

Using Commercial FEA Solvers in Reliability Analysis

Koorosh Gopal



CEPRO
Ohio Center of Excellence for
Product Reliability and Optimization

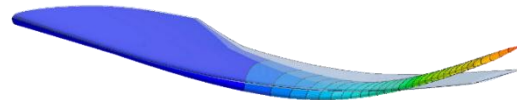
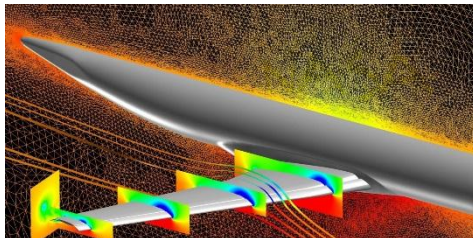
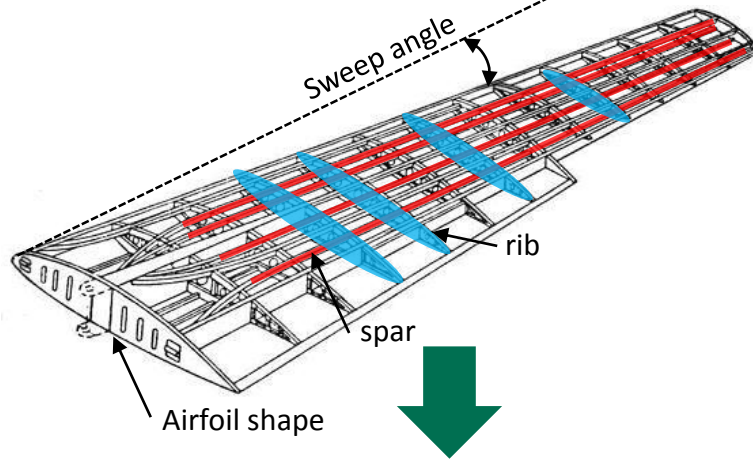
- **Why learn this?**
 - **Reliability analysis**
 - FORM, SORM, Monte Carlo
 - **Design optimization**
 - Gradient based, Evolutionary algorithms (EA)
 - **Design space exploration**
- **Background**
- **ABAQUS/ANSYS**
 - Run in Batch mode
 - Input file structure
 - Request output
 - Pre/post processing using MATLAB
- **Example problem**
 - Reliability analysis using Monte Carlo simulation



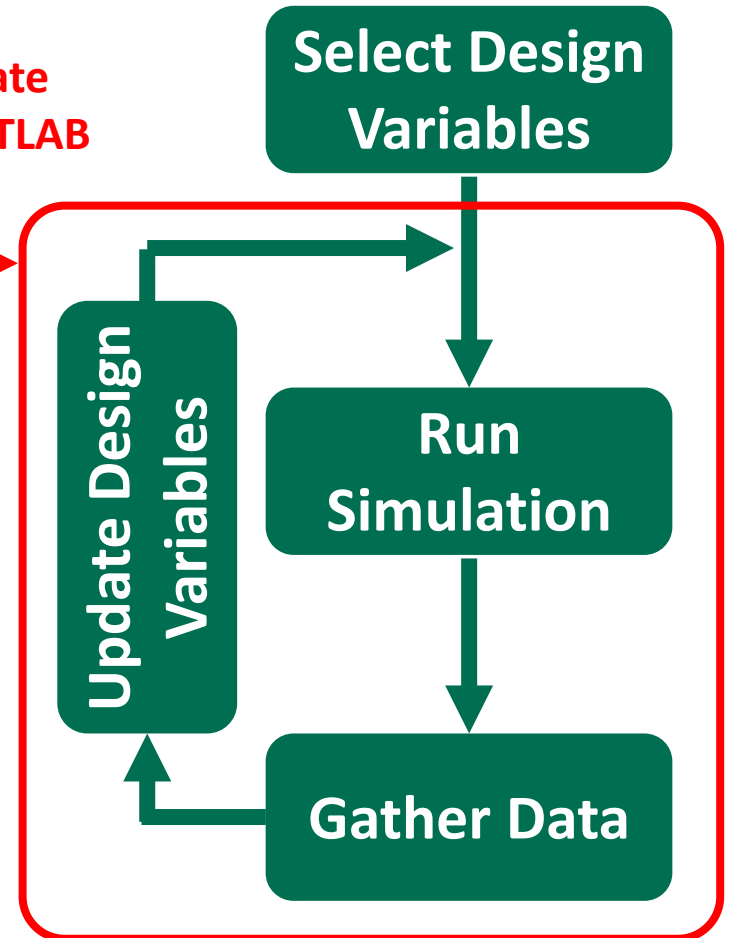
- **Design analysis using commercial packages often requires multiple simulations**
 - **Design variables**
 - Size: Cross-sectional area, length (does not affect computational mesh)
 - Shape: Curvature, slope, (affects computational mesh)
 - Probabilistic distributions: Mean and standard deviation
 - **Effect of changing design variables on the response of system**
 - Sensitivity analysis
 - First order reliability method (FORM)
 - Design space exploration (Kriging, Response surface)
 - Monte Carlo simulation

Design space exploration

- Design space exploration (DSE) is the activity of exploring design alternatives before implementation.



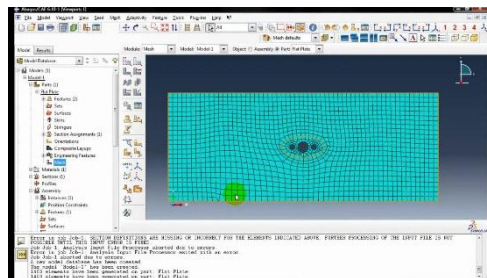
Automate
using MATLAB





- **Interactive mode**
 - Create analysis model and procedure using GUI (ABAQUS CAE)
 - Advantage: No need to remember commands
 - Disadvantage: No automatic procedure for modifying model
- **Analysis using batch mode**
 - ABAQUS generates analysis input file (`*.inp`) to run your simulation
 - ABAQUS reads this input file
 - It is possible to modify this easily

- **Abaqus/CAE is a software application for the pre and post processing the finite element analysis result**
 - Define geometry, mesh, physical properties, boundary conditions, solution type, output results
 - Write all the information to a text file
- **Abaqus Solver reads and solves the governing equations**
 - Export data (stress, displacement) to output file



Writes
Input File

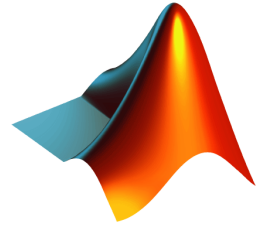
```
1 *Heading
2 ** Job name: truss_analysis Model name: Model-1
3 ** Generated by: Abaqus/CAE 6.13-1
4 *Preprint, echo=NO, model=NO, history=NO, contact=NO
5 **
6 ** PARTS
7 **
8 *Part, name=truss
9 *Node
10 1,720.,360.
11 2,720.,0.
12 3,360.,360.
13 4,360.,0.
14 5,0.,360.
15 6,0.,0.
16 *Element, type=T2D2
17 1,5,3
18 2,3,1
19 3,6,4
20 4,4,2
21 5,3,4
22 6,1,2
23 7,6,3
24 8,5,4
25 9,3,2
26 10,4,1
27 *Nset, nset=PickedSet12, internal, generate
28 1,6,1
29 *Elset, elset=PickedSet12, internal, generate
30 1,10,1
```

Input File

Abaqus Call

```
301
302
303 THE FOLLOWING TABLE IS PRINTED FOR ALL ELEMENTS WITH TYPE T2D2 AT THE INTEGRATION POINTS
304
305 ELEMENT FT FOOT- S11
306 NOTE
307
308 1 1 3.9073E+04
309 2 1 8025.
310 3 1 -4.0927E+04
311 4 1 -1.1975E+04
312 5 1 7098.
313 6 1 8025.
314 7 1 -2.6973E+04
315 8 1 2.9595E+04
316 9 1 1.6335E+04
317 10 1 -1.1349E+04
318
319 MAXIMUM 3.9073E+04
320 ELEMENT 1
321
322 MINIMUM -4.0927E+04
323 ELEMENT 3
324
325
326
327
328
329
330
331
332
333
334 THE FOLLOWING TABLE IS PRINTED FOR ALL NODES
335
336 NODE FOOT- U1 U2
337 NOTE
338
339 1 1.636 -7.590
340 2 -1.904 -7.679
341 3 1.407 -3.349
```

Output File



Define `input_params.mat`

Write `Input_file.inp` based on `input_params.mat`

CALL Abaqus with `Input_file.inp`

 **ABAQUS**

READ `Input_file.inp`

SOLVE the system of equations

WRITE `Output_file.dat`

Post process `Output_file.dat`

- **You to the directory where your input file is**
 - SHIFT+RIGHTCLICK in current directory then click on
open_command_window_here

- **Command to run ABAQUS**

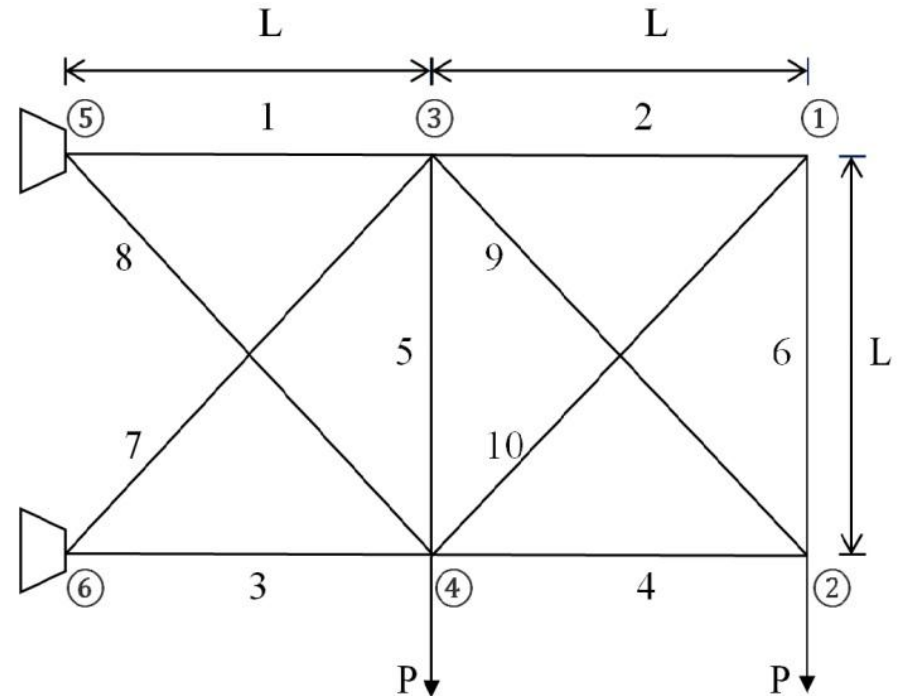
```
abaqus job=jobName input=inputFileName interactive
```

- **The interactive keyword should always be used**
 - Otherwise the job will be backgrounded and computer will think the job has finished before it really has.

Example problem

- **10 Bar Truss**

- $L = 360$ inches
- $P = 100$ kips
- $E = 10^4$ ksi
- Density = 0.1 lb/in^3
- Cross sectional area = 5.0 in^2
- Pinned at the node 5 and 6



- **We want to generate the input file and run it from command line**

Write input file

- **Easiest way to make your inputfile**

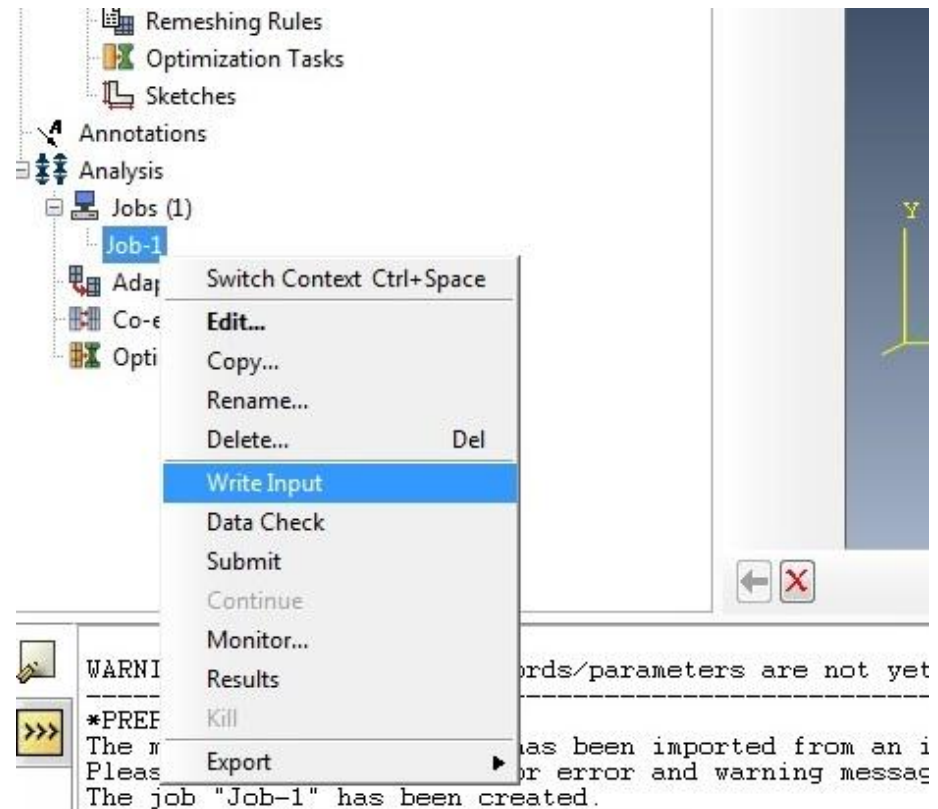
- **Open ABAQUS CAE**

- shift+rightclick in current directory then click on open_command_window_here
 - Type abaqus cae

- **Generate you model in ABAQUS CAE**

- **Rightclick on Analysis>>Job>>Jobname and choose write input**

- **The input file will be created in your working directory**



- **Heading**
 - Put the title and some other info (comments mostly)
- **Bulk data**
 - nodes, elements, parts definition and so on
- **Execution part**
 - STEPS definition, here is where you tell it what to do with your geometry.

- **Define node by using keyword** *Node
 - **On next line** <node number>, <x coordinate>, <y coordinate>
- **Define elements by using keyword** *Element
 - **Followed by** type=<element type>
 - **On next line define elements**
<element number>, <first node>, <second node>
- **Assign element properties using** *Solid section
 - **Followed by** <element set>, material = <material>
- **Define material properties** *Material, name=<material>
 - **On next lines define**
*Density
<value for density>,
*Elastic
<Young's modulus>, <Poisson ratio>

- Usually we are interested in displacements/stresses
 - Add the following lines at the end of input file

```
*NODE PRINT
```

```
U, ← Request displacement at all the nodes
```

```
*EL PRINT
```

```
S, ← Request stresses at all the elements
```

- The output is stored in `*.DAT` file in the working directory
- This file can be read inside a loop

SAMPLE INPUT/DATA FILE

- **To run ABAQUS from command line use**

```
command = ['abaqus job=<job name> ' ...  
          'input=<input file name>.inp ' ...  
          'interactive'];  
[status,~] = dos(command)
```

- **status is 0 for a successful run and 1 for failed attempt**

- **Good practice to check if your run was successful**

- Not enough licenses

- Error in input file

- **Easier debugging**

- **For big simulations writing the output file may take significant time**

- **Make sure the writing is done before postprocessing**

- Data is stored in *.DAT file
- Read nth line of *.DAT file using this command

```
fid=fopen('<file name>.dat');  
linenum = 308; % Or whichever line you wish to read  
fileLine = textscan(fid, '%s', 1, 'delimiter', '\n',  
                    'headerlines', linenum-1);  
fileLine = char(fileLine{1})  
number = str2double(fileLine(15:end))
```

- This data is used by the your algorithm
 - DOE
 - Constraint
 - Cost function

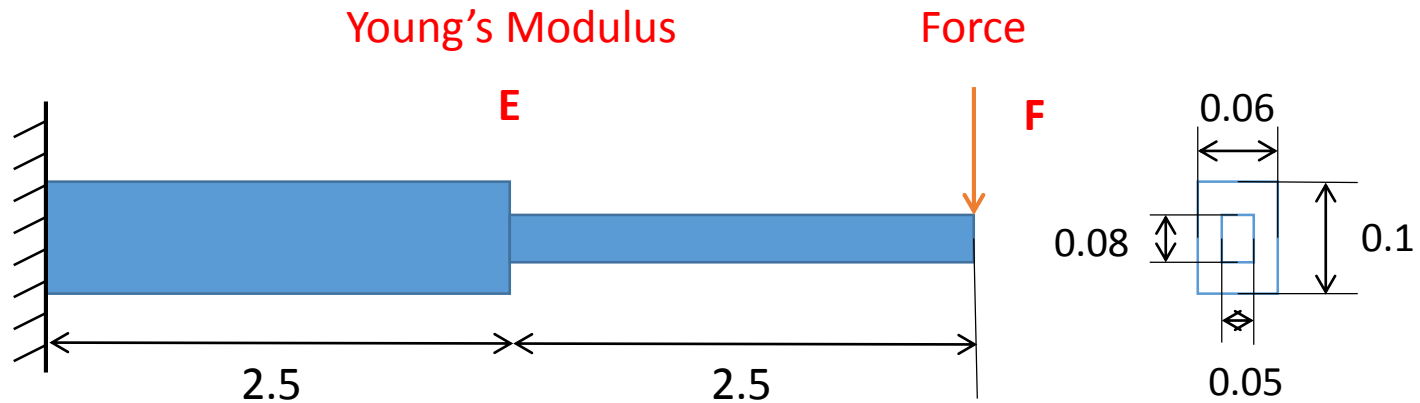
- **Define a template** <temp_input>.inp **file**
- **Read your template inputfile line by line**
 - **Modify the lines you need**
 - **You need to know the line number**

- **This code can be used**

```
fin = fopen('inp.txt','r');  
fout = fopen('out.txt','w');  
idk=0;  
while ~feof(fin)  
    idk=idk+1;  
    s = fgetl(fin);  
    if idk==250  
        s = <new value at line 250>;  
    end  
    fprintf(fout,'%s\n',s);  
end  
fclose(fin);  
fclose(fout);
```



ANSYS®



Tip Displacement d ?



```
function test_Simulation
X=[4000 2e11];%[F E]
response=Simulation(X);
end
```

- **Interactive Mode**

- exchange information with the computer continuously
- execute a command by selecting its menu path in the GUI

- **Batch Mode**

- submit a file of commands to the ANSYS program
- you can run a batch job in the background while doing other work on the computer

Interactive Mode

The image displays the ANSYS Academic Teaching Advanced Utility Menu (0101) interface. The main window shows a beam model with a coordinate system (X, Y) and a displacement plot. The plot shows a beam of length 1.0, with a displacement of 0.194669 at the right end. The beam is labeled with nodes 1, 2, and 3. The displacement is indicated by a red arrow pointing downwards.

The ANSYS Main Menu is visible on the left, with the following options:

- Preferences
- Preprocessor
- Solution
- General Postproc
- TimeHist Postpro
- Topological Opt
- ROM Tool
- DesignXplorer
- Design Opt
- Prob Design
- Radiation Opt
- Run-Time Stats
- Session Editor
- Finish

The PRNSOL Command window is open in the bottom left, showing the following output:

```
File
PRINT U  NODAL SOLUTION PER NODE

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP= 1  SUBSTEP= 1
TIME= 1.0000  LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

NODE  UY
1  0.0000
2  -0.52003E-01
3  -0.19467

MAXIMUM ABSOLUTE VALUES
NODE  3
VALUE -0.19467
```










ANSYS Input File (.log)

```

/BATCH
/COM,ANSYS RELEASE 12.1      UP20091102      14:27:21      02/12/2014
/input,menust,tmp,'',,,,,,,,,,,,,,1
/GRA,POWER
/GST,ON
/PLO,INFO,3
/GRO,CURL,ON
/CPLANE,1
/REPLOT,RESIZE
WPSTYLE,,,,,,,,,0
!*
/NOPR
/PMETH,OFF,0
KEYW,PR_SET,1
KEYW,PR_STRUC,1
KEYW,PR_THERM,0
KEYW,PR_FLUID,0
KEYW,PR_ELMAG,0
KEYW,MAGNOD,0
KEYW,MAGEDG,0
KEYW,MAGHFE,0
KEYW,MAGELC,0
KEYW,PR_MULTI,0
KEYW,PR_CFD,0
/GO
!*
/COM,
/COM,Preferences for GUI filtering have been set to display:
/COM, Structural
!*
/PREP7
!*
ET,1,BEAM3
!*
R,1,0.006,5e-6,0.1, , , ,
!*
R,2,0.004,2.133e-6,0.08, , , ,
!*
!*
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,2e11
MPDATA,PRXY,1,,
N,1,0,0,,,,
N,2,2.5,0,,,,
N,3,5,0,,,,
TYPE, 1
MAT, 1
REAL, 1
ESYS, 0
SECNUM,
TSHAP,LINE
  
```

```

!*
FLST,2,2,1
FITEM,2,1
FITEM,2,2
E,P51X
TYPE, 1
MAT, 1
REAL, 2
ESYS, 0
SECNUM,
TSHAP,LINE
!*
FLST,2,2,1
FITEM,2,2
FITEM,2,3
E,P51X
FLST,2,1,1,ORDE,1
FITEM,2,1
!*
/GO
D,P51X, , , , , ,ALL, , , , ,
FLST,2,1,1,ORDE,1
FITEM,2,3
!*
/GO
F,P51X,FY,-4000
FINISH
/SOL
/STATUS,SOLU
SOLVE
FINISH
  
```

 01.BCS
 01.emat
 01.err
 01.esav
 01.full
 01.log
 01.mntr
 01.rst
 01.stat

ANSYS Input Template File (.log)

```
/BATCH
/COM,ANSYS RELEASE 12.1    UP20091102    14:27:21    02/12/2014
/input,menust,tmp,'',,,,,,,,,,,,,,1
/GRA,POWER
/GST,ON
/PLO,INFO,3
/GRO,CURL,ON
/CPLANE,1
/REPLOT,RESIZE
WPSTYLE,,,,,,,,,0
!*
/NOPR
/PMETH,OFF,0
KEYW,PR_SET,1
KEYW,PR_STRUC,1
KEYW,PR_THERM,0
KEYW,PR_FLUID,0
KEYW,PR_ELMAG,0
KEYW,MAGNOD,0
KEYW,MAGEDG,0
KEYW,MAGHFE,0
KEYW,MAGELC,0
KEYW,PR_MULTI,0
KEYW,PR_CFD,0
/GO
!*
/COM,
/COM,Preferences for GUI filtering have been set to display:
/COM, Structural
!*
/PREP7
!*
ET,1,BEAM3
!*
R,1,0.006,5e-6,0.1, , , ,
!*
R,2,0.004,2.133e-6,0.08, , , ,
!*
!*
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,2e11
MPDATA,PRXY,1,,
N,1,0,0,,,,
N,2,2.5,0,,,,
N,3,5,0,,,,
TYPE, 1
MAT, 1
REAL, 1
ESYS, 0
SECNUM,
TSHAP,LINE
```

Young's
Modulus

```
!*
FLST,2,2,1
FITEM,2,1
FITEM,2,2
E,P51X
TYPE, 1
MAT, 1
REAL, 2
ESYS, 0
SECNUM,
TSHAP,LINE
!*
FLST,2,2,1
FITEM,2,2
FITEM,2,3
E,P51X
FLST,2,1,1,ORDE,1
FITEM,2,1
!*
/GO
D,P51X, , , , ,ALL, , , , ,
FLST,2,1,1,ORDE,1
FITEM,2,3
!*
/GO
F,P51X,FY,-4000 |
FINISH
/SOL
/STATUS,SOLU
SOLVE
FINISH
/POST1
!*
/output,output_ansys,out
prnsol,u,y
/out
/EXIT,ALL
```

Force

Save ansys output
as a file called
output_ansys.out

input_ansys.log
input_ansys_template.log
output_ansys.out
Simulation.m
test_Simulation.m

Update Input for ANSYS

```
function response=Simulation(DesignVar)
%Update input for ANSYS
fin = fopen('input_ansys_template.log','rt');
fout = fopen('input_ansys.log','wt');

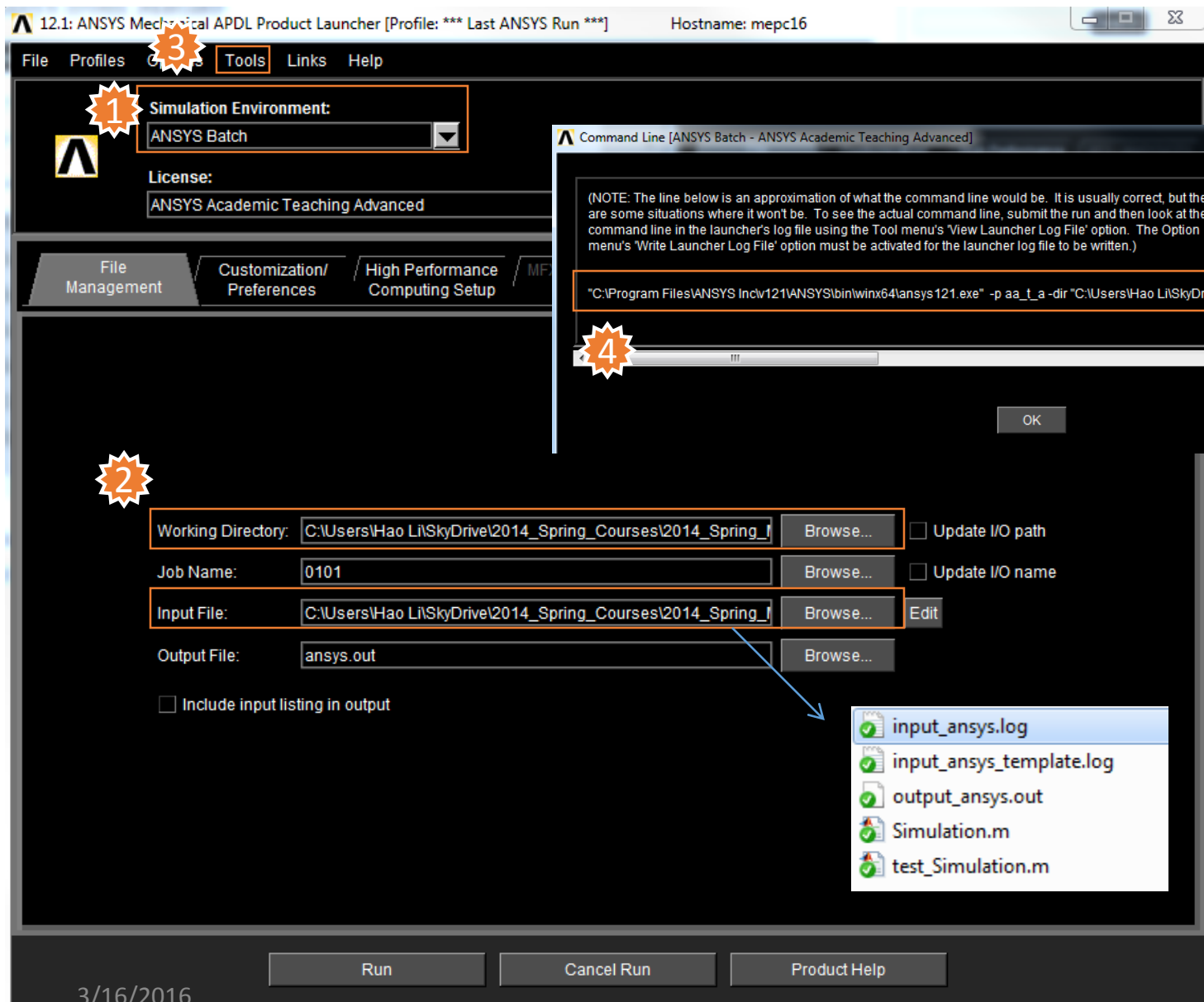
while ~feof(fin)
    s = fgetl(fin);
    s = strrep(s, 'F,P51X,FY,-4000',['F,P51X,FY,-' num2str(DesignVar(1))]);
    s = strrep(s, 'MPDATA,EX,1,,2e11',['MPDATA,EX,1,, ' num2str(DesignVar(2))]);
    fprintf(fout,'%s\n',s);
    disp(s)
end

fclose(fin)
fclose(fout)

%Call Ansys by command line
dos('"C:\Program Files\ANSYS Inc\v121\ANSYS\bin\winx64\ansys121.exe" -p aa_t_a -

%Read ANSYS output
fid=fopen('output_ansys_out','rt');
```


- Run Ansys “Mechanical APDL **Product Launcher**”
- In the section “Simulation environment”, select “**Ansys batch**”
- In the File Management tab, select you **working directory**
- In the File Management tab, select your **input file** (this is you file .log)
- Go to **Tools > Display command line**
- Copy the command line **#cmd**
- Open Matlab and run:
dos('#cmd')



ANSYS Output (output_ansys.out)

PRINT U NODAL SOLUTION PER NODE

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP= 0 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

NODE	UY
1	0.0000
2	-0.52083E-01
3	-0.19467

Tip
Displacement

MAXIMUM ABSOLUTE VALUES

NODE 3
VALUE -0.19467

Read ANSYS Output

```
%Call Ansys by command line
dos('"C:\Program Files\ANSYS Inc\v121\ANSYS\bin\winx64\ansys121.exe'

%Read ANSYS output
fid=fopen('output_ansys.out','rt');
flag_string='NODE          UY';
while ~feof(fid)
    s = fgetl(fid);
    flag=strfind(s,flag_string);
    if flag
        s = fgetl(fid);
        s = fgetl(fid);
        s = fgetl(fid);
        break
    end
end
s=str2num(strtrim(s));
response=-s(2)
fclose(fid)
```

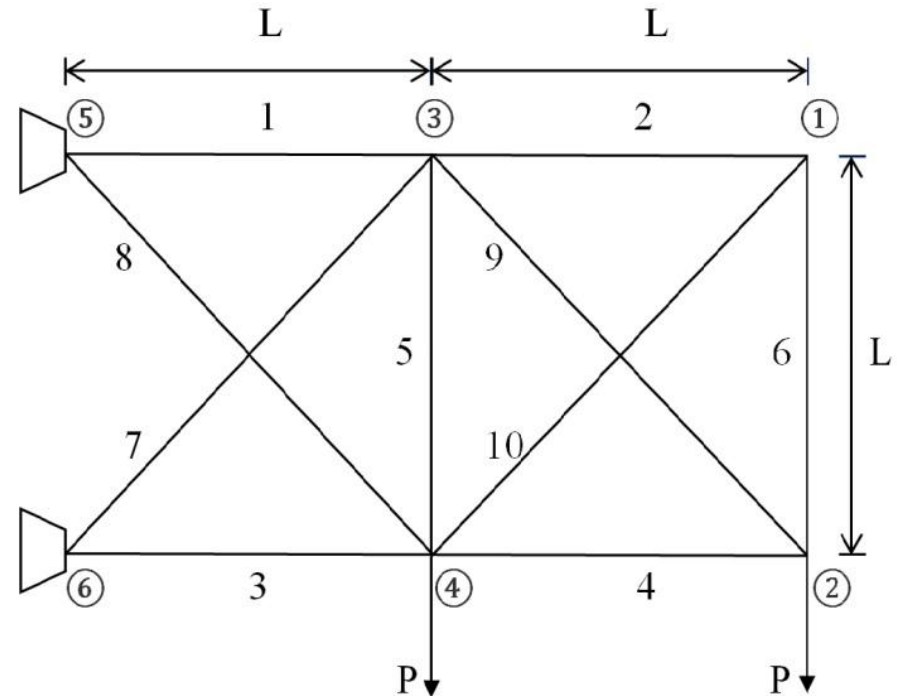
Monte Carlo Simulation on 10Bar Truss

- **10 Bar Truss**

- $L = 360$ inches
- $P = 100$ kips
- $E = 10^4$ ksi
- Cross sectional area = 5.0 in^2
- Pinned at the node 5 and 6

- $P_4(\mu = -10^5, \sigma = 1000)$

- What is the distribution of vertical displacement at node 2?



Input file structure

BEGIN INPUT

%% NODE COORDINATE

%% NODE_ID, X, Y

BEGIN NODE_COORDINATE

1, 720., 360.

2, 720., 0.

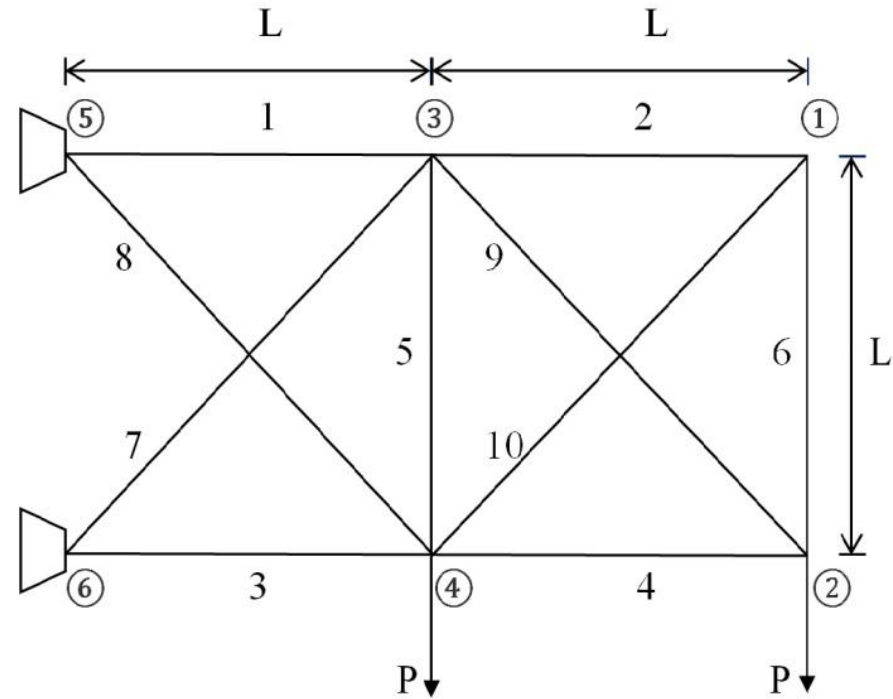
3, 360., 360.

4, 360., 0.

5, 0., 360.

6, 0., 0.

END NODE_COORDINATE



Input file structure

%% ELEMENTS

%% ELEMENT_ID, NODE_START, NODE_END

BEGIN ELEMENTS

1,5,3

2,3,1

3,6,4

4,4,2

5,3,4

6,1,2

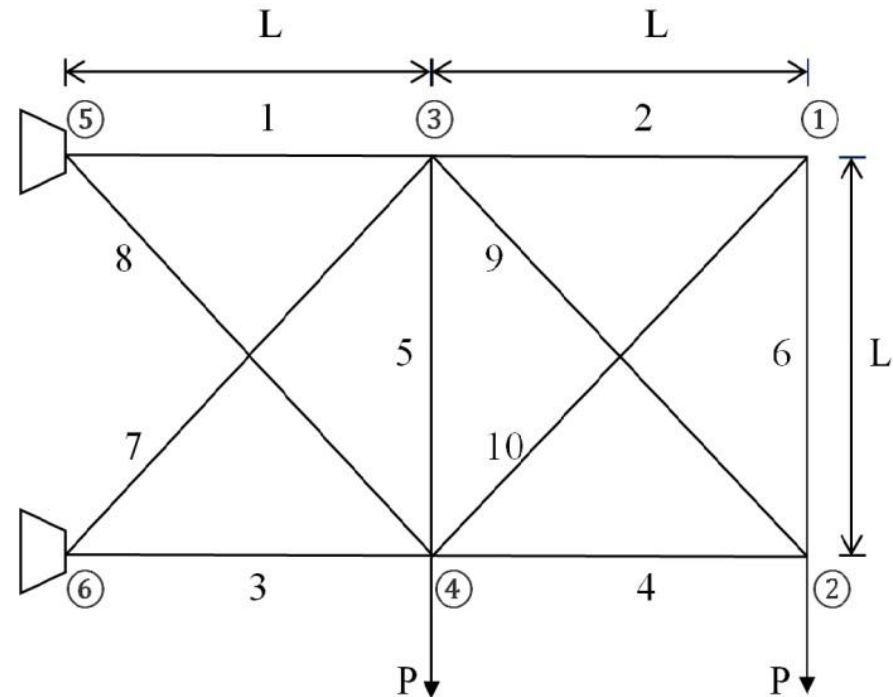
7,6,3

8,5,4

9,3,2

10,4,1

END ELEMENTS



Input file structure

%% ELEMENT AREA

%% ELEMENT_ID, CROSS_SECTIONAL_AREA

BEGIN ELEMENT_AREA

1, 5

2, 5

3, 5

4, 5

5, 5

6, 5

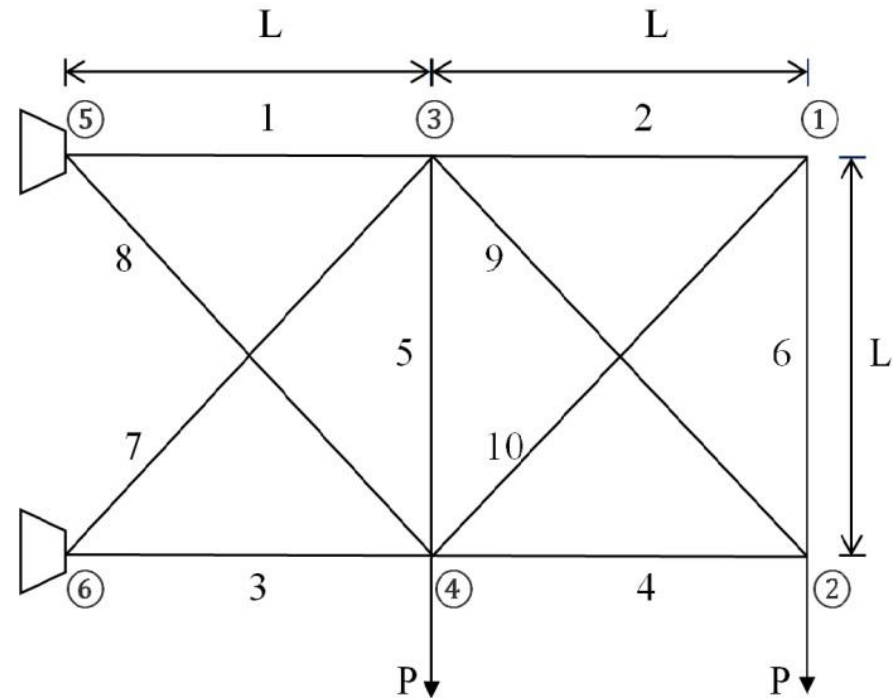
7, 5

8, 5

9, 5

10, 5

END ELEMENT_AREA



Input file structure

%% MODULUS OF ELASTICITY

%% ELEMENT_ID, MODULUS_OF_ELASTICITY

BEGIN E

1, 1e7

2, 1e7

3, 1e7

4, 1e7

5, 1e7

6, 1e7

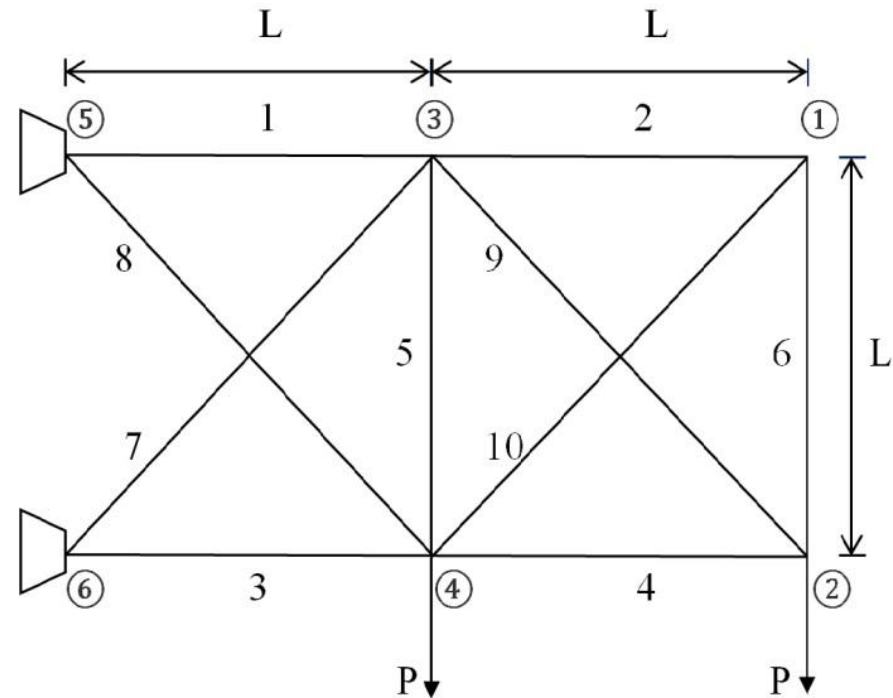
7, 1e7

8, 1e7

9, 1e7

10, 1e7

END E



Input file structure

%% DISPLACEMENT BOUNDARY CONDITION

%% NODE_ID, IS_X_FIXED, IS_Y_FIXED

BEGIN DISP_BC

5, 1, 1

6, 1, 1

END DISP_BC

%% FORCE BOUNDARY CONDITION

%% NODE_ID, FX, FY

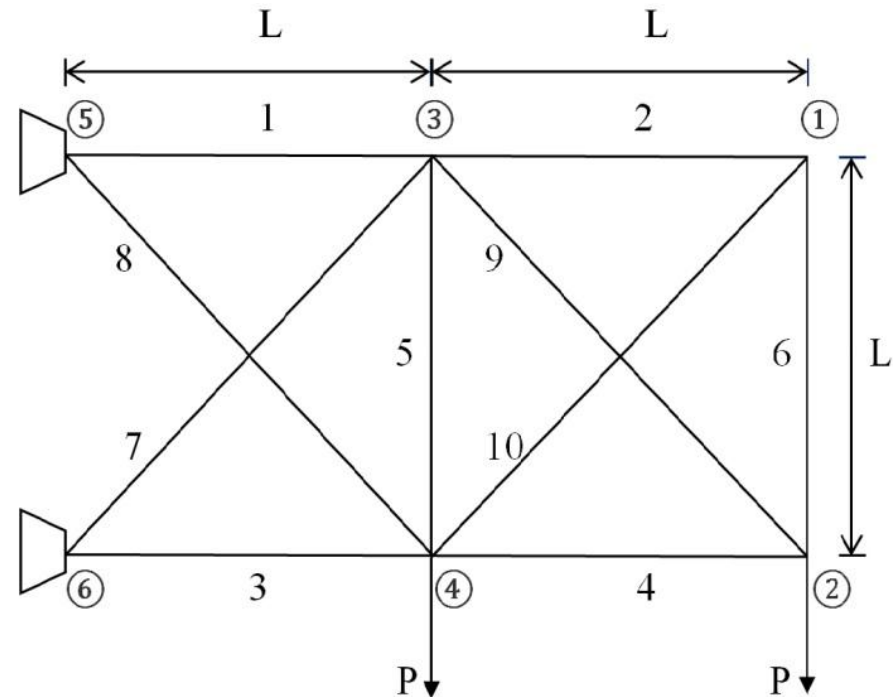
BEGIN F_BC

4, 0, -100000

2, 0, -100000

END F_BC

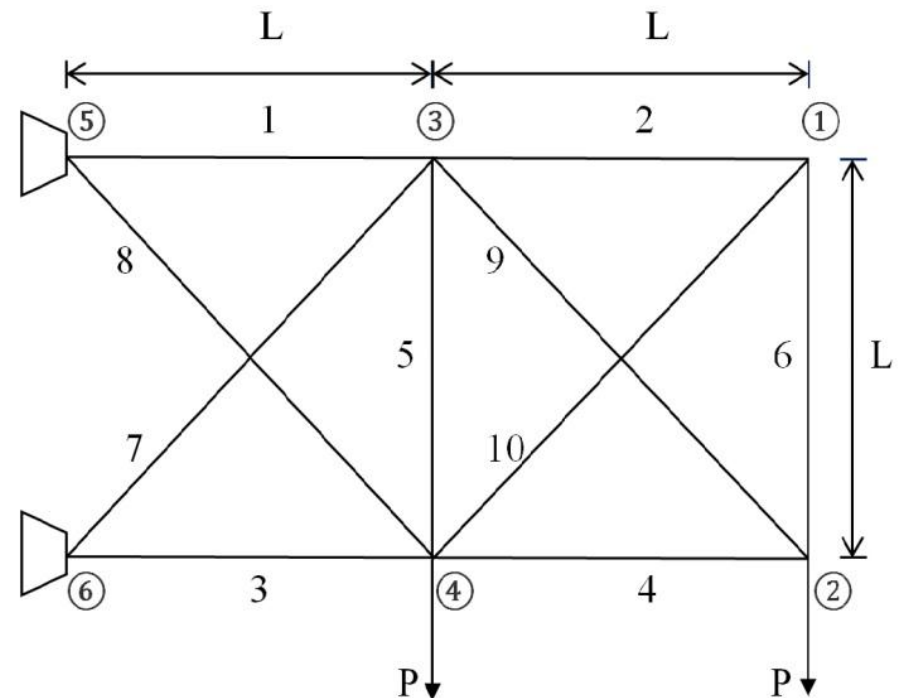
END INPUT



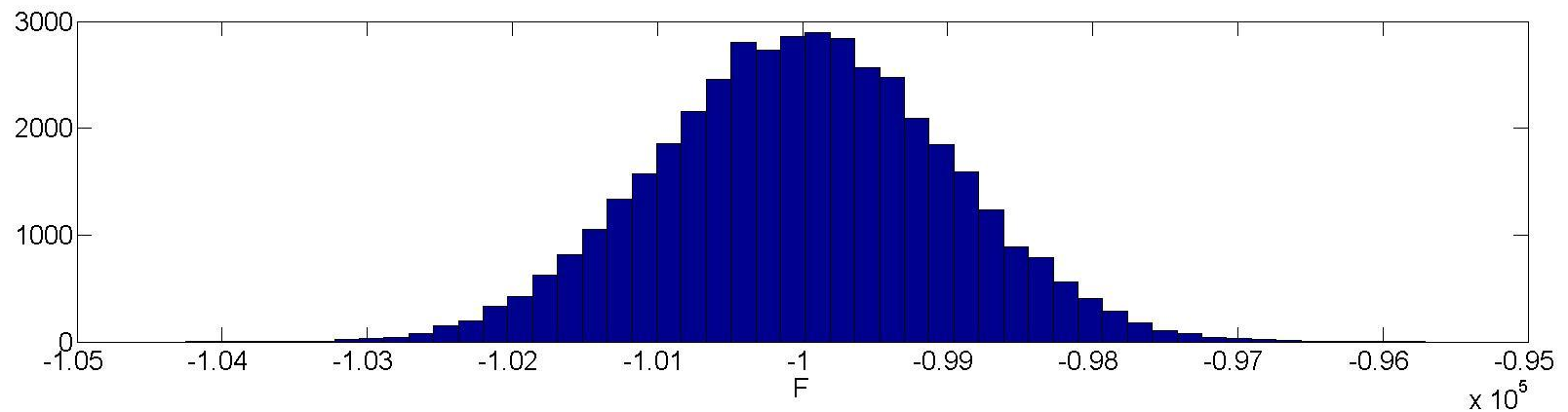
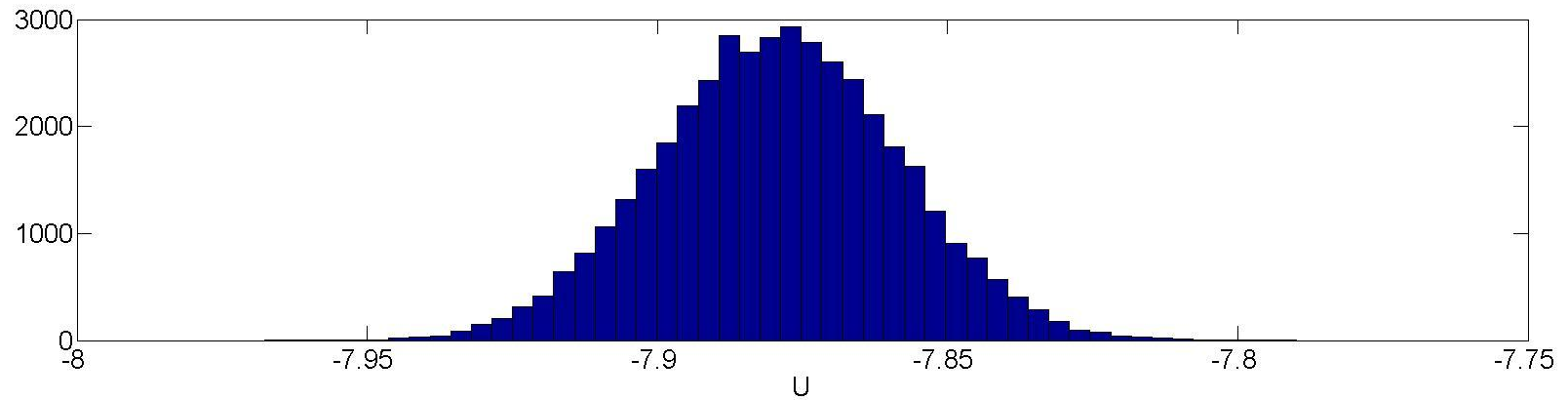
- Command to run the FEA solver: `staticSolve`

- `Input_file.inp` needs to be in the same directory
- Writes `U.out`
 - Displacement at all the degrees of freedom

1: 1.6944
2: -7.5821
3: -1.9028
4: -7.8712
5: 1.4054
6: -3.344
7: -1.4718
8: -3.5984
9: 0
10: 0
11: 0
12: 0



Results



**Thank You
Questions?**