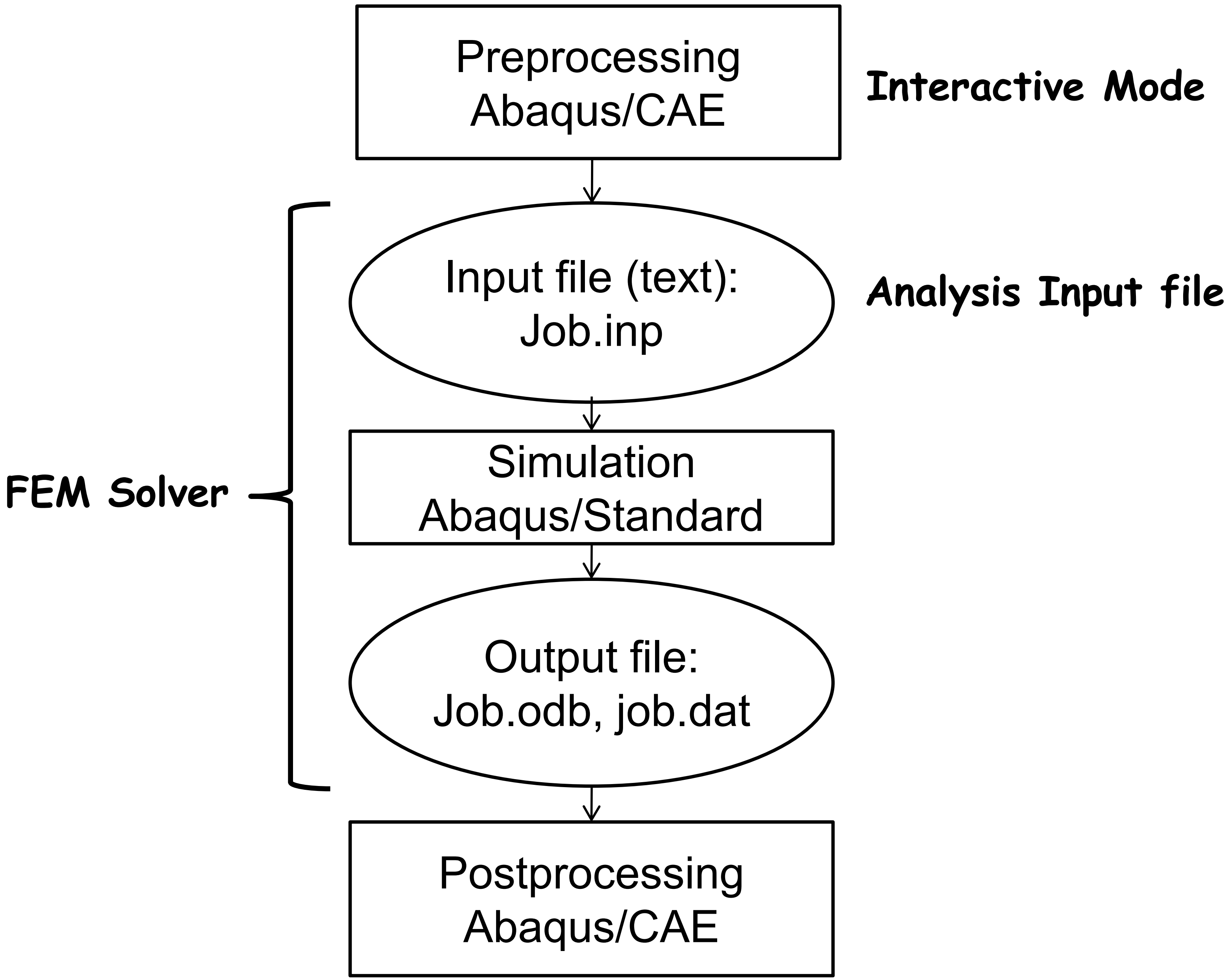
Tutorial 2:

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: Hoist.inp

```
*HEADING
** Job name: Hoist Model name: Model-1
*Preprint, echo=NO, model=NO, history=NO
**
** PARTS
*Part, name=Hoist
*Node
      1, -0.491506338,  0.792820334
      2,  0.508493662,  0.792820334
      3,   1.00849366, -0.0732050836
      4, 0.00849364884, -0.0732050836
      5, -0.991506338, -0.0732050836
*Element, type=T2D2
1, 1, 2
2, 3, 2
3, 4, 3
4, 2, 4
5, 4, 1
6, 1, 5
7, 5, 4
*Nset, nset=Set-1, generate
 1,  5,  1
*Elset, elset=Set-1, generate
 1,  7,  1
** Section: Circular_section
*Solid Section, elset=Set-1, material=Steel
1.96344e-05,
*End Part
```

```
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=Hoist-1, part=Hoist
*End Instance
**
*Nset, nset=Set-1, instance=Hoist-1
 5,
*Nset, nset=Set-2, instance=Hoist-1
 3,
*Nset, nset=Set-3, instance=Hoist-1
 4,
*End Assembly
**
** MATERIALS
**
*Material, name=Steel
*Elastic
 2e+11, 0.3
```

Input File: Hoist.inp cont.

```
**
** BOUNDARY CONDITIONS
**
** Name: Pin-joint
*Boundary
Set-1, 1, 1
Set-1, 2, 2
** Name: Sliding-joint
*Boundary
Set-2, 2, 2
** ----------------------------------------
**
** STEP: Vertical_force
**
*Step, name=Vertical_force, perturbation
*Static
**
** LOADS
**
** Name: Load-1   Type: Concentrated force
*Cload
Set-3, 2, -10000.
**
```

```
**
** OUTPUT REQUESTS
**
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
```

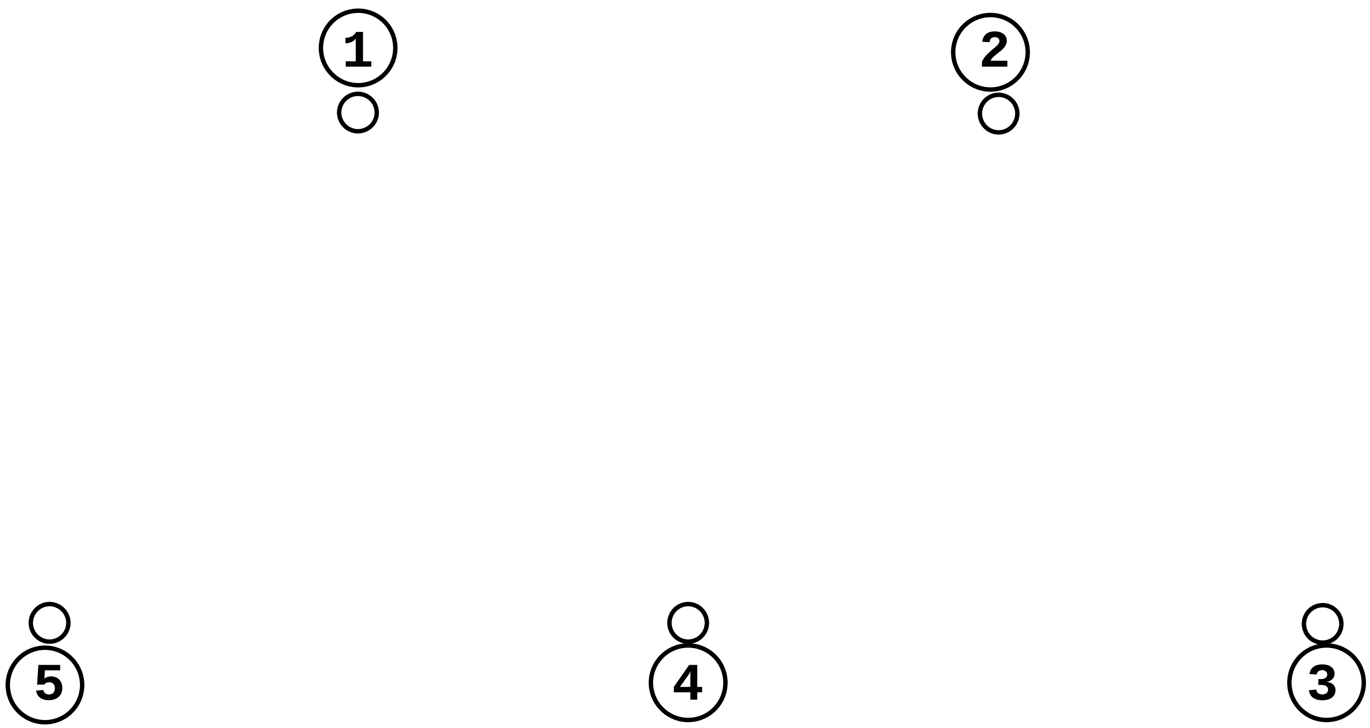
Format of Input File

- 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 **)
- 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.

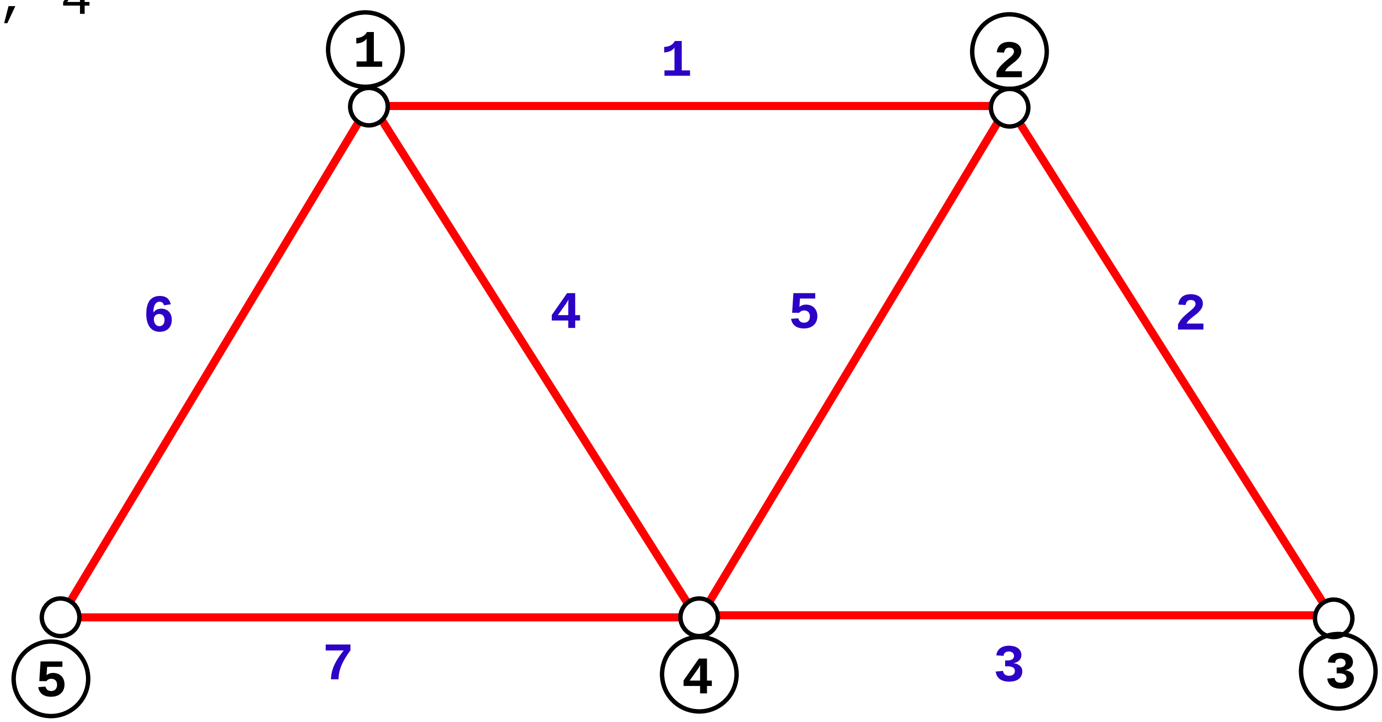
Data line - Keyword line usually followed by data lines

```
*NODE
 1, -0.491506338,  0.792820334
 2,  0.508493662,  0.792820334
 3,   1.00849366, -0.0732050836
 4, 0.00849364884, -0.0732050836
 5, -0.991506338, -0.0732050836
```



Format of Input File cont.

```
*ELEMENT
1, 1, 2
2, 3, 2
3, 4, 3
4, 2, 4
5, 4, 1
6, 1, 5
7, 5, 4
```



Format of Input File cont.

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

```
*HEADING
** Job name: Hoist Model name: Model-1
** Generated by: Abaqus/CAE 2019
```

- Data file printing options
  - Input file echo

```
*Preprint, echo=NO, model=NO, history=NO, contact=NO
```

- Comments

```
**
** PARTS
**
```

Format of Input File cont.

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

```
*Element, type=T2D2
1, 1, 2
2, 3, 2
3, 4, 3
4, 2, 4
5, 4, 1
6, 1, 5
7, 5, 4
```

- Node and element sets

```
*Elset, elset=Set-1, generate
 1,  7,  1
```

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

```
** Section: Circular_section
*Solid Section, elset=Set-1, material=Steel
1.96344e-05,
```

Format of Input File cont.

- Assembly
  - Keyword *Instance copies a part into assembly

```
*Assembly, name=Assembly
**
*Instance, name=Hoist-1, part=Hoist
*End Instance
```

- Node and element sets for boundary/load conditions

```
*Nset, nset=Set-1, instance=Hoist-1
 5,
*Nset, nset=Set-2, instance=Hoist-1
 3,
*Nset, nset=Set-3, instance=Hoist-1
 4,
*End Assembly
```

Format of Input File cont.

- Material properties
  - Keyword *MATERIAL followed by various suboptions

```
*Material, name=Steel
*Elastic
 2e+11, 0.3
```

- Boundary conditions
  - Keyword *BOUNDARY
  - (UX, UY, UZ, UR1, UR2, URS) = (1, 2, 3, 4, 5, 6)
  - Format: Node number, first dof, last dof, displ value

```
*Boundary
Set-1, 1, 1
Set-1, 2, 2
** Name: Sliding-joint Type: Displacement/Rotation
*Boundary
Set-2, 2, 2
```

Format of Input File cont.

- Creating Steps
  - Starts with keyword *STEP, followed by the title of the step
    Use `*STATIC` immediately after *STEP
    ```
    *Step, name=Vertical_force, nlgeom=NO, perturbation
    *Static
    ```
  - Define loads within the step
    ```
    *Cload
    Set-3, 2, -10000.
    ```
  - Field output request
    ```
    *Output, field, variable=PRESELECT
    *Output, history, variable=PRESELECT
    *End Step
    ```

Modifying Input File

- Multiple Sections (FRAME1 and FRAME2)
  - Assign new section to element 6

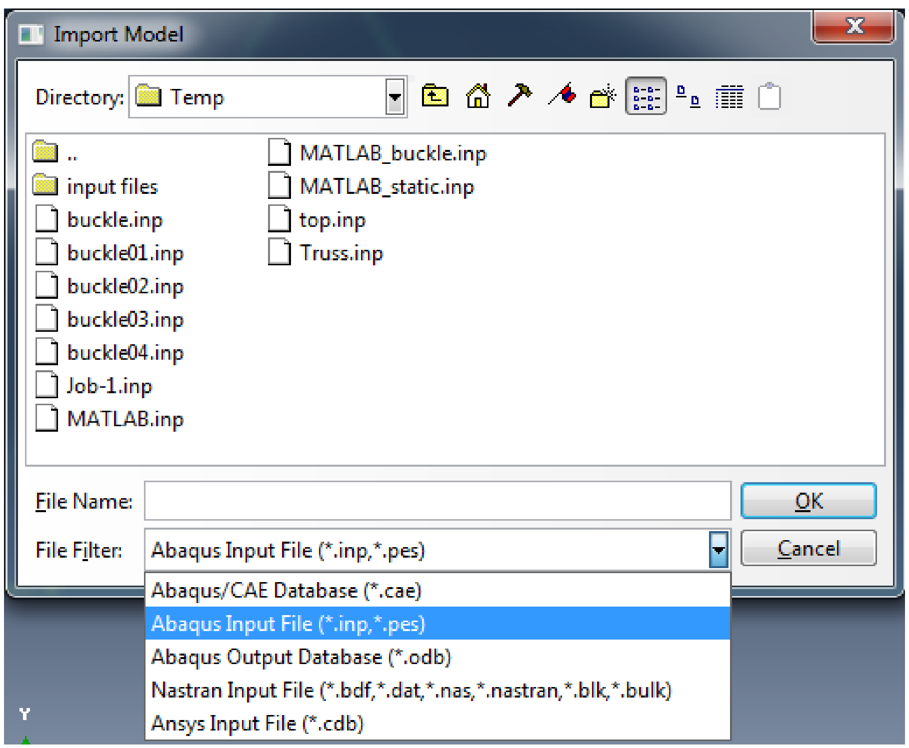
```
*ELEMENT, TYPE=T2D2, ELSET=FRAME1
11, 101, 102
12, 102, 103
13, 101, 104
14, 102, 104
15, 102, 105
16, 103, 105
*ELEMENT, TYPE=T2D2, ELSET=FRAME2
17, 104, 105
*SOLID SECTION, ELSET=FRAME1, MATERIAL=STEEL
** diameter = 5mm --> area = 1.963E-5 m^2
1.963E-5,
*SOLID SECTION, ELSET=FRAME2, MATERIAL=STEEL
2.0E-5,
```

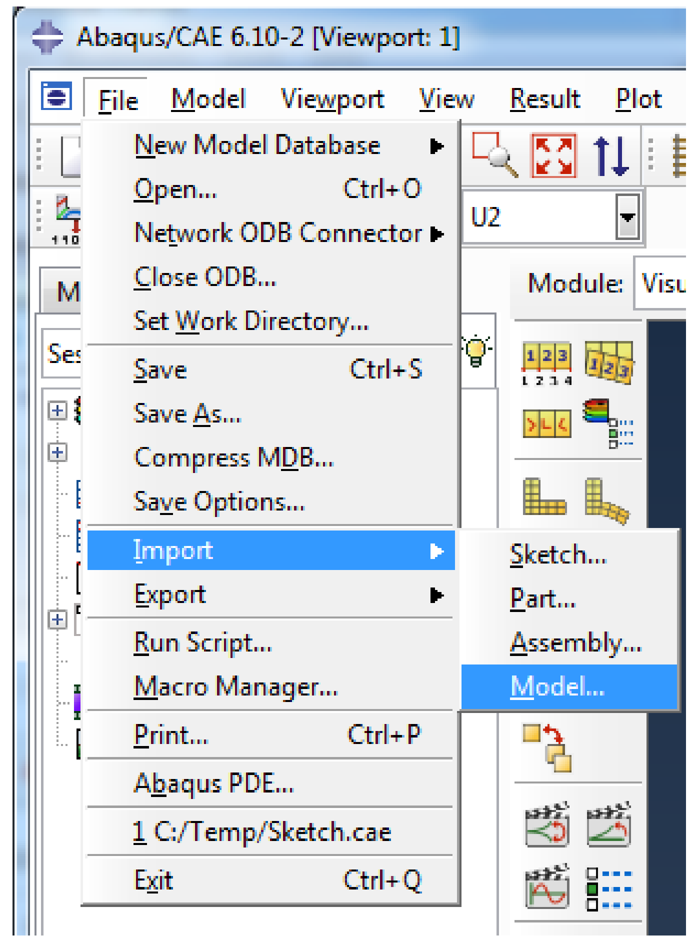
Modifying Input File (Made by ABAQUS)

- 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

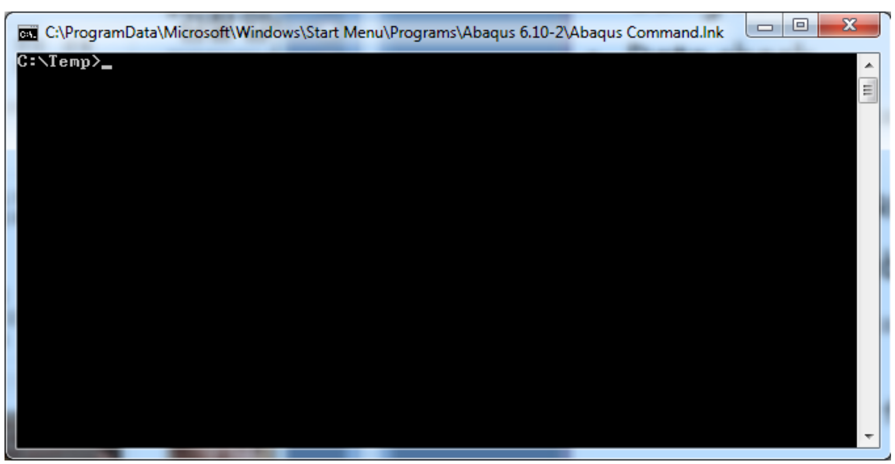
- Using Command Prompt
- Data check

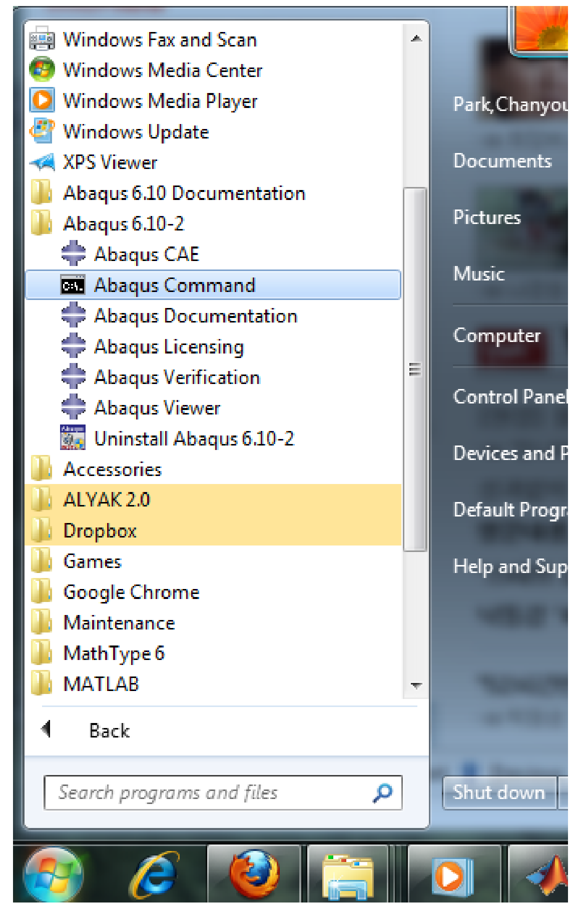
  abaqus job=frame datacheck interactive

  - 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)