Finite Element Analysis

Using ABAQUS 2

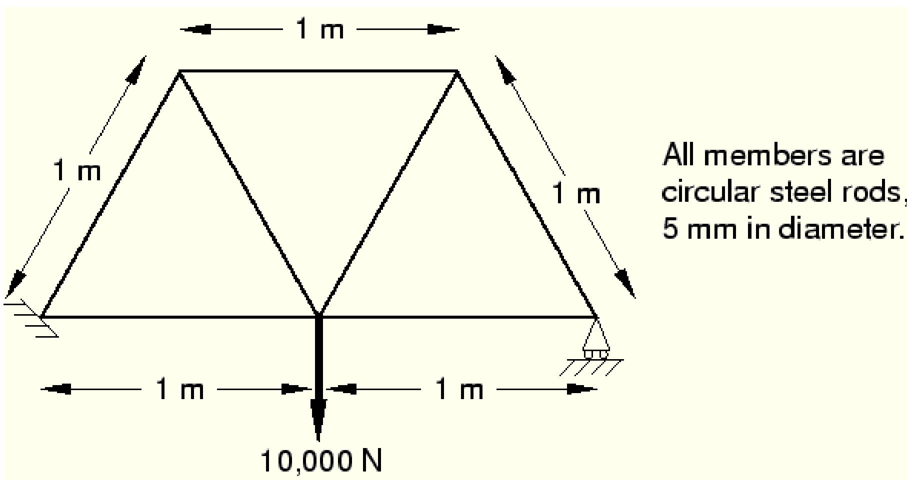
Methods of Analysis in ABAQUS

- 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



Material properties

General properties:

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

Elastic properties:

$E = 200 \times 10^9 \text{ Pa}$

$\nu = 0.3$

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

2D Solid (Continuum) Elements

- Plane strain
  - CPE3 3-node linear
  - CPE4 4-node bilinear
  - CPE6 6-node quadratic
  - CPE8 8-node biquadratic
- Plane stress
  - CPS3 3-node linear
  - CPS4 4-node bilinear
  - CPS6 6-node quadratic
  - CPS8 8-node biquadratic
- Distributed body forces (*DLOAD)
- Surface forces (*DSLOAD)