Finite Element Analysis

Using ABAQUS

EGM 6352 (Spring 2017)

Instructor: Nam-Ho Kim (nkim@ufl.edu)

Web: http://www.mae.ufl.edu/nkim/egm6352

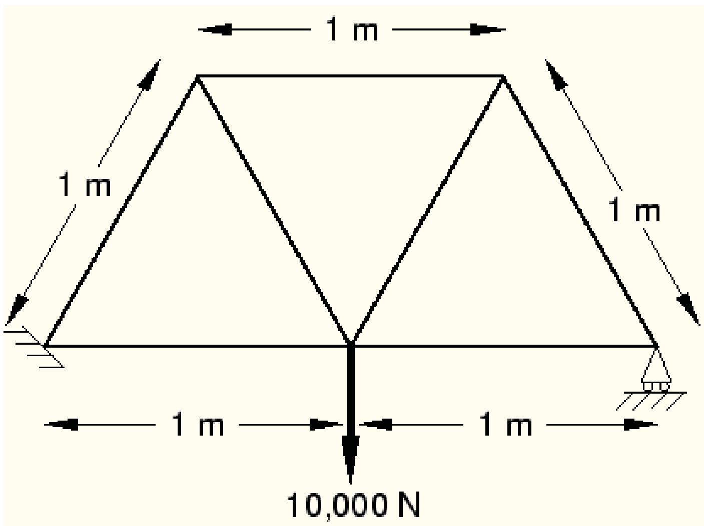
Methods of Analysis in ABAQUS

- Interactive mode
  - Create analysis model and procedure using GUI
  - Advantage: No need to remember commands
  - Disadvantage: No automatic procedure for changing model or parameters
- Python script
  - All GUI user actions will be saved as Python script
  - Advantage: User can repeat the same command procedure
  - Disadvantage: Need to learn Python language
- Analysis input file
  - At the end, ABAQUS generates analysis input file (text file)
  - ABAQUS solver reads analysis input file
  - It is possible to manually create analysis input file

Components in ABAQUS Model

- Creating nodes and elements (discretized geometry)
- Element section properties (area, moment of inertia, etc)
- Material data (linear/nonlinear, elastic/plastic, isotropic/orthotropic, etc)
- Loads and boundary conditions (nodal force, pressure, gravity, fixed displacement, joint, relation, etc)
- Analysis type (linear/nonlinear, static/dynamic, etc)
- Output requests

Example: Overhead Hoist



All members are circular steel rods, 5 mm in diameter.

Material properties

General properties:

$\rho = 7800 \text{ kg/m}^3$

Elastic properties:

$E = 200 \times 10^9$ Pa

$\nu = 0.3$

Input File: frame.inp

```
*HEADING
Two-dimensional overhead hoist frame
SI units (kg, m, s, N)
1-axis horizontal, 2-axis vertical
*PREPRINT, ECHO=YES, MODEL=YES, HISTORY=YES
**
** Model definition
**
*NODE, NSET=NALL
101, 0.,  0.,    0.
102, 1.,  0.,    0.
103, 2.,  0.,    0.
104, 0.5, 0.866, 0.
105, 1.5, 0.866, 0.
*ELEMENT, TYPE=T2D2, ELSET=FRAME
11, 101, 102
12, 102, 103
13, 101, 104
14, 102, 104
15, 102, 105
16, 103, 105
17, 104, 105
*SOLID SECTION, ELSET=FRAME, MATERIAL=STEEL
** diameter = 5mm --> area = 1.963E-5 m^2
1.963E-5,
*MATERIAL, NAME=STEEL
*ELASTIC
200.E9, 0.3
**
** History data
**
*STEP, PERTURBATION
10kN central load
*STATIC
*BOUNDARY
101, ENCASTRE
103, 2
*CLOAD
102, 2, -10.E3
*NODE PRINT
U,
RF,
*EL PRINT
S,
**********************************
** OUTPUT FOR ABAQUS QA PURPOSES
**********************************
*EL FILE
S,
*NODE FILE
U, RF
*END STEP
```


Format of Input File

- Model data section
  - Information required to define the structure being analyzed
- History data section
  - Type of simulation (static, dynamics, etc)
  - The sequence of loading or events for which the response of the structure is required
  - Divided into a sequence of steps
  - Output request
- Input file
  - Composed of a number of option blocks (describing a part of the model)
  - Each option block begins with a keyword line (starting with *), which is usually followed by one or more data lines.

Format of Input File cont.

- Keyword line
  - *ELEMENT, TYPE = T2D2, ELSET = FRAME
    - Element set FRAME is 2-dimensional truss element
  - *NODE, NSET=PART1
    - All nodes below belong to a set PART1
  - *ELEMENT, TYPE = T2D2,
    ELSET = FRAME
    - Maximum 256 characters per line
- Data line - Keyword line usually followed by data lines

```
*NODE
101, 0., 0., 0.
102, 1., 0., 0.
103, 2., 0., 0.
104, 0.5, 0.866, 0.
105, 1.5, 0.866, 0.
```


Format of Input File cont.

- Model data
- Heading
  - The first option in any Abaqus input file must be *HEADING
  - Description of the problem

```
*HEADING
Two-dimensional overhead hoist frame
SI units (kg, m, s, N)
1-axis horizontal, 2-axis vertical
```

- Data file printing options
  - Input file echo

```
*PREPRINT, ECHO=YES, MODEL=YES, HISTORY=YES
```

- Comments

```
**
** Model definition
**
```

Format of Input File cont.

- Element connectivity
  - Keyword *ELEMENT specifies element type, element set

```
*ELEMENT, TYPE=T2D2, ELSET=FRAME
11, 101, 102
12, 102, 103
13, 101, 104
14, 102, 104
15, 102, 105
16, 103, 105
17, 104, 105
```

- Section properties
  - Keyword *SOLID SECTION specifies area, I, etc

```
*SOLID SECTION, ELSET=FRAME, MATERIAL=STEEL
** diameter = 5mm --> area = 1.963E-5 m^2
1.963E-5,
```


Format of Input File cont.

- Material properties
  - Keyword *MATERIAL followed by various suboptions

```
*MATERIAL, NAME=STEEL
*ELASTIC
200.E9, 0.3
```

- History data
  - Starts with keyword *STEP, followed by the title of the step

```
*STEP, PERTURBATION
10kN central load
```

- Analysis procedure
  - Use `*STATIC` immediately after *STEP
- Boundary conditions
  - Keyword *BOUNDARY
  - (UX, UY, UZ, UR1, UR2, URS) = (1, 2, 3, 4, 5, 6)

Format of Input File cont.

- Boundary conditions cont.
  - Format: Node number, first dof, last dof, displ value

```
103, 2,2, 0.0
103, 2,2
103, 2
101, 1
101, 2
```

  - Built in constraints
    - ENCASTRE: Constraint on all displacements and rotations at a node
    - PINNED: Constraint on all translational degrees of freedom
    - XSYMM: Symmetry constraint about a plane of constant
    - YSYMM: Symmetry constraint about a plane of constant
    - ZSYMM: Symmetry constraint about a plane of constant
    - XASYMM: Antisymmetry constraint about a plane of constant
    - YASYMM: Antisymmetry constraint about a plane of constant
    - ZASYMM: Antisymmetry constraint about a plane of constant

Format of Input File cont.

- Applied loads
  - concentrated loads, pressure loads, distributed traction loads, distributed edge loads and moment on shells, nonzero boundary conditions, body loads, and temperature

```
*CLOAD
102, 2, -10.E3
```

- Output request
  - neutral binary file (.odb), printed text file (.dat), restart file (.res), binary result file (.fil)

```
*NODE PRINT
U,
RF,
*EL PRINT
S,
```

- End of step

```
*END STEP
```

Run ABAQUS

- Data check

  `abaqus job=frame datacheck interactive`

  - Show frame.dat file
  - Check for **ERROR ot **WARNING

- Solving the problem

  `abaqus job=frame continue interactive`

  - Show frame.dat file

Postprocessing

- Graphical postprocessing

  `abaqus viewer`

  - open frame.odb
  - Show labels using Options> Common> Labels
  - Plot> Deformed shape
  - Change deformation scale factor using Options> Common> Basic

NONLINEAR ANALYSIS USING ABAQUS

Nonlinear Analysis Using ABAQUS

- Geometric nonlinear (St. Venant-Kirchhoff material)

```
*STEP, NLGEOM=YES, INC=150
```

  - Large deformation on, maximum No. of increments = 150
- Time control

```
*STATIC
0.1, 1.0, 0.0001, 1.5
```

  - initial time increment, final time, min increment, max increment

Tutorial: Bending of Cantilevered Beam

- Create a part
  - Parts → Name: Beam → 2D Planar → Deformable → Shell → Approximate size = 20
  - Create lines: Rectangle → (0,0) → (10, 0.1478) → Done
  - Materials → Name:Material-1 → Mechanical → Elasticity → Elastic → Young's modulus = 1E8 → Poisson's ratio = 0 → OK
  - Sections → Name: Section-1 → Solid → Homogeneous → Continue → Material-1 → Plane stress/strain thickness = 0.1 → OK
  - Parts → Beam → Section Assignments → Select Beam → Done → Section: Section-1 → OK

Tutorial: Bending of Cantilevered Beam

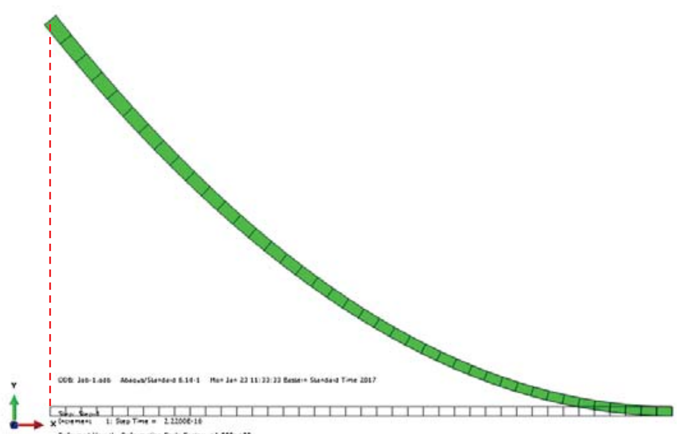
- Mesh control (Parts → Beam – Mesh (Empty)
  - Menu Mesh → Element Type → Select Beam → Done
  - Family: Plane Strain, Check Incompatible modes → OK
  - Menu Seed → Edge → Select top and bottom lines → Done → Method: By number → Number of elements = 40 → OK
  - Menu Mesh → Part → Yes
- Assembly and Steps
  - Assembly → Instance → Beam → OK
  - Steps → Name: Linear → Procedure type: Linear perturbation → Continue → OK
  - Constraints → Cancel → RP → (0, 0.0739)
  - Constraint → Coupling → Continue → Select RP-1 → Done → Surface → Select left line of the beam → Done → Coupling type: continuous distributing → OK

Tutorial: Bending of Cantilevered Beam

- Load and BC
  - BCs → Step: Initial → Displacement/Rotation → Select tie right edge of the Beam → Done → Click U1 & U2 → OK
  - Loads → Step: Step-1 → Moment → Continue → Select RP-1 → Done → CM3: -338.478 → OK
- Create job and solve the model
  - Jobs → Continue → OK
  - Right click on Job-1 → Submit → OK
- Postprocessing
  - Right click on Job-1 → Results
  - Common options → Deformation scale factor → Uniform → Value: 1 → OK
  - Plot deformed shape

Tutorial: Bending of Cantilevered Beam

- Deformed shape from linear perturbation analysis (Scale factor = 1.0)



Tutorial: Bending of Cantilevered Beam

- Nonlinear simulation
  - Go back to Model
  - Delete Step-1 (linear perturbation)
- Create a nonlinear step and submit the job
  - Steps → Procedure type: General → Static, General → Continue
  - Nlgeom: on
  - Incrementation: Max No of increments: 200 → Initial: 0.05 → Maximum: 1.0 → OK
  - Loads → Step: Step-1 → Moment → Continue → Select RP-1 → Done → CM3: -338.478 → OK
  - Submit job → OK

Tutorial: Bending of Cantilevered Beam

- Deformed shape from nonlinear static analysis (Scale factor = 1.0)