Solid Face to Solid Face
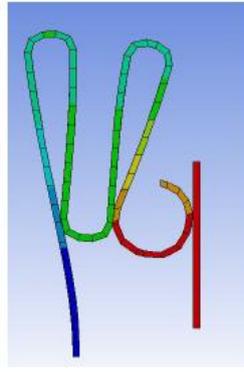
Surface Body Face to Surface body (or Solid body) Face
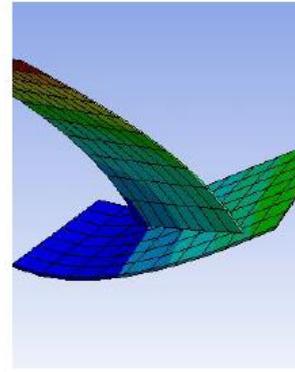
Surface Body Edge to Surface Body (or Solid) Face

DAY-5

Fifth day of workshop begun with linear perturbation, buckling analysis and post buckling analysis. Instructor explained eigen value buckling procedure. Load control and displacements control methods were studied to solve buckling of component problems. After applying load and constraint to the model, instructor explained on varying numbers of buckling mode and checking buckling mode shapes and load multipliers. He also explained the importance of post buckling behavior in failure analysis. To solve such problems non linear stabilization technique is used which is relatively new method introduced in ANSYS

Dr.Santosh Rane thanked the expert speaker for his valuable presence and sharing his expert knowledge, imparting a specialized training using ANSYS and making one-day workshop memorable one. The workshop ended with a group photo session.

Dr. SantoshRane, Dr. Sudhakar Umale with program instructor and students