

Contents

1	Introduction.....	1
1.1	Concept	1
1.2	Nodes.....	3
1.3	Elements.....	3
1.4	Direct Approach	4
1.4.1	Linear Spring.....	5
1.4.2	Heat Flow	6
1.4.3	Assembly of the Global System of Equations.....	7
1.4.4	Solution of the Global System of Equations	11
1.4.5	Boundary Conditions	12
2	Fundamentals of ANSYS.....	15
2.1	Useful Definitions	15
2.2	Before an ANSYS Session	16
2.2.1	Analysis Discipline.....	16
2.2.2	Time Dependence.....	17
2.2.3	Nonlinearity.....	18
2.2.4	Practical Modeling Considerations	19
2.3	Organization of ANSYS Software	25
2.4	ANSYS Analysis Approach.....	25
2.4.1	ANSYS Preprocessor	26
2.4.2	ANSYS Solution Processor.....	26
2.4.3	ANSYS General Postprocessor	26
2.4.4	ANSYS Time History Postprocessor	26
2.5	ANSYS File Structure	26
2.5.1	Database File	27
2.5.2	Log File	27
2.6	Error File	27
2.6.1	Results Files	28
2.7	Description of ANSYS Menus and Windows	28
2.7.1	Utility Menu	28
2.7.2	Main Menu	30

2.7.3	Toolbar	30
2.7.4	Input Field	30
2.7.5	Graphics Window	30
2.7.6	Output Window	30
2.8	Using the ANSYS Help System	31
2.8.1	Help Contents	31
2.8.2	Help Index	32
2.8.3	Search in Help	32
2.8.4	Verification Manual	33
3	Fundamentals of Discretization	35
3.1	Local and Global Numbering	35
3.2	Approximation Functions	35
3.3	Coordinate Systems	40
3.3.1	Generalized Coordinates	40
3.3.2	Global Coordinates	40
3.3.3	Local Coordinates	40
3.3.4	Natural Coordinates	41
3.4	Shape Functions	47
3.4.1	Linear Line Element with Two Nodes	50
3.4.2	Quadratic Line Element with Three Nodes: Centroidal Coordinate	52
3.4.3	Linear Triangular Element with Three Nodes: Global Coordinate	53
3.4.4	Quadratic Triangular Element with Six Nodes	55
3.4.5	Linear Quadrilateral Element with Four Nodes: Centroidal Coordinate	57
3.5	Isoparametric Elements: Curved Boundaries	59
3.6	Numerical Evaluation of Integrals	63
3.6.1	Line Integrals	63
3.6.2	Triangular Area Integrals	67
3.6.3	Quadrilateral Area Integrals	69
3.7	Problems	72
4	ANSYS Preprocessor	75
4.1	Fundamentals of Modeling	75
4.2	Modeling Operations	75
4.2.1	Title	76
4.2.2	Elements	76
4.2.3	Real Constants	82
4.2.4	Material Properties	84
4.2.5	Element Attributes	86
4.2.6	Interaction with the Graphics Window: Picking Entities	87
4.2.7	Coordinate Systems	90

4.2.8	Working Plane	92
4.3	Solid Modeling	95
4.3.1	Bottom-up Approach: Entities	96
4.3.2	Top-Down Approach: Primitives	102
4.4	Boolean Operators	107
4.4.1	Adding	108
4.4.2	Subtracting	108
4.4.3	Overlap	110
4.4.4	Gluings	110
4.4.5	Dividing	111
4.5	Additional Operations	113
4.5.1	Extrusion	113
4.5.2	Moving and Copying	117
4.5.3	Keeping/Deleting Original Entities	119
4.5.4	Listing Entities	120
4.5.5	Deleting Entities	120
4.6	Viewing a Model	121
4.6.1	Plotting: PanZoom, and Rotate Functions	121
4.6.2	Plotting/Listing Entities	123
4.6.3	Numbers in the Graphics Window	124
4.7	Meshing	125
4.7.1	Automatic Meshing	125
4.7.2	Manipulation of the Mesh	132
4.8	Selecting and Components	133
4.8.1	Selecting Operations	133
4.8.2	Components	136
5	ANSYS Solution and Postprocessing	139
5.1	Overview	139
5.2	Solution	140
5.2.1	Analysis Options/Solution Controls	141
5.2.2	Boundary Conditions	143
5.2.3	Initial Conditions	144
5.2.4	Body Loads	144
5.2.5	Solution in Single and Multiple Load Steps	145
5.2.6	Failure to Obtain Solution	148
5.3	Postprocessing	149
5.3.1	General Postprocessor	150
5.3.2	Time History Postprocessor	150
5.3.3	Read Results	151
5.3.4	Plot Results	153
5.3.5	Element Tables	156
5.3.6	List Results	159
5.4	Example: One-dimensional Transient Heat Transfer	159

6	Finite Element Equations	177
6.1	Method of Weighted Residuals	177
6.1.1	Example: One-Dimensional Differential Equation with Line Elements	179
6.1.2	Example: Two-Dimensional Differential Equation with Linear Triangular Elements.....	187
6.1.3	Example: Two-Dimensional Differential Equation with Linear Quadrilateral Elements	203
6.2	Principle of Minimum Potential Energy	221
6.2.1	Example: One-Dimensional Analysis with Line Elements	228
6.2.2	Two-Dimensional Structural Analysis	234
6.3	Problems.....	272
7	Use of Commands in ANSYS	281
7.1	Basic ANSYS Commands.....	281
7.1.1	Operators and Functions	285
7.1.2	Defining Parameters.....	286
7.2	A Typical Input File.....	290
7.3	Selecting Operations	292
7.4	Extracting Information from ANSYS	297
7.5	Programming with ANSYS.....	300
7.5.1	DO Loops.....	300
7.5.2	IF Statements.....	302
7.5.3	/OUTPUT and *VWRITE Commands	304
7.6	Macro Files.....	306
7.7	Useful Resources.....	308
7.7.1	Using the Log File for Programming.....	308
7.7.2	Using the Verification Problems for Programming.....	310
8	Linear Structural Analysis	313
8.1	Static Analysis.....	313
8.1.1	Trusses.....	313
8.1.2	Beams.....	320
8.1.3	Three-Dimensional Problems.....	325
8.1.4	Two-Dimensional Idealizations.....	329
8.1.5	Plates and Shells.....	355
8.2	Linear Buckling Analysis.....	383
8.3	Thermomechanical Analysis	392
8.4	Fracture Mechanics Analysis	400
8.5	Dynamic Analysis	411
8.5.1	Modal Analysis.....	413
8.5.2	Harmonic Analysis.....	423
8.5.3	Transient Analysis	437

9 Linear Analysis of Field Problems.....	455
9.1 Heat Transfer Problems.....	455
9.1.1 Steady-state Analysis	456
9.1.2 Transient Analysis	497
9.1.3 Radiation Analysis	519
9.2 Moisture Diffusion	525
10 Nonlinear Structural Analysis	539
10.1 Geometric Nonlinearity.....	542
10.1.1 Large Deformation Analysis of a Plate	543
10.1.2 Post-buckling Analysis of a Plate with a Hole	546
10.2 Material Nonlinearity	551
10.2.1 Plastic Deformation of an Aluminum Sphere	552
10.2.2 Plastic Deformation of an Aluminum Cylinder	555
10.2.3 Stress Analysis of a Reinforced Viscoelastic Cylinder	562
10.2.4 Viscoplasticity Analysis of a Eutectic Solder Cylinder.....	566
10.2.5 Combined Plasticity and Creep.....	572
10.3 Contact	579
10.3.1 Contact Analysis of a Block Dropping on a Beam	581
10.3.2 Simulation of a Nano-Indentation Test	587
11 Advanced Topics in ANSYS.....	595
11.1 Coupled Degrees of Freedom.....	595
11.2 Constraint Equations	597
11.3 Submodeling.....	603
11.4 Substructuring: Superelements.....	609
11.4.1 Generation Pass.....	612
11.4.2 Use Pass.....	618
11.4.3 Expansion Pass.....	620
11.5 Interacting with External Files	621
11.5.1 Reading an Input File	622
11.5.2 Writing Data to External ASCII Files	622
11.5.3 Executing an External File	625
11.5.4 Modifying ANSYS Results	627
11.6 Modifying the ANSYS GUI.....	628
11.6.1 GUI Development Demonstration	635
11.6.2 GUI Modification for Obtaining a Random Load Profile	643
11.6.3 Function Block for Selecting Elements Using a Pick Menu	648
Erratum to: The Finite Element Method and Applications in Engineering Using ANSYS®	E1
References	651
Index.....	653