Table of Contents

Finite Element Analysis (FEA)

Finite_element_method

FEA Theory and Fundamentals

- Introduction to the Finite Element Method (FEM)
- Stiffness Matrix and Degrees of Freedom (DOF)
- Shape Functions and Interpolation
- Mesh Convergence, Error Analysis, and Verification

ANSYS APDL (Parametric Design Language)

APDL Basics and Geometry

- APDL Interface (GUI), Command Logic, and File Structure (.db, .rst, .mac)
- Geometry Modeling: Keypoints, Lines, Areas, Volumes, and Boolean Operations

Elements, Materials, and Meshing

- Element Types (BEAM, SHELL, SOLID) and Real Constants
- Material Models (Linear Elastic, Hyperelastic, Plasticity)
- Meshing: Mapped/Free Mesh, Size Control, and Sweep

Loads, Solution, and Postprocessing

- Boundary Conditions (DOF) and Loads (Force, Pressure, Thermal, Body Loads)
- Solution Phase: Static, Modal, Transient, Harmonic
- Postprocessing (POST1 / POST26): Contour Plots and Data Extraction

APDL Programming (Scripting)

- Variables, *GET Command, Loops (*DO), and Macro (.mac) Writing

Case Studies

- Seat Belt Analysis

ANSYS Mechanical (Workbench)

- Workbench Interface, Engineering Data, and Geometry Links
- Linear Static Analysis Setup and Solution

- Contact Mechanics: Bonded, Frictional, No Separation
- Modal Analysis and Vibration Characteristics
- Steady-State and Transient Thermal Analyses
- Fatigue Analysis: S-N Curves and Life Calculation
- Running APDL Codes within Workbench using Command Snippets

Advanced Analyses and Special Topics

- Non-linear Analyses (Geometric, Material, and Contact Non-linearities)
- Explicit Dynamics: Crash, Blast, and Drop Tests (LS-DYNA/Autodyn)
- Topology Optimization and Weight Reduction Strategies
- Submodeling: Accurate Stress Analysis in Localized Regions

The information in this document is cited from UCH Wiki.