

Introduction to Finite Element Method

ANSYS Design Modeler

- **1D**
 - Line modeling
 - Truss element
 - Beam element
 - Pipe element
- **2D**
 - Surface modeling
 - Plane stress element
 - Plane strain element
 - Shell element
 - Membrane element
- **3D**
 - Solid modeling
 - Solid elements

Material models

- **Elastic models**
 - isotropic
 - orthotropic
 - anisotropic
- **Plastic models**
 - Kinematic hardening
 - Isotropic hardening

Mesh generation

- **1D - 2D -3D mesh generation**
 - Methods
 - algorithms
 - mesh setting

Static structural analysis

- **Governing equations**
- **Boundary conditions**
- **Linear vs. non-linear analysis**
 - Analysis settings
 - Contact tools
- **Non-linear analysis**
 - Newton-Raphson algorithm
 - NL Material
 - NL geom
 - NL contact
 - Post-processing

Thermal analysis

- **Steady-state thermal analysis**
 - Governing equations
 - Boundary conditions
 - Dirichlet
 - Neuman
 - Material considerations
 - Mesh considerations
 - Linear vs non-linear thermal analysis
 - Material
 - Radiation
 - Heat transfer coefficient
 - Post-processing
- **Thermal stress analysis**
 - coupled
 - uncoupled

Buckling analysis

- **stability and instability**
- **Eigenvalue buckling**
 - P-delta graph
 - Bifurcation point

- Post processing
 - Mode shape
 - Load multiplier
- Result interpretation
- **Intro to non-linear buckling**
 - Imperfection
 - UPGEOM command
 - Algorithms
 - Newton-Raphson
 - Stabilize
- Arc-Length