Serial No:	Day Wise Topic
DAY : 1	**Introduction** ❖ Product Design cycle. **Before you start using Ansys** ❖ CAD/CAM/CAE ❖ History/need/advantages/limitations of FEM/FEA ❖ Introduction to the Ansys GUI
DAY : 2	**Introduction of FEM** ❖ Basics of FEM ❖ Different types of analysis using Ansys software. ❖ General Steps of the Finite Element Method ❖ Explanation of 1D, 2D and 3D Elements with examples of ANSYS
DAY : 3	**About ANSYS** ❖ ANSYS Family of products with their capabilities ❖ Product Launcher ❖ Launcher Tasks ❖ Launcher Menu Options ❖ Introduction to Ansys interface ❖ Introduction about coordinate system.
DAY-4	**Preprocessor:** **Solid Modeling** ❖ Basic geometrical entities creation in solid modeling like key points, lines, areas.
DAY : 5	❖ Working with Boolean operations for 1d objects like add, subtract, intersect, overlap, glue, etc.
DAY : 6	❖ Introduction about APDL Language ❖ Using the session editor ❖ Saving and resuming the file in Ansys environment.

	❖ Pan/zoom/rotate for model ❖ Offset the UCS in Ansys environment ❖ Rotate the UCS in Ansys environment
DAY : 7	❖ Assigning Element Attributes before meshing. ❖ Using mesh Controls for generating 1d mesh.
DAY : 8	**Element Type:** ❖ Define element type according to analysis **Real Constant:** ❖ Define real constant according to element if required **Material Properties:** ❖ Assigning material properties to solid model according to requirement **Boundary and loading Conditions:** ❖ Types of Loads ❖ Applying Supports
DAY : 9	**Structural Analysis:** ❖ Linear Static analysis with taking link elements. ❖ Linear Static analysis with taking beam element.
DAY : 10	❖ Generation of Area with key point, lines and 2d primitives. ❖ Working with Boolean operation Like add, subtract, intersect and divide, etc. for 2d entities.
DAY : 11	❖ Assigning Element Attributes before meshing. ❖ Using mesh Controls for generating 2d mesh. ❖ Generating free mesh for 2d elements by

	using smart size option	
DAY : 12	❖ Defining no. of elements on line or element size for meshing 2d model ❖ Generate mapped mesh for 2d model	
DAY : 13	❖ Linear Static analysis of trusses. ❖ Linear Static analysis by taking plane element.	
DAY : 14	❖ Linear Static analysis by taking plane element with thickness option. ❖ Linear Static analysis by taking plane element with axis symmetry option.	
DAY : 15	**Introduction of Modelling** ❖ Working with Boolean operations for 3d objects like add, subtract, intersect, overlap, glue, etc.	
DAY : 16	❖ Generating model by using 3d operation like extrude, revolve, sweep. ❖ Creation of nodes and elements directly	
DAY : 17	❖ Generating model exercise-1	
DAY : 18	❖ Generating model exercise-2	
DAY : 19	❖ Generating model exercise-3	
DAY : 20	**Introduction of Meshing** ❖ Assigning Element Attributes before meshing. ❖ Mesh Controls for	

	defining 3d models ❖ Meshing for 3d elements.	
DAY : 21	❖ Meshing by using manual option. ❖ Meshing by using The ANSYS Mesh Tool ❖ Free Meshing by using smart size option	
DAY : 22	❖ Mapped meshing with concatenate command ❖ Mapped meshing for 3d models by using meshing controls.	
DAY : 23	❖ Linear Static analysis by taking solid element with tetrahedral element ❖ Linear Static analysis by importing model from cad environment	
DAY : 24	**Thermal Analysis:** ❖ Conduction analysis	
DAY : 25	❖ Convection analysis	
DAY : 26	❖ Radiation analysis	
DAY : 27	**Post-processing** ❖ Contour Plot Viewing ❖ List of results ❖ Query result on node or elements ❖ Defining element table for beam analysis. ❖ Contour plot results for element table for SFD & BMD with beam element.	
DAY : 28	**Report Generation** ❖ Report Generator	
DAY : 29-30	Major Project	