FE Analysis – What goes in and what comes out

Pankaj Pankaj
The University of Edinburgh
School of Engineering

Input to and output from an FE program

Structural analysis

- ★ Given a structure described by
  - ► geometry
  - ► physical characteristics
  - ► boundary conditions
- ★ subjected to loads

Input to a FE program

- ★ evaluate the response quantities
  - ► displacements
  - ► stresses
  - ► strains
  - ► support reactions etc.

Output from a FE program

The above is required for any kind of structural analysis and not just for FE

Input to and output from an FE program

- All FE programs require similar input – so if you understand one you know others
- The manner in which input is provided varies from one program to another
- We will one of the most powerful commercial FE package that we use both for teaching and research ABAQUS
- ABAQUS can solve many complex problems. The documentation in printed form exceeds 15000 pages. Online documentation is available at:

  http://www.see.ed.ac.uk/it/online/abaqus-681/v6.8/index.html

We will discuss ABAQUS with reference to a simple problem that can easily be solved by hand

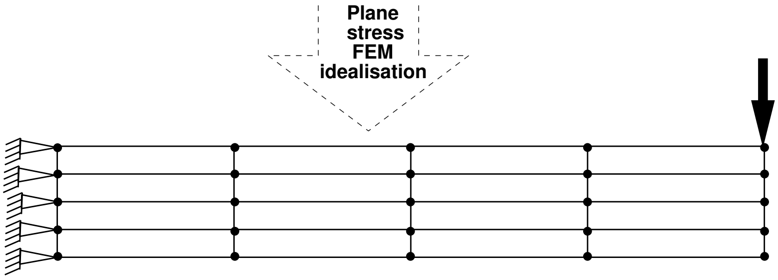
The cantilever problem

Consider a simple problem. For this problem, using elementary theory of beams you can easily work out:

- Displacements – e.g. end displacement is given by $PL^3/3EI$
- Moments
- Stresses and strains
- Reactions at the support

We will consider what is required to solve this using ABAQUS

FE input for the cantilever problem



We will:

- ★ Use plane stress idealisation (to be discussed in detail)
- ★ Discretise the continuum into 16 elements – choice arbitrary
- ★ Use 4-noded elements

FE input for the cantilever problem

We must now prepare a datafile (using any text editor on the computer). All ABAQUS datafiles must be of the form ***filename*.inp** where, *filename* is any name chosen by the analyst. Lets choose the name **cantbeam.inp** for this problem. The following lines show the contents of this file.

```
*HEADING                                                       LINE 1
CANTILEVER WITH CONTINUUM ELEMENTS--END SHEAR, CPS4I, 4 X 4 MESH    LINE 2
*NODE                                                          LINE 3
1,0.,0.                                                        LINE 4
17,6.,0.                                                       LINE 5
401,0.,.2                                                      LINE 6
417,6.,.2                                                      LINE 7
*NGEN,NSET=FIX                                                 LINE 8
1,401,100                                                      LINE 9
*NGEN,NSET=END                                                 LINE 10
17,417,100                                                     LINE 11
*NFILL                                                         LINE 12
FIX,END,4,4                                                    LINE 13
```

These are here for identification and are not actually input

FE input for the cantilever problem

```
*ELEMENT,TYPE=CPS4I                                  LINE 14
1,1,5,105,101                                        LINE 15
*ELGEN,ELSET=EALL                                    LINE 16
1,4,4,1,4,100,4                                      LINE 17
*MATERIAL,NAME=A1                                    LINE 18
*ELASTIC                                             LINE 19
1.E7,0.                                              LINE 20
*SOLID SECTION,MATERIAL=A1,ELSET=EALL                LINE 21
.1                                                   LINE 22
*PREPRINT, ECHO=YES, MODEL=YES, HISTORY=YES          LINE 23
*STEP,PERTURBATION                                   LINE 24
*STATIC                                              LINE 25
*BOUNDARY                                            LINE 26
FIX,1,2                                              LINE 27
*CLOAD                                               LINE 28
17,2,-.5                                             LINE 29
417,2,-.5                                            LINE 30
```

These are here for identification and are not actually input

FE input for the cantilever problem

```
*EL PRINT                                            LINE 31
COORD                                                LINE 32
S                                                    LINE 33
E                                                    LINE 34
*EL PRINT ,POSITION=AVERAGED AT NODES                LINE 35
S                                                    LINE 36
*NODE PRINT                                          LINE 37
U                                                    LINE 38
RF                                                   LINE 39
*END STEP                                            LINE 40
```

These are here for identification and are not actually input

We will now consider what these input lines mean

FE input for the cantilever problem

LINES 1 and 2

- ★ All lines beginning with a single * are keyword lines.
- ★ The first line indicates that a heading is to follow.
- ★ The heading is for our own convenience so that we can identify the problem at a later date.

Indicates a heading or a title is to follow

```
*HEADING
CANTILEVER WITH CONTINUUM ELEMENTS--END SHEAR, CPS4I, 4 X 4 MESH
```

A title of our choice

FE input for the cantilever problem

LINES 3 to 7

- ★ The *NODE keyword in line 3 indicates that data related to nodes is to follow.
- ★ Lines 4 to 7 have a node number followed by the co-ordinates of the node.

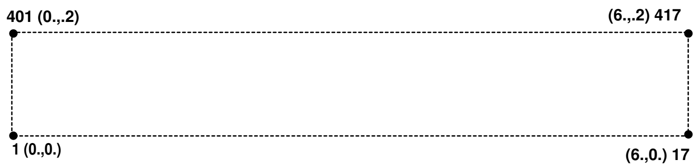
```
*NODE                                                        LINE 3
1,0.,0.                                                      LINE 4
17,6.,0.                                                     LINE 5
401,0.,.2                                                    LINE 6
417,6.,.2                                                    LINE 7
```

Indicates node coordinates follow

y-coordinate

x-coordinate

node number



Information provided so far

FE input for the cantilever problem

LINES 8 to 11

- ★ The `*NGEN` keyword generates nodes.
- ★ Parameters are sometimes required on keyword lines. Many parameters are optional.
- ★ On line 8 an optional parameter `NSET=FIX` has been used. This implies that the generated nodes will form a set called `FIX`.
- ★ In line 9 we ask ABAQUS to generate nodes from node no. `1` to node no. `401` with an increment of `100`.

Generate a set of nodes and call them FIX

```
*NGEN,NSET=FIX                                       LINE 8
1,401,100                                            LINE 9
*NGEN,NSET=END                                       LINE 10
17,417,100                                           LINE 11
```

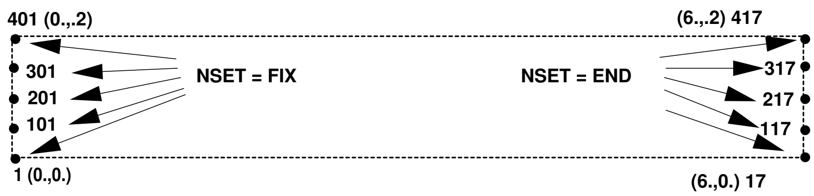
Generate a set of nodes and call them END

Node no. increments

End node number

Start node number

FE input for the cantilever problem



Information provided so far

FE input for the cantilever problem

LINES 12 and 13

- ★ The `*NFILL` keyword fills nodes between two bounds.
- ★ The first two entries on line 12 give the names of the node sets defining the first and the second bounds of the region respectively.
- ★ The third entry gives the number of intervals between bounding nodes, which is 4 in our case.
- ★ The fourth entry gives the increment in node numbers from the node number at the first bound set.

```
*NFILL                 LINE 12
FIX,END,4,4            LINE 13
```

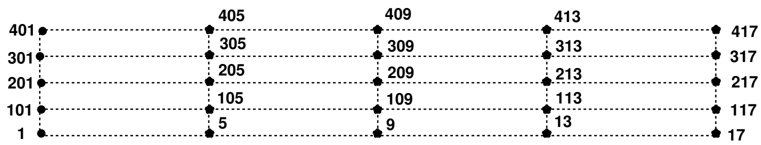
Fill nodes between bounds

Node no. increments

No. of intervals between bounding nodes

2nd bound

1st bound



FE input for the cantilever problem

Lines 14 and 15

- ★ The `*ELEMENT` is a keyword used to define an element directly by specifying its nodes
- ★ This has a required parameter `TYPE` that is required to be set equal to the element type. In our case we shall use a 4-noded element that has a name `CPS4I` in ABAQUS.
- ★ In line 15 one element is defined. The first entry on this line gives this element a number and entries 2 to 5 give it a "connectivity". Connectivity refers to the set of nodes connected to the element given in a specific order

Indicates that an element is to be specified

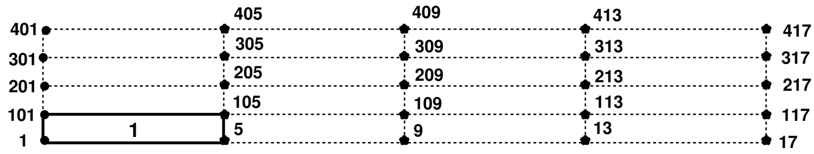
Specifies the element type. `CPS4I` is ABAQUS terminology

```
*ELEMENT,TYPE=CPS4I    LINE 14
1,1,5,105,101          LINE 15
```

Connectivity i.e. node numbers comprising the element in order

Element number

FE input for the cantilever problem



Information provided so far

FE input for the cantilever problem

LINES 16 and 17

The `*ELGEN` keyword generates elements. We use an optional parameter `ELSET` to give the set of generated elements a name (we call them `EALL`). The entries on line 17 are:

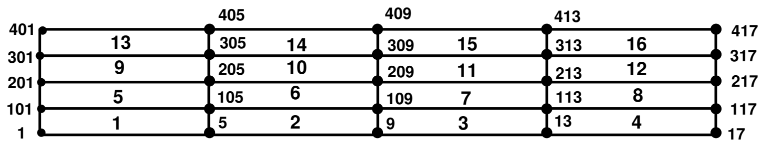
1. Master element number — 1 in our case as this is the master element that we shall use to generate other similar elements.
2. Number of elements to be defined in the first row generated, including the master element — 4 in our problem.
3. Increment in node numbers of corresponding nodes from element to element in the row — 4 in our case.
4. Increment in element numbers in the row — let us choose 1.
5. Number of rows to be defined including the master row — we need 4.
6. Increment in node numbers of corresponding nodes from row to row — 100 in our case.
7. Increment in element numbers of corresponding elements from row to row — we use 4.

```
*ELGEN,ELSET=EALL                                                   LINE 16
1,4,4,1,4,100,4                                                     LINE 17
```

FE input for the cantilever problem

```
*ELGEN,ELSET=EALL                                    LINE 16
1,4,4,1,4,100,4                                      LINE 17
```

- `*ELGEN,ELSET=EALL`: Generate elements and call the set EALL
- `4` (7th entry): Increment in corresponding element nos. from row to row
- `100`: Increment in corresponding node nos. from row to row
- `4` (5th entry): No. of rows to be defined including the master row
- `1`: Increment in element nos. in the row
- `4` (3rd entry): Increment in corresponding node nos. in the row
- `4` (2nd entry): Number of elements to be defined in the first row generated, including the master element
- `1` (1st entry): Master element number



All elements in ELSET = EALL

FE input for the cantilever problem

LINES 18 to 22

- ★ The `*MATERIAL` keyword indicates the start of a material definition and has a required parameter `NAME` associated to it – let us call it `A1`.
- ★ The `*ELASTIC` option in line 19 is used to define linear elastic moduli to the material `A1`. Young's modulus, E and Poisson's ratio, ν are specified in line 20.
- ★ The `*SOLID SECTION` keyword has been used here to define thickness – 0.1m specified in line 22). This thickness is valid for all elements `ELSET=EALL` and these elements have elastic properties of `MATERIAL=A1`.

```
*MATERIAL,NAME=A1                                    LINE 18
*ELASTIC                                             LINE 19
1.E7,0.                                              LINE 20
*SOLID SECTION,MATERIAL=A1,ELSET=EALL                LINE 21
.1                                                   LINE 22
```

- `*MATERIAL,NAME=A1`: We are defining a material called A1
- `*ELASTIC`: The elastic properties of A1 follow
- `1.E7,0.`: Young's modulus and Poisson's ratio
- `*SOLID SECTION,MATERIAL=A1,ELSET=EALL`: Thickness of A1 to follow. All elements in EALL are of material A1
- `.1`: Thickness

Model now fully defined. Loading and BCs to be specified

FE input for the cantilever problem

LINE 23

After you have run ABAQUS the numerical results are printed in the ***filename*.dat** file. This lines indicates that you would like your input file (***filename*.inp**)**, the history data and the model data to be included in the results file.**

```
*PREPRINT, ECHO=YES, MODEL=YES, HISTORY=YES                    LINE 23
```

LINES 24 and 40

These two lines indicate the beginning and the end of a step. We have already provided the model data. The program now requires boundary conditions and loading. The parameter `PERTURBATION` indicates that we are conducting a linear analysis in which we shall be providing load and boundary changes.

```
*STEP,PERTURBATION                                             LINE 24
.
.
.
*END STEP                                                      LINE 40
```

FE input for the cantilever problem

LINES 25 to 27

- ★ `*STATIC` indicates that the step should be analysed as a static load step.
- ★ `*BOUNDARY` (line 26) is used to prescribe boundary conditions.
- ★ The first entry on data line (line 27) indicates that the boundary conditions relate to node set `FIX`. The following two entries on line 27 restrain degrees of freedom 1 and 2. Thus all nodes of the node set `FIX` are now restrained from displacement in both x- and y-directions.

```
*STATIC                                                        LINE 25
*BOUNDARY                                                      LINE 26
FIX,1,2                                                        LINE 27
```

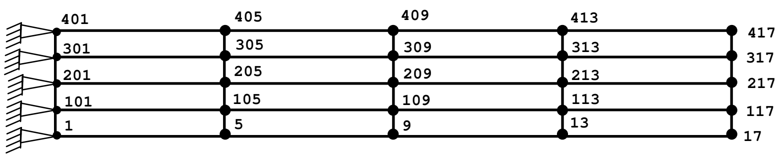
Load step is static

Boundary conditions follow

Restrained in direction 2 – y direction

Restrained in direction 1 – x direction

Boundary conditions relate to all nodes in NSET called FIX



FE input for the cantilever problem

LINES 28 to 30

- ★ `*CLOAD` indicates concentrated loads are being specified.
- ★ In lines 29 and 30 concentrated loads have been applied on nodes 17 and 417, in the y-direction (degree of freedom – 2), and the magnitude of these loads is 0.5. The minus sign indicates that the load acts downwards (y is positive upwards).

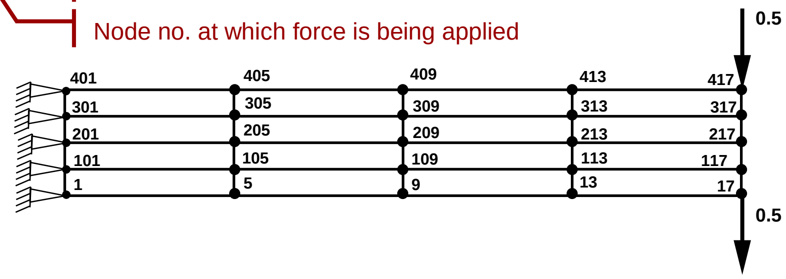
```
*CLOAD                                                   LINE 28
17,2,-.5                                                 LINE 29
417,2,-.5                                                LINE 30
```

Concentrated loads being specified

Force applied

Force direction is 2 – y direction

Node no. at which force is being applied



FE input for the cantilever problem

LINES 31 to 39

- ★ `*EL PRINT` is used to define output requests for element variables such as stresses and strains. FE does all calculations at integration points within elements.
- ★ Line 32 to 34 request output of the coordinates of integration points (`COORD`), stresses (`S`), and strains (`E`). By default option `*EL PRINT` outputs values at integration points. If we want averages of values extrapolated at the nodes, we can use the parameter `POSITION=AVERAGED AT NODES`. Here we have requested stresses (line 36).
- ★ `*NODE PRINT` is used to define output requests for nodal variables such as displacements and reactions
- ★ Here we want nodal displacements which are indicated by the letter `U` in line 38 and reaction forces indicated by `RF` in line 39.

FE input for the cantilever problem

```
*EL PRINT                                      LINE 31
COORD                                          LINE 32
S                                              LINE 33
E                                              LINE 34
*EL PRINT ,POSITION=AVERAGED AT NODES          LINE 35
S                                              LINE 36
*NODE PRINT                                    LINE 37
U                                              LINE 38
RF                                             LINE 39
```

Print request for element variables

Print integration point coordinates

Print stresses

Print strains

Print element variables extrapolated at nodes

Print stresses

Print displacements

Print reactions at restrained nodes

ABAQUS outputs results in a number of different files.
filename.dat is a readable file in which you can see the above output

FE output for the cantilever problem

Consider partial results from the *filename*.dat file

E L E M E N T   D E F I N I T I O N S

NUMBER	TYPE	PROPERTY REFERENCE	NODES FORMING ELEMENT			
1	CPS4I	1	1	5	105	101
2	CPS4I	1	5	9	109	105
3	CPS4I	1	9	13	113	109
.						
.						
.						

Element connectivity

Element property

Element type

Element nos.

FE output for the cantilever problem

```
M A T E R I A L   D E S C R I P T I O N

 MATERIAL NAME: A1

    ELASTIC          YOUNG'S    POISSON'S
                     MODULUS      RATIO

                   1.00000E+07 0.00000E+00
```

Values for material A1 as input

FE output for the cantilever problem

```
N O D E   D E F I N I T I O N S

   NODE            COORDINATES                         SINGLE POINT CONSTRAINTS
  NUMBER                                                 TYPE     PLUS     DOF

       1    0.00000E+00 0.00000E+00 0.00000E+00                   1 2
       5     1.5000      0.00000E+00 0.00000E+00
       9     3.0000      0.00000E+00 0.00000E+00
      13     4.5000      0.00000E+00 0.00000E+00
       .
       .
       .
```

Nodal coordinates as input or generated by ABAQUS in accordance to our instructions in the input file. Note the constraints included.

FE output for the cantilever problem

```
B O U N D A R Y   C O N D I T I O N S

  NODE       DOF      AMP.     MAGNITUDE                NODE       DOF      AMP.     MAGNITUDE
                      REF.                                                  REF.

       1      1      (RAMP)    0.00000E+00                   1      2      (RAMP)    0.00000E+00
     101      1      (RAMP)    0.00000E+00                 101      2      (RAMP)    0.00000E+00
     201      1      (RAMP)    0.00000E+00                 201      2      (RAMP)    0.00000E+00
     301      1      (RAMP)    0.00000E+00                 301      2      (RAMP)    0.00000E+00
     401      1      (RAMP)    0.00000E+00                 401      2      (RAMP)    0.00000E+00

    - (RAMP) OR (STEP) - INDICATE USE OF DEFAULT AMPLITUDES ASSOCIATED WITH THE STEP


C O N C E N T R A T E D   L O A D S

        NODE    DOF      AMP.     AMPLITUDE            NODE    DOF      AMP.     AMPLITUDE

          17      2              -0.50000                417      2              -0.50000
```

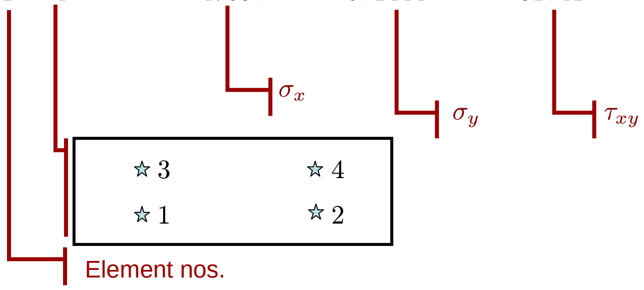
FE output for the cantilever problem

```
E L E M E N T   O U T P U T

  THE FOLLOWING TABLE IS PRINTED FOR ALL ELEMENTS
  WITH TYPE CPS4I AT THE INTEGRATION POINTS

 ELEMENT  PT FOOT-       COORD1       COORD2
             NOTE
       1   1           0.3170      1.0566E-02
       1   2            1.183      1.0566E-02
       1   3           0.3170      3.9434E-02
       1   4            1.183      3.9434E-02
```

Integration points are points within the elements. You can verify from the above coordinates that for element 1 the points are approximately located as:

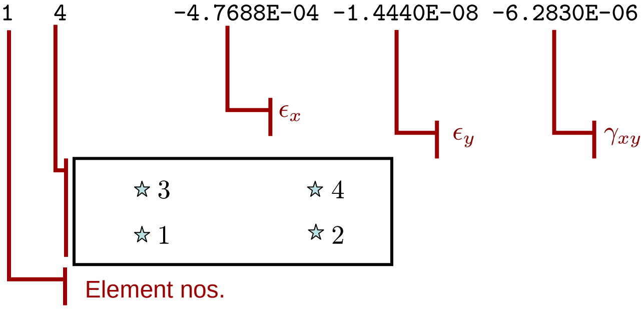
FE output for the cantilever problem

```
THE FOLLOWING TABLE IS PRINTED FOR ALL ELEMENTS WITH
TYPE CPS4I    AT THE INTEGRATION POINTS

 ELEMENT  PT FOOT-      S11         S22         S12
             NOTE
       1   1          -7044.     -3.8691E-02  -31.41
       1   2          -7044.     -0.1444      -31.41
       1   3          -4769.     -3.8691E-02  -31.41
       1   4          -4769.     -0.1444      -31.41
```



FE output for the cantilever problem

```
THE FOLLOWING TABLE IS PRINTED FOR ALL ELEMENTS WITH
TYPE CPS4I    AT THE INTEGRATION POINTS

ELEMENT  PT FOOT-      E11         E22         E12
             NOTE
       1   1         -7.0440E-04 -3.8691E-09 -6.2830E-06
       1   2         -7.0440E-04 -1.4440E-08 -6.2830E-06
       1   3         -4.7688E-04 -3.8691E-09 -6.2830E-06
       1   4         -4.7688E-04 -1.4440E-08 -6.2830E-06
```



FE output for the cantilever problem

THE FOLLOWING TABLE IS PRINTED FOR ALL ELEMENTS WITH TYPE CPS4I
AVERAGED AT THE NODES

NODE	FOOT-NOTE	S11	S22	S12
1		-7877.	7.8724E-11	-31.41
5		-6750.	-0.1831	-31.27
9		-4500.	0.7042	-31.23
13		-2250.	-2.615	-31.27

σ_x σ_y τ_{xy}

Node nos.

FE output for the cantilever problem

N O D E   O U T P U T

THE FOLLOWING TABLE IS PRINTED FOR ALL NODES

NODE	FOOT-NOTE	U1	U2
5		-1.1815E-03	-8.8765E-03
9		-2.0250E-03	-3.2940E-02
13		-2.5315E-03	-6.7129E-02
17		-2.7000E-03	-0.1064

y-direction displacement,
in theory we shall call this q_y

x-direction displacement,
in theory we shall call this q_x

Node nos.

FE output for the cantilever problem

NODE	FOOT-NOTE	RF1	RF2
1		18.76	8.3114E-02
101		22.47	0.2474
201		-8.5938E-12	0.3389
301		-22.47	0.2474
401		-18.76	8.3114E-02

Reaction in y-direction

Reaction in x-direction

Node nos.

What should the total reactions in the two directions be?

THE ANALYSIS HAS BEEN COMPLETED