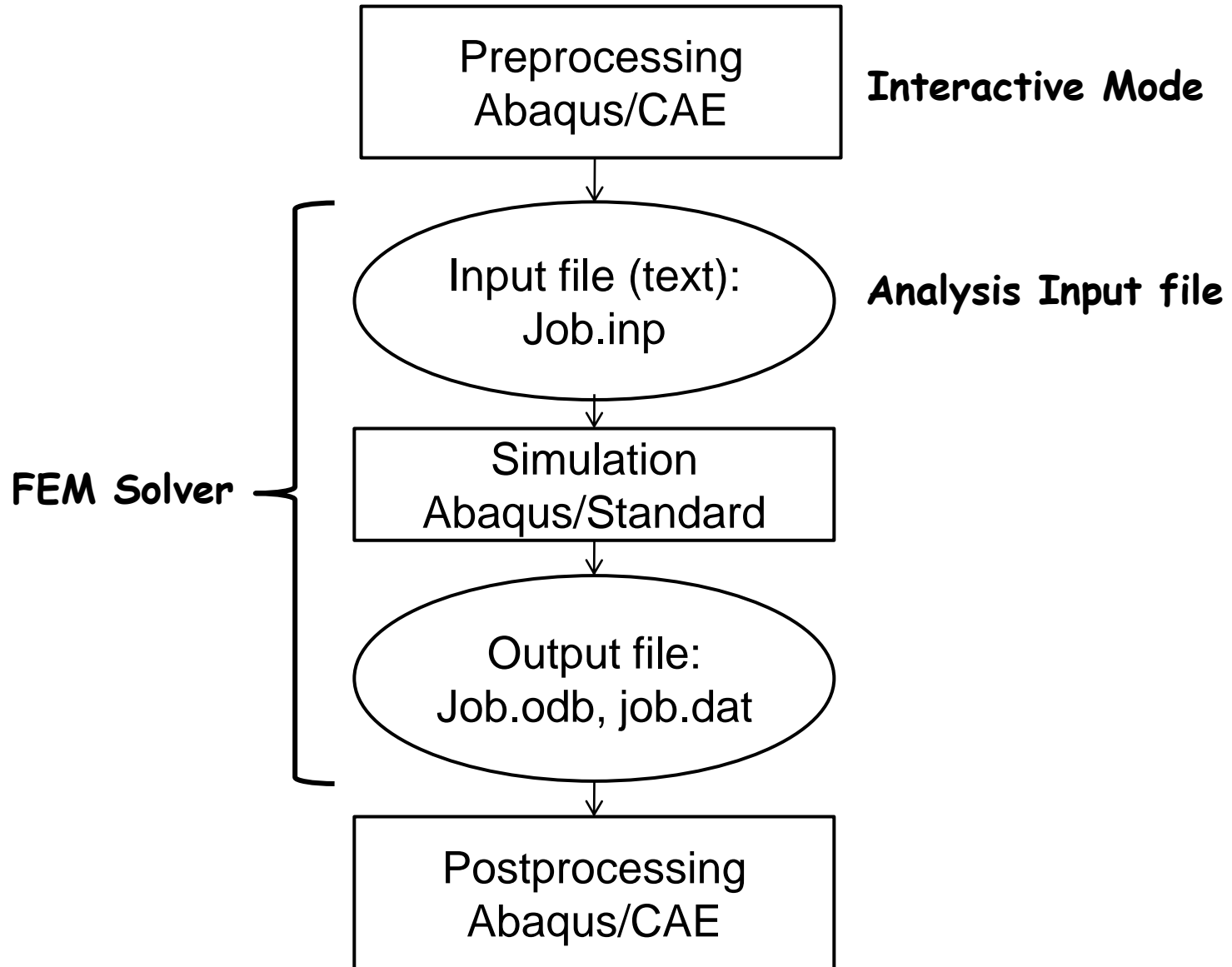


Tutorial 2:

Abaqus with Analysis Input File

Abaqus Basics



Why do I go with input files?

- Analysis with input files
 - ABAQUS solver reads the analysis input file
 - Advantage:
 - User can change model directly without GUI
 - FASTER than analysis using GUI
 - Useful for minor modification (GUI automatically create an input file)
 - Disadvantage:
 - No visual information (should use GUI to check model layout)
 - User has to discretize model

Input File: frame.inp

*HEADING

**

** Model definition

**

*NODE, NSET=NALL

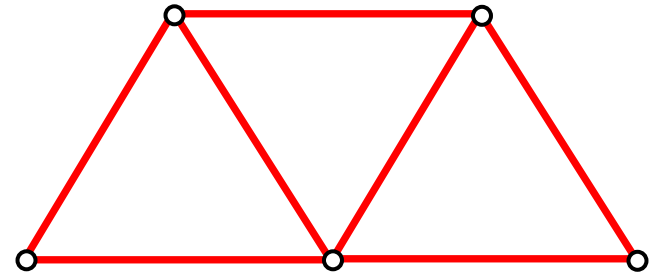
*ELEMENT, TYPE=T2D2, ELSET=FRAME

*SOLID SECTION, ELSET=FRAME, MATERIAL=STEEL

*MATERIAL, NAME=STEEL

*ELASTIC

200.E9, 0.3



— Truss element
└ Solid section
└ Elastic material

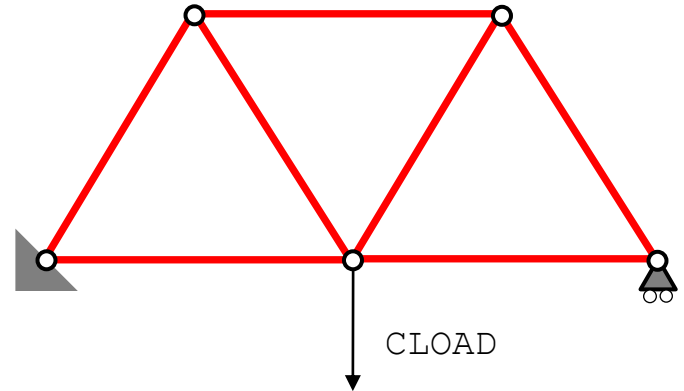
Input File: frame.inp

```
**
** History data
**
*STEP, PERTURBATION

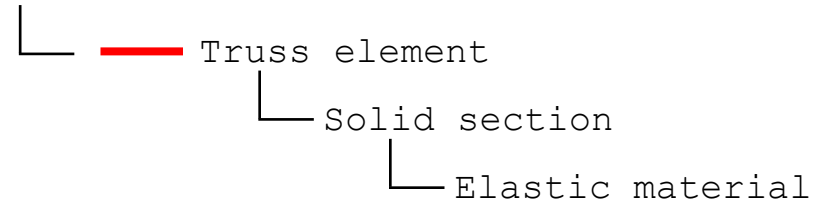
*STATIC
*BOUNDARY

*CLOAD
```

```
*****
** OUTPUT FOR ABAQUS QA PURPOSES
*****
```



PERTURBATION STATIC



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.
 - Description for the data lines (starting with **)

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

Format of Input File cont.

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.

104 ○

105 ○

101 ○

102 ○

103 ○

Format of Input File cont.

*ELEMENT

11, 101, 102

12, 102, 103

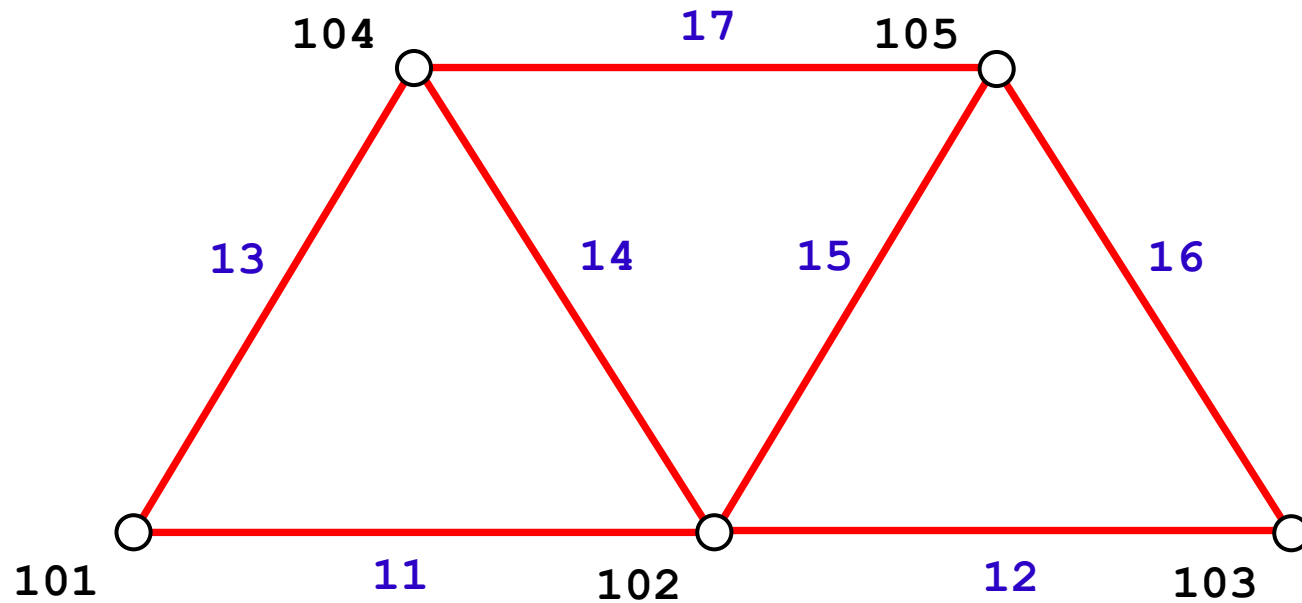
13, 101, 104

14, 102, 104

15, 102, 105

16, 103, 105

17, 104, 105



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)

```
*EL PRINT
```

```
S, E
```

```
*NODE PRINT
```

```
U,
```

```
RF,
```

- End of step

```
*END STEP
```

Modifying Input File

- Multiple Sections (**FRAME1** and **FRAME2**)

- Assign new section to element 6

```
*ELEMENT, TYPE=T2D2, ELSET=FRAME1
```

```
11, 101, 102
```

```
12, 102, 103
```

```
13, 101, 104
```

```
14, 102, 104
```

```
15, 102, 105
```

```
16, 103, 105
```

```
*ELEMENT, TYPE=T2D2, ELSET=FRAME2
```

```
17, 104, 105
```

```
*SOLID SECTION, ELSET=FRAME1, MATERIAL=STEEL
```

```
** diameter = 5mm --> area = 1.963E-5 m^2
```

```
1.963E-5,
```

```
*SOLID SECTION, ELSET=FRAME2, MATERIAL=STEEL
```

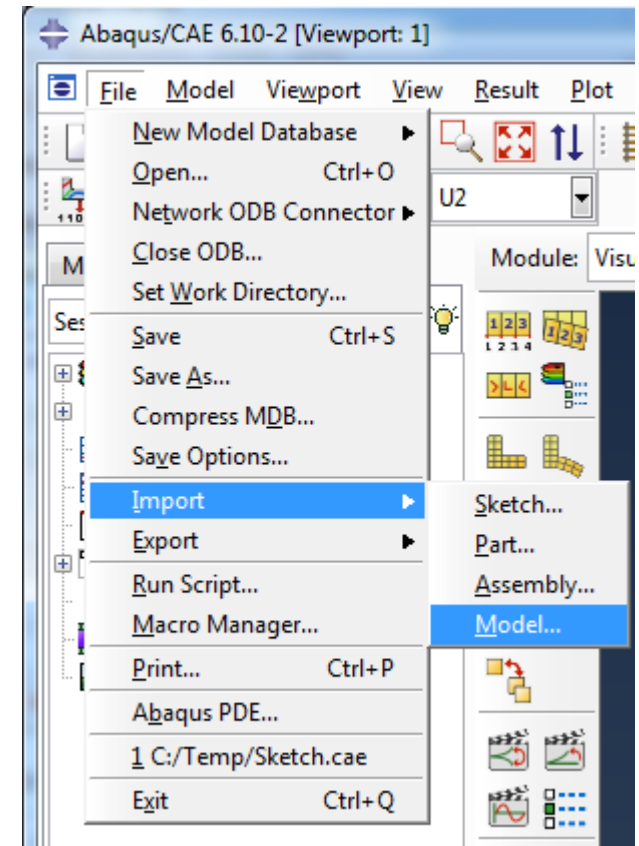
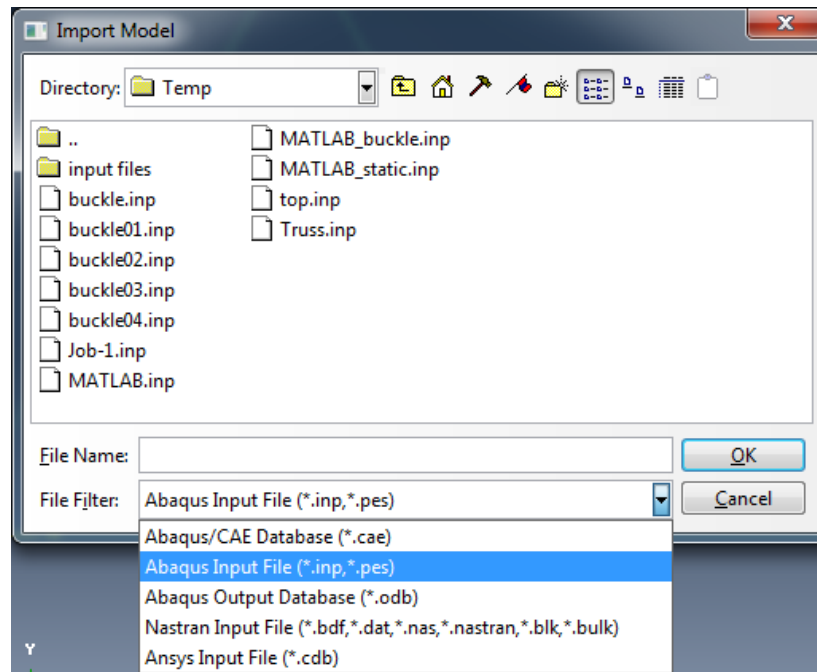
```
2.0E-5,
```


Modifying Input File (Made by ABAQUS)

- Input files made by GUI
 - Find the files in the work directory
(to check where the directory is: Files > Set Work Directory)
 - Automatically made by GUI when users submit a model
(ex: [Jobname].inp)
 - Edit the existing input file

Run ABAQUS

- Using Abaqus/CAE
 - Import the input model
 - Advantage: visually check FEM model
 - Disadvantage: A couple of commands do not work (ex: text out request commands)



Run ABAQUS

- Using Command Prompt
- Data check

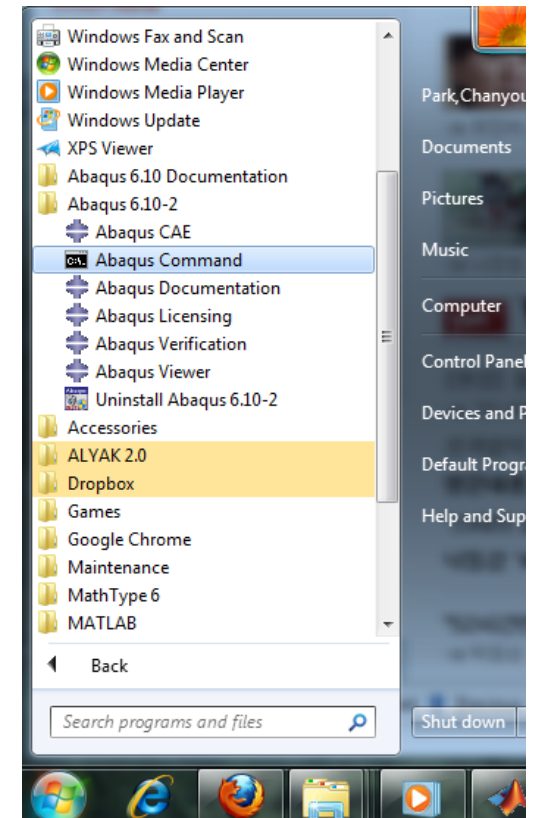
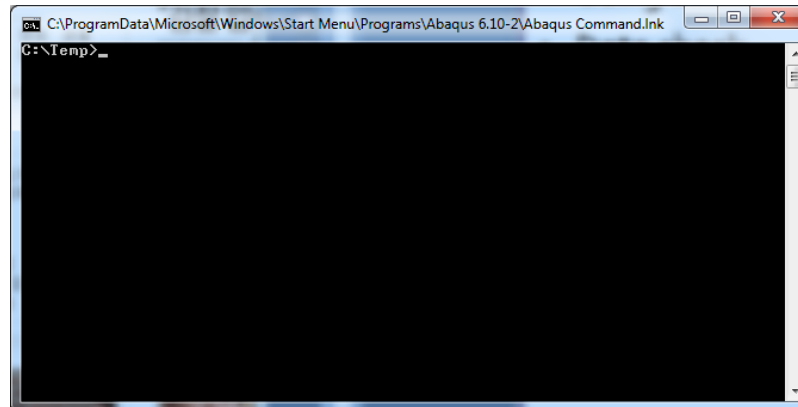
`abaqus job=frame datacheck interactive`

- Check for ****ERROR** or ****WARNING**

- Solving the problem

`abaqus job=frame continue interactive`

- Show frame.dat file



Run ABAQUS

- Basic commands in command prompt

`cd [directory name]` : change directory
to new directory

(ex: `cd test`)

`cd /` : change directory to root at
once

`dir` : see available files in current
directory

Batch Test

- Running many jobs
 - Useful to run many jobs at the same time
 - Create a batch file (ex: multirun.bat)

```
(  
abaqus job=frame-1 interactive  
abaqus job=frame-2 interactive  
abaqus job=frame-3 interactive  
abaqus job=frame-4 interactive  
)
```
- Making a batch file
 - Make an empty text file and write a list of files
 - Change the file name and extension
(ex: newname.txt -> multirun.bat)