

ANSYS Introductory Mechanical Training

About ANSYS Mechanical

ANSYS Mechanical provides solutions for many types of analyses including structural, thermal, modal, linear buckling and shape optimization studies.

ANSYS Mechanical is an intuitive mechanical analysis tool that allows geometry to be imported from a number of different CAD systems. It can be used to verify product performance and integrity from the concept phase through the various product design and development phases.

The use of ANSYS Mechanical accelerates product development by providing rapid feedback on multiple design scenarios, which reduces the need for multiple prototypes and product testing iterations.

About the Training

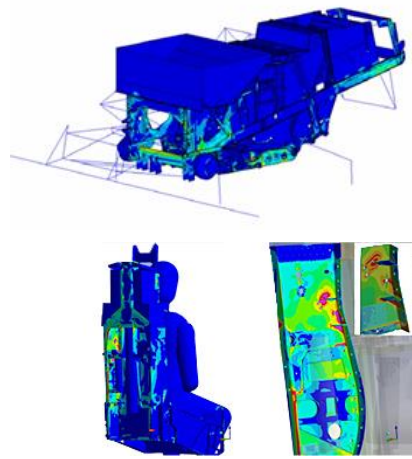
This three-days training course teaches students how to effectively use ANSYS Mechanical to build a mechanical simulation model, analyze it and interpret the results.

Who Should Attend

This course is designed for new users that wants to get use to basic control and usage of ANSYS Workbench Mechanical

Pre-requisites

A technical education and background is recommended but an engineering degree is not required.



Topics Covered

- Introduction
- Mechanical Basics
- General Preprocessing
- Static Structural Analysis
- Vibration Analysis
- Thermal Analysis
- Linear Buckling Analysis
- Results Postprocessing
- CAD & Parameters

Each course chapter is followed by "hands-on" workshops and exercises.

HRDF Claimable

Seminar Title	ANSYS Introductory Mechanical Training (3 days)
Date	Please refer to our website here
Time	9.00 am to 5.00 pm
Venue	CAD-IT Consultants (M) Sdn Bhd 607, Block A, Kelana Centre Point, Jln SS7/19, Kelana Jaya, 47301, Petaling Jaya, Selangor.

For more information, please call 03-7806 4758 or send your enquiry to eventsma1@cadit.com.sg

CAD-IT Consultants (M) Sdn Bhd

www.cadit.com.sg

Bringing you tomorrow's technology... today!