Lehigh University

Advanced Technology for Large Structural Systems

CEE 466 – Advanced Finite Element Methods

Project 2A: Stochastic FE code – User Manual

By: Amirali Shojaeian

Instructor: Dr. Paolo Bocchini

November 2016

The information regarding the geometry, material properties, loads, BCs, etc. of the problem must be entered in the input file in the format below. The input file is in the format of a MATLAB script. The main code calls the input file as a function (e.g. Input10.m) and stores all the information that is required for the stochastic analysis.

Note that all the data provided in the input file must be consistent in the units. For instance, if SI system is used for the geometrical properties of the problem, the same system of units must be used for the material properties and so forth.

In this manual the input data of a sample problem shown below is entered in the specific format in the input file. This is a cantilever beam of length L = *10 m*, with a concentrated load of Q = 1 *N* (for the sake of simplicity) at the end tip (point *B*). We are interested in the deflection of the end point of the beam as the output. The material properties are as follows.

$$E_i = E_0(1 + \alpha_i)$$

in which $E_0 = 200$ *GPa* and α_i is a random variable with mean = *0* and standard deviation of *0.15*. The correlation coefficient among two generic α_i and α_j is given by the equation:

$$\rho_{ij} = exp(-\frac{|\Delta x_{ij}|}{0.8L})$$

in which is the distance between the midpoints of elements *i* and *j*. A deterministic value of G = 76.92 *GPa* is assumed for the shear modulus. The cross section is a square of 10×10 cm.

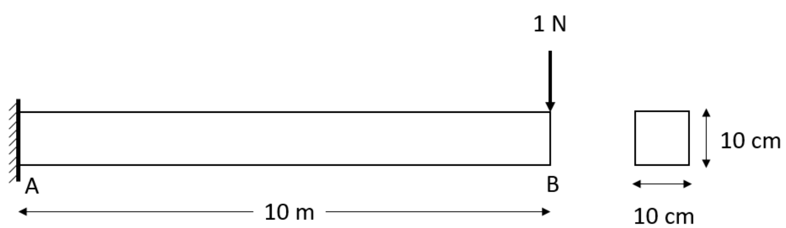

Figure 1 - Problem statement

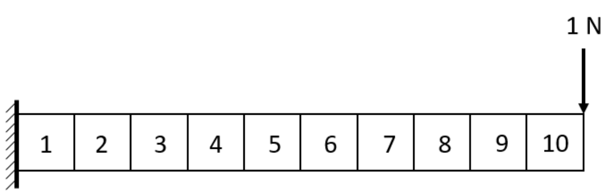

Figure 2 - Discretization of the finite element model

1- The name of the input file must be determined in order that the main code can call this input file as a function. Here the name is Input10().

```
function [Inp] = Input10()
Inp = struct('Inp',1);
```

2- The maximum size of the elements for meshing must be entered as the first parameter of the analysis. Here we want to have 10 elements over the length of the beam each of length *1000 mm*.

```
Inp.Par(1) = 1000;
```

3- The type of analysis must be determined here.

```
Inp.Par(2) = 1;
```

4- Number of samples and correlation length must be determined in this part in case stochastic FEM is to be performed. Number of samples is required specifically for Monte Carlo simulation and Polynomial approach.

```
Inp.Par(3) = 10000;
Inp.Par(4) = 8000;
```

5- A library of all the materials used in the problem must be built here in the format below. Be careful to change the units if needed for consistency. Here we are using *mm* for length and *N* for force. The values of Poisson's ratio and standard deviation are calculated as shown below.

$$\upsilon = \frac{G}{E} = \frac{76.92}{200} = 0.3846$$

$$\sigma_E = \sigma_\alpha \times E_0 = 0.15 \times 200 = 30\ Gpa$$

```
Inp.Mat = [1 200000  .3846 30000];
```

6- If a predefined correlation matrix is to be used in the stochastic analysis, it should be defined here with the format below. Here we are using correlation length to automate the covariance matrix, thus this parameter must be left empty.

```
Inp.Corr = [];
```

7- A library of all the cross sections used in the problem must be built here in the format below. Here, we have a only one cross section and we put the code 1 for it in the first column. Since the shape is rectangle, we need to enter the second parameter in the second column equal to 1, and enter the width and height of the cross section respectively in column 3 and 4 of the row containing the information of the cross section. The rest must be entered zero.

```
Inp.Sec = [1 1 100 100 0];
```

8- Next is to enter the location of the nodes. We choose 2 nodes at the two ends of the beam at points A and B. As we are interested in the displacement of the end tip for which we already have a node. Note the units used here.

```
Inp.Nd = [1 0 0
          2 10000 0];
```

9- In this part, we must determine the properties of the members in the format below. Here we are using Timoshenko beam elements for the analysis.

```
Inp.Mem = [1 1 2 1 1 3 1 1 1 1 1 1];
```

10- The Boundary conditions for the displacements must be defined in this part. We have one cantilever support at the location of point A.

```
Inp.Rst = [1 1 0
           1 2 0
           1 3 0];
```

11- For the concentrated loads at the location of the nodes (also called nodal loads) we should define the parameters here. If there is no nodal load in the problem, we just leave it empty like this.

```
Inp.CLd = [2 2 -1];
```

12- In the last part, we enter the parameters of the distributed loads over the elements. Be careful that if the load is being applied downward, a minus sign should be used to specify it.

```
Inp.DLd = [];
save('Inp')
end
```

13- After running the code for different types of analysis we have the results for the displacement of the end tip of the beam.

Table 1 - Summary of the output results

	Deterministic	MCS (with 10000 samples)	Perturbation	PRA: Method B and solution 1 (with 10000 samples)
Mean value	-0.2000 *mm*	-0.2045 *mm*	-0.2000 *mm*	-0.2041 *mm*
Standard deviation	-	0.0293 *mm*	0.0266 *mm*	0.0296 *mm*
Computational time	1.9 *s*	449.6 *s*	2.0 *s*	277.3 *s*