

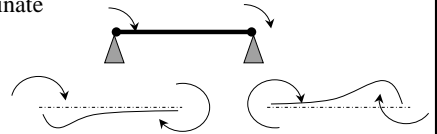
The Introduction of Finite Element Program - ABAQUS

Shao-Tang Yen
 Department of Civil Engineering
 Tamkang University
 Taiwan, R.O.C.



Structure Matrix

- Node, Coordinate
- Force, Def
- k
- K
- Stiffness Matrix



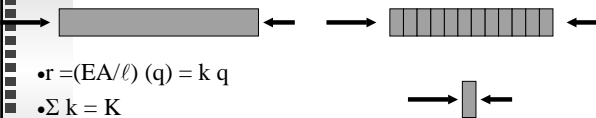
$$\begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{bmatrix} q_1 \\ q_2 \end{bmatrix} = \begin{bmatrix} R_1 \\ R_2 \end{bmatrix}$$

$$[F_i] = [K] \times [w_i]$$



Finite Element Method (1)

- The Mean of Finite Element



- $r = (EA/\ell) (q) = k q$
- $\Sigma k = K$
- $R = KQ$



Finite Element Method (2)

$$[F_i] = [K] \times [w_i]$$

$$[w_i] = [K]^{-1} \times [F_i]$$

$$\varepsilon_i = \frac{\partial w}{\partial x_i} \quad \sigma_i = E \varepsilon_i$$



Finite Element Method (3)

- Model
 - Coordinate
 - Geometry
- Material Properties
- Condition
 - Force
 - Deformation



Review ABAQUS

- Hibbitt, Karlsson & Sorensen(HKS)
- ABAQUS/Standard
- ABAQUS/Explicit
- Version
- Software
 - WINNT, Exceed, Exceed 3D
- Hard
 - Intel, Network Card



ABAQUS Basics

- See Figure 1



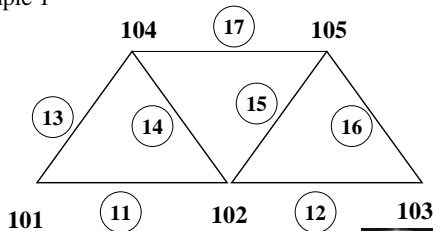
Required Model Information

- Model Data
 - Geometry -- Nodes, Elements, Connectivity
 - Element Section Properties
 - Material Data
- History Data
 - Loads and Boundary Conditions
 - Analysis Type
 - Output Requests



Input File Syntax /Example 1

- See Example 1



Input File Syntax /Model Data 1

- Title
 - *HEADING *HEADING
 - EMAPLE 1
- Syntax : <Text>
- Node
 - *NODE,NSET=<Text>
 - Syntax : <Node#>,<X>,<Y>,<Z>
 - NSET Specifies the Name of This Set of Nodes

```
*NODE,NSET=LINE1
101, 0,0,0
102, 5,0,0
103,10,0,0
```



Input File Syntax /Model Data 2

- Element
 - *ELEMENT, TYPE=<Text>, ELSET=<Text>
 - Syntax : < Element #>,<Node1>,< Node2 >,...
 - ELSET Specifies the Name of This Set of Elements, Used in *SOLID SECTION to Define the Properties for Elements (Material, Section Properties)

```
*ELEMENT, TYPE=T2D2,ELSET=FRAME
11, 101, 102
12, 102, 103
13, 101, 104
14, 102, 104
```



Input File Syntax /Model Data 3

- Section Properties
 - Syntax :
 - *SOLID SECTION, ELSET=<Text>, MATERIAL=<Text>
 - *MATERIAL, NAME=<Text>
 - *ELASTIC
 - <E>,<u>

```
*SOLID SECTION, ELSET=FRAME, MATERIAL=CONC
*MATERIAL, NAME=CONC
*ELASTIC
4.0E+6, 0.15
```



Input File Syntax /History Data 1

- Step
 - *STEP,PERTURBATION
 - *END STEP
- Static or Dynamic
 - *STATIC
- Boundary
 - *BOUNDARY
 - Syntax : <Node or Node Set>,<BC Condition>

```
*BOUNDARY
101, ENCASTRE
102, 2
LINE,1,3
```



Input File Syntax /History Data 2

- Force Loading
 - *CLOAD
 - Syntax : <Node>,<Dof>,<Magnitude>
 - Other Loading Types Available
- Output Requests
 - *RESTART, WRITE
 - *NODE PRINT
 - *EL PRINT

```
*CLOAD
102, 2,-1000
```



Output Requests

- RESTART, WRITE
 - Creates Restart File -- Post
- *NODE PRINT, [NSET= < >]
 - U - Deformation
 - RF -Result Force
- *EL PRINT, [ELSET= < >]
 - S - All Stress
 - S11 - Sxx
 - S12 - Sxy



Running ABAQUS

- Create Input File as <job-name>.inp
 - test.inp**
- Check Errors in Input File
 - abaqus job=test datacheck interactive**
 - Look at <job-name>.dat File
- Run Full Analysis
 - abaqus job=test**



Output Files

- ABAQUS Creates a Number of Output Files
 - test.dat - Full Output File
 - test.msg - Runtime Messages
 - test.log -Time
 - test.res - Post or Restart Analysis
 - test.sta - Step Processing Data
 - test.mpl - Plot File
 - test.ps - Postscript file



Post Processing 1

- Exceed (3D)
- To View Results
 - abaqus post job=<job-name>**
- Post Command
 - draw** -Show Model
 - draw, displaced** - Show Deformation
 - Set,<...>**
 - Use ? key



Post Processing

- Create Plot Files
 - set, hard copy=on, file= <filename>**
 - ...
 - ...
 - set, hard copy=off**
 - <job-name>.mpl
- Exit Post Command : **end**



Print Results

- Make a Plot File Form ABAQUS post Processing
- Create a Postscript File
 - abaqus plot job=<job-name> device=cps**
 - Answer the Questions
 - The <job-name>.ps is created
- Send File to Postscript Printer
 - copy <jobname>.inp lpt1**



Running Network ABAQUS

- IE or Netscape
 - <http://163.13.138.178>
 - User, Password
- Click the ABAQUS Program at Start Menu
- Note
 - The Network procedure Allows only One User Connect.
 - Just Running Program in Your Directory
 - Don't Erase Anything !!!

