9 Programs of Finite Element Method

■ **Contents**

9 Programs of Finite Element Method

■ **Keywords**

- **Program** 程序
- **Geometric model** 几何模型
- **Arrays** 数组
- **Modules** 模块
- **Input data** 输入数据
- **Output solutions** 输出结果

9 Programs of Finite Element Method

9.1 One-dimensional program of beam deformation

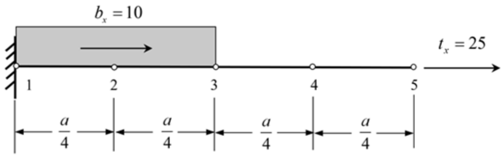
9.2 Two-dimensional program of plane strain problem

9.3 Three-dimensional program of solid compression

9.4 Exercises

9.1 One-dimensional program of beam deformation

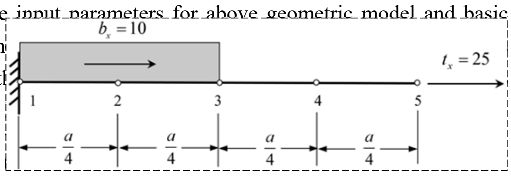
- This program is suitable for solving the Example 3.2. The geometric model of one-dimensional program of beam deformation is shown in Fig. 9.1; this program will solve the cantilever beam in Example 3.2 with the young's module E=1000, length a=10, loading t_x=25, and loading b_x=10 (0<x<a/2).



- **Figure 9.1 Geometric model of one-dimensional program of beam deformation.**

9.1 One-dimensional program of beam deformation

- The input parameters for above geometric model and basic con... wit... are...
  •



Input data	Explanations
4 1	element in *x*, material
1000.0	*EA*
2.5 2.5 2.5 2.5	element length
1	nr displacement BC
1 0	k, nf(:,k), *i*=1, nr
5	nr traction BC
1 12.5 2 25 3 12.5 4 0 5 25	k, nf(:,k), *i*=1, nr
0	fixed_freedoms

$$\begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \\ f_5 \end{Bmatrix} = \begin{Bmatrix} 12.5 \\ 25 \\ 12.5 \\ 0 \\ 25 \end{Bmatrix}$$

9.1 One-dimensional program of beam deformation

- Based on running the program of one-dimensional program of beam deformation, the output solutions for above geometric model and basic conditions are written in the file **1DFEM.res**, as shown below

Node	Disp	Element	Actions	
1	0.0000E+00	1	-0.6250E+02	0.6250E+02
2	0.1563E+00	2	-0.3750E+02	0.3750E+02
3	0.2500E+00	3	-0.2500E+02	0.2500E+02
4	0.3125E+00	4	-0.2500E+02	0.2500E+02
5	0.3750E+00			

3.5 Treatments on boundary conditions

- Formal solution

$$\mathbf{u} = \mathbf{K}^{-1}\mathbf{f}$$

$$\begin{bmatrix} 800 & -400 & & \\ -400 & 800 & -400 & \\ & -400 & 800 & -400 \\ & & -400 & 400 \end{bmatrix} \begin{Bmatrix} \hat u_2 \\ \hat u_3 \\ \hat u_4 \\ \hat u_5 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 12.5 \\ 0 \\ 25 \end{Bmatrix} \Rightarrow \begin{Bmatrix} \hat u_2 \\ \hat u_3 \\ \hat u_4 \\ \hat u_5 \end{Bmatrix} = \begin{Bmatrix} 0.15625 \\ 0.25 \\ 0.3125 \\ 0.375 \end{Bmatrix}$$

- Displacement and stress solution

$$\hat u^e(x') = N_1(x')\tilde u_1^e + N_2(x')\tilde u_2^e \qquad \hat\sigma_x^e(x') = E\frac{\partial \hat u^e(x')}{\partial x'} = E\left(\frac{\mathrm{d}N_1(x')}{\mathrm{d}x'}\tilde u_1^e + \frac{\mathrm{d}N_2(x')}{\mathrm{d}x'}\tilde u_2^e\right)$$

$$\hat u^1(x') = \left(1-\frac{x'}{h_e}\right)\times 0 + \frac{x'}{h_e}\times 0.15625 = 0.0625x'$$

$$\hat\sigma_x^1(x') = E\frac{\partial \hat u^1(x')}{\partial x'} = E\left(-\frac{1}{h_e}\times 0 + \frac{1}{h_e}\times 0.15625\right) = 62.5$$

3.5 Treatments on boundary conditions

- Displacement and stress solution

$$\hat u^e(x') = N_1(x')\tilde u_1^e + N_2(x')\tilde u_2^e \qquad \hat\sigma_x^e(x') = E\frac{\partial \hat u^e(x')}{\partial x'} = E\left(\frac{\mathrm{d}N_1(x')}{\mathrm{d}x'}\tilde u_1^e + \frac{\mathrm{d}N_2(x')}{\mathrm{d}x'}\tilde u_2^e\right)$$

$$\hat u^2(x') = \left(1-\frac{x'}{h_e}\right)\times 0.15625 + \frac{x'}{h_e}\times 0.25 = 0.09375x' + 0.15625$$

$$\hat\sigma_x^2(x') = E\frac{\partial \hat u^2(x')}{\partial x'} = E\left(-\frac{1}{h_e}\times 0.15625 + \frac{1}{h_e}\times 0.25\right) = 37.5$$

$$\hat u^3(x') = \left(1-\frac{x'}{h_e}\right)\times 0.25 + \frac{x'}{h_e}\times 0.3125 = 0.0625x' + 0.25$$

$$\hat\sigma_x^3(x') = E\frac{\partial \hat u^3(x')}{\partial x'} = E\left(-\frac{1}{h_e}\times 0.25 + \frac{1}{h_e}\times 0.3125\right) = 25$$

$$\hat u^4(x') = \left(1-\frac{x'}{h_e}\right)\times 0.3125 + \frac{x'}{h_e}\times 0.375 = 0.0625x' + 0.3125$$

$$\hat\sigma_x^4(x') = E\frac{\partial \hat u^4(x')}{\partial x'} = E\left(-\frac{1}{h_e}\times 0.3125 + \frac{1}{h_e}\times 0.375\right) = 25$$

9.1 One-dimensional program of beam deformation

- **One-dimensional program of beam deformation: 1DFEM**

- The main program of one-dimensional program of beam deformation is shown below, containing the modules of
- **dynamic arrays,**
- **input and initialisation,**
- **loop the elements to find global arrays sizes,**
- **global stiffness matrix assembly,**
- **read loads and/or displacements,**
- **equation solution, and retrieve element end actions.**

9 Programs of Finite Element Method

9.1 One-dimensional program of beam deformation

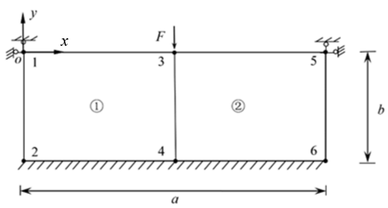
9.2 Two-dimensional program of plane strain problem

9.3 Three-dimensional program of solid compression

9.4 Exercises

9.2 Two-dimensional program of plane strain problem

- This program is suitable for solving the Example 6.1. The geometric model of two-dimensional program of plane strain problem is shown in Fig. 9.2; this program will solve the Example 6.1. The sizes of the model are *a*=30 in the horizontal *x*-direction and *b*=10 in the vertical *y*-direction.

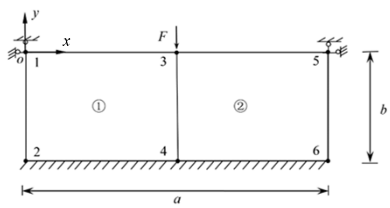


- **Figure 9.2 Geometric model of two-dimensional program of plane strain problem.**

Element number: ① and ② Node number: 1~6

9.2 Two-dimensional program of plane strain problem

- The material parameters are young's modulus E of 1×10^6 and Poisson's ratio v of 0.3. The boundary conditions are that the upper left and upper right vertices are fixed, and the lower parts are fixed. The vertical concentrated load F acting on the middle domain is -1×10^6.

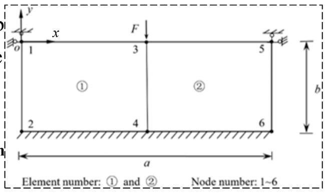


- **Figure 9.2 Geometric model of two-dimensional program of plane strain problem.**

Element number: ① and ② Node number: 1~6

9.2 Two-dimensional program of plane strain problem

- The input parameters for ab
conditions are written in the
with uniform size are used;
are listed in Table 9.2.



  - **Table 9.2 Input data and explan plane strain problem.**

Input data	Explanations
plane	element type
quadrilateral 4 y	element, node, direction
2 1 4 1	element in *x*, element in *y*, integration points, material
1.0e6 0.3	*E* *v*
0.0 15.0 30.0	*x*_coords
0.0 -10.0	*y*_coords
5	nr displacement BC
1 0 0 2 0 0 4 0 0 5 0 0 6 0 0	k, nf(:,k), *i*=1, nr
1	nr traction BC
3 0 -1.0e6	k, nf(:,k), *i*=1, nr
0	fixed_freedoms

9.2 Two-dimensional program of plane strain problem

- Based on running the program of two-dimensional program of plane strain problem, the output solutions for above geometric model and basic conditions are in the file **2DFEM.res**, as shown below

```
Node    x-disp      y-disp
  1  0.0000E+00  0.0000E+00
  2  0.0000E+00  0.0000E+00
  3  0.5855E-16 -0.6592E+00
  4  0.0000E+00  0.0000E+00
  5  0.0000E+00  0.0000E+00
  6  0.0000E+00  0.0000E+00
```


9.2 Two-dimensional program of plane strain problem

- Based on running the program of two-dimensional program of plane strain problem, the output solutions for above geometric model and basic conditions are in the file **2DFEM.res**, as shown below

```
The integration point (nip= 4) stresses are:
Element x-coord     y-coord       sig_x        sig_y       tau_xy
  1  0.3170E+01 -0.2113E+01 -0.8036E+04 -0.1875E+05 -0.1333E+05
  1  0.1183E+02 -0.2113E+01 -0.2999E+05 -0.6998E+05 -0.1333E+05
  1  0.3170E+01 -0.7887E+01 -0.8036E+04 -0.1875E+05 -0.3572E+04
  1  0.1183E+02 -0.7887E+01 -0.2999E+05 -0.6998E+05 -0.3572E+04
  2  0.1817E+02 -0.2113E+01 -0.2999E+05 -0.6998E+05  0.1333E+05
  2  0.2683E+02 -0.2113E+01 -0.8036E+04 -0.1875E+05  0.1333E+05
  2  0.1817E+02 -0.7887E+01 -0.2999E+05 -0.6998E+05  0.3572E+04
  2  0.2683E+02 -0.7887E+01 -0.8036E+04 -0.1875E+05  0.3572E+04
```


9.2 Two-dimensional program of plane strain problem

- Based on running the program of two-dimensional program of plane strain problem, the output solutions for above geometric model and basic conditions are in the file **2DFEM.res**, as shown below

```
The node point stresses are:
Element x-coord     y-coord       sig_x        sig_y       tau_xy
  1  0.0000E+00  0.0000E+00  0.5255E-11  0.2252E-11 -0.1690E+05
  1  0.1500E+02  0.0000E+00 -0.3803E+05 -0.8873E+05 -0.1690E+05
  1  0.0000E+00 -0.1000E+02  0.0000E+00  0.0000E+00  0.0000E+00
  1  0.1500E+02 -0.1000E+02 -0.3803E+05 -0.8873E+05  0.2252E-11
  2  0.1500E+02  0.0000E+00 -0.3803E+05 -0.8873E+05  0.1690E+05
  2  0.3000E+02  0.0000E+00 -0.5255E-11 -0.2252E-11  0.1690E+05
  2  0.1500E+02 -0.1000E+02 -0.3803E+05 -0.8873E+05  0.2252E-11
  2  0.3000E+02 -0.1000E+02  0.0000E+00  0.0000E+00  0.0000E+00
```


9.2 Two-dimensional program of plane strain problem

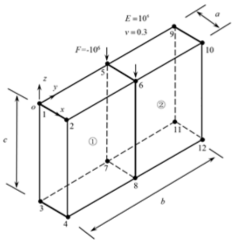
- **Two-dimensional program of plane strain problem: 2DFEM**

- The main program of two-dimensional program of plane strain problem is shown below, containing the modules of
- **dynamic arrays,**
- **input and initialisation,**
- **loop the elements to find global arrays sizes,**
- **element stiffness integration and assembly,**
- **equation solution,**
- **recover stresses at nip integrating points,**
- **recover stresses at node points.**

9 Programs of Finite Element Method

9.3 Three-dimensional program of solid compression

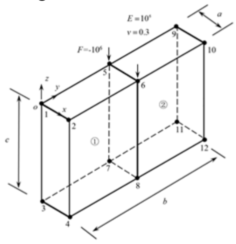
- The geometric model of three-dimensional program of solid compression is shown in Fig. 9.3. The sizes of the model are *a*=0.5 in the horizontal *x*-direction and *b*=3 in the horizontal *y*-direction and *c*=2 in the vertical *z*-direction.



- **Figure 9.3 Geometric model of three-dimensional program of solid compression.**

9.3 Three-dimensional program of solid compression

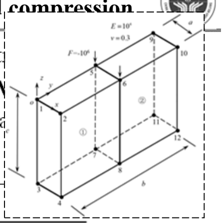
- The material parameters are young's modulus *E* of 1×10^6 and Poisson's ratio *v* of 0.3. The boundary conditions are that the upper left and upper right vertices are fixed, and the lower parts are fixed. The vertical concentrated load *F* acting on the middle domain is -1×10^6.



- **Figure 9.3 Geometric model of three-dimensional program of solid compression.**

9.3 Three-dimensional program of solid compression

- The input parameters for above geomet
conditions are written in the file **3DFEM**
with uniform size are used; the input da
are listed in Table 9.3.



- **Table 9.3 Input data and explanations of three-**
of solid compression.

Input data	Explanations
8	element-node
1 2 1 8 1	element in *x*, element in *y*, element in *z*, integration points, material
1.0e6 0.3	*E v*
0.0 0.5	*x*_coords
0.0 1.5 3.0	*y*_coords
0.0 -2.0	*z*_coords
10	nr displacement BC
1 0 0 0 2 0 0 0 3 0 0 0 4 0 0 0 7 0 0 0 8 0 0 0 9 0 0 0 10 0 0 0 11 0 0 0 12 0 0 0	k, nf(:,k), *i*=1, nr
2	nr traction BC
5 0.0 0.0 -1.0e6 6 0.0 0.0 -1.0e6	k, nf(:,k), *i*=1, nr
0	fixed_freedoms

9.3 Three-dimensional program of solid compression

- Based on running the program of three-dimensional program of solid compression, the output solutions for above geometric model and basic conditions are in the file **3DFEM.res**, as shown below

Node	x-disp	y-disp	z-disp
1	0.0000E+00	0.0000E+00	0.0000E+00
2	0.0000E+00	0.0000E+00	0.0000E+00
3	0.0000E+00	0.0000E+00	0.0000E+00
4	0.0000E+00	0.0000E+00	0.0000E+00
5	-0.3438E+00	0.3035E-16	-0.4332E+01
6	0.3438E+00	0.1720E-16	-0.4332E+01

Node	x-disp	y-disp	z-disp
7	0.0000E+00	0.0000E+00	0.0000E+00
8	0.0000E+00	0.0000E+00	0.0000E+00
9	0.0000E+00	0.0000E+00	0.0000E+00
10	0.0000E+00	0.0000E+00	0.0000E+00
11	0.0000E+00	0.0000E+00	0.0000E+00
12	0.0000E+00	0.0000E+00	0.0000E+00

9.3 Three-dimensional program of solid compression

- Based on running the program of three-dimensional program of solid compression, the output solutions for above geometric model and basic conditions are in the file **3DFEM.res**, as shown below

```
The integration point (nip= 8) stresses are:
  Element    x-coord    y-coord    z-coord
  sig_x      sig_y      sig_z      tau_xy     tau_yz     tau_zx
    1      0.3943E+00  0.1183E+01 -0.4226E+00
 0.1661E+06 -0.4920E+06 -0.1806E+07  0.4014E+05 -0.8760E+06  0.3011E+05
    1      0.3943E+00  0.1183E+01 -0.1577E+01
-0.6770E+06 -0.8533E+06 -0.2167E+07  0.1076E+05 -0.2347E+06  0.3011E+05
    1      0.3943E+00  0.3170E+00 -0.4226E+00
 0.4450E+05 -0.1318E+06 -0.4839E+06  0.4014E+05 -0.8760E+06  0.8068E+04
    1      0.3943E+00  0.3170E+00 -0.1577E+01
-0.1814E+06 -0.2286E+06 -0.5807E+06  0.1076E+05 -0.2347E+06  0.8068E+04
    1      0.1057E+00  0.1183E+01 -0.4226E+00
 0.1661E+06 -0.4920E+06 -0.1806E+07 -0.4014E+05 -0.8760E+06 -0.3011E+05
    1      0.1057E+00  0.3170E+00 -0.4226E+00
 0.4450E+05 -0.1318E+06 -0.4839E+06 -0.4014E+05 -0.8760E+06 -0.8068E+04
    1      0.1057E+00  0.1183E+01 -0.1577E+01
-0.6770E+06 -0.8533E+06 -0.2167E+07 -0.1076E+05 -0.2347E+06 -0.3011E+05
    1      0.1057E+00  0.3170E+00 -0.1577E+01
-0.1814E+06 -0.2286E+06 -0.5807E+06 -0.1076E+05 -0.2347E+06 -0.8068E+04
```


9.3 Three-dimensional program of solid compression

- Based on running the program of three-dimensional program of solid compression, the output solutions for above geometric model and basic conditions are in the file **3DFEM.res**, as shown below

```
The integration point (nip= 8) stresses are:
  Element    x-coord    y-coord    z-coord
  sig_x      sig_y      sig_z      tau_xy     tau_yz     tau_zx
    2      0.3943E+00  0.2683E+01 -0.4226E+00
 0.4450E+05 -0.1318E+06 -0.4839E+06 -0.4014E+05  0.8760E+06  0.8068E+04
    2      0.3943E+00  0.2683E+01 -0.1577E+01
-0.1814E+06 -0.2286E+06 -0.5807E+06 -0.1076E+05  0.2347E+06  0.8068E+04
    2      0.3943E+00  0.1817E+01 -0.4226E+00
 0.1661E+06 -0.4920E+06 -0.1806E+07 -0.4014E+05  0.8760E+06  0.3011E+05
    2      0.3943E+00  0.1817E+01 -0.1577E+01
-0.6770E+06 -0.8533E+06 -0.2167E+07 -0.1076E+05  0.2347E+06  0.3011E+05
    2      0.1057E+00  0.2683E+01 -0.4226E+00
 0.4450E+05 -0.1318E+06 -0.4839E+06  0.4014E+05  0.8760E+06 -0.8068E+04
    2      0.1057E+00  0.1817E+01 -0.4226E+00
 0.1661E+06 -0.4920E+06 -0.1806E+07  0.4014E+05  0.8760E+06 -0.3011E+05
    2      0.1057E+00  0.2683E+01 -0.1577E+01
-0.1814E+06 -0.2286E+06 -0.5807E+06  0.1076E+05  0.2347E+06 -0.8068E+04
    2      0.1057E+00  0.1817E+01 -0.1577E+01
-0.6770E+06 -0.8533E+06 -0.2167E+07  0.1076E+05  0.2347E+06 -0.3011E+05
```

9.3 Three-dimensional program of solid compression

- Based on running the program of three-dimensional program of solid compression, the output solutions for above geometric model and basic conditions are in the file **3DFEM.res**, as shown below

```
The node point stresses are:
  Element    x-coord    y-coord    z-coord
  sig_x      sig_y      sig_z      tau_xy     tau_yz     tau_zx
    1     0.5000E+00  0.1500E+01  0.0000E+00
 0.6018E+06 -0.4561E+06 -0.2122E+07  0.8816E+05 -0.1111E+07  0.6612E+05
    1     0.5000E+00  0.1500E+01 -0.2000E+01
-0.1250E+07 -0.1250E+07 -0.2916E+07  0.0000E+00  0.3307E-11  0.6612E+05
    1     0.5000E+00  0.0000E+00  0.0000E+00
 0.6614E-11  0.1543E-10  0.6614E-11  0.8816E+05 -0.1111E+07  0.0000E+00
    1     0.5000E+00  0.0000E+00 -0.2000E+01
 0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
    1     0.0000E+00  0.1500E+01  0.0000E+00
 0.6018E+06 -0.4561E+06 -0.2122E+07 -0.8816E+05 -0.1111E+07 -0.6612E+05
    1     0.0000E+00  0.0000E+00  0.0000E+00
 0.1167E-10  0.2724E-10  0.1167E-10 -0.8816E+05 -0.1111E+07  0.0000E+00
    1     0.0000E+00  0.1500E+01 -0.2000E+01
-0.1250E+07 -0.1250E+07 -0.2916E+07  0.0000E+00  0.5836E-11 -0.6612E+05
    1     0.0000E+00  0.0000E+00 -0.2000E+01
 0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
```

25
25

9.3 Three-dimensional program of solid compression

- Based on running the program of three-dimensional program of solid compression, the output solutions for above geometric model and basic conditions are in the file **3DFEM.res**, as shown below

```
The node point stresses are:
  Element    x-coord    y-coord    z-coord
  sig_x      sig_y      sig_z      tau_xy     tau_yz     tau_zx
    2     0.5000E+00  0.3000E+01  0.0000E+00
-0.6614E-11 -0.1543E-10 -0.6614E-11 -0.8816E+05  0.1111E+07  0.0000E+00
    2     0.5000E+00  0.3000E+01 -0.2000E+01
 0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
    2     0.5000E+00  0.1500E+01  0.0000E+00
 0.6018E+06 -0.4561E+06 -0.2122E+07 -0.8816E+05  0.1111E+07  0.6612E+05
    2     0.5000E+00  0.1500E+01 -0.2000E+01
-0.1250E+07 -0.1250E+07 -0.2916E+07  0.0000E+00  0.3307E-11  0.6612E+05
    2     0.0000E+00  0.3000E+01  0.0000E+00
-0.1167E-10 -0.2724E-10 -0.1167E-10  0.8816E+05  0.1111E+07  0.0000E+00
    2     0.0000E+00  0.1500E+01  0.0000E+00
 0.6018E+06 -0.4561E+06 -0.2122E+07  0.8816E+05  0.1111E+07 -0.6612E+05
    2     0.0000E+00  0.3000E+01 -0.2000E+01
 0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
    2     0.0000E+00  0.1500E+01 -0.2000E+01
-0.1250E+07 -0.1250E+07 -0.2916E+07  0.0000E+00  0.5836E-11 -0.6612E+05
```

26
26

9.3 Three-dimensional program of solid compression

- **Three-dimensional program of solid compression: 3DFEM**

- The main program of three-dimensional program of solid compression is shown below, containing the modules of
- **dynamic arrays,**
- **input and initialisation,**
- **loop the elements to find global arrays sizes,**
- **element stiffness integration and assembly,**
- **equation solution,**
- **recover stresses at nip integrating points,**
- **recover stresses at node points.**

27
27

The End

28