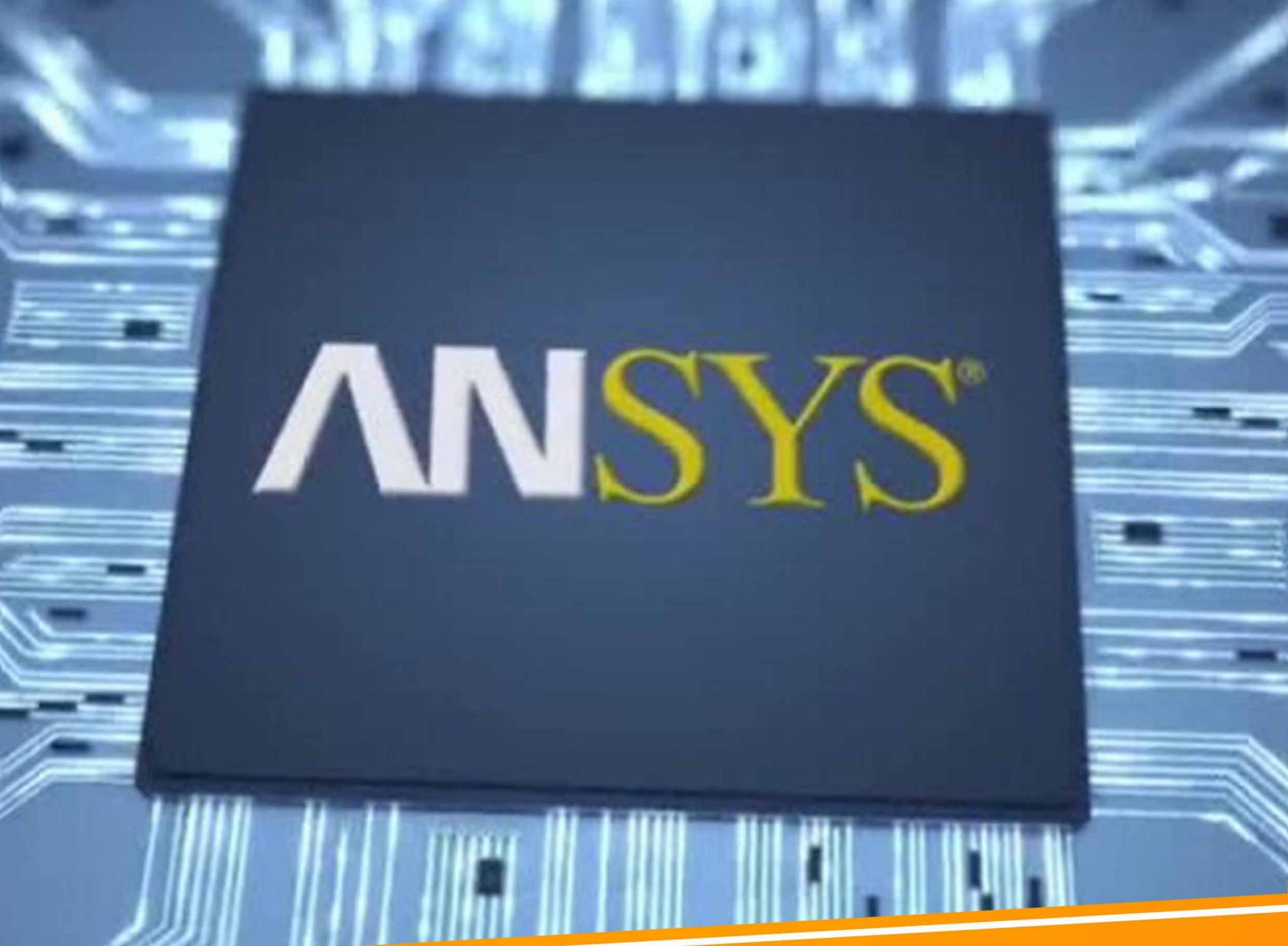




ANSYS

The image shows a dark, rectangular microchip with the ANSYS logo printed on its surface. The logo consists of the word "ANSYS" in a stylized font, with "AN" in white and "SYS" in yellow. The chip is set against a background of glowing blue circuit traces on a printed circuit board (PCB).

ANSYS[®]

About Course

ANSYS is a general-purpose, finite-element modeling package for numerically solving a wide variety of mechanical problems. These problems include static/dynamic, structural analysis, heat transfer, and fluid problems, as well as acoustic and electromagnetic problems. There are two methods to use ANSYS.

ANSYS

CURRICULUM

① Before you start using Ansys

Introduction to the Finite Element Method

What is the Finite Element Method?

General Steps of the Finite Element Method

Explanation of 1D, 2D and 3D Elements with examples of ANSYS Elements

Need of FEM

Types of analysis that can be done using ANSYS

Advantages of the Finite Element Method

Limitations of FEA

About ANSYS Inc.

ANSYS Family of products with their capabilities

Types of analysis that can be done with ANSYS.

Introduction to the Ansys GUI

Operation Modes of Ansys

Product Launcher

Launcher Menu Options

The ANSYS GUI

The Icon Toolbar Menu

Quitting Ansys

2

Selection Logic

Pan-Zoom-Rotate

Picking

Coordinate Systems

3

Solid Modeling

An Overview of Solid Modeling Operations

Working with Boolean operations

Working Plane

Importing of 3D models

4

Meshing

Free meshing or Mapped meshing

Setting Element Attributes

Selecting Element Type

Shape Function

Defining Element Types

Defining Section Properties

Assigning Element Attributes before meshing

Mesh Controls

The ANSYS Mesh Tool

Smart sizing

Meshing



5

Material Properties

Material Library

Specifying properties

6

Boundary Conditions

Types of Loads

Applying loads

7

Solvers

Types of Solvers

Solver Setup

Load Step Options

Solving Multiple Load Steps

8

Post-processing

Contour Plot Viewing

Time History Postprocessor (POST26)

Report Generator

9

Introduction to Non-Linear Analysis

10

Sample Structural Analysis



Using the Toolbar & Creating Abbreviations

Introduction to APDL

Using Parameters

Using the Start File

Using the Session Editor

Using Input Files

11

ANSYS Workbench

Introduction to ANSYS Workbench

Graphical User Interface

Static Structural Analysis

Modal Analysis

Thermal Analysis

Contact Recognition



 www.softcrayons.com

 (+91) 854 501 2345

   @softcrayons

 info@softcrayons.com

 693, Sector 14-A, Vasundhara,
Ghaziabad (U.P.), 201012