

Ansys	Total Duration : 80 Hrs
Session 1	<ul style="list-style-type: none">• Introduction to FEM/FEA concepts• Introduction to Ansys• Creating geometry• Generation of key points, lines,
Session 2	<ul style="list-style-type: none">• Areas & volumes• Extruding areas, lines & key points• Work plane management, coordinate systems.• Editing geometry• Performing Boolean operations (Add, subtract, intersect)
Session 3	<ul style="list-style-type: none">• Boolean operation- (divide, glue, partition, overlap)• Move/modify,• Copy& reflect geometries.• Check geometric properties.• Import geometry from other CAD software
Session 4	<ul style="list-style-type: none">• Material Definition• Explaining about nodes & elements• Meshing• Types of mesh-Free& mapped meshing• Explaining about mesh attributes. Size controls• Meshing key points, lines, areas & volumes.
Session 5	<ul style="list-style-type: none">• Mapped meshing.• Mesh concatenation• Mesh extrusion,• Mesh sweeping.• Hexa – tetra conversion• Define loading & boundary conditions.• Different types of loads• Different types of constraints.
Session 6	<ul style="list-style-type: none">• Introduction to Static Structural Analysis• Beam analysis,• Truss analysis• Stresses in bars•

Session 7	<ul style="list-style-type: none"> • Stresses in composite bars, strain energy calculations • Finding out reaction solutions • Modal analysis
Session 8	<ul style="list-style-type: none"> • Transient Dynamic analysis • Over view on Non-linear analysis, • Thermal analysis
Session 9	<ul style="list-style-type: none"> • Coupled field analysis • APDL Concepts • Advanced post processing • Report generation
Session 10	<ul style="list-style-type: none"> • Modal Analysis • Impulse Loading • Sinusoidal Loading • Harmonic Loading
Session 11	<ul style="list-style-type: none"> • Transient Heat Analysis
Session 12	<ul style="list-style-type: none"> • Fluid Dynamics
Session 13	<ul style="list-style-type: none"> • Test & Review.