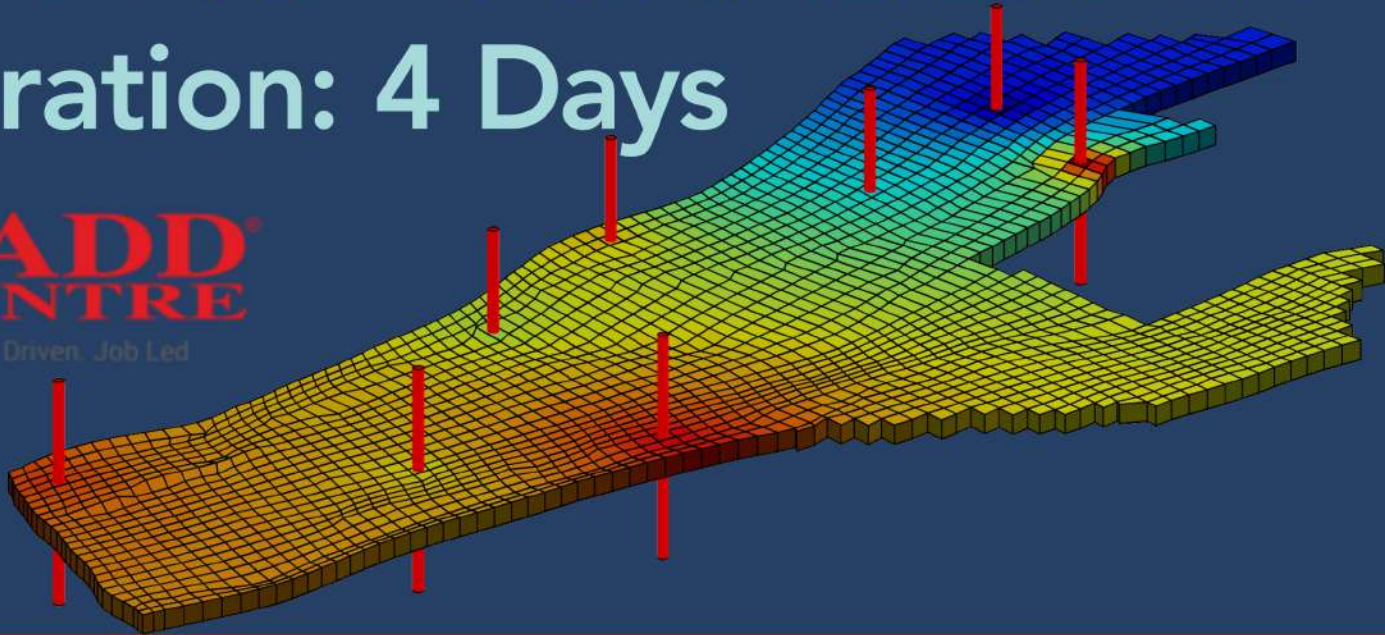


ANSYS WORKBENCH

Duration: 4 Days

**CADD
CENTRE**
Skills Driven. Job Led



ANSYS Workbench is a software for performing structural, thermal, & electromagnetic analyses. The class focuses on geometry creation & optimization, attaching existing geometry, setting up the finite element model, solving, and reviewing results.

COURSE CURRICULUM

- ✓ Theory of FEA
- ✓ Exploring the GUI
- ✓ Graphics picking
- ✓ General analysis procedure
- ✓ Solid modeling
- ✓ Defining work planes
- ✓ Coordinate systems
- ✓ Importing geometry
- ✓ Defining element attributes
- ✓ Element types
- ✓ Generating mesh
- ✓ Free meshing
- ✓ Mapped meshing
- ✓ Defining material
- ✓ Defining loads & B.C.
- ✓ Select logic
- ✓ Solvers
- ✓ Post-processing
- ✓ Structural static analysis
- ✓ Modal analysis
- ✓ Transient dynamic analysis
- ✓ Nonlinear analysis
- ✓ Thermal analysis
- ✓ Coupled field analysis

LEARNING OUTCOME

- Good knowledge in structural & static analysis
- Can identify various types of stress on a structure under various working conditions
- Can find the natural frequency of a structure
- Can get an overview on transient dynamic & non-linear analysis
- To know thermal and coupled field analysis



Books to be given:
ANSYS Reference Guide



SIX SIGMA SYNERGY SDN BHD (756845-A)

B-02-01, Ground Floor, 3 Two Square, No. 2,
Jalan 19/1, Seksyen 19, 46300 Petaling Jaya, Selangor



03-79566306, 019-3612598 ; FAX: 03-7956 6806



www.6sigmasynergy.com.my info@sixsigmasynergy.com.my

