

## Introduction to ANSYS - Part 1

**Length:** 3 Days

Recommended for anyone who performs finite element analysis (FEA) of mechanical parts or fluids and has little or no ANSYS experience, Introduction to ANSYS, Part I is a three-day course that focuses on basic linear, static analyses in any discipline.

### OBJECTIVES:

After completing the course, attendees should be able to maneuver efficiently within the ANSYS Graphical user interface (GUI), build two- and three-dimensional models, apply loads and obtain solutions as well as effectively verify the results of an analysis and display results.

### COURSE TOPICS INCLUDE:

TABLE OF CONTENTS	
<b>1. Introduction</b>	<b>1-1</b>
<b>2. FEA and ANSYS</b>	<b>2-1</b>
A. About the Company	2-2
B. About ANSYS	2-4
C. What is FEA?	2-10
D. Instructor Example	2-13
E. Workshop	2-14
<b>3. Getting Started</b>	<b>3-1</b>
A. Interactive vs Batch Mode	3-2
B. Starting ANSYS	3-3
C. Product Launcher	3-4
D. ANSYS Workbench	3-9
E. Memory	3-12
F. GUI	3-15
G. The Database and Files	3-37
H. Exiting ANSYS	3-46
I. On-Line Help	3-47
J. Workshop	3-53

<b>4. ANSYS Basics</b>	<b>4-1</b>
A. Overview	4-2
B. Plotting	4-3
C. Picking	4-10
D. Coordinate Systems	4-16
E. Select Logic	4-21
F. Components	4-28
G. Workshop	4-31
<b>5. General Analysis Procedure</b>	<b>5-1</b>
A. Preliminary Decisions	5-5
B. Preprocessing	5-8
• Create Solid Model	5-8
• Create FEA Model	5-10
• Define Material	5-12
C. Solution	5-13
• Define Loads	5-13
D. Postprocessing	5-15
• Review Results	5-15
• Verification	5-18
<b>6. Creating the Solid Model</b>	<b>6-1</b>
A. What to Model?	6-3
B. Importing Geometry	6-14
• IGES Imports	6-14
• Connection Products	6-15
• Workshops	6-19
C. ANSYS Native Commands	6-20
• Definitions	6-21
D. ANSYS Native Geometry Creation	6-24
• Working Plane	6-24
E. ANSYS Coordinate Systems	6-30
• Active Coordinate System	6-30
• Global Coordinate System	6-31
• Local Coordinate System	6-32
• Working Plane Coord. System	6-33
•	
<b>7. Creating the Finite Element Model</b>	<b>7-1</b>
A. Overview	7-2
B. Element Attributes	7-4
C. Multiple Element Attributes	7-29
D. Workshop	7-36
E. Controlling Mesh Density	7-37
F. Mesh Order Control	7-46
G. Generating the Mesh	7-47
H. Changing a Mesh	7-49
I. Mapped Meshing	7-52
J. Hex-to-Tex Meshing	7-69
K. Mesh Extrusion	7-77
L. Sweep Meshing	7-82
M. F.E. Imports	7-88
N. Workshops	7-89

<b>8. Defining the Material</b>	<b>8-1</b>
A. Units	8-3
B. ANSYS Defined Materials	8-4
C. Material Model GUI	8-6
D. Listing Defined Materials	8-11
E. Workshops	8-12
<b>9. Loading</b>	<b>9-1</b>
A. Define Loads	9-3
B. Nodal Coordinate Systems	9-6
C. Displacement Constraints	9-11
D. Concentrated Forces	9-12
E. Verifying Loads	9-16
<b>10. Solution</b>	<b>10-1</b>
A. Solvers	10-2
B. Definitions	10-10
C. Multiple Loadsteps	10-11
D. Workshops	10-19
<b>11. Structural Analysis</b>	<b>11-1</b>
A. Preprocessing	11-3
B. Solution	11-5
C. Postprocessing	11-17
D. Workshops	11-25
<b>12. Thermal Analysis</b>	<b>12-1</b>
A. Preprocessing	12-3
B. Solution	12-5
C. Postprocessing	12-15
D. Workshop	12-22
<b>13. Postprocessing</b>	<b>13-1</b>
A. Query Picking	13-3
B. Results Coordinate System	13-6
C. Path Operations	13-9
D. Error Estimation	13-16
E. Load Case Combinations	13-25
F. Workshops	13-31
G. Results Viewer	13-32
H. Variable Viewer	13-40
I. Report Generator	13-43
J. Workshop	13-51

## 14. Short Topics

- A. Toolbar & Abbreviations
- B. Start File
- C. APDL
- D. Defining Parameters
- E. Using Parameters
- F. Retrieving Database Information
- G. Batch Mode
- H. Input Files
- I. Session Editor
- J. Workshops

## 14-1

- 14-3
- 14-9
- 14-10
- 14-11
- 14-15
- 14-18
- 14-22
- 14-23
- 14-28
- 14-30

## Appendix

### A. ANSYS Native Geometry Creation

- A. Definitions
- B. Top-Down Modeling Workshop
- C. Bottom-Up Modeling Workshops
- D. Best Modeling Practices

## A-1

## A-2

- A-3
- A-6
- A-22
- A-23
- A-37
- A-38