TRAINING PROGRAM

ANSYS WORK BENCH 18.0

Ansys Inc. is an American public company based in Canonsburg, Pennsylvania. It develops and markets engineering simulation software. Ansys software is used to design products and semiconductors, as well as to create simulations that test a product's durability, temperature distribution, fluid movements, and electromagnetic properties.

Ansys was founded in 1970 by John Swanson. Ansys went public on NASDAQ in 1996. In the 2000s, Ansys made numerous acquisitions of other engineering design companies, acquiring additional technology for fluid dynamics, electronics design.

KEY FEATURES

Effective Upskilling Planned Curriculum
Team Learning Awesome Quizzes
Complete Hands on

The below Curriculum is Scheduled for 2 weeks

CURRICULUM

Module 1. Design modeler
1.1. How to create and modify geometry in preparation for analysis
1.2. How to navigate within the graphical user interface
1.3. How to generate 2D sketches and convert them into 2D or 3D models
1.4. How to modify 2D and 3D geometry
1.5. How to import existing CAD geometry
1.6. How to modify and clean up imported CAD
1.7. How to model assemblies
1.8. How to utilise parameters

Module 2. Meshing Platform
2.1. Work with ANSYS Meshing Platform,
2.2. Understand the different meshing methods available for 2D and 3D geometries
2.3. Create tetrahedral meshes
2.4. Create hexahedral meshes
2.5. Create inflation (boundary) layer meshes near to walls
2.6. Apply advanced controls to refine and coarsen the mesh in different regions of the domain
2.7. Examine the quality of the mesh

Module 3. Boundary Conditions
3.1. Types of loads
3.2. Types of supports
3.3. Material Properties

Module 4. Solving
4.1. Set solver controls
4.2. Set up solution monitors
4.3. Provide an initial solution
4.4. Display and examine results

Module 5. Result and Simulation
5.1. Result
5.2. Simulation and Animation

Module 6. Analysis Exercise
6.1. Static Analysis
6.2. Thermal Analysis
6.3. Modal Analysis
6.4. Dynamic Analysis
6.5. Computational Fluid Dynamics