

# Table of Contents

## Chapter 1 The Basics of the Finite Element method

- 1.1 Method of weighted residuals
  - 1.1.1 Sub-domain method (Finite Volume Method)
  - 1.1.2 Galarkin method
- 1.2 Rayleigh-Ritz method
- 1.3 Finite element method
- 1.4 Finite element method in two-dimensional elastostatic problems
  - 1.4.1 Elements of finite element procedures in the analysis of plane elastostatic problems
  - 1.4.2 Fundamental formulae in plane elastostatic problems
  - 1.4.3 Variational formulae in elastostatic problems; the principle of virtual work
  - 1.4.4 Formulation of the fundamental finite element equations in a plane elastostatic problem

## Chapter 2 Overview of ANSYS structure and visual capabilities

- 2.1 Starting the programme
  - 2.1.1 Preliminaries
  - 2.1.2 Constituent elements
  - 2.1.3 Saving current database and opening existing
  - 2.1.4 Organisation of files
  - 2.1.5 Exiting the programme
- 2.2 Preprocessing stage
  - 2.2.1 Types of analysis
  - 2.2.2 Element types
  - 2.2.3 Material properties
  - 2.2.4 Construction of model
  - 2.2.5 Meshing
- 2.3 Solution stage
  - 2.3.1 Constraints
  - 2.3.2 Loads
  - 2.3.3 Solution
- 2.4 Postprocessing stage
  - 2.4.1 Types of postprocessors
  - 2.4.2 General postprocessor
    - 2.4.2.1 Read results
    - 2.4.2.2 Plot results
- 2.5 Visual capabilities
  - 2.5.1 Plot controls
  - 2.5.2 Two and three dimensional primitives
  - 2.5.3 Modelling options

## Chapter 3 Application of ANSYS to stress analysis

- 3.1 Cantilever beam
- 3.2 The principle of St. Venant
- 3.3 Stress concentration due to elliptic inclusions and holes
- 3.4 Stress singularity problem
- 3.5 Two-dimensional contact stress

## Chapter 4 Application of ANSYS to mode analysis

- 4.1 Vibration modes of cantilever beam
- 4.2 Vibration modes of a shaft with concentrated mass
- 4.3 Vibration modes of a suspension for hard disc drive

## Chapter 5 Application of ANSYS to fluid flow analysis

- 5.1 Analysis of a conical diffuser flow
- 5.2 Analysis of inverse flow in a conical diffuser

## Chapter 6 Application of ANSYS to thermo mechanics

- 6.1 General characteristic of thermo mechanics problems
- 6.2 Examples of using ANSYS in thermo mechanics
  - 6.2.1 Heat convection and temperature gradients in a furnace
  - 6.2.2 Pipe intersection
  - 6.2.3 Heat dissipation from a developed surface

## Chapter 7 Application of ANSYS to contact between machine elements

- 7.1 General characteristic of contact problems
- 7.2 Examples of using ANSYS in analysis of contact problems
  - 7.2.1 Pin-in-hole interface fit
  - 7.2.2 Contact between concave surfaces
  - 7.2.3 Wheel-on-rail contact
  - 7.2.4 O-ring assembly