ANSYS Mechanical using Command Objects

Duration: 2 days

- This training class provides an overview of using Command Objects in ANSYS Mechanical to access advanced solver functionality.

- Objectives of this training course:
  - Learn how to use the Mechanical APDL interface to verify and postprocess models
  - Gain an understanding of APDL command syntax, as well as APDL parameters
  - Explore different ways in which "Commands" objects can be used to access advanced pre- and post-processing functionality

Prerequisite: ANSYS Workbench - Simulation - Structural

- It is intended for users already familiar with ANSYS Mechanical:
  - The ANSYS Mechanical GUI and analysis procedure are not covered here but in the *ANSYS Mechanical Introduction* course
  - Familiarity with ANSYS Mechanical APDL (ANSYS Parametric Design Language) is helpful but not required.

Course Topics Include:

- Introduction to Mechanical APDL
- Mechanical APDL Element Attributes
- Postprocessing in Mechanical APDL
- APDL Commands
- Using APDL in Mechanical

**Fee:** Rs 14,000/- + Applicable Taxes

**For more information / registration contact:**

**Email:** india-register@ansys.com

Phone: +91-20-66522545 ; Fax : +91-20-66522600