ANSYS Version 13

The book was developed using ANSYS version 12.1. Different versions and different computer systems will given slightly different numerical values for the results given in the examples.

ANSYS has now released version 13 . Updates are usually extensions with new items. Unfortunately, in version 13 some classical elements have been removed from the GUI menus and left undocumented in the HELP files. Principal among these are the LINK1, PLANE42, BEAM3, and BEAM4 elements. These elements are still available but undocumented, and they must be implemented using the command line. The GUI actions to select an element have to be replaced by the command line statements. Each command line statement is followed by ENTER.

An element is selected by the command

ET, *type*, *name*

where *type* is a number and *name* is LINK1, PLANE42, or BEAM3. The options for the element can be selected by the command

KEYOPT,*type*,*option-number, option key-number*

where *type* is the type number of the element. Each option is numbered numerically and the default option key-number is zero. The real constants for the element can be entered by the command

R,*nset,R1,R2,R3,R4,R5,R6*

The axial force member used to model truss elements and axially loaded rod elements is presented in Sections 1.4 and 1.5. This stiffness matrix is created in ANSYS by using the LINK1 element. The area can be entered as the first parameter on the real constant command:

ET,1,LINK1
R,1,*area*

LINK180 is a 3D line element that can be substituted for the LINK1 element but it may be necessary to constrain the third direction.

The classical beam bending element analyzed in Section 1.7 is modeled in ANSYS by the BEAM3 element. The 3D version is BEAM4. The area *A*, moment of inertia *I*, and height *h* can be entered as the first three parameters by the real constant (R) command:

ET,1,BEAM3
R, 1,*area,Izz,h*

For the default KEYOPT(9) = 0, the *x*-component of force on an element at the *I*-node and the *J*-node can be obtained from the ETABLE as SMISC = 1 and SMISC = 7. The *Y*-component of force on an element at the *I*-node and the *J*-node are obtained from the ETABLE as SMISC = 2 and SMISC = 8. The bending moment about the *Z*-axis on an element at the *I*-node and the *J*-node are obtained from the ETABLE as SMISC = 6 and SMISC = 12. The axial stress on the element at the *I*-node and the *J*-node are obtained from the ETABLE as LS = 1 and LS = 4. The bending stress on the +*Y* side of the element at the *I*-node and the *J*-node are obtained from the ETABLE as LS = 2 and LS = 5. The bending stress on the -*Y* side of the element at the *I*-node and the *J*-node are obtained from the ETABLE as LS = 3 and LS = 6.

Results for line elements can also be displayed by

GENERAL POSTPROCESSOR > LIST RESULTS
ELEMENT SOLUTION > STRESS
LINE ELEMENT RESULTS > ELEMENT RESULTS

The element BEAM 188 is a 3D 2-node element that can be used in place of BEAM3 and BEAM4. It is based on Timoshenko beam equations and therefore includes shear deformations, so that the calculated displacements will be slightly different from the result using BEAM3 or BEAM4 which neglect shear deformations. To use BEAM188, use the following steps:

C PREPROCESSOR
C ELEMENT TYPE
C ADD
C ADD in the new window
C BEAM (2 NODE 188) [BEAM3 Element]
C OK
C OPTIONS
SET K3 TO QUADRATIC [required to represent linear variation of bending moment]
SET K12 TO CONSTANT [constant cross section]
C OK
C CLOSE
SECTIONS > BEAM > COMMON SECTIONS
SUBTYPE = RECTANGLE [or other shape]
B=2
H=2
CLOSE

Note that it is necessary to set KEYOPT(3) to quadratic. The ETABLE numbers are different for this element. The LINE ELEMENT RESULTS command is not available.

The classical 4-node quadrilateral element analyzed in Section 5.1 and Section 5.2.1 is modeled in ANSYS by the PLANE42 element. The extra shape functions of Wilson-

Taylor are included by default. Use KEYOPT(2) = 1 to suppress the extra shape functions. Plane stress is the default. Use KEYOPT(3)=2 for plane strain.

Alternative steps that may be used in both version 13 and version 12 are as follows.

For applications using the LINK1 element, instead of

C PREPROCESSOR > ELEMENT TYPE > ADD > ADD
C LINK (2D SPAR 1)
C OK
C CLOSE
C REAL CONSTANTS > ADD > ADD
C OK
enter area A
C OK
C CLOSE

You can use

C PREPROCESSOR then type the command
ET,*n*,LINK1 followed by the ENTER key [where *n* is the type number]
R,*n*,*A* [where *n* is the set number and *A* is the area]

For applications using the BEAM3 element, instead of

C PREPROCESSOR
C ELEMENT TYPE
C ADD
C ADD in the new window
C BEAM (Select 2D Elastic 3)
C OK
C OPTIONS
SET K6 TO INCLUDE OUTPUT [of data on member forces and moments]
C OK
C CLOSE
C PREPROCESSOR > REAL CONSTANTS > ADD
C ADD
C OK
Enter *Area*, *IZZ*, *height*
C OK

You can use

C PREPROCESSOR [to enter the preprocessor]
ET,1,BEAM3 [ET,type, name]
KEYOPT,1,6,1 [K6 = 1 to save element data]

R,1,*A*,*Izz*,*h* [R, nset, A, I, h]

For applications using the Plane42 element, instead of

C PREPROCESSOR > ELEMENT TYPE > ADD
C ADD in the new window
C SOLID
QUAD 4node 42
C OK
C OPTIONS
C EXCLUDE on the K2 drop down menu [to not use special d.o.f]
C OK
C CLOSE

You can use

C PREPROCESSOR [to enter the preprocessor]
ET,1,PLANE42 [ET,type, name]
KEYOPT,1,2,1 [K2 = 1 to exclude extra DOF]

Plate and shell elements 43, 63, 93, and 143 have be eliminated from Version 13. You must use the 4 node 181 element or the 8 node 281 element. Since these elements serve multiple purposes, the thickness is not input as a Real Constant (R command). You must use the "Sections > Shell > Lay-up > Add" GUI path to input the thickness, or the commands "SECTYPE,,Shell" and "SECDATA,thickness".

Following are changes by section including typos and changes in version 13.

15.3

Page 356**,** second line after **f**, change CLOSE to C FILE > CLOSE.
Page 356**,** fourth line after **g**, change CLOSE to FILE > CLOSE.

15.6

Page 363, second line above **e**, change LINES to NODES.

15.9

Page 371, fifth line, change STRUCTURAL to STRUCTURAL > DISPLACEMENT.

15.10

Page 372, line 11 after **b**, change SURF to STRUCTURAL.

15.11

Page 377, line 4, change SURF to STRUCTURAL.
Page 377, line 6 after **d**, change NDIV to SIZE.
Page 378, line 9 after **e**, change STRUCTURAL to STRUCTURAL > DISPLACEMENT.

15.14

Page 390, third line before **I**, the expected result should read UY = 0.007157.

15.16

Page 395, first line after **b**, change LINK1 to LINK.
Page 396, line 5 after **d**, add "and UZ = 0".

15.21

Page 405, line 3 after **b**, change HYPERELASTIC to SOLID.

15.22

Page 409, line 3 after **b**, change HYPERELASTIC to SOLID.
Page 409, line 4 after **b**, change 3D to BRICK.

15.24

Page 416, part **e**, change 100 steps to 50 steps if convergence fails.

15.26

Page 421, replace the instructions for part **b** by

PREPROCESSOR > ELEMENT TYPE > ADD
ADD
SHELL
3D 4 node 181
OK
CLOSE
MATERIAL PROP > MATERIAL MODELS
STRUCTURAL
LINEAR
ELASTIC
ISOTROPIC
enter 10.92 for EX parameter $[E = 12(1 - \nu^2)]$
enter 0.3 for PRXY parameter
OK

MATERIAL > EXIT
SECTIONS > SHELL > LAY-UP > ADD
enter 1 for Thickness
OK

Page 422, before **f**, insert

SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL >
DISPLACEMENT > ON NODES
C on lower left corner
APPLY
C UX and UY
C on box for VALUES
T 0 for the value [to prevent rigid body motion in plane]
APPLY
C on lower right corner
OK
C UY
C on box for VALUES
T 0 for the value [to prevent rigid body motion in plane]
OK

15.27

Page 423, replace the instructions in **b** by

PREPROCESSOR > ELEMENT TYPE > ADD
ADD
SHELL
8 node 281
OK
CLOSE
MATERIAL PROP > MATERIAL MODELS
STRUCTURAL
LINEAR
ELASTIC
ISOTROPIC
enter 1092 for EX parameter [D = 100]
enter 0.3 for PRXY parameter
OK
MATERIAL > EXIT
SECTIONS > SHELL > LAY-UP > ADD
enter 1 for Thickness
OK

15.28

Page 425, replace the instructions in **b** by

PREPROCESSOR > ELEMENT TYPE > ADD
ADD
SHELL
4 node 181
OK
CLOSE
PREPROCESSOR >MATERIAL PROPS > MATERIAL MODELS
STRUCTURAL
LINEAR
ELASTIC
ISOTROPIC
EX = 3e6
NUXY = 0.
OK
DENSITY
DENS = .208
OK
MATERIAL > EXIT
SECTIONS > SHELL > LAY-UP > ADD
enter 3 for Thickness
OK

Page 428, second line before **g**, should read "J toward K",
Page 428, in **h**, insert OK after line 2. The expected result is UY = 0.546.

15.29

Page 430, replace the instructions in **b** by

PREPROCESSOR > ELEMENT TYPE > ADD
ADD in the new window
SHELL
Select 8 NODE 281
OK
CLOSE
PREPROCESSOR > MATERIAL PROPS > MATERIAL MODELS
STRUCTURAL
LINEAR
ELASTIC
ISOTROPIC
enter 30e6 for EX parameter
C in PRXY box
enter 0.3 for PRXY parameter

OK
MATERIAL > EXIT
SECTIONS > SHELL > LAY-UP > ADD
enter 1 for Thickness
OK

15.32

Page 441, insert OK after ALPX = 6.67e-6.

15.33

Page 443, change LINK1 to LINK180 in three places.
Page 444, change LINK1 to LINK180.
Page 445, delete OK in line 6.

15.34

Page 448, replace

WORK PLANE > CHANGE ACTIVE CS TO > GLOBAL
CYLINDRICALMODELING > CREATE > KEYPOINTS > IN
ACTIVE CS

by

WORK PLANE > CHANGE ACTIVE CS TO > GLOBAL CYLINDRICAL
MODELING > CREATE > KEYPOINTS > IN ACTIVE CS

Replace
WORK PLANE > CHANGE ACTIVE CS TO > GLOBAL CARTESIAN
MODELING > CREATE > AREAS > ARBITRARY > THROUGH
KPs

by

WORK PLANE > CHANGE ACTIVE CS TO > GLOBAL CARTESIAN
MODELING > CREATE > AREAS > ARBITRARY > THROUGH KPs

15.36

Page 457, change 2D SPAR 1 to 3D finit stn 180.
Page 459, in **e**, add

LOADS > DEFINE LOADS > APPLY
STRUCTURAL > DISPLACEMENT > ON NODES
PICK NODES ON THE ROD BY A BOX
OK
UZ
VALUE = 0
OK

15.37
Page 461, line 3 after **b**, change HYPERELASTIC to SOLID.

15.38

Page 465, line 3 after **b**, change HYPERELASTIC to SOLID.