AB AQUS Tutorial

by Shi-Yu Xu
Creating the Mesh

*NODE, NSET=WholeModel
**10, 0.0, 0.00, 0.0
20, 0.0, 0.00, 0.0
30, 0.0, 3.048, 0.0
** node#, X, Y, Z

*NGEN, NSET=BENT_COL
20, 30, 2
** 1st node#, 2nd node#, increment in the #

*NSET, NSET=Enforced_node
30

*ELEMENT, TYPE=B31
20, 20, 22
** ele #, 1st node, 2nd node

*ELGEN, ELSET=BENTCOL_M_phi
20, 5, 2, 2
** Master ele #, # of eles to be defined, Increment in node #, Increment in ele #
Data lines to define the node:

First line:
1. Node number.
2. First coordinate of the node.
3. Second coordinate of the node.
4. Third coordinate of the node.
5. First direction cosine of the normal at the node (optional).
6. Second direction cosine of the normal at the node (optional). For nodes entered in a cylindrical or spherical system, this entry is an angle given in degrees.
7. Third direction cosine of the normal at the node (optional). For nodes entered in a spherical system, this entry is an angle given in degrees.

The normal will be used only for element types with rotational degrees of freedom. See Part VI, “Elements,” of the Abaqus Analysis User's Manual.

Repeat this data line as often as necessary.

Figure 14.8–1 Coordinate systems.
Defining the Nonlinear Beam-Column Element

Set SECTION=GENERAL (default) for linear beam;
Set SECTION=NONLINEAR GENERAL for NL beam.

*BEAM GENERAL SECTION, DENSITY=2405.0, SECTION=NONLINEAR GENERAL, ELSET=BENTCOL_M_phi

1.1678, 0.1085, 0.0, 0.1085, 0.2169
0.0, 0.0, -1.0

*AIAL, LINEAR
2.899E10,

*M1
0.0000E+00, 0.0000E-00
2.7937E+05, 8.2008E-05
5.5857E+05, 1.6406E-04

......

*M2

......

*TORQUE, LINEAR
2.244E9,

*TRANSVERSE SHEAR STIFFNESS
1.075E10, 1.075E10

A, I_{11}, I_{12}, I_{22}, J
Direction cosines of t_1 axis

Axial stiffness

M

Torsion stiffness

Torque, linear

Shear stiffness K_{23}, K_{13}
Spring and Dashpot Elements

*ELEMENT, TYPE=SPRING2, ELSET=S2
201, 10, 20

*SPRING, ELSET=S2
2, 2
1.E14,

*ELEMENT, TYPE=DASHPOT2, ELSET=C1
300, 10, 20
301, 110, 120
302, 210, 220
303, 310, 320

*DASHPOT, ELSET=C1
1, 1
1.5589E7,
**User-Defined Element (UEL)**

*USER ELEMENT*, NODES=2, TYPE=U1, PROPERTIES=3, I PROPERTIES=1, COORDINATES=3, VARIABLES=14

1, 2, 3

*ELEMENT*, TYPE=U1, ELSET=NLSpring

102, 1, 2

*UEL PROPERTY*, ELSET=NLSpring

1.84E+9, 1.84E+8, 1.79676E+7, 1

**NODES=2**: number of nodes connected to the UEL

**TYPE=U1**: label of the UEL

**PROPERTIES=3**: number of floating point number parameters required by the UEL

**I PROPERTIES=1**: number of integer number parameters required by the UEL

**COORDINATES=3**: maximum number of active DOFs at each node

**VARIABLES=14**: number of solution-dependent state variables used by the UEL
**DAMPING**, ALPHA=0.6641, BETA=0.001884

**ELEMENT**, TYPE=MASS, ELSET=M1

801, 30

**MASS**, ELSET=M1

389571.

**AMPLITUDE**, NAME=linear_pushover, DEFINITION=TABULAR

0.00, 0.00, 10.00, 1.00,

*AMPLITUDE, NAME=accel_ctrl, INPUT=ElCentro033x_ABAQUS.txt

*BASELINE CORRECTION*
Data Output

*NSET, NSET=ColTOP
30,
*NSET, NSET=ColBOTTOM
20,
*NSET, NSET=FNDNbase
20,
*NSET, NSET=Nopt1
ColTOP, ColBOTTOM, FNDNbase
*ELSET, ELSET=ELopt1
20,
*OUTPUT, FIELD, FREQ=1
*NODE FILE, FREQUENCY=1, NSET =Nopt1
U
*NODE FILE, FREQUENCY=1, NSET =FNDNbase
RF
*OUTPUT, HISTORY, FREQUENCY=1
*NODE OUTPUT, NSET=Nopt1
U1, U2, U3
*NODE OUTPUT, NSET=FNDNbase
RF1, RF2, RF3, RM1, RM2, RM3
*ELEMENT OUTPUT, ELSET=ELopt1
SF1, SF2, SF3, SM1, SM2, SM3

Output to .fil file
- Must compose Fortran subroutine to read.

Output to .obd file
- Can be post-processed in CAE.
Applying Gravity Load

*NSET, NSET=ColTOP
30,
*NSET, NSET=FNDNbase
20,

*STEP
STEP 0 - GRAVITY LOAD
*STATIC
1.0E-10, 10.0,
*BOUNDARY
COLTOP, 3, 5
FNDNbase, 1, 6
*DLOAD
, GRAV, 9.81, 0, -1,0
*END STEP

**Applying gravity load in 10 secs.
Initial time increment, duration of step

Fix DOFs 3 ~ 5 at node set “COLTOP”
Fix DOFs 1 ~ 6 at node set “FNDNbase”
Apply gravity load in -Y direction
**Displacement-Controlled Pushover**

*NSET, NSET=Enforced_node

30

*STEP, INC=8000

Maximum # of increments to run.

STEP 1 - Static Pushover

*STATIC, DIRECT

Constant time increment

0.05, 10.0,

Initial time increment, duration of step

** 0.05, 10.0, 1.E-15, 0.05

Same, same, min increment, max increment

*BOUNDARY, OP=NEW

COLTOP, 3, 5

Fix DOFs 3 ~ 5 at node set “COLTOP”

FNDNbase, 1, 6

Fix DOFs 1 ~ 6 at node set “FNDNbase”

*BOUNDARY, OP=NEW, TYPE=DISPLACEMENT, AMPLITUDE=linear_pushover

Enforced_node, 1, 1, 0.4

Apply displacement in DOF 1 with SF=0.4

*END STEP
Time-History Analysis

*NSET, NSET=ColTOP
30,
*NSET, NSET=ColBOTTOM
20,
*NSET, NSET=FNDNbase
10,

*STEP, INC=10000
Maximum # of increments to run.
STEP 1-1 - EARTHQUAKE EXCITATION with gravity load
*DYNAMIC, DIRECT
Constant time increment
0.004, 25.0,
Initial time increment, duration of step
*BOUNDARY, OP=NEW
COL, 3, 5
Fix DOFs 3 ~ 5 at node set “COL”
ColTOP, 3, 5
Fix DOFs 3 ~ 5 at node set “COLTOP”
FNDNbase, 2, 6
Fix DOFs 2 ~ 6 at node set “FNDNbase”
*BOUNDARY, OP=NEW, TYPE=ACCELERATION, AMPLITUDE=accel_ctrl
FNDNbase, 1, 1, 9.81
Apply acceleration in DOF 1
*END STEP
Running the Analysis

- Change to ABAQUS working directory first.
- Under the command prompt, type:

```
D:\ABAQUS\Work>abaqus analysis job=Abaqus_Column_UserEle_Dynamic user=UELspring
```

- The string after the “job” parameter is the *script filename* (w/o extension).
- The string after the “user” parameter is the user-defined subroutine (optional).
Variables Need to Be Defined

\[ M\ddot{u}(x) + C\dot{u}(x) + ku(x) - f_{ext} = RHS \]

**RHS**
- Contribution of the UEL to the Right-Hand-Side of the equation of motion.
- Size of the RHS array is NRHS.

**AMATRIX**
- Contribution of the UEL to the Jacobian of the system.

**SVARS**
- User-defined solution-dependent state variables associated with the UEL.
- Size of the SVARS array is NSVARS.

**ENERGY**
- Energy quantities associated with the UEL.
  - ENERGY(1): Kinetic energy.
  - ENERGY(2): Elastic strain energy.
  - ENERGY(5): Viscous dissipation.
  - ENERGY(8): Incremental work done by loads applied within the user element.
Passed-in Variables (1)

**NDOFEL**
Number of DOFs associated with the UEL.

**NNODE**
Number of nodes in the UEL.

**PROPS**
User-defined floating point number parameters. Size of the PROPS array is NPROPS.

**JPROPS**
User-defined integer parameters. Size of the JPROPS array is NJPROPS.

**COORDS**
Coordinates of the nodes of the UEL (undeformed configuration).

**U, DU, V, A**
Current estimates of the solution variables at the end of the increment.
U: Total displacement.
DU: Incremental displacement.
V: Velocities.
A: Accelerations.
Passed-in Variables (2)

**JTYPE**
- User-assigned integer number for a specific type of UEL.
  (i.e., *ELEMENT, TYPE=U1)

**TIME(1)**
- Current step time.

**TIME(2)**
- Current total time.

**DTIME**
- Time increment.

**KSTEP**
- Current step number.

**KINC**
- Current increment number.

**JELEM**
- User-assigned element number.
Passed-in Variables (3)

**PARAMS**

Constants used in the Hilber-Hughes-Taylor time integration scheme (i.e., The $\alpha$-Method).

PARAMS(1) = $\alpha$ ; PARAMS(2) = $\beta$ ; PARAMS(3) = $\gamma$

**LFLAGS**

An array defining the tasks that the UEL should do when being called.

LFLAGS(1): Defines the procedure type.

1 = Static, automatic incrementation
2 = Static, direct incrementation
11 = Implicit dynamic, half-step residual tolerance given
12 = Implicit dynamic, fixed time increments
41 = Eigenvalue frequency extraction

LFLAGS(3) = 1: Normal implicit time incrementation procedure. Define the RHS and AMATRX arrays.
LFLAGS(3) = 2: Define the current stiffness matrix only.
LFLAGS(3) = 3: Define the current damping matrix only.
LFLAGS(3) = 4: Define the current mass matrix only.
LFLAGS(3) = 5: Define the current residual vector only.
LFLAGS(3) = 6: Define the current mass matrix and residual vector.
LFLAGS(3) = 100: Define perturbation quantities for output.
LFLAGS(4) = 0: General step.
LFLAGS(4) = 1: Linear perturbation step.
Compiling the UEL

- Use “make” command:

```
D:\ABAQUS\Work>make library=uelbilinear
Begin Compiling ABAQUS/Standard User Subroutines
1/20/2011 2:20:24 AM
uelbilinear.for
End Compiling ABAQUS/Standard User Subroutines
1/20/2011 2:20:25 AM
Begin Linking ABAQUS/Standard User Subroutines
1/20/2011 2:20:25 AM
  Creating library standardU.lib and object standardU.exp
End Linking ABAQUS/Standard User Subroutines
1/20/2011 2:20:25 AM

D:\ABAQUS\Work>analysis job=xxxx user=uelbilinear-std
```
Example

*USER ELEMENT, NODES=2, TYPE=U1, PROPERTIES=4, COORDINATES=3, VARIABLES=12
1, 2, 3

*ELEMENT, TYPE=U1
101, 101, 102

*ELGEN, ELSET=UTRUSS
101, 5

*UEL PROPERTY, ELSET=UTRUSS
0.002, 2.1E11, 0.3, 7200.
Question?