ABAQUS 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

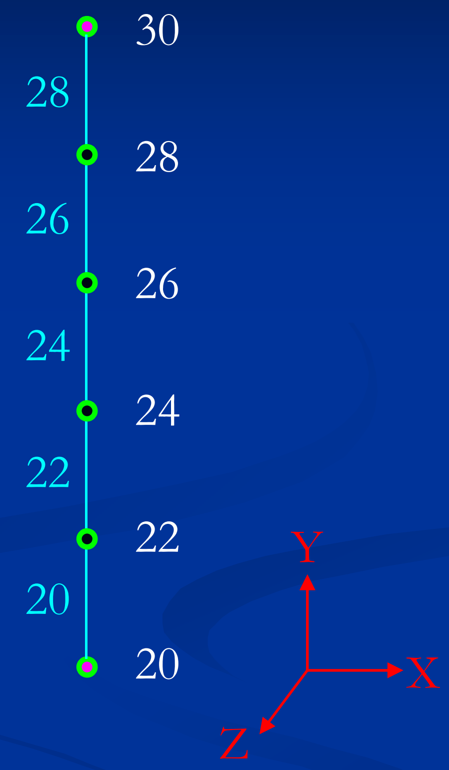
20, 20, 22

** 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:

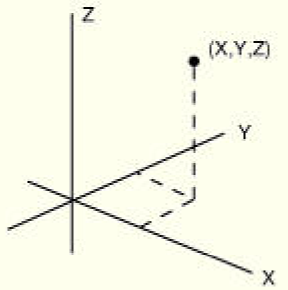
First line:

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.



Rectangular Cartesian
(default)

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 — $A, I_{11}, I_{12}, I_{22}, J$

0.0, 0.0, -1.0 — Direction cosines of t_1 axis

*AXIAL, LINEAR

2.899E10, — Axial stiffness

*M1

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

*M2

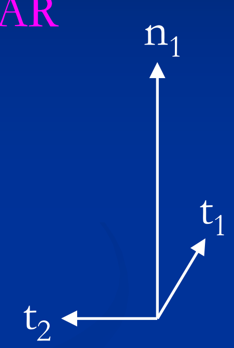
……

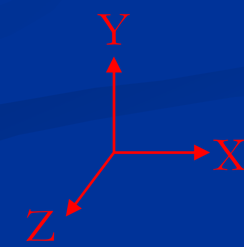
*TORQUE, LINEAR

2.244E9, — Torsion stiffness

*TRANSVERSE SHEAR STIFFNESS

1.075E10, 1.075E10 — Shear stiffness K_{23}, K_{13}

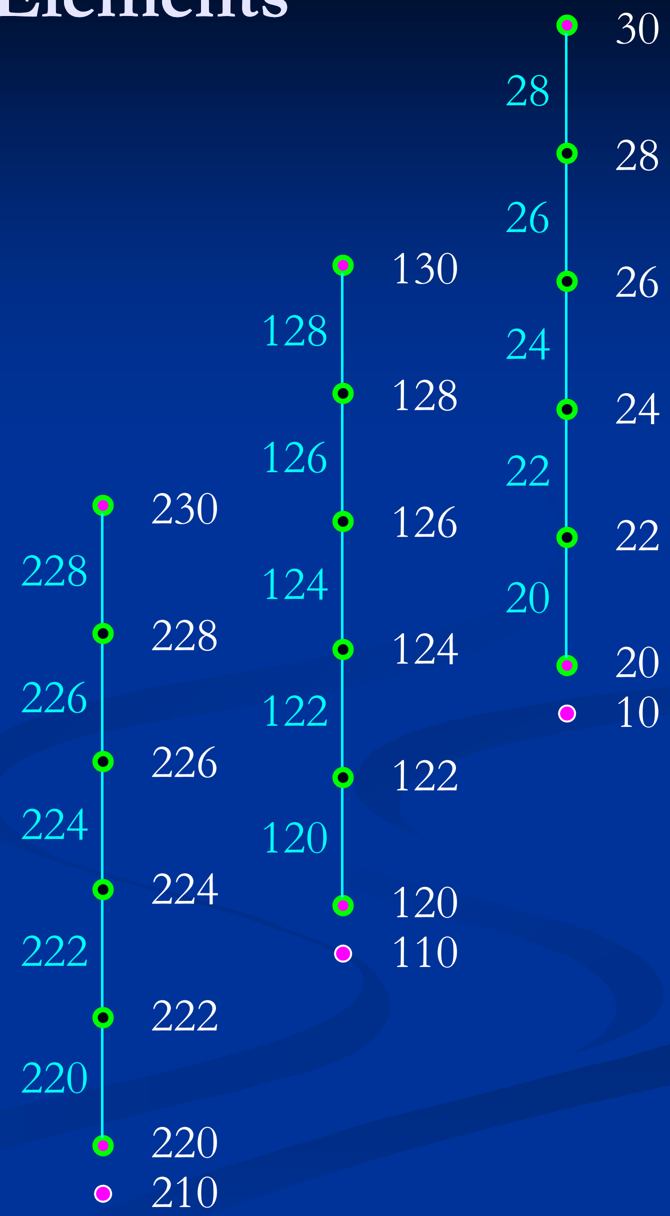




Spring and Dashpot Elements

```
*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

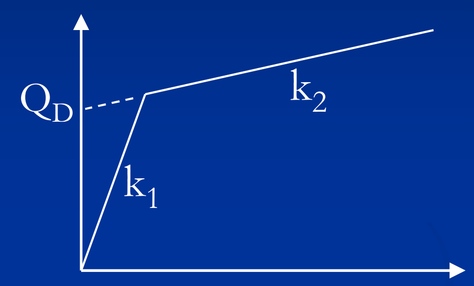
102, 1, 2

*UEL PROPERTY, ELSET=NLSPRING

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

PROPERTIES

I PROPERTIES



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

Miscellaneous

*DAMPING, ALPHA=0.6641, BETA=0.001884

*ELEMENT, TYPE=MASS, ELSET=M1

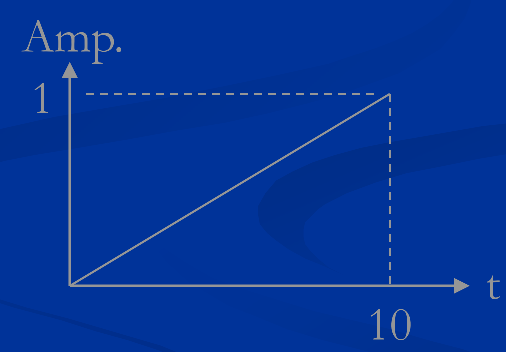
801, 30

*MASS, ELSET=M1

389571.

*AMPLITUDE, NAME=linear_pushover, DEFINITION=TABULAR

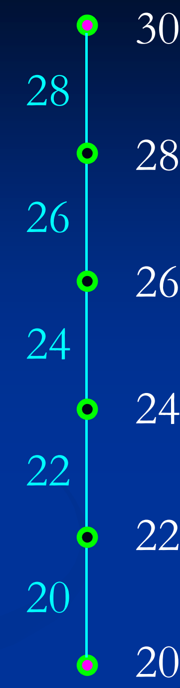
0.00, 0.00, 10.00, 1.00,



*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
```

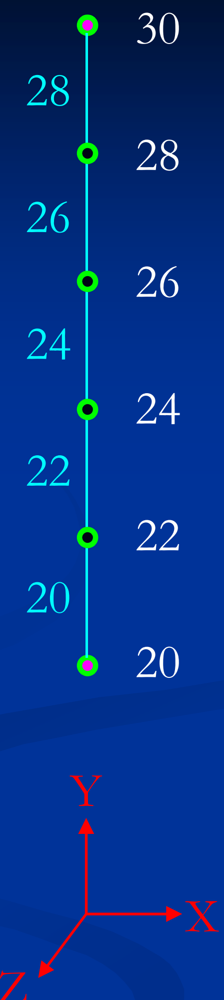
Output to .fil file
➔Must compose Fortran subroutine to read.

Output to .obd file
➔Can be post-processed in CAE.

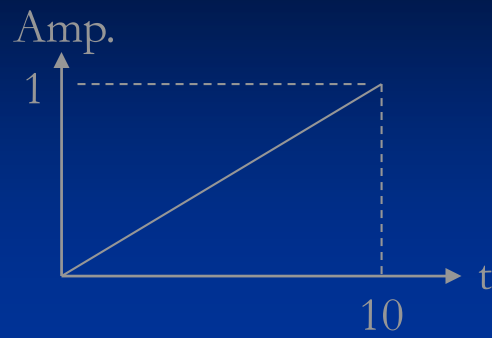
Applying Gravity Load

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

*STEP
 STEP 0 - GRAVITY LOAD
*STATIC                  **Applying gravity load in 10 secs.
 1.0E-10, 10.0,          Initial time increment, duration of step
*BOUNDARY
COLTOP, 3, 5             Fix DOFs 3 ~ 5 at node set "COLTOP"
FNDNbase  , 1, 6         Fix DOFs 1 ~ 6 at node set "FNDNbase"
*DLOAD
, GRAV, 9.81, 0, -1,0    Apply gravity load in -Y direction
*END STEP
```



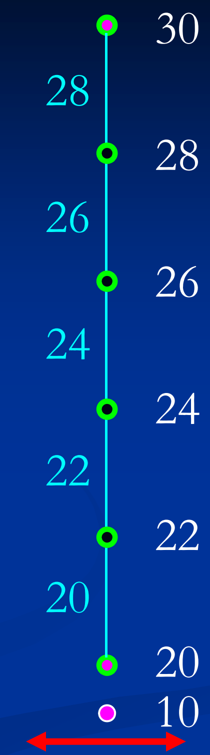
Displacement-Controlled Pushover




```
*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

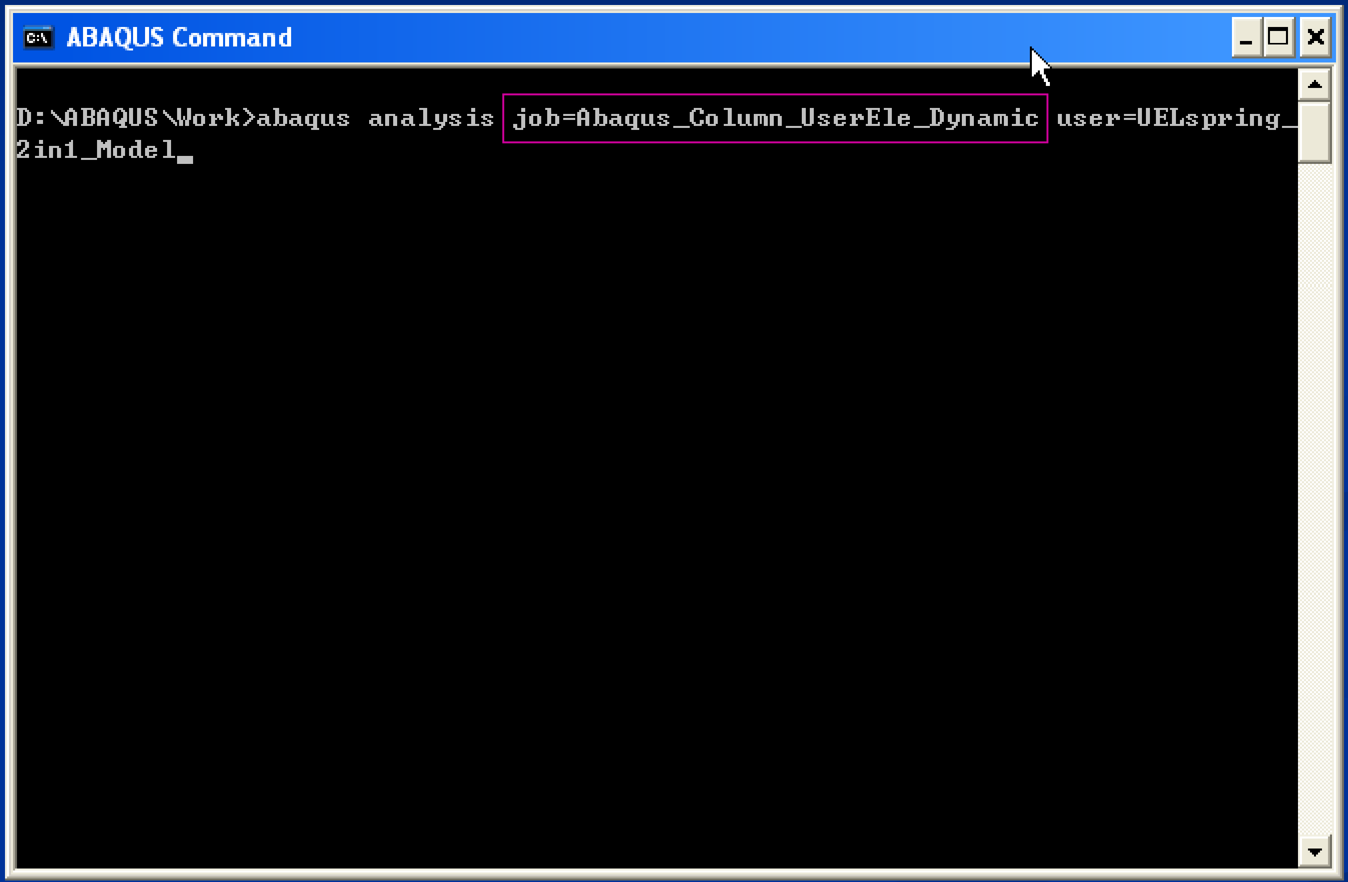


*NSET, NSET=ColTOP
30,
*NSET, NSET=ColBOTTOM
20,
*NSET, NSET=FNDNbase
10,

*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:



- 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.

AMATRX

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(4): Plastic dissipation.

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 α-Method).

PARAMS(1)=α ; PARAMS(2)=β ; PARAMS(3)=γ

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:

ABAQUS Command

```
D:\ABAQUS\Work>abaqus 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>abaqus 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

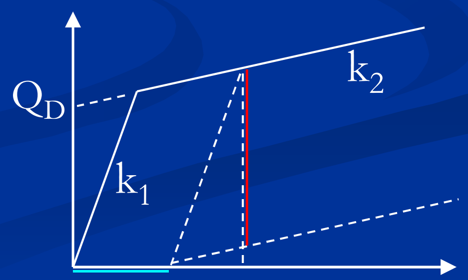
101, 101, 102

*ELGEN, ELSET=UTRUSS

101, 5

*UEL PROPERTY, ELSET=UTRUSS

0.002, 2.1E11, 0.3, 7200.



Question?