

Contents

1 Elastic Beam Element Under Static Loading	3
1.1 Simulate the cantilever by 1 elastic beam element	3
2 Elastic Beam Element under Dynamic Loading	5
2.1 Cantilever, One Elastic Beam Element	5
2.2 Cantilever, 5 Elastic Beam Elements	7
3 Cantilever, one 27 Node Brick Element, Dynamic Loading	12
3.1 Simulate the cantilever using one 27NodeBrick element	12
3.2 Simulate Cantilever Using five 27 Node Brick Elements	16
4 Elastic Beam Element under Dynamic Loading with concentrated mass	22
4.1 Simulate Cantilever Using 1 Elastic Beam Element	22
5 Elastic Beam, 27 Node Brick Model With Concentrated Mass	26
5.1 Simulate the cantilever by one 27NodeBrick element	26
6 Elastic Beam Element, Dynamic Loading, Viscous (Rayleigh/Caughey) and Numerical (Newmark/HHT) Damping	31
6.1 Single Beam Element	31
7 Elastic Beam Element for a Simple Frame Structure	37
8 27NodeBrick Cantilever Beam for the static load	39
9 4NodeANDES Cantilever Beams Under the Force Perpendicular to Plane	41
10 4NodeANDES Cantilever Beams under the In-Plane Force	43
11 27NodeBrick Cantilever Beams for Dynamic Input	44
12 4NodeANDES Square Plate with Four Edges Clamped	47
13 One Dimensional DRM Model	50
14 Three Dimensional DRM Model	55
15 ShearBeam Element for Pisano Materials	57
16 8NodeBrickLT Element for Drucker Prager Armstrong Federick Material	59
17 Beam theory	63
18 Verification of 8NodeBrick elements	65
18.1 Verification of 8NodeBrick cantilever beams	65
18.2 Verification of 8NodeBrick cantilever beam for different Poisson's ratio	67
18.3 Test of irregular shaped 8NodeBrick cantilever beams	72
18.4 Verification of 8NodeBrick stress in cantilever beams	80
18.5 Verification of 8NodeBrick square plate with four edges clamped	83
18.6 Verification of 8NodeBrick square plate with four edges simply supported	88

18.7 Verification of 8NodeBrick circular plate with all edges clamped	93
18.8 Verification of 8NodeBrick circular plate with all edges simply supported	98
19 Verification of 27NodeBrick elements	103
19.1 Verification of 27NodeBrick cantilever beams	103
19.2 Verification of 27NodeBrick cantilever beam for different Poisson's ratio	105
19.3 Test of irregular shaped 27NodeBrick cantilever beams	111
19.4 Verification of 27NodeBrick stress in cantilever beams	119
19.5 Verification of 27NodeBrick square plate with four edges clamped	122
19.6 Verification of 27NodeBrick square plate with four edges simply supported	127
19.7 Verification of 27NodeBrick circular plate with all edges clamped	132
19.8 Verification of 27NodeBrick circular plate with all edges simply supported	137
20 Verification of 4NodeANDES elements	142
20.1 Verification of 4NodeANDES cantilever beams	142
20.2 Verification of 4NodeANDES cantilever beam for different Poisson's ratio	146
20.3 Test of irregular shaped 4NodeANDES cantilever beams	155
20.4 Verification of 4NodeANDES square plate with four edges clamped	165
20.5 Verification of 4NodeANDES square plate with four edges simply supported	170
20.6 Verification of 4NodeANDES circular plate with all edges clamped	175
20.7 Verification of 4NodeANDES circular plate with all edges simply supported	180
21 G/Gmax plot by Drucker-Prager Armstrong Frederick	184
21.1 Plot stress-strain	184
21.2 Plot G/Gmax	185
21.3 Plot damping ratio	187
21.4 Appendix (Code)	189
21.4.1 Framework: A cpp main function to test the materials directly	189
21.4.2 The function to plot the results with 51 loops	193
21.4.3 The function to plot the G/Gmax with one loading only	196

1 Elastic Beam Element Under Static Loading

1.1 Simulate the cantilever by 1 elastic beam element

This is a simple beam example under static loading in three directions. The diagram below shows the loading in one bending direction.

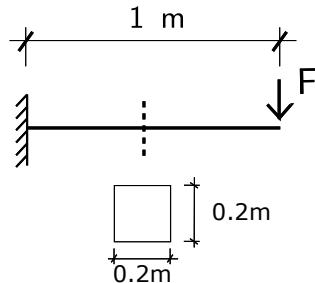


Figure 1: The cantilever model

```

ESSI model fei file:
1 model name "beam_1element" ;
2 // define the node coordinates
3 add node # 1 at ( 0.0*m , 0.0*m, 0.0*m) with 6 dofs;
4 add node # 2 at ( 1.0*m , 0.0*m, 0.0*m) with 6 dofs;
5 // Geometry: width and height. Help the beam definition.
6 b=0.2*m;
7 h=0.2*m;
8 I=b*h^3/12.0;
9 // define the beam element
10 add element # 1 type beam_elastic with nodes (1,2)
11   cross_section = b*h
12   elastic_modulus = 1e9*N/m^2
13   shear_modulus = 5e8*N/m^2
14   torsion_Jx = 0.33*b*h^3
15   bending_Iy = I
16   bending_Iz = I
17   mass_density = 0*kg/m^3
18   xz_plane_vector = ( 1, 0, 1)
19   joint_1_offset = (0*m, 0*m, 0*m)
20   joint_2_offset = (0*m, 0*m, 0*m);
21 // add boundary condition
22 fix node # 1 dofs all;
23 // axial loading
24 new loading stage "axial";
25 add load # 1 to node # 2 type linear Fx = 1*N;
26 define load factor increment 1;
27 define algorithm With_no_convergence_check ;
28 define solver ProfileSPD;
29 simulate 1 steps using static algorithm;
30 // bending in one direction
31 new loading stage "bending1";
32 remove load # 1;
33 add load # 2 to node # 2 type linear Fy = 1*N;
```

```
34 define load factor increment 1;
35 define algorithm With_no_convergence_check ;
36 define solver ProfileSPD;
37 simulate 1 steps using static algorithm;
38 // bending in the other direction
39 new loading stage "bending2";
40 remove load # 2;
41 add load # 3 to node # 2 type linear Fz = 1*N;
42 define load factor increment 1;
43 define algorithm With_no_convergence_check ;
44 define solver ProfileSPD;
45 simulate 1 steps using static algorithm;
46
47 bye;
```

The ESSI model fei files for this example can be downloaded [here](#).

2 Elastic Beam Element under Dynamic Loading

2.1 Cantilever, One Elastic Beam Element

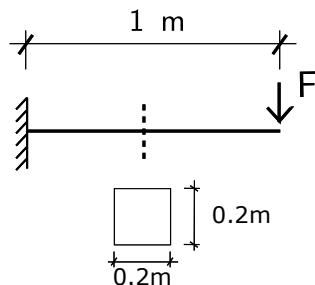


Figure 2: The cantilever model.

Problem description:

ESSI model fei file:

```

1 model name "beam_1element" ;
2
3 // add node
4 add node # 1 at ( 0.0*m , 0.0*m, 0.0*m) with 6 dofs;
5 add node # 2 at ( 1.0*m , 0.0*m, 0.0*m) with 6 dofs;
6   // Geometry: width and height
7 b=0.2*m;
8 h=0.2*m;
9 // Materials: properties
10 natural_period = 1*s;
11 natural_frequency = 2*pi/natural_period;
12 elastic_constant = 1e9*N/m^2;
13 I=b*h^3/12.0;
14 A=b*h;
15 L=1*m;
16 rho = (1.8751)^4*elastic_constant*I/(natural_frequency^2*L^4*A);
17 possion_ratio=0.3;
18 // add elements
19 add element # 1 type beam_elastic with nodes (1,2)
20   cross_section = b*h
21   elastic_modulus = elastic_constant
22   shear_modulus = elastic_constant/2/(1+possion_ratio)
23   torsion_Jx = 0.33*b*h^3
24   bending_Iy = b*h^3/12
25   bending_Iz = b*h^3/12
26   mass_density = rho
27   xz_plane_vector = ( 1, 0, 1)
28   joint_1_offset = (0*m, 0*m, 0*m)
29   joint_2_offset = (0*m, 0*m, 0*m);
30
31 // add boundary condition

```

```
32 fix node #      1 dofs all;
33
34 // // -----
35 // // --slowLoading-----
36 // // add load in 180 seconds. (Slow)
37 // // -----
38 // new loading stage "slowLoading";
39 // add load # 1 to node # 2 type path_time_series
40 // Fz = 1.*N
41 // series_file = "slowLoading.txt" ;
42 // define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
43 // define algorithm With_no_convergence_check ;
44 // define solver ProfileSPD;
45 // simulate 2000 steps using transient algorithm
46 // time_step = 0.1*s;
47
48 // // -----
49 // // --fastLoading-----
50 // // add load in 0.6 seconds (Fast)
51 // // -----
52 // remove load # 1;
53 // new loading stage "fastLoading";
54 // add load # 2 to node # 2 type path_time_series
55 // Fz = 1.*N
56 // series_file = "fastLoading.txt" ;
57 // define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
58 // define algorithm With_no_convergence_check ;
59 // define solver ProfileSPD;
60 // simulate 1000 steps using transient algorithm
61 // time_step = 0.01*s;
62
63 // // -----
64 // // --freeVibration-----
65 // // add a load and then release to free vibration
66 // // -----
67 // remove load # 2;
68 new loading stage "freeVibration";
69 add load # 3 to node # 2 type path_time_series
70   Fz = 1.*N
71   series_file = "freeVibration.txt" ;
72 define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
73 define algorithm With_no_convergence_check ;
74 define solver ProfileSPD;
75 simulate 2000 steps using transient algorithm
76   time_step = 0.01*s;
77
78 bye;
```

Displacement Results The ESSI model fei files for this example can be downloaded [here](#).

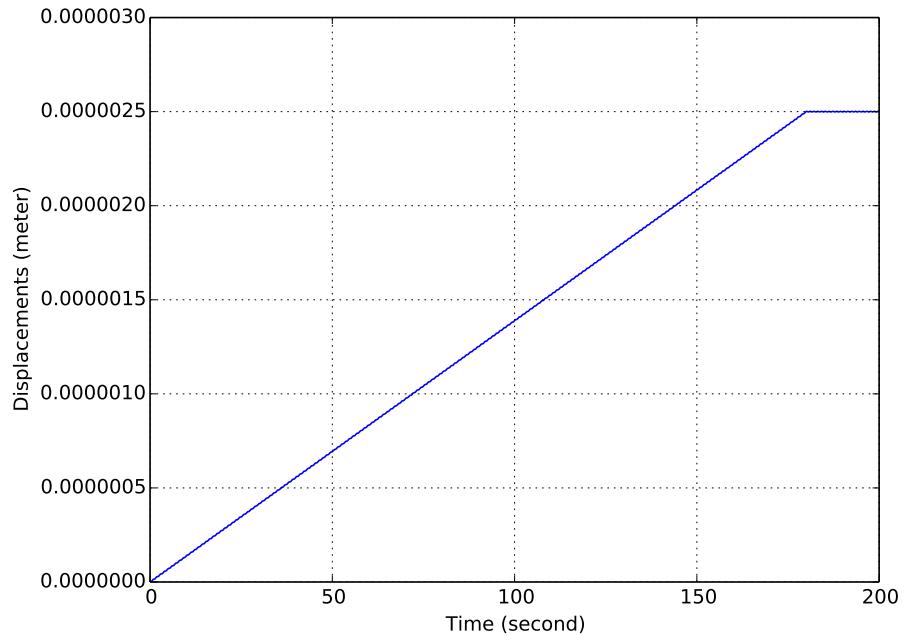


Figure 3: Slow loading condition, vertical displacements or the cantilever tip.

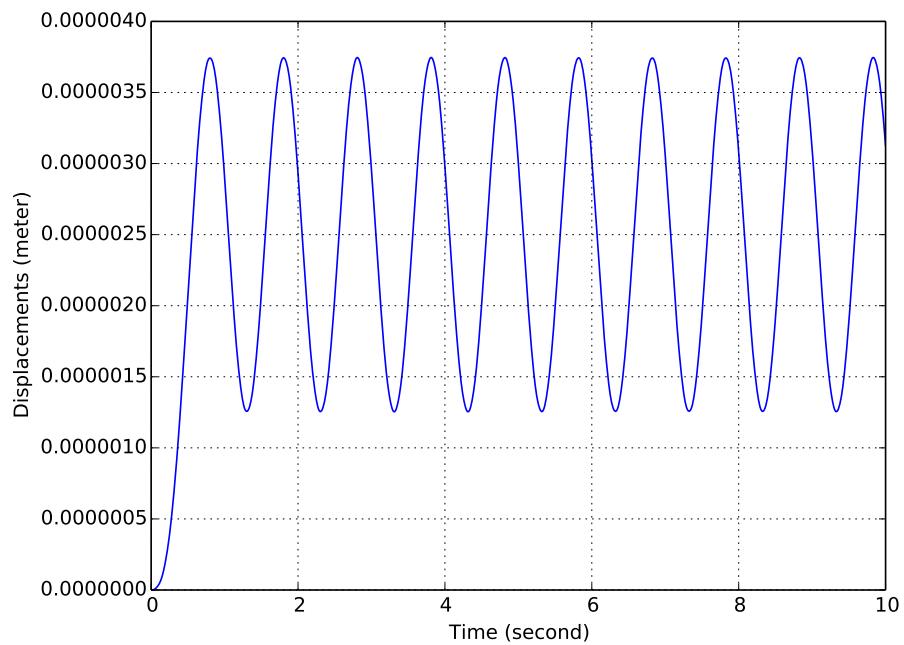


Figure 4: Fast loading condition, vertical displacements of the cantilever tip.

2.2 Cantilever, 5 Elastic Beam Elements

Problem description:

ESSI model fei file:

```
1 model name "beam_5element" ;
```

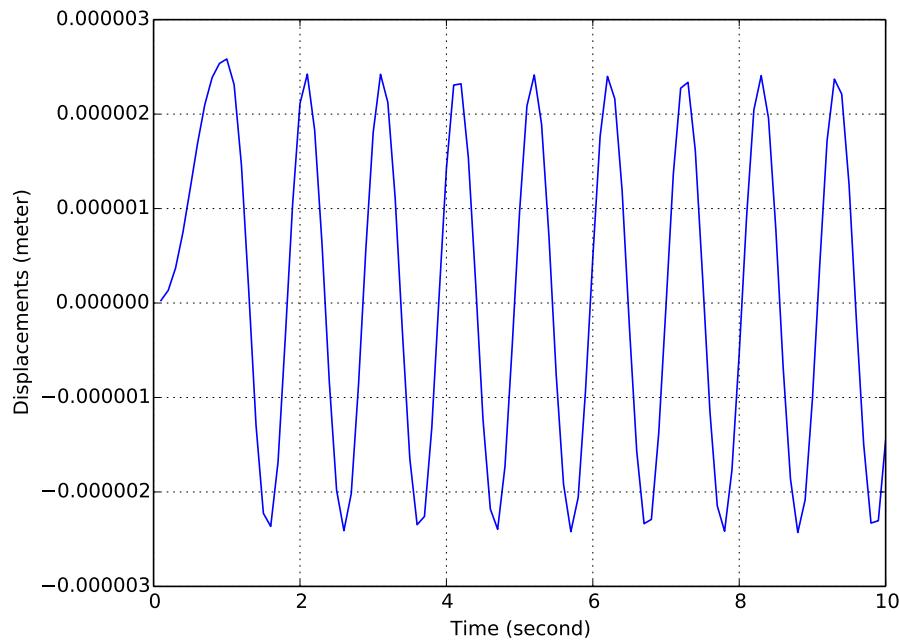


Figure 5: Free vibration, vertical displacements of the cantilever tip.

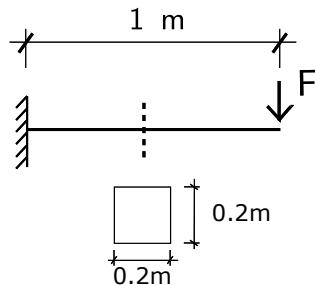


Figure 6: The cantilever model.

```

2
3 // add node
4 add node # 1 at ( 0.0*m , 0.0*m, 0.0*m) with 6 dofs;
5 add node # 2 at ( 0.2*m , 0.0*m, 0.0*m) with 6 dofs;
6 add node # 3 at ( 0.4*m , 0.0*m, 0.0*m) with 6 dofs;
7 add node # 4 at ( 0.6*m , 0.0*m, 0.0*m) with 6 dofs;
8 add node # 5 at ( 0.8*m , 0.0*m, 0.0*m) with 6 dofs;
9 add node # 6 at ( 1.0*m , 0.0*m, 0.0*m) with 6 dofs;
10
11 // Geometry: width and height
12 b=0.2*m;
13 h=0.2*m;
14
15 // Materials: properties
16 natural_period = 1*s;
17 natural_frequency = 2*pi/natural_period;
18 elastic_constant = 1e9*N/m^2;
19 I=b*h^3/12.0;

```

```

20 A=b*h;
21 L=1*m;
22 rho = (1.8751)^4*elastic_constant*I/(natural_frequency^2*L^4*A);
23 possion_ratio=0.3;
24
25 // Cross section geometry: width and height
26 b=0.2*m;
27 h=0.2*m;
28
29 // add elements
30 ii=1;
31 while (ii<6) {
32   add element # ii type beam_elastic with nodes (ii,ii+1)
33   cross_section = b*h
34   elastic_modulus = elastic_constant
35   shear_modulus = elastic_constant/2/(1+possion_ratio)
36   torsion_Jx = 0.33*b*h^3
37   bending_Iy = b*h^3/12
38   bending_Iz = b*h^3/12
39   mass_density = rho
40   xz_plane_vector = ( 1, 0, 1)
41   joint_1_offset = (0*m, 0*m, 0*m)
42   joint_2_offset = (0*m, 0*m, 0*m);
43   ii+=1;
44 }
45
46 // add boundary condition
47 fix node # 1 dofs all;
48
49 // // -----
50 // // --slowLoading-----
51 // // add load in 180 seconds.
52 // // -----
53 // new loading stage "slowLoading";
54 // add load # 1 to node # 6 type path_time_series
55 // Fz = 1.*N
56 // series_file = "slowLoading.txt" ;
57 // define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
58 // define algorithm With_no_convergence_check ;
59 // define solver ProfileSPD;
60 // simulate 2000 steps using transient algorithm
61 // time_step = 0.1*s;
62
63 // // -----
64 // // --fastLoading-----
65 // // add load in 0.6 seconds.
66 // // -----
67 // remove load # 1;
68 // new loading stage "fastLoading";
69 // add load # 2 to node # 6 type path_time_series
70 // Fz = 1.*N
71 // series_file = "fastLoading.txt" ;
72 // define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
73 // define algorithm With_no_convergence_check ;
74 // define solver ProfileSPD;

```

```

75 // simulate 1000 steps using transient algorithm
76 // time_step = 0.01*s;
77
78 // // -----
79 // // --freeVibration-
80 // // add a load and then release for free vibration
81 // // -----
82 // remove load # 2;
83 new loading stage "freeVibration";
84 add load # 3 to node # 6 type path_time_series
85   Fz = 1.*N
86   series_file = "freeVibration.txt" ;
87 define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
88 define algorithm With_no_convergence_check ;
89 define solver ProfileSPD;
90 simulate 100 steps using transient algorithm
91   time_step = 0.1*s;
92
93 bye;

```

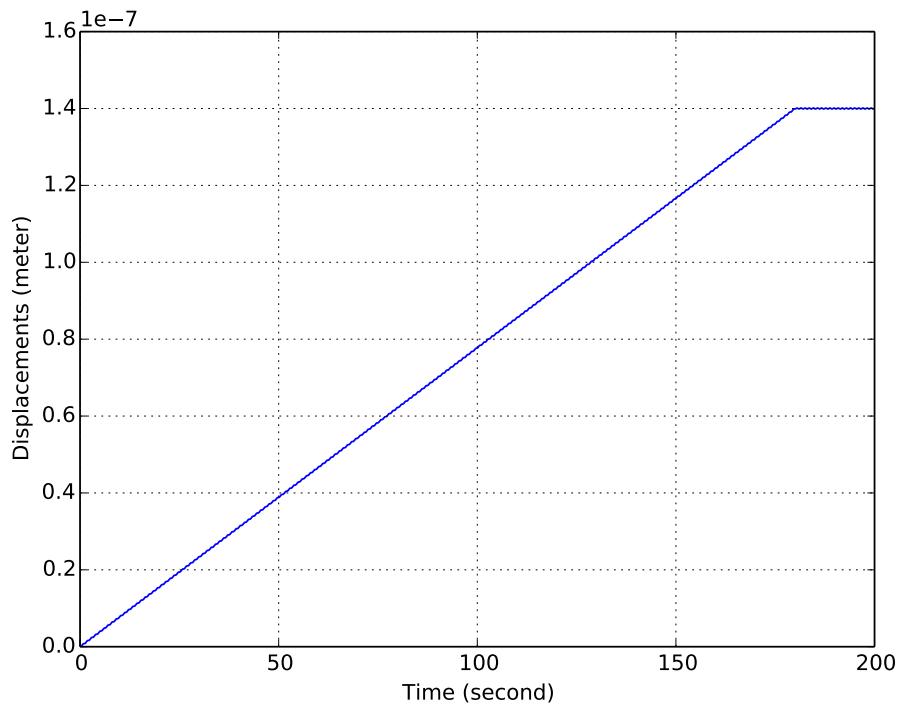


Figure 7: Slow loading condition, vertical displacements of the cantilever tip.

Displacement results The ESSI model fei files for this example can be downloaded [here](#).

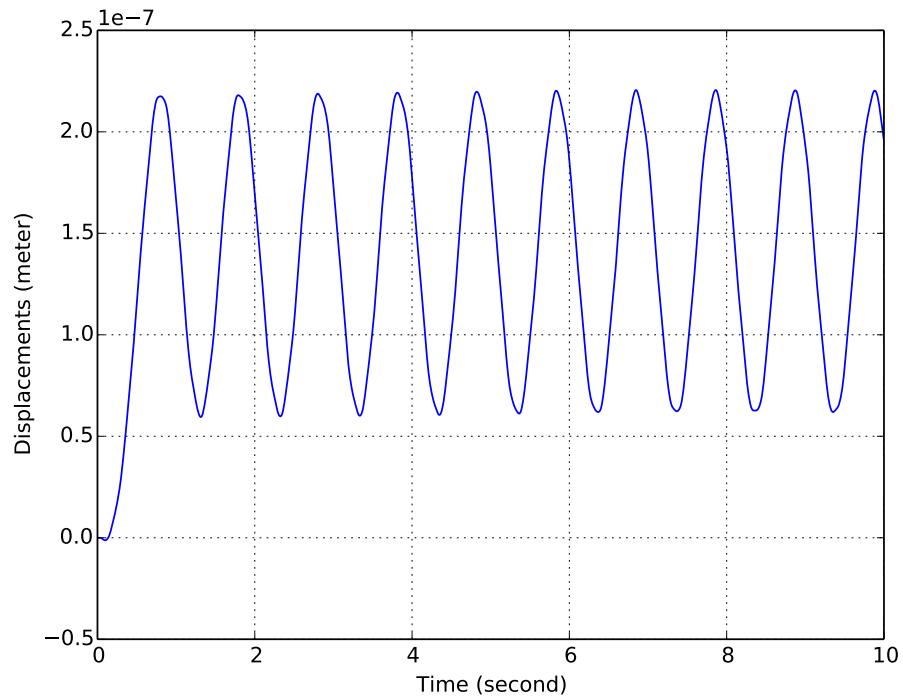


Figure 8: Fast loading condition, vertical displacements of the cantilever tip.

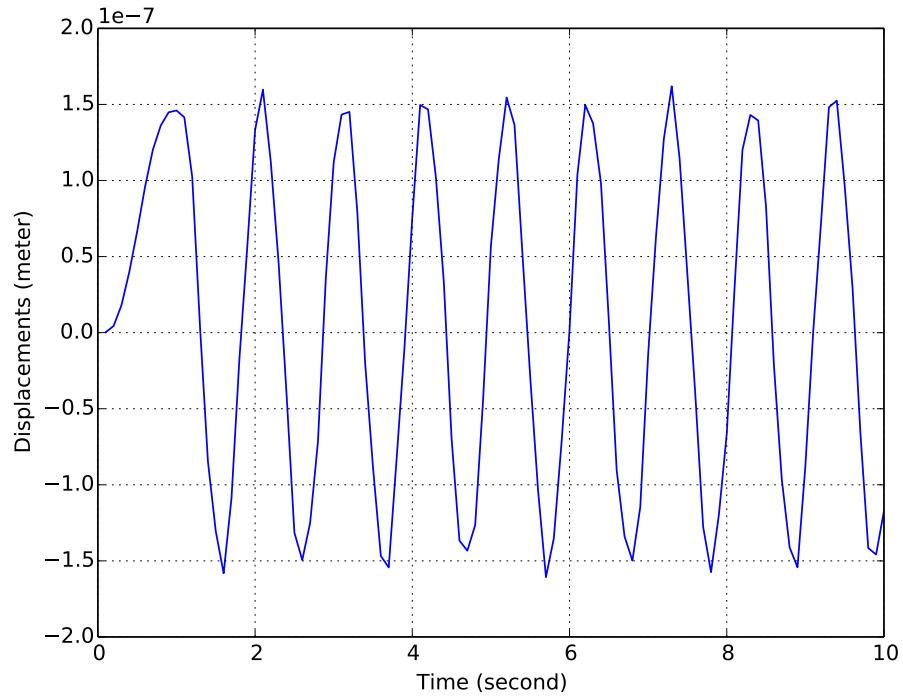


Figure 9: Free vibration condition, vertical displacements of the cantilever tip.

3 Cantilever, one 27 Node Brick Element, Dynamic Loading

3.1 Simulate the cantilever using one 27NodeBrick element

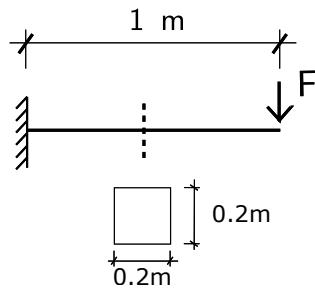


Figure 10: The cantilever model.

Problem description:

ESSI model fei file:

```

1 model name "brick_1element" ;
2
3 // Geometry: width and height
4 b=0.2*m;
5 h=0.2*m;
6
7 // Materials: properties
8 natural_period = 1*s;
9 natural_frequency = 2*pi/natural_period;
10 elastic_constant = 1e9*N/m^2;
11 I=b*h^3/12.0;
12 A=b*h;
13 L=1*m;
14 rho = (1.8751)^4*elastic_constant*I/(natural_frequency^2*L^4*A);
15 possion_ratio=0.3;
16
17
18 add material # 1 type linear_elastic_isotropic_3d_LT
19   mass_density = rho
20   elastic_modulus = elastic_constant
21   poisson_ratio = possion_ratio;
22
23 add node #      1 at ( 0.0000 *m, 0.2000 *m, 0.0000 *m) with 3 dofs;
24 add node #      2 at ( 0.0000 *m, 0.0000 *m, 0.0000 *m) with 3 dofs;
25 add node #      3 at ( 1.0000 *m, 0.2000 *m, 0.0000 *m) with 3 dofs;
26 add node #      4 at ( 1.0000 *m, 0.0000 *m, 0.0000 *m) with 3 dofs;
27 add node #      5 at ( 0.0000 *m, 0.0000 *m, 0.2000 *m) with 3 dofs;
28 add node #      6 at ( 1.0000 *m, 0.0000 *m, 0.2000 *m) with 3 dofs;
29 add node #      7 at ( 1.0000 *m, 0.2000 *m, 0.2000 *m) with 3 dofs;
30 add node #      8 at ( 0.0000 *m, 0.2000 *m, 0.2000 *m) with 3 dofs;
31 add node #      9 at ( 0.0000 *m, 0.1000 *m, 0.0000 *m) with 3 dofs;
32 add node #     10 at ( 0.5000 *m, 0.2000 *m, 0.0000 *m) with 3 dofs;
33 add node #     11 at ( 1.0000 *m, 0.1000 *m, 0.0000 *m) with 3 dofs;
```

```

34 | add node #    12 at ( 0.5000 *m, 0.0000 *m, 0.0000 *m) with 3 dofs;
35 | add node #    13 at ( 0.0000 *m, 0.1000 *m, 0.2000 *m) with 3 dofs;
36 | add node #    14 at ( 0.5000 *m, 0.2000 *m, 0.2000 *m) with 3 dofs;
37 | add node #    15 at ( 1.0000 *m, 0.1000 *m, 0.2000 *m) with 3 dofs;
38 | add node #    16 at ( 0.5000 *m, 0.0000 *m, 0.2000 *m) with 3 dofs;
39 | add node #    17 at ( 0.0000 *m, 0.0000 *m, 0.1000 *m) with 3 dofs;
40 | add node #    18 at ( 0.0000 *m, 0.2000 *m, 0.1000 *m) with 3 dofs;
41 | add node #    19 at ( 1.0000 *m, 0.2000 *m, 0.1000 *m) with 3 dofs;
42 | add node #    20 at ( 1.0000 *m, 0.0000 *m, 0.1000 *m) with 3 dofs;
43 | add node #    21 at ( 0.5000 *m, 0.1000 *m, 0.1000 *m) with 3 dofs;
44 | add node #    22 at ( 0.0000 *m, 0.1000 *m, 0.1000 *m) with 3 dofs;
45 | add node #    23 at ( 0.5000 *m, 0.2000 *m, 0.1000 *m) with 3 dofs;
46 | add node #    24 at ( 1.0000 *m, 0.1000 *m, 0.1000 *m) with 3 dofs;
47 | add node #    25 at ( 0.5000 *m, 0.0000 *m, 0.1000 *m) with 3 dofs;
48 | add node #    26 at ( 0.5000 *m, 0.1000 *m, 0.0000 *m) with 3 dofs;
49 | add node #    27 at ( 0.5000 *m, 0.1000 *m, 0.2000 *m) with 3 dofs;

50
51 | add element #      1 type 27NodeBrickLT with nodes( 2,           1,           3,
52 |                           8,           7,           6,           9,           10,          11,          12,          13,          14,          15,          16,
53 |                           17,          18,          19,          20,          21,          22,          23,          24,          25,          26,          27)
54 | use material #      1;

55
56 | fix node # 1 dofs all;
57 | fix node # 2 dofs all;
58 | fix node # 5 dofs all;
59 | fix node # 8 dofs all;
60 | fix node # 9 dofs all;
61 | fix node # 13 dofs all;
62 | fix node # 17 dofs all;
63 | fix node # 18 dofs all;
64 | fix node # 22 dofs all;

65 // // -----
66 // // --slowLoading-----
67 // new loading stage "slowLoading";
68 // add load # 1 to node # 4 type path_time_series Fz=1/36.0*N series_file = "slowLoading.txt"
69 // ;
70 // add load # 2 to node # 6 type path_time_series Fz=1/36.0*N series_file = "slowLoading.txt"
71 // ;
72 // add load # 3 to node # 3 type path_time_series Fz=1/36.0*N series_file = "slowLoading.txt"
73 // ;
74 // add load # 4 to node # 7 type path_time_series Fz=1/36.0*N series_file = "slowLoading.txt"
75 // ;
76 // add load # 5 to node # 20 type path_time_series Fz=1/9.0*N series_file = "slowLoading.txt"
77 // ;
78 // add load # 6 to node # 11 type path_time_series Fz=1/9.0*N series_file = "slowLoading.txt"
79 // ;
80 // add load # 7 to node # 15 type path_time_series Fz=1/9.0*N series_file = "slowLoading.txt"
81 // ;
82 // add load # 8 to node # 19 type path_time_series Fz=1/9.0*N series_file = "slowLoading.txt"
83 // ;
84 // add load # 9 to node # 24 type path_time_series Fz=4/9.0*N series_file = "slowLoading.txt"
85 // ;

```

```
77 // // add algorithm and solver
78 // define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
79 // define algorithm With_no_convergence_check ;
80 // define solver ProfileSPD;
81 // simulate 2000 steps using transient algorithm
82 // time_step = 0.1*s;
83
84 // // -----
85 // // --fastLoading
86 // // -----
87 // new loading stage "fastLoading";
88 // add load # 101 to node # 4 type path_time_series Fz=1/36.0*N series_file =
89 //   "fastLoading.txt" ;
90 // add load # 102 to node # 6 type path_time_series Fz=1/36.0*N series_file =
91 //   "fastLoading.txt" ;
92 // add load # 103 to node # 3 type path_time_series Fz=1/36.0*N series_file =
93 //   "fastLoading.txt" ;
94 // add load # 104 to node # 7 type path_time_series Fz=1/36.0*N series_file =
95 //   "fastLoading.txt" ;
96 // add load # 105 to node # 20 type path_time_series Fz=1/9.0*N series_file =
97 //   "fastLoading.txt" ;
98 // add load # 106 to node # 11 type path_time_series Fz=1/9.0*N series_file =
99 //   "fastLoading.txt" ;
100 // add load # 107 to node # 15 type path_time_series Fz=1/9.0*N series_file =
101 //   "fastLoading.txt" ;
102 // add load # 108 to node # 19 type path_time_series Fz=1/9.0*N series_file =
103 //   "fastLoading.txt" ;
104 // // add algorithm and solver
105 // define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
106 // define algorithm With_no_convergence_check ;
107 // define solver ProfileSPD;
108 // simulate 1000 steps using transient algorithm
109 // time_step = 0.01*s;
110
111 // // -----
112 // // --freeVibration
113 // // -----
114 new loading stage "freeVibration";
115 add load # 201 to node # 4 type path_time_series Fz=1/36.0*N series_file =
116 //   "freeVibration.txt" ;
117 add load # 202 to node # 6 type path_time_series Fz=1/36.0*N series_file =
118 //   "freeVibration.txt" ;
119 add load # 203 to node # 3 type path_time_series Fz=1/36.0*N series_file =
120 //   "freeVibration.txt" ;
121 add load # 204 to node # 7 type path_time_series Fz=1/36.0*N series_file =
122 //   "freeVibration.txt" ;
123 add load # 205 to node # 20 type path_time_series Fz=1/9.0*N series_file =
124 //   "freeVibration.txt" ;
125 add load # 206 to node # 11 type path_time_series Fz=1/9.0*N series_file =
126 //   "freeVibration.txt" ;
127 add load # 207 to node # 15 type path_time_series Fz=1/9.0*N series_file =
128 //   "freeVibration.txt" ;
129 add load # 208 to node # 19 type path_time_series Fz=1/9.0*N series_file =
```

```
116 "freeVibration.txt" ;
add load # 209 to node # 24 type path_time_series Fz=4/9.0*N series_file =
117 "freeVibration.txt" ;
// add algorithm and solver
118 define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
119 define algorithm With_no_convergence_check ;
120 define solver ProfileSPD;
121 simulate 10000 steps using transient algorithm
122 time_step = 0.001*s;
123
124 // end
125 bye;
```

Displacement results against time series

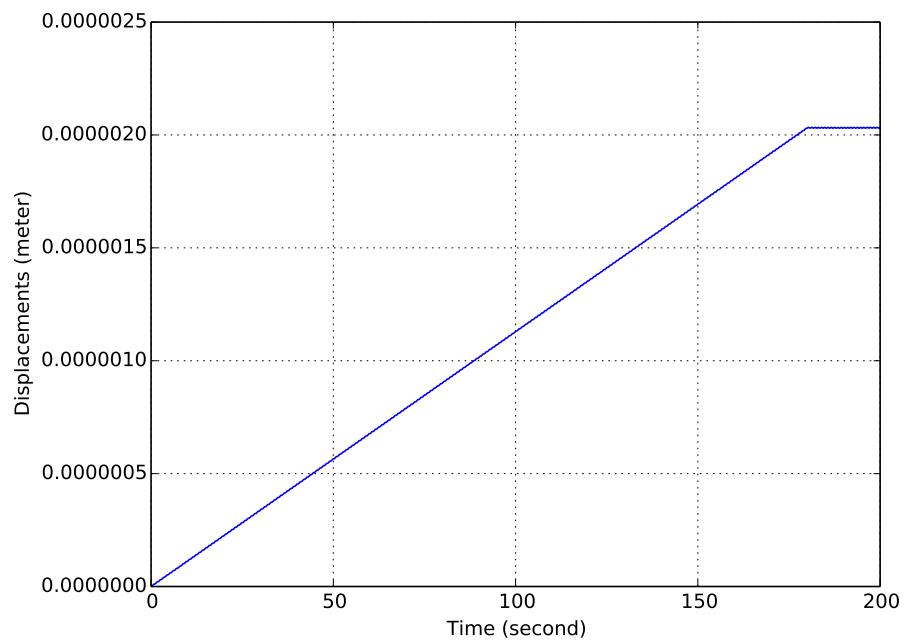


Figure 11: Slow loading condition, vertical displacements of the cantilever tip.

The ESSI model fei files for this example can be downloaded [here](#).

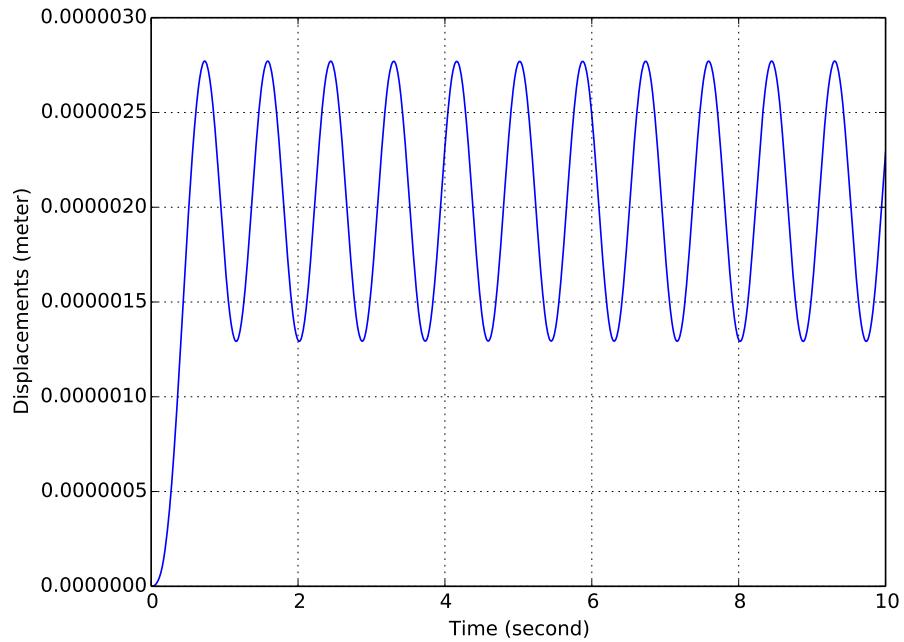


Figure 12: Fast loading condition, vertical displacements of the cantilever tip.

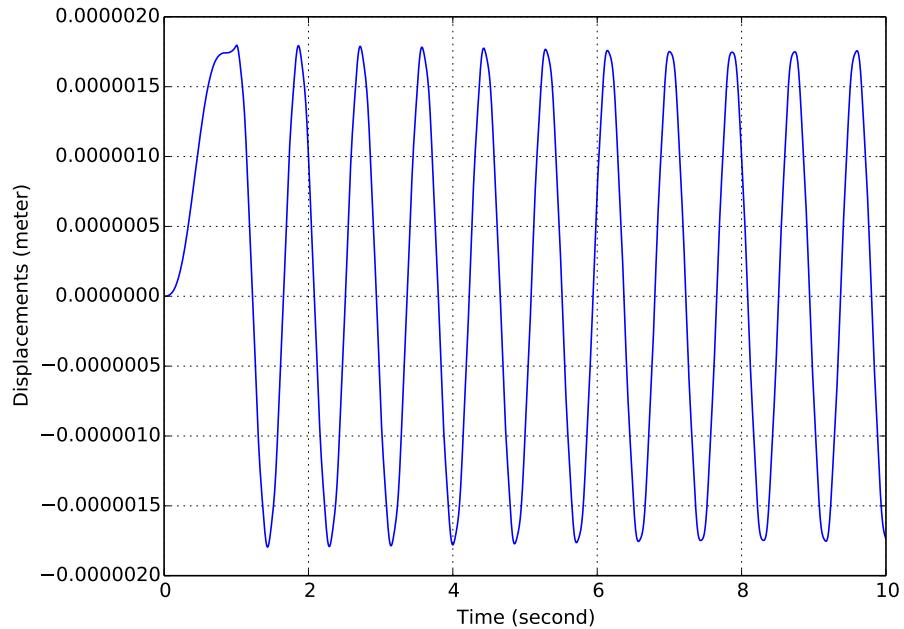


Figure 13: Free vibration condition, vertical displacements of the cantilever tip.

3.2 Simulate Cantilever Using five 27 Node Brick Elements

Problem description:

ESSI model fei file:

```
1 model name "brick_5element" ;
```

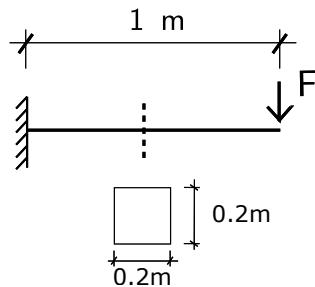


Figure 14: The cantilever model.

```

2
3 // Geometry: width and height
4 b=0.2*m;
5 h=0.2*m;
6
7 // Materials: properties
8 natural_period = 1*s;
9 natural_frequency = 2*pi/natural_period;
10 elastic_constant = 1e9*N/m^2;
11 I=b*h^3/12.0;
12 A=b*h;
13 L=1*m;
14 rho = (1.8751)^4*elastic_constant*I/(natural_frequency^2*L^4*A);
15 possion_ratio=0.3;
16
17
18 add material # 1 type linear_elastic_isotropic_3d_LT
19   mass_density = rho
20   elastic_modulus = elastic_constant
21   poisson_ratio = possion_ratio;
22
23 add node # 1 at (0.0*m, 0.0*m , 0.0*m) with 3 dofs;
24 add node # 2 at (0.1*m, 0.0*m , 0.0*m) with 3 dofs;
25 add node # 3 at (0.2*m, 0.0*m , 0.0*m) with 3 dofs;
26 add node # 4 at (0.0*m, 0.1*m , 0.0*m) with 3 dofs;
27 add node # 5 at (0.1*m, 0.1*m , 0.0*m) with 3 dofs;
28 add node # 6 at (0.2*m, 0.1*m , 0.0*m) with 3 dofs;
29 add node # 7 at (0.0*m, 0.2*m , 0.0*m) with 3 dofs;
30 add node # 8 at (0.1*m, 0.2*m , 0.0*m) with 3 dofs;
31 add node # 9 at (0.2*m, 0.2*m , 0.0*m) with 3 dofs;
32
33 fix node No 1 dofs ux uy uz;
34 fix node No 2 dofs ux uy uz;
35 fix node No 3 dofs ux uy uz;
36 fix node No 4 dofs ux uy uz;
37 fix node No 5 dofs ux uy uz;
38 fix node No 6 dofs ux uy uz;
39 fix node No 7 dofs ux uy uz;
40 fix node No 8 dofs ux uy uz;
41 fix node No 9 dofs ux uy uz;
42 e = 0;

```

```
43 hh = 0*m;
44 NBricks=5;
45 dz = 0.2*m;
46 while ( e < NBricks)
47 {
48     hh += dz;
49     add node # 10+18*e at (0.0*m, 0.0*m , hh - 0.5*dz) with 3 dofs;
50     add node # 11+18*e at (0.1*m, 0.0*m , hh - 0.5*dz) with 3 dofs;
51     add node # 12+18*e at (0.2*m, 0.0*m , hh - 0.5*dz) with 3 dofs;
52     add node # 13+18*e at (0.0*m, 0.1*m , hh - 0.5*dz) with 3 dofs;
53     add node # 14+18*e at (0.1*m, 0.1*m , hh - 0.5*dz) with 3 dofs;
54     add node # 15+18*e at (0.2*m, 0.1*m , hh - 0.5*dz) with 3 dofs;
55     add node # 16+18*e at (0.0*m, 0.2*m , hh - 0.5*dz) with 3 dofs;
56     add node # 17+18*e at (0.1*m, 0.2*m , hh - 0.5*dz) with 3 dofs;
57     add node # 18+18*e at (0.2*m, 0.2*m , hh - 0.5*dz) with 3 dofs;
58
59     add node # 19+18*e at (0.0*m, 0.0*m , hh) with 3 dofs;
60     add node # 20+18*e at (0.1*m, 0.0*m , hh) with 3 dofs;
61     add node # 21+18*e at (0.2*m, 0.0*m , hh) with 3 dofs;
62     add node # 22+18*e at (0.0*m, 0.1*m , hh) with 3 dofs;
63     add node # 23+18*e at (0.1*m, 0.1*m , hh) with 3 dofs;
64     add node # 24+18*e at (0.2*m, 0.1*m , hh) with 3 dofs;
65     add node # 25+18*e at (0.0*m, 0.2*m , hh) with 3 dofs;
66     add node # 26+18*e at (0.1*m, 0.2*m , hh) with 3 dofs;
67     add node # 27+18*e at (0.2*m, 0.2*m , hh) with 3 dofs;
68
69     add element # e+1 type 27NodeBrickLT with nodes
70     (
71         21+18*e,
72         27+18*e,
73         25+18*e,
74         19+18*e,
75
76         3+18*e,
77         9+18*e,
78         7+18*e,
79         1+18*e,
80
81         24+18*e,
82         26+18*e,
83         22+18*e,
84         20+18*e,
85
86         6+18*e,
87         8+18*e,
88         4+18*e,
89         2+18*e,
90
91         12+18*e,
92         18+18*e,
93         16+18*e,
94         10+18*e,
95
96         14+18*e,
97         15+18*e,
```

```
98     17+18*e,
99     13+18*e,
100    11+18*e,
101    23+18*e,
102    5+18*e
103 )
104 use material # 1;
105
106 e += 1;
107 };
108
109 e = e -1;
110
111
112 // // -----
113 // // --slowLoading-----
114 // // add the 1 Newton load in 180 seconds.
115 // // -----
116 // new loading stage "slowLoading";
117 // add load # 1 to node # (19+18*e) type path_time_series Fx=1/36.0*N series_file =
118 //   "slowLoading.txt";
119 // add load # 2 to node # (20+18*e) type path_time_series Fx=1/9.0*N series_file =
120 //   "slowLoading.txt";
121 // add load # 3 to node # (21+18*e) type path_time_series Fx=1/36.0*N series_file =
122 //   "slowLoading.txt";
123 // add load # 4 to node # (22+18*e) type path_time_series Fx=1/9.0*N series_file =
124 //   "slowLoading.txt";
125 // add load # 5 to node # (23+18*e) type path_time_series Fx=4/9.0*N series_file =
126 //   "slowLoading.txt";
127 // add load # 6 to node # (24+18*e) type path_time_series Fx=1/9.0*N series_file =
128 //   "slowLoading.txt";
129 // add load # 7 to node # (25+18*e) type path_time_series Fx=1/36.0*N series_file =
130 //   "slowLoading.txt";
131 // add load # 8 to node # (26+18*e) type path_time_series Fx=1/9.0*N series_file =
132 //   "slowLoading.txt";
133 // add load # 9 to node # (27+18*e) type path_time_series Fx=1/36.0*N series_file =
134 //   "slowLoading.txt";
135 // // add algorithm and solver
136 // define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
137 // define algorithm With_no_convergence_check ;
138 // define solver ProfileSPD;
139 // simulate 2000 steps using transient algorithm
140 // time_step = 0.1*s;
141
142 // // -----
143 // // --fastLoading-----
144 // // add the 1 Newton load in 0.6 seconds.
145 // // -----
146 // new loading stage "fastLoading";
147 // add load # 101 to node # (19+18*e) type path_time_series Fx=1/36.0*N series_file =
148 //   "fastLoading.txt" ;
149 // add load # 102 to node # (20+18*e) type path_time_series Fx=1/9.0*N series_file =
150 //   "fastLoading.txt" ;
151 // add load # 103 to node # (21+18*e) type path_time_series Fx=1/36.0*N series_file =
```

```
        "fastLoading.txt" ;
142 // add load # 104 to node # (22+18*e) type path_time_series Fx=1/9.0*N series_file =
143 //   "fastLoading.txt" ;
144 // add load # 105 to node # (23+18*e) type path_time_series Fx=4/9.0*N series_file =
145 //   "fastLoading.txt" ;
146 // add load # 106 to node # (24+18*e) type path_time_series Fx=1/9.0*N series_file =
147 //   "fastLoading.txt" ;
148 // add load # 107 to node # (25+18*e) type path_time_series Fx=1/36.0*N series_file =
149 //   "fastLoading.txt" ;
150 // add load # 108 to node # (26+18*e) type path_time_series Fx=1/9.0*N series_file =
151 //   "fastLoading.txt" ;
152 // add load # 109 to node # (27+18*e) type path_time_series Fx=1/36.0*N series_file =
153 //   "fastLoading.txt" ;
154 // // add algorithm and solver
155 // define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
156 // define algorithm With_no_convergence_check ;
157 // define solver ProfileSPD;
158 // simulate 1000 steps using transient algorithm
159 // time_step = 0.01*s;
160
161 // // -----
162 // --freeVibration-----
163 // // add a load and then release to free vibration
164 // // -----
165
166 new loading stage "freeVibration";
167 add load # 201 to node # (19+18*e) type path_time_series Fx=1/36.0*N series_file =
168 //   "freeVibration.txt" ;
169 add load # 202 to node # (20+18*e) type path_time_series Fx=1/9.0*N series_file =
170 //   "freeVibration.txt" ;
171 add load # 203 to node # (21+18*e) type path_time_series Fx=1/36.0*N series_file =
172 //   "freeVibration.txt" ;
173 add load # 204 to node # (22+18*e) type path_time_series Fx=1/9.0*N series_file =
174 //   "freeVibration.txt" ;
175 add load # 205 to node # (23+18*e) type path_time_series Fx=4/9.0*N series_file =
176 //   "freeVibration.txt" ;
177 add load # 206 to node # (24+18*e) type path_time_series Fx=1/9.0*N series_file =
178 //   "freeVibration.txt" ;
179 add load # 207 to node # (25+18*e) type path_time_series Fx=1/36.0*N series_file =
180 //   "freeVibration.txt" ;
181 add load # 208 to node # (26+18*e) type path_time_series Fx=1/9.0*N series_file =
182 //   "freeVibration.txt" ;
183 add load # 209 to node # (27+18*e) type path_time_series Fx=1/36.0*N series_file =
184 //   "freeVibration.txt" ;
185 // add algorithm and solver
186 define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
187 define algorithm With_no_convergence_check ;
188 define solver ProfileSPD;
189 simulate 100 steps using transient algorithm
190   time_step = 0.1*s;
191
192 // end
193 bye;
```

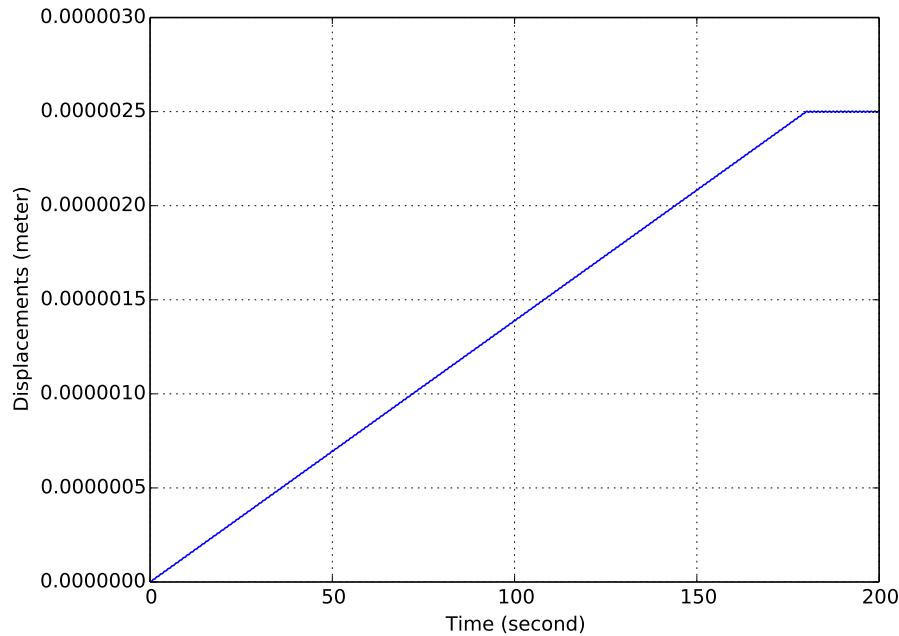


Figure 15: Slow loading condition, vertical displacements of the cantilever tip.

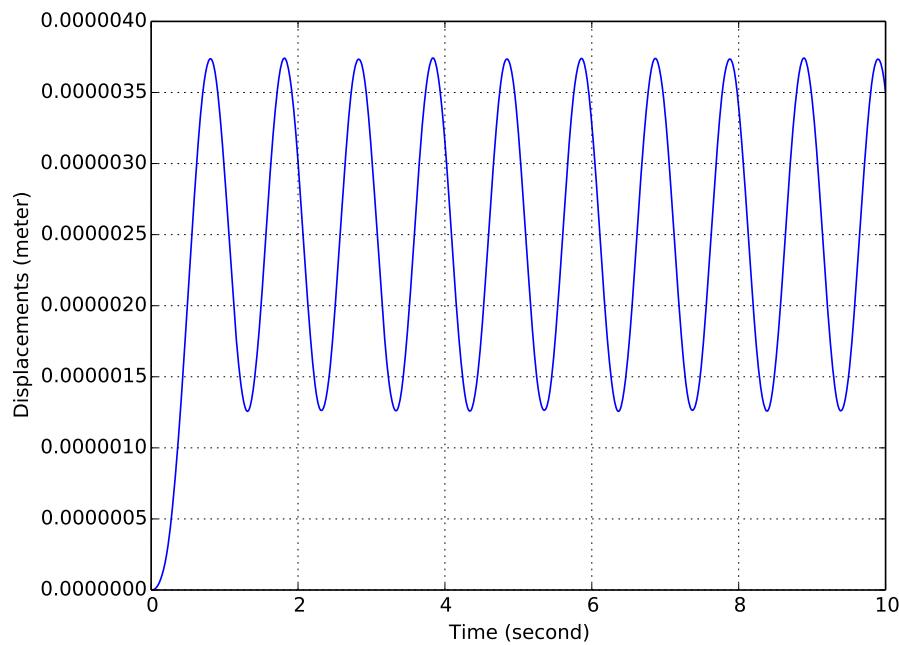


Figure 16: Fast loading condition, vertical displacements of the cantilever tip.

Displacement Results. The ESSI model fei files for this example can be downloaded [here](#).

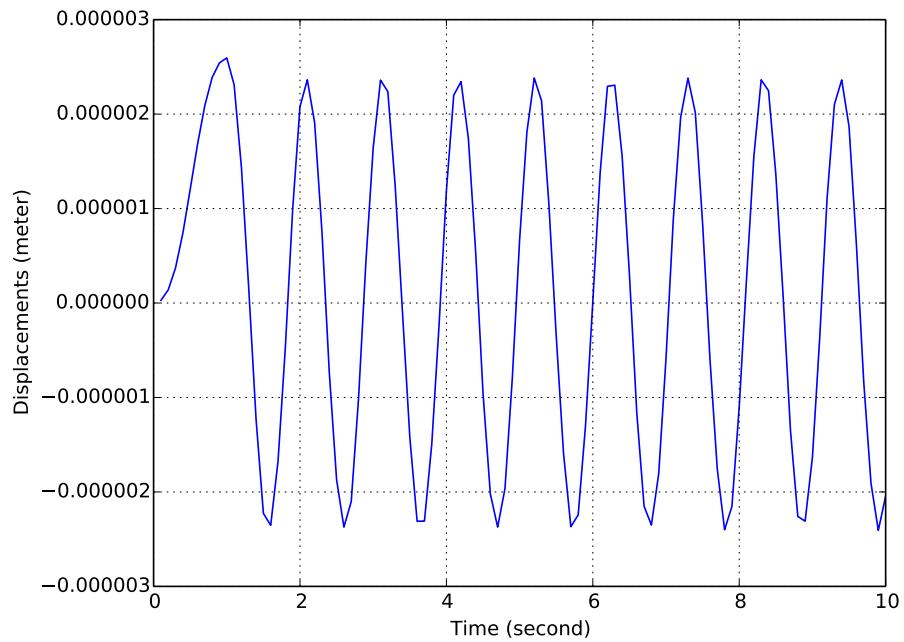


Figure 17: Free vibration condition, vertical displacements of the cantilever tip.

4 Elastic Beam Element under Dynamic Loading with concentrated mass

4.1 Simulate Cantilever Using 1 Elastic Beam Element

Problem description:

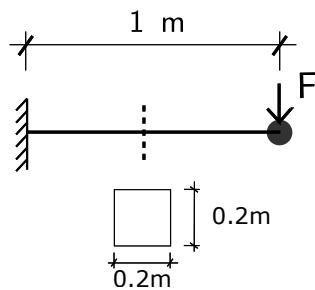


Figure 18: The cantilever-mass model.

ESSI model fei file:

```

1 model name "beam-mass_1element" ;
2
3 // add node
4 add node # 1 at ( 0.0*m , 0.0*m, 0.0*m) with 6 dofs;
5 add node # 2 at ( 1.0*m , 0.0*m, 0.0*m) with 6 dofs;

```

```
6 // Geometry: width and height
7 b=0.2*m;
8 h=0.2*m;
9
10 // Materials: properties
11 natural_period = 1*s;
12 natural_frequency = 2*pi/natural_period;
13 elastic_constant = 1e9*N/m^2;
14 I=b*h^3/12.0;
15 A=b*h;
16 L=1*m;
17 rho = (1.8751)^4*elastic_constant*I/(natural_frequency^2*L^4*A);
18 possion_ratio=0.3;
19
20 // add elements
21 add element # 1 type beam_elastic with nodes (1,2)
22   cross_section = b*h
23   elastic_modulus = elastic_constant
24   shear_modulus = elastic_constant/2/(1+possion_ratio)
25   torsion_Jx = 0.33*b*h^3
26   bending_Iy = b*h^3/12
27   bending_Iz = b*h^3/12
28   mass_density = rho
29   xz_plane_vector = ( 1, 0, 1)
30   joint_1_offset = (0*m, 0*m, 0*m)
31   joint_2_offset = (0*m, 0*m, 0*m);
32
33 // add boundary condition
34 fix node #     1 dofs all;
35
36 // add mass
37 beamMass=rho*A*L;
38 add mass to node # 2
39   mx = beamMass
40   my = beamMass
41   mz = beamMass
42   Imx = 0*beamMass*L^2
43   Imy = 0*beamMass*L^2
44   Imz = 0*beamMass*L^2;
45
46 // // -----
47 // // --slowLoading-
48 // // -----
49 // new loading stage "slowLoading";
50 // add load # 1 to node # 2 type path_time_series
51 // Fz = 1.*N
52 // series_file = "slowLoading.txt" ;
53 // define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
54 // define algorithm With_no_convergence_check ;
55 // define solver ProfileSPD;
56 // simulate 2000 steps using transient algorithm
57 // time_step = 0.1*s;
58
59 // // -----
60
```

```

61 // // --fastLoading-----
62 // // -----
63 // remove load # 1;
64 // new loading stage "fastLoading";
65 // add load # 2 to node # 2 type path_time_series
66 // Fz = 1.*N
67 // series_file = "fastLoading.txt" ;
68 // define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
69 // define algorithm With_no_convergence_check ;
70 // define solver ProfileSPD;
71 // simulate 1000 steps using transient algorithm
72 // time_step = 0.01*s;
73
74 // // -----
75 // // --freeVibration-----
76 // // -----
77 // remove load # 2;
78 new loading stage "freeVibration";
79 add load # 3 to node # 2 type path_time_series
80   Fz = 1.*N
81   series_file = "freeVibration.txt" ;
82 define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
83 define algorithm With_no_convergence_check ;
84 define solver ProfileSPD;
85 simulate 1000 steps using transient algorithm
86   time_step = 0.01*s;
87
88 bye;

```

Displacement results against time series

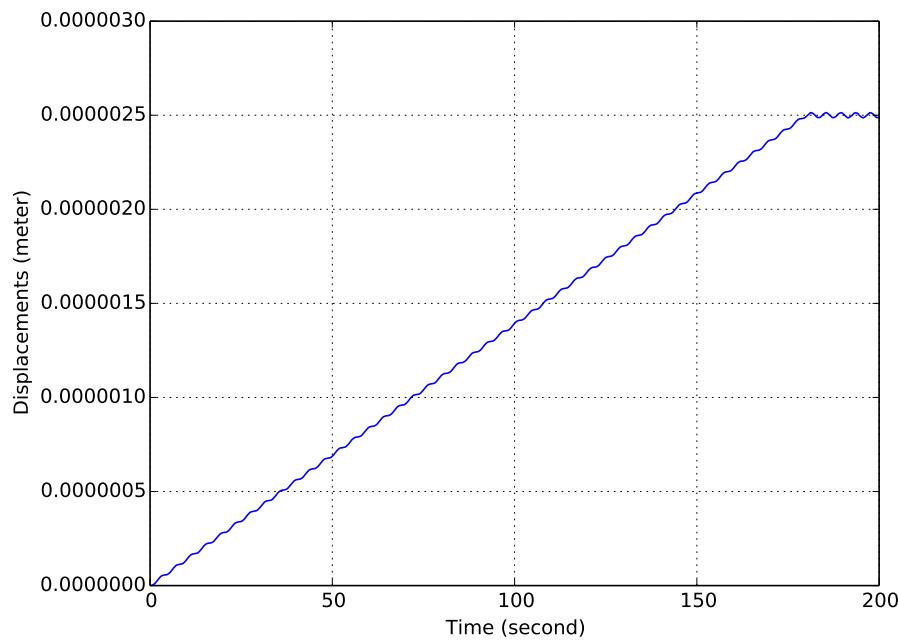


Figure 19: Slow loading condition, vertical displacements of the cantilever tip.

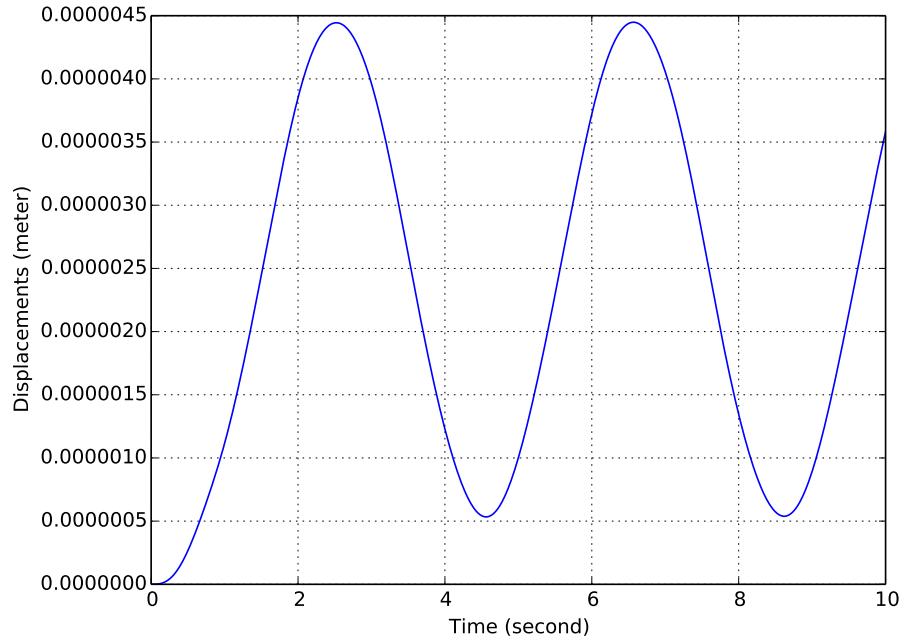


Figure 20: Fast loading condition, vertical displacements of the cantilever tip.

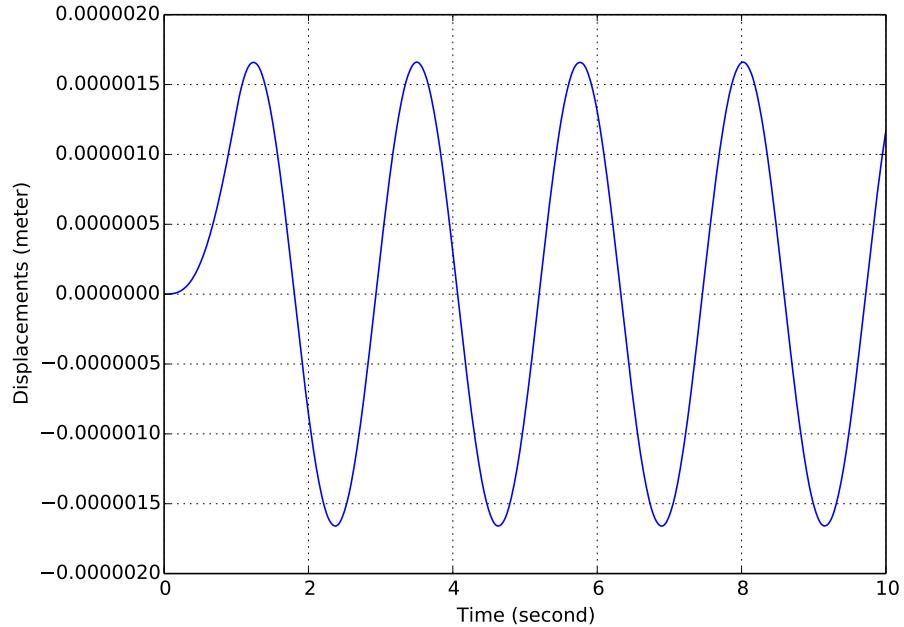


Figure 21: Free vibration condition, vertical displacements of the cantilever tip.

The ESSI model fei files for this example can be downloaded [here](#).

5 Elastic Beam, 27 Node Brick Model With Concentrated Mass

5.1 Simulate the cantilever by one 27NodeBrick element

Problem description:

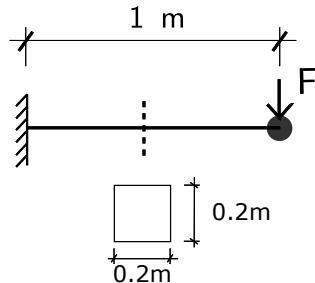


Figure 22: The cantilever-mass model.

ESSI model fei file:

```

1 model name "brick-mass_1element" ;
2
3 // Geometry: width and height
4 b=0.2*m;
5 h=0.2*m;
6
7 // Materials: properties
8 natural_period = 1*s;
9 natural_frequency = 2*pi/natural_period;
10 elastic_constant = 1e9*N/m^2;
11 I=b*h^3/12.0;
12 A=b*h;
13 L=1*m;
14 rho = (1.8751)^4*elastic_constant*I/(natural_frequency^2*L^4*A);
15 possion_ratio=0.3;
16
17 add material # 1 type linear_elastic_isotropic_3d_LT
18   mass_density = rho
19   elastic_modulus = elastic_constant
20   poisson_ratio = possion_ratio;
21
22 add node #      1 at ( 0.0000 *m, 0.2000 *m, 0.0000 *m) with 3 dofs;
23 add node #      2 at ( 0.0000 *m, 0.0000 *m, 0.0000 *m) with 3 dofs;
24 add node #      3 at ( 1.0000 *m, 0.2000 *m, 0.0000 *m) with 3 dofs;
25 add node #      4 at ( 1.0000 *m, 0.0000 *m, 0.0000 *m) with 3 dofs;
26 add node #      5 at ( 0.0000 *m, 0.0000 *m, 0.2000 *m) with 3 dofs;
27 add node #      6 at ( 1.0000 *m, 0.0000 *m, 0.2000 *m) with 3 dofs;
28 add node #      7 at ( 1.0000 *m, 0.2000 *m, 0.2000 *m) with 3 dofs;
29 add node #      8 at ( 0.0000 *m, 0.2000 *m, 0.2000 *m) with 3 dofs;
30 add node #      9 at ( 0.0000 *m, 0.1000 *m, 0.0000 *m) with 3 dofs;
31 add node #     10 at ( 0.5000 *m, 0.2000 *m, 0.0000 *m) with 3 dofs;
32 add node #     11 at ( 1.0000 *m, 0.1000 *m, 0.0000 *m) with 3 dofs;
33 add node #     12 at ( 0.5000 *m, 0.0000 *m, 0.0000 *m) with 3 dofs;
```

```

34 | add node #    13 at ( 0.0000 *m, 0.1000 *m, 0.2000 *m) with 3 dofs;
35 | add node #    14 at ( 0.5000 *m, 0.2000 *m, 0.2000 *m) with 3 dofs;
36 | add node #    15 at ( 1.0000 *m, 0.1000 *m, 0.2000 *m) with 3 dofs;
37 | add node #    16 at ( 0.5000 *m, 0.0000 *m, 0.2000 *m) with 3 dofs;
38 | add node #    17 at ( 0.0000 *m, 0.0000 *m, 0.1000 *m) with 3 dofs;
39 | add node #    18 at ( 0.0000 *m, 0.2000 *m, 0.1000 *m) with 3 dofs;
40 | add node #    19 at ( 1.0000 *m, 0.2000 *m, 0.1000 *m) with 3 dofs;
41 | add node #    20 at ( 1.0000 *m, 0.0000 *m, 0.1000 *m) with 3 dofs;
42 | add node #    21 at ( 0.5000 *m, 0.1000 *m, 0.1000 *m) with 3 dofs;
43 | add node #    22 at ( 0.0000 *m, 0.1000 *m, 0.1000 *m) with 3 dofs;
44 | add node #    23 at ( 0.5000 *m, 0.2000 *m, 0.1000 *m) with 3 dofs;
45 | add node #    24 at ( 1.0000 *m, 0.1000 *m, 0.1000 *m) with 3 dofs;
46 | add node #    25 at ( 0.5000 *m, 0.0000 *m, 0.1000 *m) with 3 dofs;
47 | add node #    26 at ( 0.5000 *m, 0.1000 *m, 0.0000 *m) with 3 dofs;
48 | add node #    27 at ( 0.5000 *m, 0.1000 *m, 0.2000 *m) with 3 dofs;
49 |
50 | add element #      1 type 27NodeBrickLT with nodes( 2,           1,           3,           4,           5,
51 |                           8,           7,           6,           9,          10,          11,          12,          13,          14,          15,          16,
52 |                           17,          18,          19,          20,          21,          22,          23,          24,          25,          26,          27)
53 |   use material #    1;
54 |
55 | fix node # 1 dofs all;
56 | fix node # 2 dofs all;
57 | fix node # 5 dofs all;
58 | fix node # 8 dofs all;
59 | fix node # 9 dofs all;
60 | fix node # 13 dofs all;
61 | fix node # 17 dofs all;
62 | fix node # 18 dofs all;
63 | fix node # 22 dofs all;
64 |
65 // Mapping from 3 dofs to 6 dofs.
66 | add node #    1003 at ( 1.0000 *m, 0.2000 *m, 0.0000 *m) with 6 dofs;
67 | add node #    1004 at ( 1.0000 *m, 0.0000 *m, 0.0000 *m) with 6 dofs;
68 | add node #    1006 at ( 1.0000 *m, 0.0000 *m, 0.2000 *m) with 6 dofs;
69 | add node #    1007 at ( 1.0000 *m, 0.2000 *m, 0.2000 *m) with 6 dofs;
70 |
71 // And connect the nodes at the same location.
72 | add constraint equal dof with master node # 3 and slave node # 1003 dof to constrain ux uy uz;
73 | add constraint equal dof with master node # 4 and slave node # 1004 dof to constrain ux uy uz;
74 | add constraint equal dof with master node # 6 and slave node # 1006 dof to constrain ux uy uz;
75 | add constraint equal dof with master node # 7 and slave node # 1007 dof to constrain ux uy uz;
76 |
77 | add mass to node # 24 mx = rho*A*L my = rho*A*L mz = rho*A*L;
78 |
79 // add 6 beams to connect the mass
80 smallb=0.01*m;
81 smallh=0.01*m;
82 smallE = 1e9*N/m^2;
83 smallnu=0.3;
84 smallrho=0*kg/m^3;
85 smallI=smallb*smallh^3/12.0;
86 | add element # 11 type beam_elastic with nodes (1003,1004)
87 |   cross_section = smallb*smallh
88 |   elastic_modulus = smallE

```

```
86 shear_modulus = smallE/2/(1+smallnu)
87 torsion_Jx = 0.33*smallb*smallh^3
88 bending_Iy = smallI
89 bending_Iz = smallI
90 mass_density = smallrho
91 xz_plane_vector = ( 1, 0, 1)
92 joint_1_offset = (0*m, 0*m, 0*m)
93 joint_2_offset = (0*m, 0*m, 0*m);
94 add element # 12 type beam_elastic with nodes (1003,1006)
95 cross_section = smallb*smallh
96 elastic_modulus = smallE
97 shear_modulus = smallE/2/(1+smallnu)
98 torsion_Jx = 0.33*smallb*smallh^3
99 bending_Iy = smallI
100 bending_Iz = smallI
101 mass_density = smallrho
102 xz_plane_vector = ( 1, 0, 1)
103 joint_1_offset = (0*m, 0*m, 0*m)
104 joint_2_offset = (0*m, 0*m, 0*m);
105 add element # 13 type beam_elastic with nodes (1003,1007)
106 cross_section = smallb*smallh
107 elastic_modulus = smallE
108 shear_modulus = smallE/2/(1+smallnu)
109 torsion_Jx = 0.33*smallb*smallh^3
110 bending_Iy = smallI
111 bending_Iz = smallI
112 mass_density = smallrho
113 xz_plane_vector = ( 1, 0, 1)
114 joint_1_offset = (0*m, 0*m, 0*m)
115 joint_2_offset = (0*m, 0*m, 0*m);
116 add element # 14 type beam_elastic with nodes (1004,1006)
117 cross_section = smallb*smallh
118 elastic_modulus = smallE
119 shear_modulus = smallE/2/(1+smallnu)
120 torsion_Jx = 0.33*smallb*smallh^3
121 bending_Iy = smallI
122 bending_Iz = smallI
123 mass_density = smallrho
124 xz_plane_vector = ( 1, 0, 1)
125 joint_1_offset = (0*m, 0*m, 0*m)
126 joint_2_offset = (0*m, 0*m, 0*m);
127 add element # 15 type beam_elastic with nodes (1004,1007)
128 cross_section = smallb*smallh
129 elastic_modulus = smallE
130 shear_modulus = smallE/2/(1+smallnu)
131 torsion_Jx = 0.33*smallb*smallh^3
132 bending_Iy = smallI
133 bending_Iz = smallI
134 mass_density = smallrho
135 xz_plane_vector = ( 1, 0, 1)
136 joint_1_offset = (0*m, 0*m, 0*m)
137 joint_2_offset = (0*m, 0*m, 0*m);
138 add element # 16 type beam_elastic with nodes (1006,1007)
139 cross_section = smallb*smallh
140 elastic_modulus = smallE
```

```
141 shear_modulus = smallE/2/(1+smallnu)
142 torsion_Jx = 0.33*smallb*smallh^3
143 bending_Iy = smallI
144 bending_Iz = smallI
145 mass_density = smallrho
146 xz_plane_vector = ( 1, 0, 1)
147 joint_1_offset = (0*m, 0*m, 0*m)
148 joint_2_offset = (0*m, 0*m, 0*m);
149
150
151 // // -----
152 // // --slowLoading-----
153 // // add the 1 Newton load in 180 seconds.
154 // //
155 // new loading stage "slowLoading";
156 // add load # 1 to node # 4 type path_time_series Fz=1/36.0*N series_file = "slowLoading.txt"
157 // ;
158 // add load # 2 to node # 6 type path_time_series Fz=1/36.0*N series_file = "slowLoading.txt"
159 // ;
160 // add load # 3 to node # 3 type path_time_series Fz=1/36.0*N series_file = "slowLoading.txt"
161 // ;
162 // add load # 4 to node # 7 type path_time_series Fz=1/36.0*N series_file = "slowLoading.txt"
163 // ;
164 // add load # 5 to node # 20 type path_time_series Fz=1/9.0*N series_file = "slowLoading.txt"
165 // ;
166 // add load # 6 to node # 11 type path_time_series Fz=1/9.0*N series_file = "slowLoading.txt"
167 // ;
168 // add load # 7 to node # 15 type path_time_series Fz=1/9.0*N series_file = "slowLoading.txt"
169 // ;
170 // add load # 8 to node # 19 type path_time_series Fz=1/9.0*N series_file = "slowLoading.txt"
171 // ;
172 // // add algorithm and solver
173 // define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
174 // define algorithm With_no_convergence_check ;
175 // define solver ProfileSPD;
176 // simulate 2000 steps using transient algorithm
177 // time_step = 0.1*s;
178
179
180 // // -----
181 // // --fastLoading-----
182 // // add the 1 Newton load in 0.6 seconds.
183 // //
184 // new loading stage "fastLoading";
185 // add load # 101 to node # 4 type path_time_series Fz=1/36.0*N series_file =
186 // "fastLoading.txt" ;
187 // add load # 102 to node # 6 type path_time_series Fz=1/36.0*N series_file =
188 // "fastLoading.txt" ;
189 // add load # 103 to node # 3 type path_time_series Fz=1/36.0*N series_file =
190 // "fastLoading.txt" ;
191 // add load # 104 to node # 7 type path_time_series Fz=1/36.0*N series_file =
192 // "fastLoading.txt" ;
193 // add load # 105 to node # 20 type path_time_series Fz=1/9.0*N series_file =
194 // "fastLoading.txt" ;
```

```

182 // add load # 106 to node # 11 type path_time_series Fz=1/9.0*N series_file =
183 // "fastLoading.txt" ;
184 // add load # 107 to node # 15 type path_time_series Fz=1/9.0*N series_file =
185 // "fastLoading.txt" ;
186 // add load # 108 to node # 19 type path_time_series Fz=1/9.0*N series_file =
187 // "fastLoading.txt" ;
188 // add load # 109 to node # 24 type path_time_series Fz=4/9.0*N series_file =
189 // "fastLoading.txt" ;
190 // // add algorithm and solver
191 // define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
192 // define algorithm With_no_convergence_check ;
193 // define solver ProfileSPD;
194 // simulate 1000 steps using transient algorithm
195 // time_step = 0.01*s;
196
197 // // -----
198 // // --freeVibration-
199 // // -----
200 new loading stage "freeVibration";
201 add load # 201 to node # 4 type path_time_series Fz=1/36.0*N series_file =
202 // "freeVibration.txt" ;
203 add load # 202 to node # 6 type path_time_series Fz=1/36.0*N series_file =
204 // "freeVibration.txt" ;
205 add load # 203 to node # 3 type path_time_series Fz=1/36.0*N series_file =
206 // "freeVibration.txt" ;
207 add load # 204 to node # 7 type path_time_series Fz=1/36.0*N series_file =
208 // "freeVibration.txt" ;
209 add load # 205 to node # 20 type path_time_series Fz=1/9.0*N series_file =
210 // "freeVibration.txt" ;
211 add load # 206 to node # 11 type path_time_series Fz=1/9.0*N series_file =
212 // "freeVibration.txt" ;
213 add load # 207 to node # 15 type path_time_series Fz=1/9.0*N series_file =
214 // "freeVibration.txt" ;
215 add load # 208 to node # 19 type path_time_series Fz=1/9.0*N series_file =
216 // "freeVibration.txt" ;
217 add load # 209 to node # 24 type path_time_series Fz=4/9.0*N series_file =
218 // "freeVibration.txt" ;
219 // add algorithm and solver
220 define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
221 define algorithm With_no_convergence_check ;
222 define solver ProfileSPD;
223 simulate 100 steps using transient algorithm
224 time_step = 0.1*s;
225
226 // end
227 bye;

```

Displacement Results. The ESSI model fei files for this example can be downloaded [here](#).

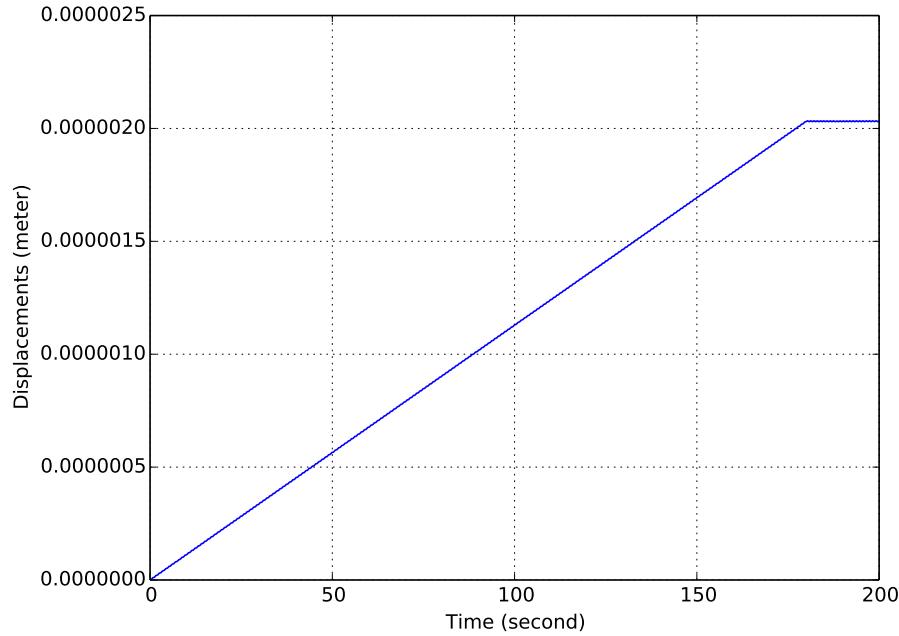


Figure 23: Slow loading condition, vertical displacements of the cantilever tip.

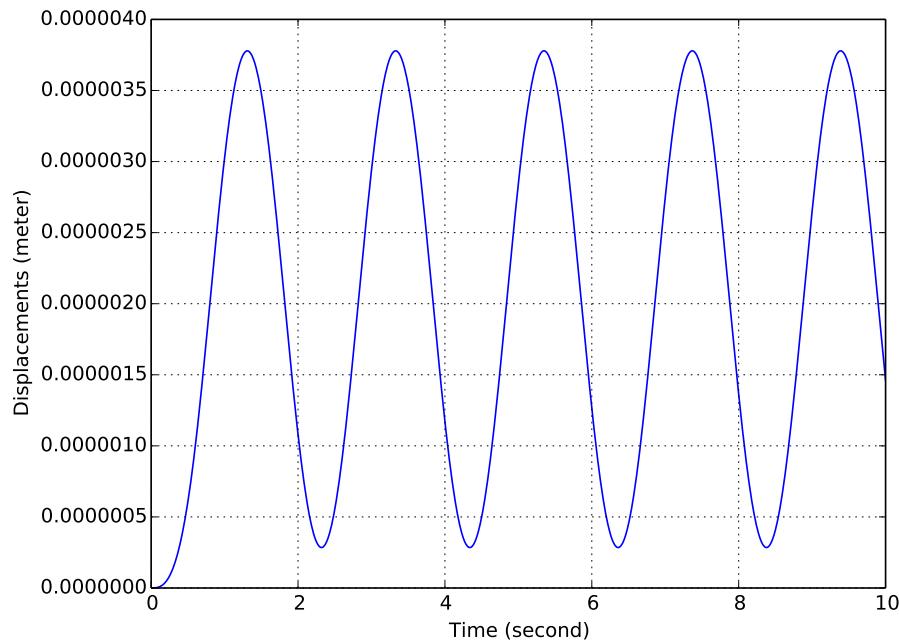


Figure 24: Fast loading condition, vertical displacements of the cantilever tip.

6 Elastic Beam Element, Dynamic Loading, Viscous (Rayleigh/Caughey) and Numerical (Newmark/HHT) Damping

6.1 Single Beam Element

Problem description:

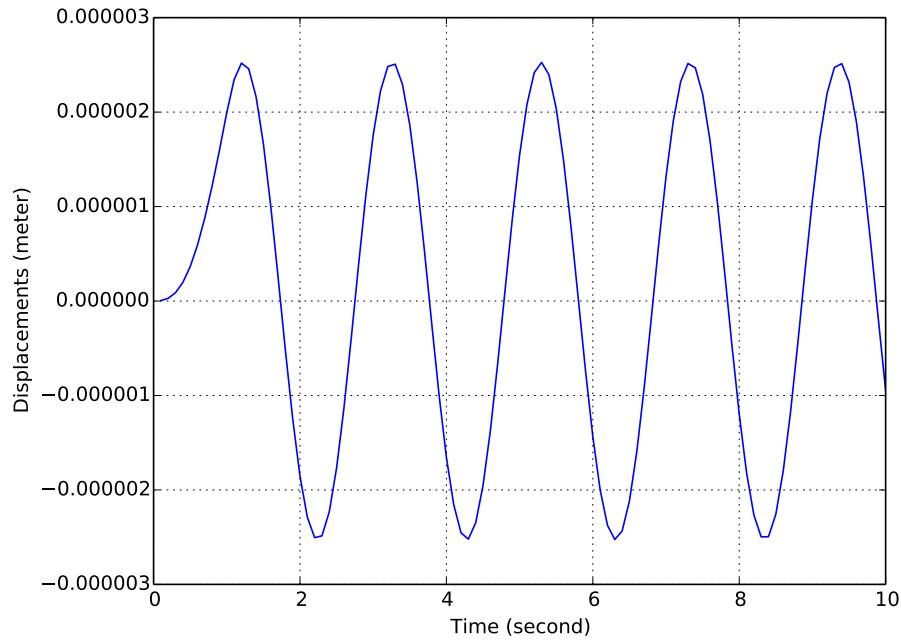


Figure 25: Free vibration condition, vertical displacements of the cantilever tip.

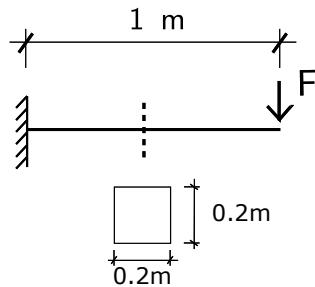


Figure 26: The cantilever-mass model.

ESSI model fei file:

```

1 model name "beam_1element" ;
2
3 // add node
4 add node # 1 at ( 0.0*m , 0.0*m, 0.0*m) with 6 dofs;
5 add node # 2 at ( 1.0*m , 0.0*m, 0.0*m) with 6 dofs;
6
7 // Geometry: width and height
8 b=0.2*m;
9 h=0.2*m;
10
11 // Materials: properties
12 natural_period = 1*s;
13 natural_frequency = 2*pi/natural_period;
14 elastic_constant = 1e9*N/m^2;
15 I=b*h^3/12.0;
16 A=b*h;

```

```

17 L=1*m;
18 rho = (1.8751)^4*elastic_constant*I/(natural_frequency^2*L^4*A);
19 possion_ratio=0.3;
20
21 // add elements
22 add element # 1 type beam_elastic with nodes (1,2)
23 cross_section = b*h
24 elastic_modulus = elastic_constant
25 shear_modulus = elastic_constant/2/(1+possion_ratio)
26 torsion_Jx = 0.33*b*h^3
27 bending_Iy = b*h^3/12
28 bending_Iz = b*h^3/12
29 mass_density = rho
30 xz_plane_vector = ( 1, 0, 1)
31 joint_1_offset = (0*m, 0*m, 0*m)
32 joint_2_offset = (0*m, 0*m, 0*m);
33
34 // add boundary condition
35 fix node # 1 dofs all;
36
37 // // -----
38 // // --no-damping-
39 // // -----
40 // new loading stage "no-damping";
41 // add load # 1 to node # 2 type path_time_series
42 // Fz = 1.*N
43 // series_file = "freeVibration.txt" ;
44 // define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
45 // define algorithm With_no_convergence_check ;
46 // define solver ProfileSPD;
47 // simulate 100 steps using transient algorithm
48 // time_step = 0.1*s;
49
50 // // -----
51 // // --Newmark-damping-
52 // // -----
53 // remove load # 2;
54 // new loading stage "Newmark-damping";
55 // add load # 3 to node # 2 type path_time_series
56 // Fz = 1.*N
57 // series_file = "freeVibration.txt" ;
58 // define dynamic integrator Newmark with gamma = 0.6 beta = 0.3025;
59 // define algorithm With_no_convergence_check ;
60 // define solver ProfileSPD;
61 // simulate 100 steps using transient algorithm
62 // time_step = 0.1*s;
63
64 // // -----
65 // // --HHT-damping-
66 // // -----
67 // remove load # 3;
68 // new loading stage "HHT-damping";
69 // add load # 4 to node # 6 type path_time_series
70 // Fz = 1.*kN
71 // series_file = "freeVibration.txt" ;
// define dynamic integrator Hilber_Hughes_Taylor with alpha = -0.20;

```

```
72 // define algorithm With_no_convergence_check ;
73 // define solver ProfileSPD;
74 // simulate 300 steps using transient algorithm
75 // time_step = 0.1*s;
76 // // -----
77 // // --Rayleigh-damping-----
78 // // -----
79 // remove load # 4;
80 // simulate using eigen algorithm number_of_modes = 2;
81 f1=0.996807/s;
82 f2=0.996807/s;
83 w1 = 2*pi*f1;
84 w2 = 2*pi*f2;
85 xi=0.05;
86 rayl_a1 = 2*xi/(w1 + w2);
87 rayl_a0 = rayl_a1*w1*w2;
88
89 add damping # 1 type Rayleigh with
90   a0 = rayl_a0
91   a1 = rayl_a1
92   stiffness_to_use = Initial_Stiffness;
93 add damping # 1 to element # 1;
94
95 new loading stage "Rayleigh-damping";
96 add load # 5 to node # 2 type path_time_series
97   Fz = 1.*N
98   series_file = "freeVibration.txt" ;
99 define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
100 define algorithm With_no_convergence_check ;
101 define solver ProfileSPD;
102 simulate 100 steps using transient algorithm
103   time_step = 0.1*s;
104
105 // // -----
106 // // --Caughey3rd-damping-----
107 // // -----
108 // add damping # 2 type Caughey3rd with
109 //   a0 = 0.560523/s
110 //   a1 = 0.0730746*s
111 //   a2 = 0.000361559*s^3
112 //   stiffness_to_use = Last_Committed_Stiffness;
113 // kk=1;
114 // while (kk<6) {
115 //   add damping # 2 to element # kk;
116 //   kk+=1;
117 // }
118 // new loading stage "Caughey3rd-damping";
119 // add load # 6 to node # 6 type path_time_series
120 //   Fz = 10.*kN
121 //   series_file = "freeVibration.txt" ;
122 // define dynamic integrator Newmark with gamma = 0.6 beta = 0.3025;
123 // define algorithm With_no_convergence_check ;
124 // define solver ProfileSPD;
125 // simulate 100 steps using transient algorithm
126 //   time_step = 0.2*s;
```

```

127
128
129 // // -----
130 // // --Caughey4th-damping-
131 // // -----
132 // add damping # 2 type Caughey4th with
133 // a0 = 0.560523/s
134 // a1 = 0.0756472*s
135 // a2 = 0.000517195*s^3
136 // a3 = 1.20005*10^(-6)*s^5
137 // stiffness_to_use = Last_Committed_Stiffness;
138 // kk=1;
139 // while (kk<6) {
140 //   add damping # 2 to element # kk;
141 //   kk+=1;
142 }
143 // new loading stage "Caughey4th-damping";
144 // add load # 6 to node # 6 type path_time_series
145 // Fz = 10.*kN
146 // series_file = "freeVibration.txt" ;
147 // define dynamic integrator Newmark with gamma = 0.6 beta = 0.3025;
148 // define algorithm With_no_convergence_check ;
149 // define solver ProfileSPD;
150 // simulate 100 steps using transient algorithm
151 // time_step = 0.2*s;
152
153 bye;

```

Displacement results against time series

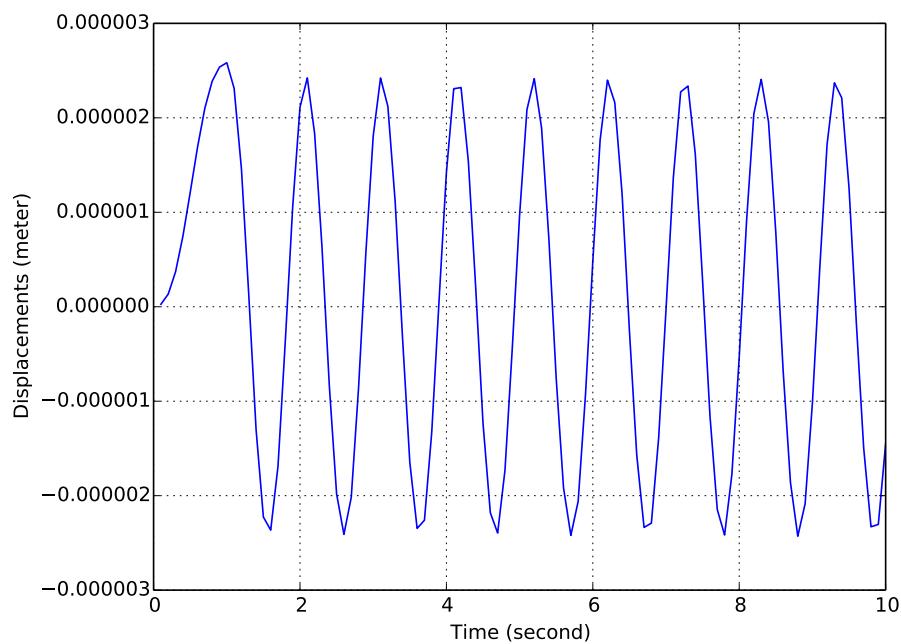


Figure 27: Free vibration condition, no damping, vertical displacements of the cantilever tip.

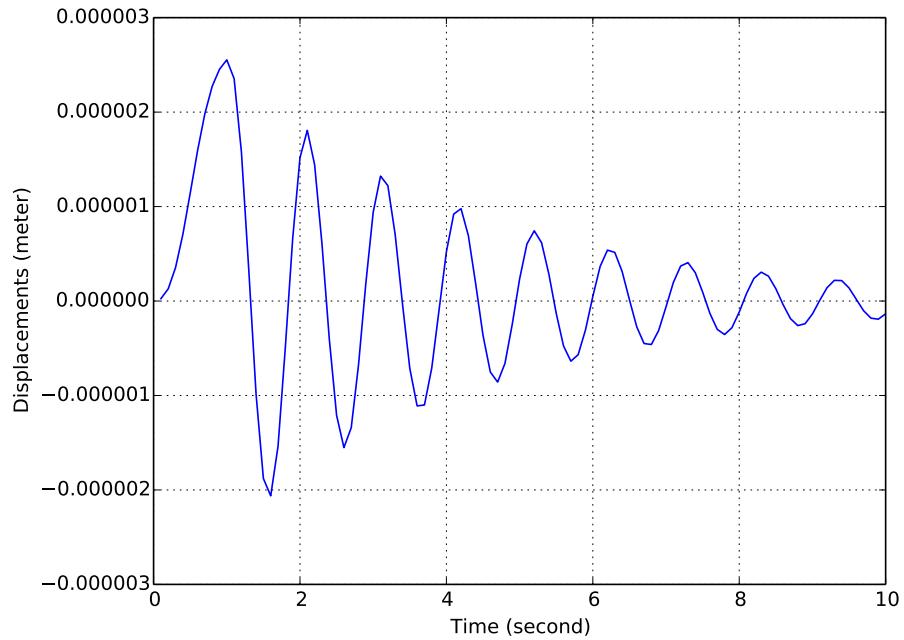


Figure 28: Free vibration condition, viscous (Rayleigh) damping, vertical displacements of the cantilever tip.

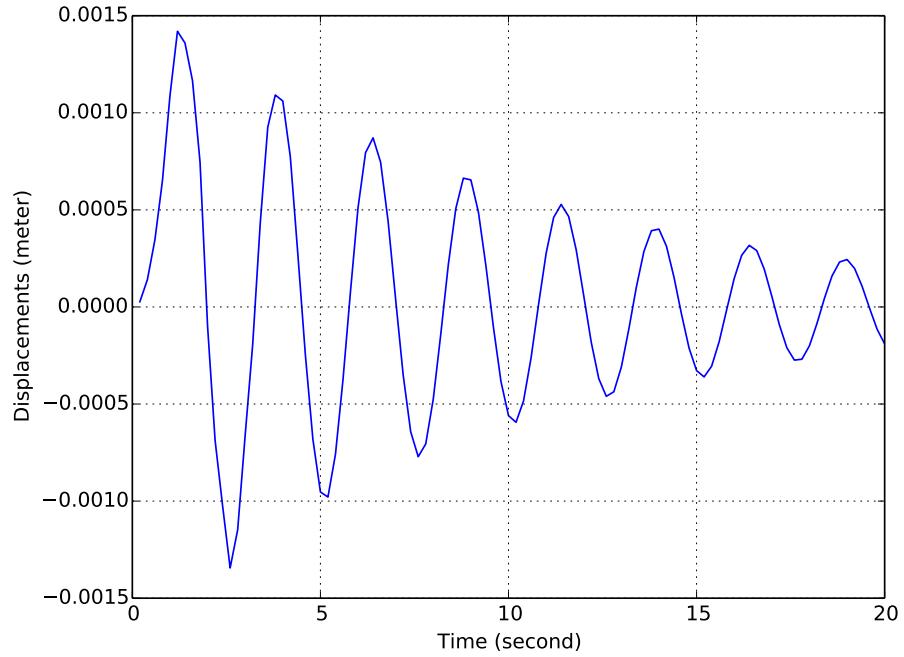


Figure 29: Free vibration condition, viscous (Caughey3rd) damping, vertical displacements of the cantilever tip.

The ESSI model fei files for this example can be downloaded [here](#).

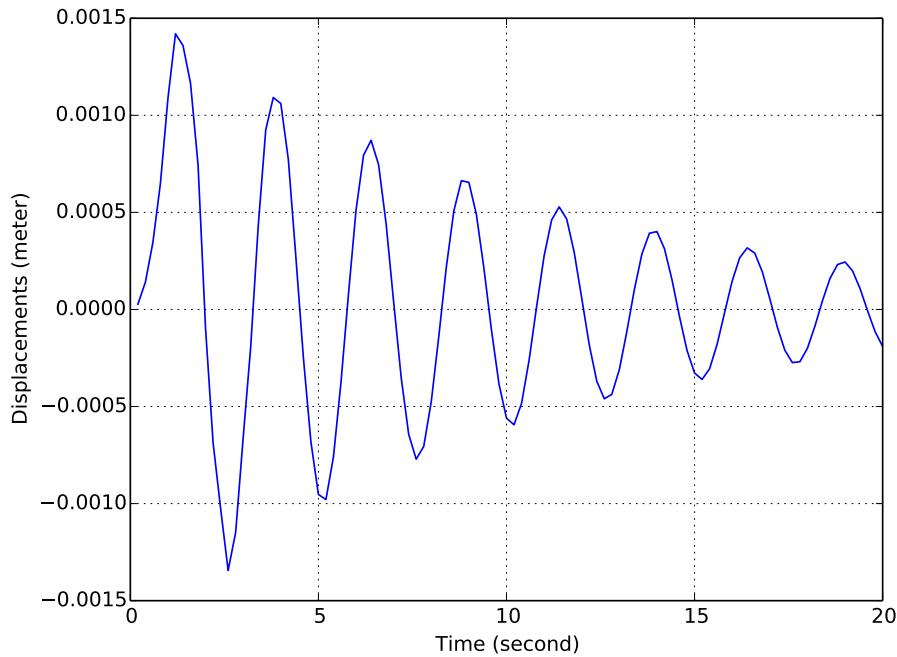


Figure 30: Free vibration condition, viscous (Caughey4th) damping, vertical displacements of the cantilever tip.

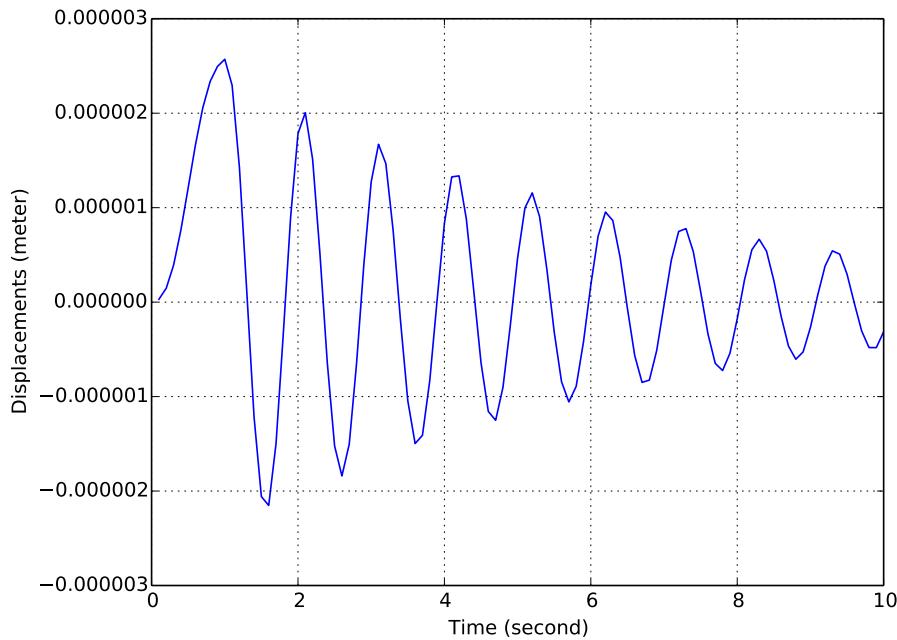


Figure 31: Free vibration condition, numerical (Newmark) damping, vertical displacements of the cantilever tip.

7 Elastic Beam Element for a Simple Frame Structure

Problem Description

- Dimensions: hidth=6m, height=6m, force=100N

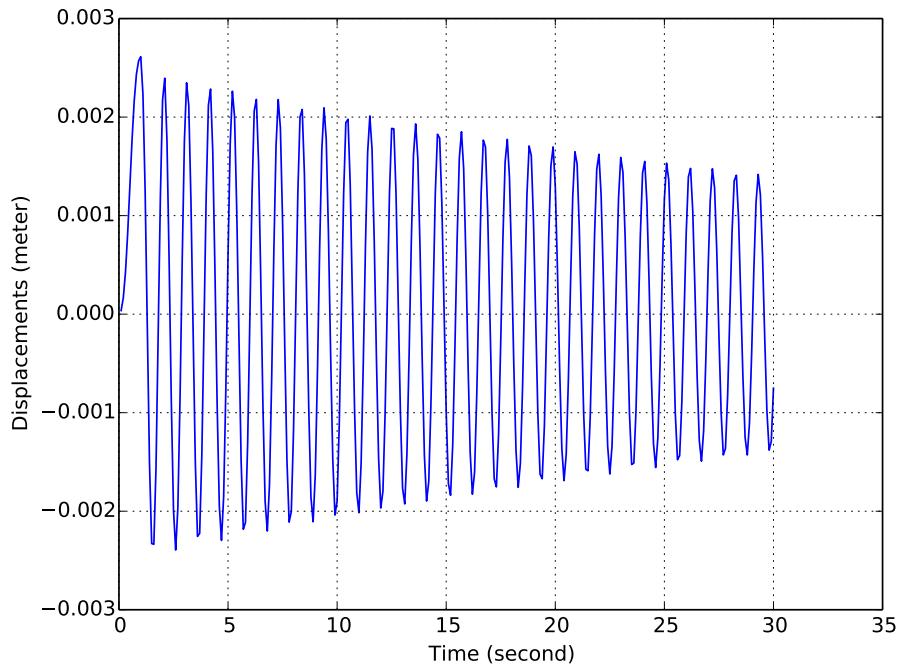


Figure 32: Free vibration condition, numerical (HHT) damping, vertical displacements of the cantilever tip.

- Element dimensions: length=6m, cross section width=1m, cross section height=1m, mass density $\rho = 0.0\text{kN/m}^3$, Young's modulus $E = 1E8 \text{ Pa}$, Poisson's ratio $\nu = 0.0$.

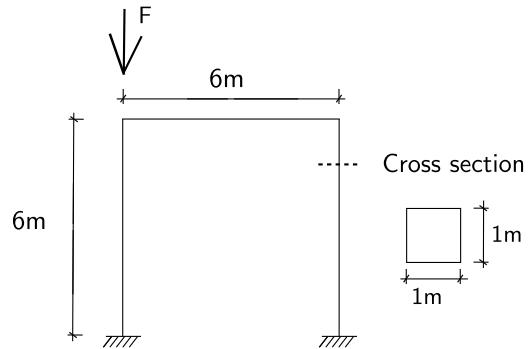


Figure 33: Elastic frame with *beam_elastic* elements.

ESSI model fei file:

```

1 model name "beam_element_presentation" ;
2
3 add node # 1 at ( 0.00*m, 0.00*m, 0.00*m) with 6 dofs;
4 add node # 2 at ( 0.00*m, 0.00*m, 6.00*m) with 6 dofs;
5 add node # 3 at ( 6.00*m, 0.00*m, 6.00*m) with 6 dofs;

```

```

6 add node # 4 at ( 6.00*m, 0.00*m, 0.00*m) with 6 dofs;
7
8 elastic_constant = 1e8*N/m^2;
9 b=1*m;
10 h=1*m;
11 rho = 0*kg/m^3; // Mass density
12
13 add element # 1 type beam_elastic with nodes (1, 2)
14 cross_section = b*h elastic_modulus = elastic_constant
15 shear_modulus = elastic_constant/2
16 torsion_Jx = 0.33*b*h^3 bending_Iy = b*h^3/12 bending_Iz = h*b^3/12
17 mass_density = rho xz_plane_vector = (1, 0, 1)
18 joint_1_offset = (0*m, 0*m, 0*m) joint_2_offset = (0*m, 0*m, 0*m);
19
20 add element # 2 type beam_elastic with nodes (2,3)
21 cross_section = b*h elastic_modulus = elastic_constant
22 shear_modulus = elastic_constant/2
23 torsion_Jx = 0.33*b*h^3 bending_Iy = b*h^3/12 bending_Iz = h*b^3/12
24 mass_density = rho xz_plane_vector = (1, 0, 1)
25 joint_1_offset = (0*m, 0*m, 0*m) joint_2_offset = (0*m, 0*m, 0*m);
26
27 add element # 3 type beam_elastic with nodes (3,4)
28 cross_section = b*h elastic_modulus = elastic_constant
29 shear_modulus = elastic_constant/2
30 torsion_Jx = 0.33*b*h^3 bending_Iy = b*h^3/12 bending_Iz = h*b^3/12
31 mass_density = rho xz_plane_vector = (1, 0, 1)
32 joint_1_offset = (0*m, 0*m, 0*m) joint_2_offset = (0*m, 0*m, 0*m);
33
34 fix node #1 dofs all;
35 fix node #4 dofs all;
36
37 new loading stage "Fz";
38
39 add load # 1 to node # 2 type linear Fz=50*N;
40
41 define algorithm With_no_convergence_check;
42 define solver ProfileSPD;
43 define load factor increment 1;
44 simulate 1 steps using static algorithm;
45
46 bye;

```

The ESSI model fei files for this example can be downloaded [here](#).

8 27NodeBrick Cantilever Beam for the static load

Problem description:

Length=6m, Width=1m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.0$. The force direction is shown in Figure (112).

Numerical model:

The 27NodeBrick elements for cantilever beams is shown in Figure (113):

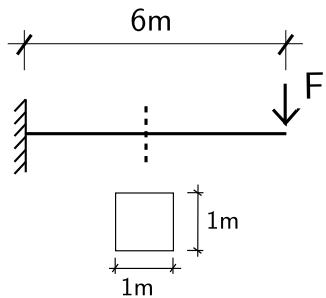


Figure 34: Problem description for cantilever beam.

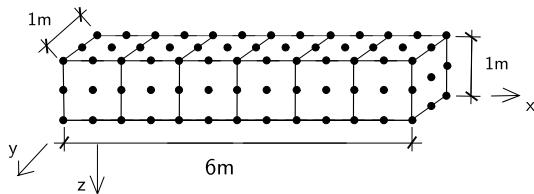


Figure 35: 27NodeBrick elements for cantilever beams made of solid elements.

ESSI model fei file:

```

1 model name "6meter_cantilever_27brick" ;
2
3 add material # 1 type linear_elastic_isotropic_3d
4   mass_density = 0*kg/m^3
5   elastic_modulus = 1e8*N/m^2
6   poisson_ratio = 0.0;
7
8 add node # 1 at ( 0.00 *m, 1.00 *m, 0.00 *m) with 3 dofs;
9 add node # 2 at ( 0.00 *m, 0.00 *m, 0.00 *m) with 3 dofs;
10 add node # 3 at ( 6.00 *m, 1.00 *m, 0.00 *m) with 3 dofs;
11 add node # 4 at ( 5.00 *m, 1.00 *m, 0.00 *m) with 3 dofs;
12 add node # 5 at ( 4.00 *m, 1.00 *m, 0.00 *m) with 3 dofs;
13 add node # 6 at ( 3.00 *m, 1.00 *m, 0.00 *m) with 3 dofs;
14 ...
15 ...
16 add node #117 at ( 5.50 *m, 0.50 *m, 1.00 *m) with 3 dofs;
17
18 add element # 1 type 27NodeBrickLT with nodes( 2, 10, 8, 1, 15, 17, 28, 23, 29, 30, 31,
19   32, 33, 34, 35, 36, 37, 38, 39, 40, 41, 42, 43, 44, 45, 46, 47) use material # 1;
20 add element # 2 type 27NodeBrickLT with nodes( 10, 11, 7, 8, 17, 18, 27, 28, 48, 49, 50,
21   30, 51, 52, 53, 34, 38, 54, 55, 39, 56, 57, 58, 59, 43, 60, 61) use material # 1;
22 add element # 3 type 27NodeBrickLT with nodes( 11, 12, 6, 7, 18, 19, 26, 27, 62, 63, 64,
23   49, 65, 66, 67, 52, 54, 68, 69, 55, 70, 71, 72, 73, 58, 74, 75) use material # 1;
24 add element # 4 type 27NodeBrickLT with nodes( 12, 13, 5, 6, 19, 20, 25, 26, 76, 77, 78,
25   63, 79, 80, 81, 66, 68, 82, 83, 69, 84, 85, 86, 87, 72, 88, 89) use material # 1;
26 add element # 5 type 27NodeBrickLT with nodes( 13, 14, 4, 5, 20, 21, 24, 25, 90, 91, 92,
27   77, 93, 94, 95, 80, 82, 96, 97, 83, 98, 99, 100, 101, 86, 102, 103) use material # 1;
```

```

23 add element # 6 type 27NodeBrickLT with nodes( 14, 9, 3, 4, 21, 16, 22, 24, 104, 105, 106,
      91, 107, 108, 109, 94, 96, 110, 111, 97, 112, 113, 114, 115, 100, 116, 117) use material #
      1;
24
25 fix node # 1 dofs all;
26 fix node # 2 dofs all;
27 fix node # 15 dofs all;
28 fix node # 23 dofs all;
29 fix node # 32 dofs all;
30 fix node # 36 dofs all;
31 fix node # 37 dofs all;
32 fix node # 40 dofs all;
33 fix node # 45 dofs all;
34
35 new loading stage "Fz";
36 add load # 1 to node # 13 type linear Fz=2.777778*N;
37 add load # 2 to node # 24 type linear Fz=2.777778*N;
38 add load # 3 to node # 3 type linear Fz=2.777778*N;
39 add load # 4 to node # 34 type linear Fz=2.777778*N;
40 add load # 5 to node # 182 type linear Fz=11.111111*N;
41 add load # 6 to node # 177 type linear Fz=11.111111*N;
42 add load # 7 to node # 180 type linear Fz=11.111111*N;
43 add load # 8 to node # 183 type linear Fz=11.111111*N;
44 add load # 9 to node # 186 type linear Fz=44.444444*N;
45
46 define algorithm With_no_convergence_check ;
47 define solver UMFPack;
48 define load factor increment 1;
49 simulate 1 steps using static algorithm;
50
51 bye;

```

The ESSI model fei files for this example can be downloaded [here](#).

9 4NodeANDES Cantilever Beams Under the Force Perpendicular to Plane

Problem description:

Length=6m, Width=1m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.0$.

Numerical model:

For a force direction perpendicular to the plane, only the bending deformation is present. The model is shown in Figure (166).

ESSI model fei file:

```

1 model name "6meter_cantilever_4NodeANDES" ;
2
3 add material # 1 type linear_elastic_isotropic_3d
4   mass_density = 0*kg/m^3
5   elastic_modulus = 1e8*N/m^2

```

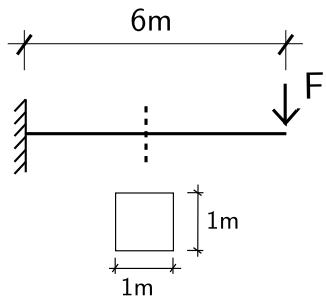


Figure 36: Cantilever beams

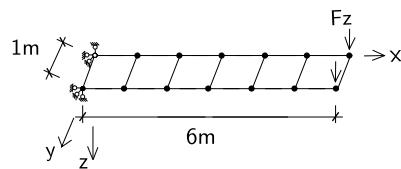


Figure 37: 4NodeANDES elements for cantilever beams under force perpendicular to plane.

```

6   poisson_ratio = 0.0;
7
8 add node # 1 at ( 0.0*m, 0.0*m, 0.0*m) with 6 dofs;
9 add node # 2 at ( 6.0*m, 0.0*m, 0.0*m) with 6 dofs;
10 add node # 3 at ( 1.0*m, 0.0*m, 0.0*m) with 6 dofs;
11 add node # 4 at ( 2.0*m, 0.0*m, 0.0*m) with 6 dofs;
12 add node # 5 at ( 3.0*m, 0.0*m, 0.0*m) with 6 dofs;
13 add node # 6 at ( 4.0*m, 0.0*m, 0.0*m) with 6 dofs;
14 add node # 7 at ( 5.0*m, 0.0*m, 0.0*m) with 6 dofs;
15 add node # 8 at ( 6.0*m, 1.0*m, 0.0*m) with 6 dofs;
16 add node # 9 at ( 0.0*m, 1.0*m, 0.0*m) with 6 dofs;
17 add node # 10 at ( 5.0*m, 1.0*m, 0.0*m) with 6 dofs;
18 add node # 11 at ( 4.0*m, 1.0*m, 0.0*m) with 6 dofs;
19 add node # 12 at ( 3.0*m, 1.0*m, 0.0*m) with 6 dofs;
20 add node # 13 at ( 2.0*m, 1.0*m, 0.0*m) with 6 dofs;
21 add node # 14 at ( 1.0*m, 1.0*m, 0.0*m) with 6 dofs;
22
23 h = 1*m;
24 add element # 1 type 4NodeShell_ANDES with nodes (1,3,14,9) use material # 1 thickness = h ;
25 add element # 2 type 4NodeShell_ANDES with nodes (3,4,13,14) use material # 1 thickness = h ;
26 add element # 3 type 4NodeShell_ANDES with nodes (4,5,12,13) use material # 1 thickness = h ;
27 add element # 4 type 4NodeShell_ANDES with nodes (5,6,11,12) use material # 1 thickness = h ;
28 add element # 5 type 4NodeShell_ANDES with nodes (6,7,10,11) use material # 1 thickness = h ;
29 add element # 6 type 4NodeShell_ANDES with nodes (7,2,8,10) use material # 1 thickness = h ;
30
31 fix node # 1 dofs all ;
32 fix node # 9 dofs all ;
33
34 new loading stage "Fz";
35 add load # 1 to node # 8 type linear Fz=50*N;

```

```

36 add load # 2 to node # 2 type linear Fz=50*N;
37
38 define algorithm With_no_convergence_check ;
39 define solver ProfileSPD;
40 define load factor increment 1;
41 simulate 1 steps using static algorithm;
42
43 bye;

```

The ESSI model fei files for this example can be downloaded [here](#).

10 4NodeANDES Cantilever Beams under the In-Plane Force

Problem description:

Length=6m, Width=1m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.0$.

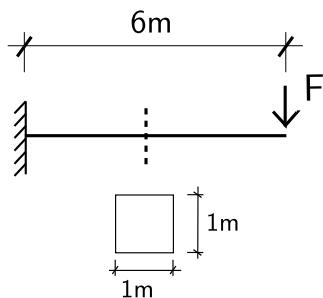


Figure 38: Problem description for cantilever beams with in plane force

Numerical model:

The 4NodeANDES elements under in-plane force is shown in Figure (167).

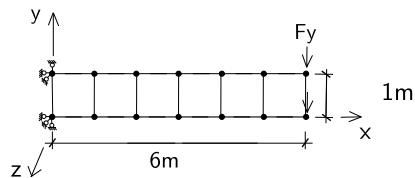


Figure 39: 4NodeANDES elements for cantilever beams under in-plane force

ESSI model fei file:

```

1 model name "6meter_cantilever_4NodeANDES" ;
2
3 add material # 1 type linear_elastic_isotropic_3d

```

```

4   mass_density = 0*kg/m^3
5   elastic_modulus = 1e8*N/m^2
6   poisson_ratio = 0.0;
7
8   add node # 1 at ( 0.00*m, 0.00*m, 0.00*m) with 6 dofs;
9   add node # 2 at ( 6.00*m, 0.00*m, 0.00*m) with 6 dofs;
10  add node # 3 at ( 1.00*m, 0.00*m, 0.00*m) with 6 dofs;
11  add node # 4 at ( 2.00*m, 0.00*m, 0.00*m) with 6 dofs;
12  add node # 5 at ( 3.00*m, 0.00*m, 0.00*m) with 6 dofs;
13  add node # 6 at ( 4.00*m, 0.00*m, 0.00*m) with 6 dofs;
14  add node # 7 at ( 5.00*m, 0.00*m, 0.00*m) with 6 dofs;
15  add node # 8 at ( 6.00*m, 1.00*m, 0.00*m) with 6 dofs;
16  add node # 9 at ( 0.00*m, 1.00*m, 0.00*m) with 6 dofs;
17  add node # 10 at ( 5.00*m, 1.00*m, 0.00*m) with 6 dofs;
18  add node # 11 at ( 4.00*m, 1.00*m, 0.00*m) with 6 dofs;
19  add node # 12 at ( 3.00*m, 1.00*m, 0.00*m) with 6 dofs;
20  add node # 13 at ( 2.00*m, 1.00*m, 0.00*m) with 6 dofs;
21  add node # 14 at ( 1.00*m, 1.00*m, 0.00*m) with 6 dofs;
22
23 h     = 1*m;
24 add element # 1 type 4NodeShell_ANDES with nodes (1,3,14,9) use material # 1 thickness = h ;
25 add element # 2 type 4NodeShell_ANDES with nodes (3,4,13,14) use material # 1 thickness = h ;
26 add element # 3 type 4NodeShell_ANDES with nodes (4,5,12,13) use material # 1 thickness = h ;
27 add element # 4 type 4NodeShell_ANDES with nodes (5,6,11,12) use material # 1 thickness = h ;
28 add element # 5 type 4NodeShell_ANDES with nodes (6,7,10,11) use material # 1 thickness = h ;
29 add element # 6 type 4NodeShell_ANDES with nodes (7,2,8,10) use material # 1 thickness = h ;
30
31 fix node # 1 dofs all;
32 fix node # 9 dofs all;
33
34 new loading stage "Fy";
35 add load # 1 to node # 8 type linear Fy=50*N;
36 add load # 2 to node # 2 type linear Fy=50*N;
37
38 define algorithm With_no_convergence_check ;
39 define solver ProfileSPD;
40 define load factor increment 1;
41 simulate 1 steps using static algorithm;
42
43 bye;

```

The ESSI model fei files for this example can be downloaded [here](#).

11 27NodeBrick Cantilever Beams for Dynamic Input

Problem description:

Length=20m, Width=1m, Height=1m, E=504MPa, $\nu = 0.4$.

All degree of freedoms at the bottom nodes are fixed.

The load is a self weight with a dynamic displacement of supports.

Numerical model:

The numerical model applied 27NodeBrick to simulate the 1D motion.

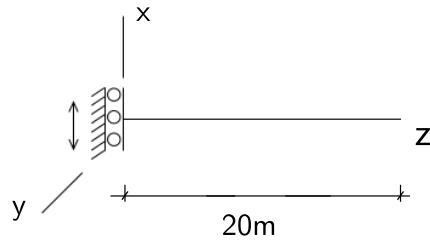


Figure 40: Problem description for one simple dynamic example

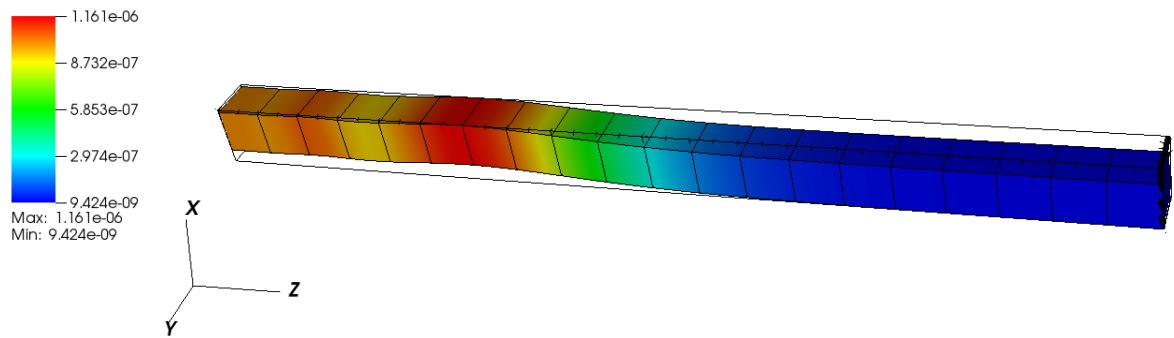


Figure 41: Numerical model for one simple dynamic example

ESSI model fei file:

```

1 model name "dynamic_example";
2
3 add material # 1 type linear_elastic_isotropic_3d_LT
4 mass_density    = 2000*kg/m^3
5 elastic_modulus = 504000000.00*Pa
6 poisson_ratio   = 0.4;
7
8 add node No 1 at (0*m, 0*m, 0*m) with 3 dofs;
9 add node No 2 at (0*m, 0.5*m, 0*m) with 3 dofs;
10 add node No 3 at (0*m, 1*m, 0*m) with 3 dofs;
11 add node No 4 at (0.5*m, 0*m, 0*m) with 3 dofs;
12 add node No 5 at (0.5*m, 0.5*m, 0*m) with 3 dofs;
13 add node No 6 at (0.5*m, 1*m, 0*m) with 3 dofs;
14 ...
15 ...
16 add node No 369 at (1*m, 1*m, 20*m) with 3 dofs;
17
18 add element # 1 type 27NodeBrickLT with nodes
19      (27,21,19,25,9,3,1,7,24,20,22,26,6,2,4,8,18,12,10,16,14,15,11,13,17,23,5) use material # 1
        ;
      add element # 2 type 27NodeBrickLT with nodes
        (45,39,37,43,27,21,19,25,42,38,40,44,24,20,22,26,36,30,28,34,32,33,29,31,35,41,23) use
          material # 1 ;

```

```

20 add element # 3 type 27NodeBrickLT with nodes
  (63,57,55,61,45,39,37,43,60,56,58,62,42,38,40,44,54,48,46,52,50,51,47,49,53,59,41) use
    material # 1 ;
21 add element # 4 type 27NodeBrickLT with nodes
  (81,75,73,79,63,57,55,61,78,74,76,80,60,56,58,62,72,66,64,70,68,69,65,67,71,77,59) use
    material # 1 ;
22 add element # 5 type 27NodeBrickLT with nodes
  (99,93,91,97,81,75,73,79,96,92,94,98,78,74,76,80,90,84,82,88,86,87,83,85,89,95,77) use
    material # 1 ;
23 ...
24 ...
25 add element # 20 type 27NodeBrickLT with nodes
  (369,363,361,367,351,345,343,349,366,362,364,368,348,
  344,346,350,360,354,352,358,356,357,353,355,359,365,347) use material # 1 ;
26
27
28 add acceleration field # 1 ax = 0*g ay = 0*g az = -1*g ;
29 add load # 1 to element # 1 type self_weight use acceleration field # 1;
30 add load # 2 to element # 2 type self_weight use acceleration field # 1;
31 add load # 3 to element # 3 type self_weight use acceleration field # 1;
32 add load # 4 to element # 4 type self_weight use acceleration field # 1;
33 add load # 5 to element # 5 type self_weight use acceleration field # 1;
34 add load # 6 to element # 6 type self_weight use acceleration field # 1;
35 ...
36 ...
37 add load # 20 to element # 20 type self_weight use acceleration field # 1;
38
39 fix node No 1 dofs uy uz;
40 fix node No 2 dofs uy uz;
41 fix node No 3 dofs uy uz;
42 fix node No 4 dofs uy uz;
43 fix node No 5 dofs uy uz;
44 fix node No 6 dofs uy uz;
45 ...
46 ...
47 fix node No 369 dofs uy uz;
48
49 zeta = 0.0166667;
50 fq1 = 3.75;
51 fq2 = 11.25;
52 omega1 = 2*pi*fq1;
53 omega2 = 2*pi*fq2;
54 zeta1 = zeta;
55 zeta2 = zeta;
56 alpha1 = 2*omega1*omega2*(zeta1*omega2-zeta2*omega1)/(omega2*omega2-omega1*omega1);
57 beta1 = 2*          (zeta2*omega2-zeta1*omega1)/(omega2*omega2-omega1*omega1);
58 add damping # 1
  type Rayleigh
  with
    a0 = alpha1/s
    a1 = beta1*s
    stiffness_to_use = Initial_Stiffness;
59
60
61
62
63
64
65 add damping # 1 to element # 1;
66 add damping # 1 to element # 2;
67 add damping # 1 to element # 3;

```

```

68 add damping # 1 to element # 4;
69 add damping # 1 to element # 5;
70 add damping # 1 to element # 6;
71 ...
72 ...
73 add damping # 1 to element # 20;
74
75 new loading stage "impose_motion";
76
77 add imposed motion # 1001 to node # 1 dof ux
78 displacement_scale_unit = 1*m    displacement_file = "dis.txt"
79 velocity_scale_unit   = 1*m/s   velocity_file     = "vel.txt"
80 acceleration_scale_unit = 1*m/s^2 acceleration_file = "acc.txt";
81
82 add imposed motion # 1002 to node # 2 dof ux
83 displacement_scale_unit = 1*m    displacement_file = "dis.txt"
84 velocity_scale_unit   = 1*m/s   velocity_file     = "vel.txt"
85 acceleration_scale_unit = 1*m/s^2 acceleration_file = "acc.txt";
86
87 add imposed motion # 1003 to node # 3 dof ux
88 displacement_scale_unit = 1*m    displacement_file = "dis.txt"
89 velocity_scale_unit   = 1*m/s   velocity_file     = "vel.txt"
90 acceleration_scale_unit = 1*m/s^2 acceleration_file = "acc.txt";
91 ...
92 ...
93 add imposed motion # 1009 to node # 9 dof ux
94 displacement_scale_unit = 1*m    displacement_file = "dis.txt"
95 velocity_scale_unit   = 1*m/s   velocity_file     = "vel.txt"
96 acceleration_scale_unit = 1*m/s^2 acceleration_file = "acc.txt";
97
98 define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
99 define algorithm With_no_convergence_check;
100 define solver ProfileSPD;
101 simulate 50 steps using transient algorithm time_step = 0.005*s;
102
103 bye;

```

The ESSI model fei files for this example can be downloaded [here](#).

12 4NodeANDES Square Plate with Four Edges Clamped

Problem description:

Length=20m, Width=20m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.3$.

The four edges are **clamped**.

The load is a self weight.

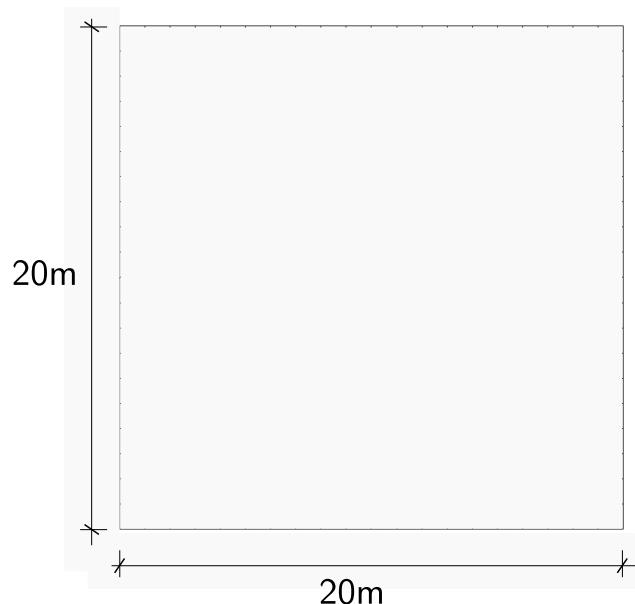


Figure 42: Square plate with four edges clamped

Numerical model:

The element side length is 1 meter.

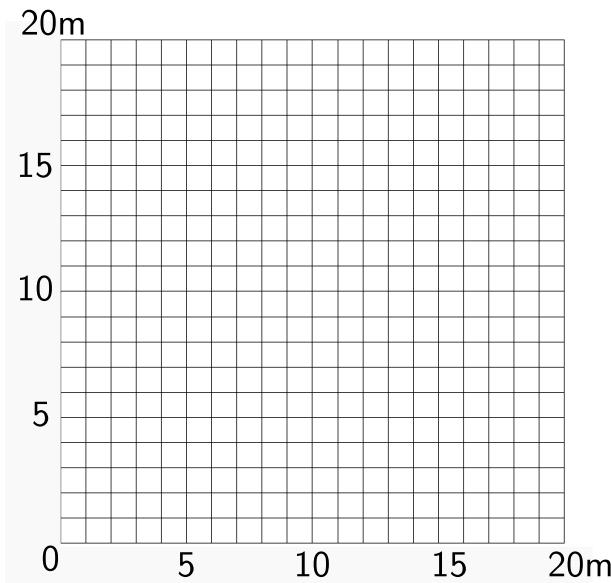


Figure 43: 4NodeANDES edge clamped square plate with element side length 1m

ESSI model fei file:

```

1 model name "square_plate" ;
2
3 add material # 1 type linear_elastic_isotropic_3d
4   mass_density = 1e2*kg/m^3 elastic_modulus = 1e8*N/m^2 poisson_ratio = 0.3;
5
6 add node # 1 at ( 0.00*m, 0.00*m, 0.00*m) with 6 dofs;
7 add node # 2 at ( 20.00*m, 0.00*m, 0.00*m) with 6 dofs;
8 add node # 3 at ( 1.00*m, 0.00*m, 0.00*m) with 6 dofs;
9 add node # 4 at ( 2.00*m, 0.00*m, 0.00*m) with 6 dofs;
10 add node # 5 at ( 3.00*m, 0.00*m, 0.00*m) with 6 dofs;
11 add node # 6 at ( 4.00*m, 0.00*m, 0.00*m) with 6 dofs;
12 ...
13 ...
14 add node # 441 at ( 19.00*m, 19.00*m, 0.00*m) with 6 dofs;
15
16 h      = 1*m;
17 add element #    1 type 4NodeShell_ANDES with nodes( 1, 3, 81, 80) use material # 1
18   thickness=h;
19 add element #    2 type 4NodeShell_ANDES with nodes( 3, 4, 100, 81) use material # 1
20   thickness=h;
21 add element #    3 type 4NodeShell_ANDES with nodes( 4, 5, 119, 100) use material # 1
22   thickness=h;
23 add element #    4 type 4NodeShell_ANDES with nodes( 5, 6, 138, 119) use material # 1
24   thickness=h;
```

```

21 add element #      5 type 4NodeShell_ANDES with nodes( 6, 7, 157, 138) use material # 1
22   thickness=h;
23 add element #      6 type 4NodeShell_ANDES with nodes( 7, 8, 176, 157) use material # 1
24   thickness=h;
25 ...
26 ...
27 add element #    400 type 4NodeShell_ANDES with nodes( 441, 41, 22, 43) use material # 1
28   thickness=h;
29
30 fix node #      1 dofs all  ;
31 fix node #      2 dofs all  ;
32 fix node #      3 dofs all  ;
33 fix node #      4 dofs all  ;
34 fix node #      5 dofs all  ;
35 fix node #      6 dofs all  ;
36 ...
37 ...
38
39 new loading stage "self_weight";
40 add acceleration field # 1 ax = 0*g ay = 0*g az = 1*m/s^2;
41 add load # 1 to element # 1 type self_weight use acceleration field # 1;
42 add load # 2 to element # 2 type self_weight use acceleration field # 1;
43 add load # 3 to element # 3 type self_weight use acceleration field # 1;
44 add load # 4 to element # 4 type self_weight use acceleration field # 1;
45 add load # 5 to element # 5 type self_weight use acceleration field # 1;
46 add load # 6 to element # 6 type self_weight use acceleration field # 1;
47 ...
48 ...
49 add load # 400 to element # 400 type self_weight use acceleration field # 1;
50
51
52 define algorithm With_no_convergence_check ;
53 define solver ProfileSPD;
54 define load factor increment 1;
55 simulate 1 steps using static algorithm;
56
57 bye;

```

The ESSI model fei files for this example can be downloaded [here](#).

13 One Dimensional DRM Model

Problem description:

A simple 1D DRM model is shown in Fig.(44). The "DRM element", "Exterior node" and "Boundary node" are required to be designated in the DRM HDF5 input. The format and script for the HDF5 input is available in DSL/input manual.

Numerical model:

ESSI model fei file:

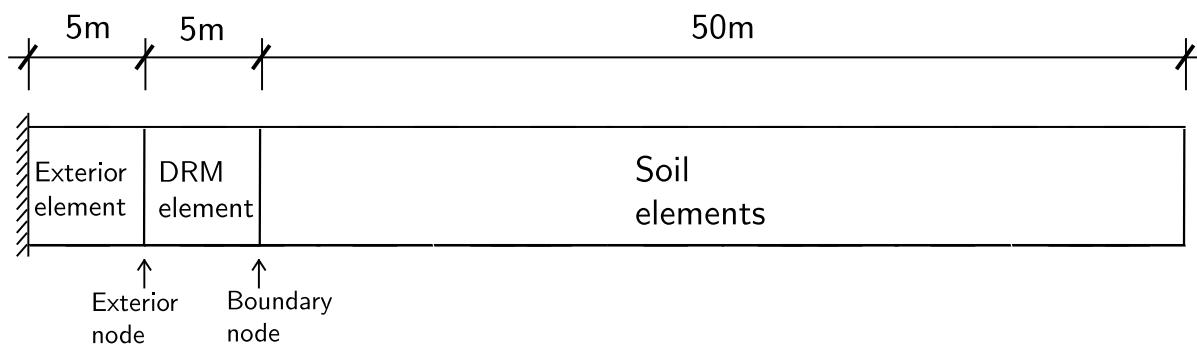
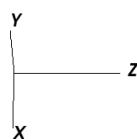
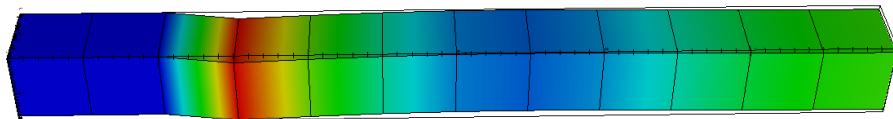


Figure 44: 1D DRM model.

DB: DRM_1D.h5.feioutput
Time:2.87

Mesh
Var: ESSI Domain Mesh

Pseudocolor
Var: Generalized Displacements_magnitude
0.001175
0.0008821
0.0005896
0.0002971
4.677e-06
Max: 0.001175
Min: 4.677e-06



User: yuan
Sat Nov 7 11:34:02 2015

Figure 45: 1D DRM model.

```

1 model name "DRM" ;
2
3 //Material for soil
4 add material # 1 type linear_elastic_isotropic_3d_LT
5 mass_density = 2000*kg/m^3
6 elastic_modulus = 1300*MPa
7 poisson_ratio = 0.3;
8

```

```

9 //Material for DRM layer
10 add material # 2 type linear_elastic_isotropic_3d_LT
11   mass_density = 2000*kg/m^3
12   elastic_modulus = 1300*MPa
13   poisson_ratio = 0.3;
14
15 //Material for exterior layer
16 add material # 3 type linear_elastic_isotropic_3d_LT
17   mass_density = 2000*kg/m^3
18   elastic_modulus = 1300*MPa
19   poisson_ratio = 0.3;
20 //
21 add node # 1 at ( 0.00*m, 0.00*m, 0.00*m) with 3 dofs;
22 add node # 2 at ( 5.00*m, 0.00*m, 0.00*m) with 3 dofs;
23 add node # 3 at ( 5.00*m, 5.00*m, 0.00*m) with 3 dofs;
24 add node # 4 at ( 0.00*m, 5.00*m, 0.00*m) with 3 dofs;
25 add node # 5 at ( 5.00*m, 0.00*m, 50.00*m) with 3 dofs;
26 add node # 6 at ( 5.00*m, 0.00*m, 5.00*m) with 3 dofs;
27 ...
28 ...
29 add node # 52 at ( 0.00*m, 5.00*m, -5.00*m) with 3 dofs;
30
31 //
32 add element #      1 type 8NodeBrickLT with nodes( 1, 4, 3, 2, 24, 44, 34, 6) use material # 1;
33 add element #      2 type 8NodeBrickLT with nodes( 24, 44, 34, 6, 23, 43, 33, 7) use material
34   # 1;
35 ...
36 add element #      12 type 8NodeBrickLT with nodes( 48, 47, 45, 46, 52, 51, 49, 50) use
37   material # 3;
38 //
39 fix node # 1 dofs uy ;
40 fix node # 1 dofs uz ;
41 fix node # 2 dofs uy ;
42 fix node # 2 dofs uz ;
43 fix node # 3 dofs uy ;
44 fix node # 3 dofs uz ;
45 fix node # 4 dofs uy ;
46 fix node # 4 dofs uz ;
47 ...
48 fix node # 51 dofs ux ;
49
50 new loading stage "1D";
51 add domain reduction method loading # 1
52   hdf5_file = "input.hdf5";
53
54 define algorithm With_no_convergence_check ;
55 define solver ProfileSPD;
56 define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
57 simulate 999 steps using transient algorithm time_step = 0.01*s;
58
59 bye;

```

The ESSI model fei files for this example can be downloaded [here](#).

The same model for this example with 27NodeBrickLT can be downloaded [here](#).

Long 1D DRM model 1000:1

To show the wave propagation explicitly, a long 1D model (1000:1) similar to the 1D DRM model above was made in this section.

The model description is same to Fig.(44) except this model use far more soil elements.

The general view is shown in Fig.(46) below.

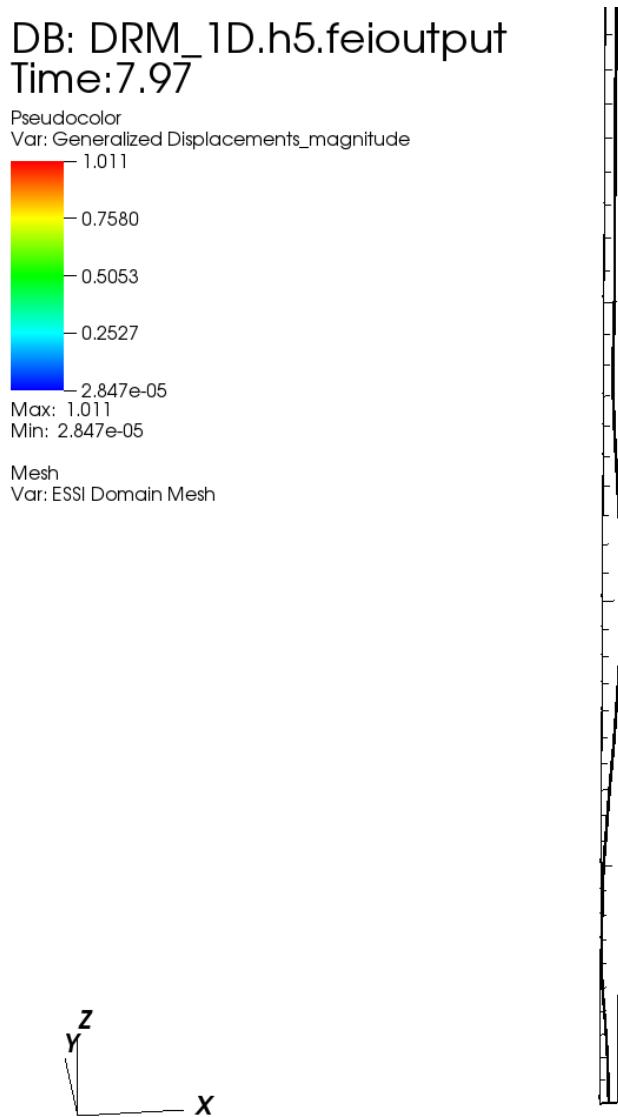


Figure 46: Long 1D DRM model

There is still now outgoing waves at the exterior layers, which is shown in Fig(47).

The ESSI model fei files for this example can be downloaded [here](#).

The results can also be seen from this [video](#).

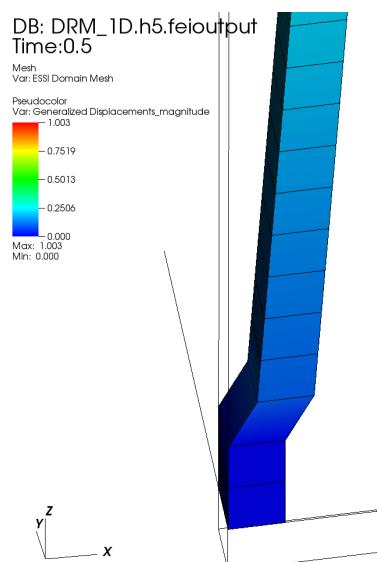


Figure 47: Long 1D DRM model: exterior layer

14 Three Dimensional DRM Model

Problem description:

As shown in Fig.(48), the DRM layer is used to add the earthquake motion.

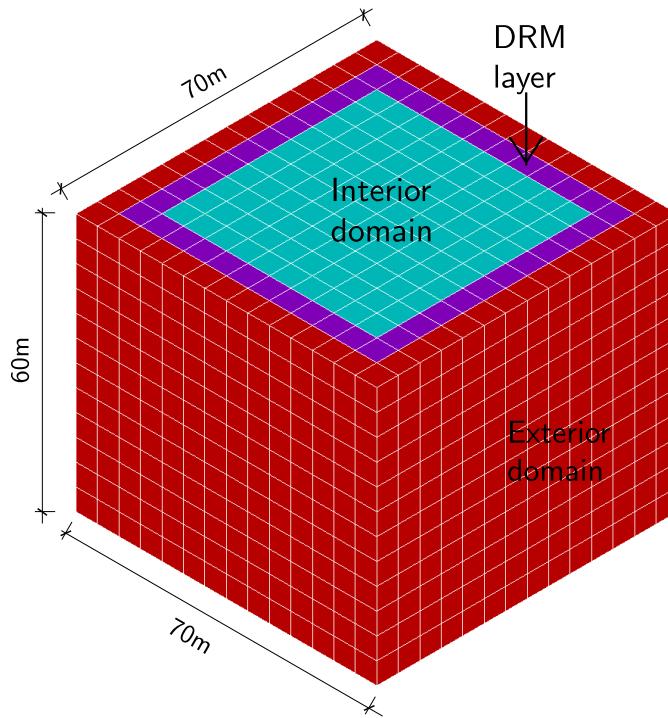


Figure 48: The diagram for 3D Domain Reduction Method example.

Numerical result:

ESSI model fei file:

```

1 model name "DRM" ;
2
3 //Material for soil
4 add material # 1 type linear_elastic_isotropic_3d_LT
5   mass_density = 2000*kg/m^3
6   elastic_modulus = 1300*MPa
7   poisson_ratio = 0.3;
8
9 //Material for DRM layer
10 add material # 2 type linear_elastic_isotropic_3d_LT
11   mass_density = 2000*kg/m^3
12   elastic_modulus = 1300*MPa
13   poisson_ratio = 0.3;
14
15 //Material for exterior layer
16 add material # 3 type linear_elastic_isotropic_3d_LT
17   mass_density = 2000*kg/m^3

```

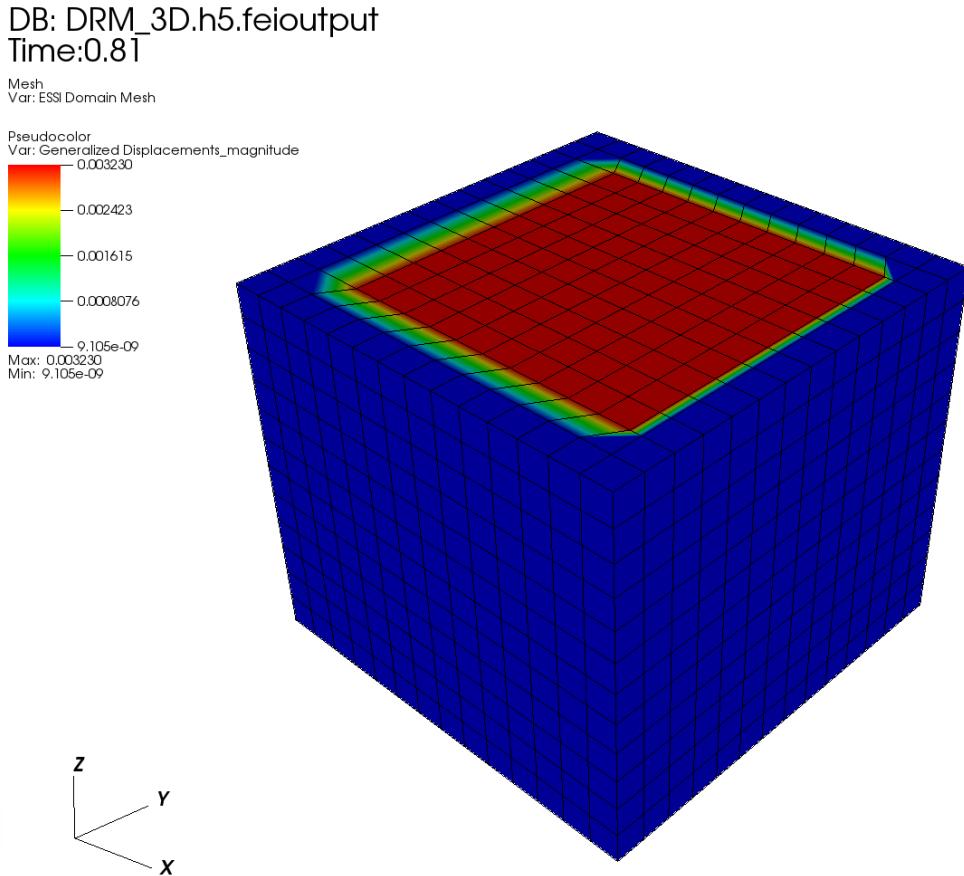


Figure 49: Diagram for the 3D DRM model.

```

18 elastic_modulus = 1300*MPa
19 poisson_ratio = 0.3;
20
21 // 
22 add node # 1 at ( 0.00*m, 0.00*m, 0.00*m) with 3 dofs;
23 add node # 2 at ( 50.00*m, 0.00*m, 0.00*m) with 3 dofs;
24 add node # 3 at ( 5.00*m, 0.00*m, 0.00*m) with 3 dofs;
25 add node # 4 at ( 10.00*m, 0.00*m, 0.00*m) with 3 dofs;
26 add node # 5 at ( 15.00*m, 0.00*m, 0.00*m) with 3 dofs;
27 add node # 6 at ( 20.00*m, 0.00*m, 0.00*m) with 3 dofs;
28 add node # 7 at ( 25.00*m, 0.00*m, 0.00*m) with 3 dofs;
29 ...
30 ...
31 add node # 2925 at ( 55.00*m, 55.00*m, -5.00*m) with 3 dofs;
32
33 // 
34 add element # 1 type 8NodeBrickLT with nodes( 1, 40, 41, 3, 150, 441, 603, 151) use material
# 1;
35 add element # 2 type 8NodeBrickLT with nodes( 3, 41, 50, 4, 151, 603, 684, 160) use material
# 1;
36 ...
37 add element # 2352 type 8NodeBrickLT with nodes( 2925, 2924, 2922, 2923, 2921, 2920, 2918,
2919) use material # 3;

```

```

38
39 // 
40 fix node # 1332 dofs all ;
41 fix node # 1334 dofs all ;
42 ...
43 ...
44 fix node # 2924 dofs all ;
45
46 new loading stage "3D";
47 add domain reduction method loading # 1
48   hdf5_file = "input.hdf5";
49
50 define algorithm With_no_convergence_check ;
51 define solver ProfileSPD;
52 define dynamic integrator Newmark with gamma = 0.5 beta = 0.25;
53
54 simulate 999 steps using transient algorithm time_step = 0.01*s;
55
56 bye;

```

The ESSI model fei files for this example can be downloaded [here](#).

The same model for this example with 27NodeBrickLT can be downloaded [here](#).

15 ShearBeam Element for Pisano Materials

Problem description:

In the element type "ShearBeamLT", only one Gauss point exists. ShearBeamLT element was used here to test the Pisano material model.

Vertical force F_z was used to apply confinement to the element. Then, cyclic force F_x is used to load. point.

Results

Resulting stress-strain relationship is shown in Fig.(51).

ESSI model fei file:

```

1 model name "pisanoLT";
2
3 add node # 1 at (0*m,0*m,0*m) with 3 dofs;
4 add node # 2 at (0*m,0*m,1*m) with 3 dofs;
5
6 fix node # 1 dofs all;
7 fix node # 2 dofs uy;
8
9 add material # 1 type New_PisanoLT
10 mass_density = 2000*kg/m^3
11 elastic_modulus_1atm = 325*MPa poisson_ratio = 0.3
12 M_in = 1.4 kd_in = 0.0 xi_in = 0.0 h_in = 700 m_in = 0.7
13 initial_confining_stress = 0*kPa n_in = 0 a_in = 0.0 eplcum_cr_in = 1e-6;
14
15 add element # 1 type ShearBeamLT with nodes (1, 2) \
16   cross_section = 1*m^2 use material # 1;

```

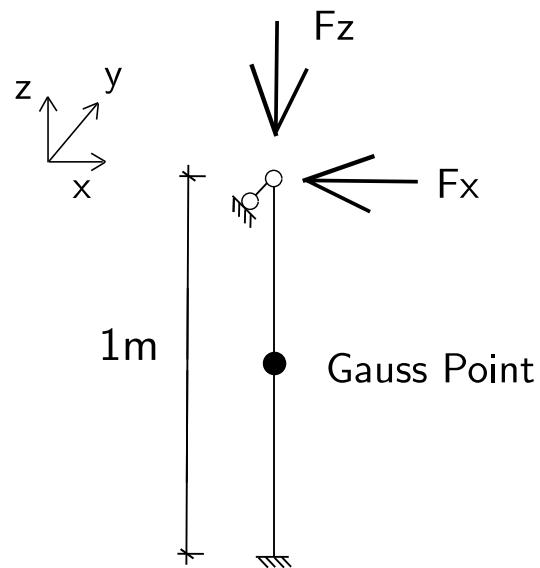


Figure 50: ShearBeam element.

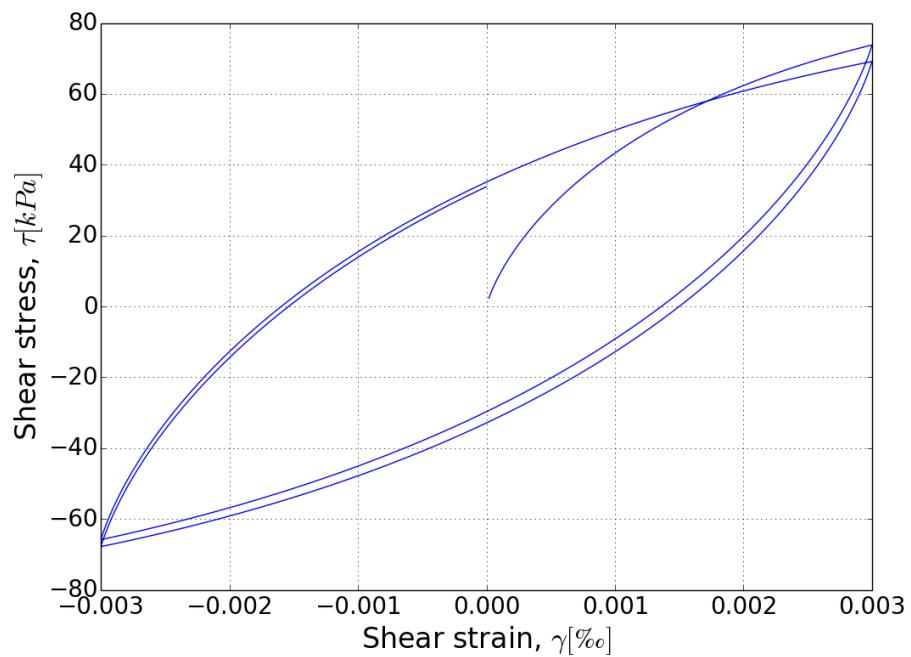


Figure 51: Shear stress-strain response.

```

17
18 new loading stage "confinement";
19
20 add load # 1 to node # 2 type linear Fz = -200*kN;

```

```

21 define load factor increment 0.01;
22 define algorithm With_no_convergence_check ;
23 define solver UMFPack;
24 simulate 100 steps using static algorithm;
25
26 new loading stage "test01";
27 gamma_max = 3e-3;
28 add imposed motion # 2 to node # 2 dof ux
29 displacement_scale_unit = gamma_max*m displacement_file = "input_sine.txt"
30 velocity_scale_unit = gamma_max*m/s velocity_file = "input_sine.txt"
31 acceleration_scale_unit = gamma_max*m/s^2 acceleration_file = "input_sine.txt";
32
33 define load factor increment 0.0005;
34 define algorithm With_no_convergence_check;
35 define solver UMFPack;
36 simulate 2000 steps using static algorithm;
37
38 bye;

```

The ESSI model fei files for this example can be downloaded [here](#).

16 8NodeBrickLT Element for Drucker Prager Armstrong Frederick Material

Problem description:

This example is used to test the materials properties, such as G/Gmax against strains. The element type is 8NodeBrickLT. And there are two stages of loading. The first loading stage is confinement and the second loading stage is shearing.

The boundary condition is specially designed such that each Gauss point has the same stress state.

Results

Resulting stress-strain relationship is shown in Fig.(52).

ESSI model fei file:

```

1 // Drucker Prager Armstrong Frederick
2 // This model is created by Jose.
3 model name "druckeraf";
4
5 // Parameters:
6 phi    = 5;
7 ha     = 1000;
8 cr     = 973;
9 gam    = 0.01;
10 Ncyc   = 5;
11 Nsteps  = 1000;
12 H=1;
13 vp=1000*m/s;
14 vs=500*m/s;
15 rho=2000*kg/m^3;
16 p0 = 250*kPa;
17 G = rho*vs^2;

```

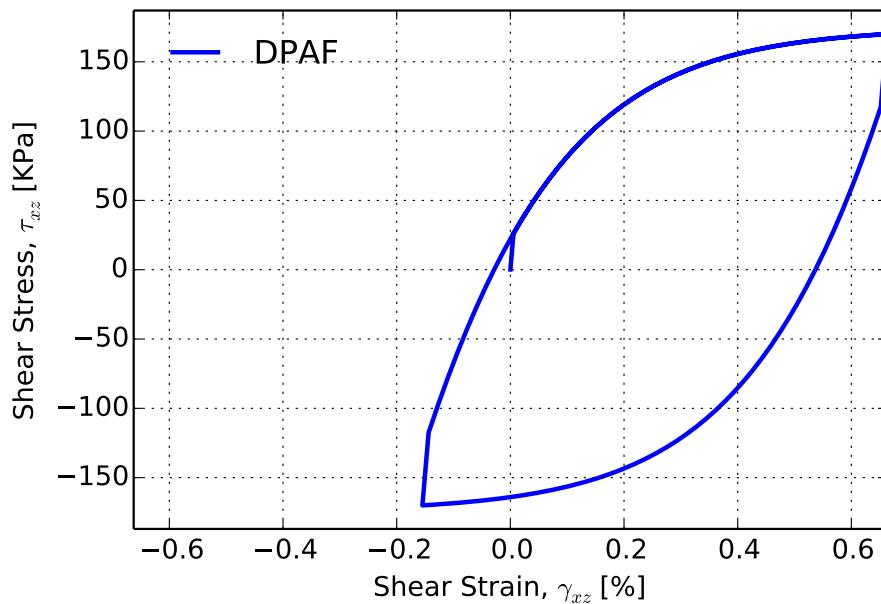


Figure 52: Shear stress-strain response.

```

18 M = rho*vp^2;
19
20 //From wiki (https://en.wikipedia.org/wiki/Elastic\_modulus)
21 E = G*(3*M-4*G)/(M-G);
22 nu = (M-2*G)/(2*M-2*G);
23
24 K0 = 1.0;
25 phirad = pi*phi/180;
26 M = 6*sin(phirad)/(3-sin(phirad));
27
28 // Define the material:
29 add material # 1 type DruckerPragerArmstrongFrederickLT
30     mass_density = 0*kg/m^3
31     elastic_modulus = E
32     poisson_ratio = nu
33     druckerprager_k = M
34     armstrong_frederick_ha = ha*Pa
35     armstrong_frederick_cr = cr*Pa
36     isotropic_hardening_rate = 0*E
37     initial_confining_stress = 1*Pa;
38
39 // define the node:
40 add node # 1 at (0*m,0*m,1*m) with 3 dofs;
41 add node # 2 at (1*m,0*m,1*m) with 3 dofs;
42 add node # 3 at (1*m,1*m,1*m) with 3 dofs;
43 add node # 4 at (0*m,1*m,1*m) with 3 dofs;
44
45 add node # 5 at (0*m,0*m,0*m) with 3 dofs;
46 add node # 6 at (1*m,0*m,0*m) with 3 dofs;
47 add node # 7 at (1*m,1*m,0*m) with 3 dofs;
48 add node # 8 at (0*m,1*m,0*m) with 3 dofs;

```

```

49 // add equal degree of freedom in three directions
50 add constraint equal dof with master node # 2 and slave node # 3 dof to constrain ux;
51 add constraint equal dof with master node # 2 and slave node # 6 dof to constrain ux;
52 add constraint equal dof with master node # 2 and slave node # 7 dof to constrain ux;
53
54 add constraint equal dof with master node # 3 and slave node # 4 dof to constrain uy;
55 add constraint equal dof with master node # 3 and slave node # 8 dof to constrain uy;
56 add constraint equal dof with master node # 3 and slave node # 7 dof to constrain uy;
57
58 add constraint equal dof with master node # 1 and slave node # 2 dof to constrain uz;
59 add constraint equal dof with master node # 1 and slave node # 3 dof to constrain uz;
60 add constraint equal dof with master node # 1 and slave node # 4 dof to constrain uz;
61
62 // Define the element.
63 add element # 1 type 8NodeBrickLT with nodes (1, 2,3 , 4, 5, 6,7, 8) use material # 1;
64
65
66 new loading stage "confinement";
67 fix node # 1 dofs ux uy;
68 fix node # 2 dofs uy;
69 fix node # 4 dofs ux;
70
71 fix node # 5 dofs ux uy uz;
72 fix node # 6 dofs uy uz;
73 fix node # 7 dofs uz;
74 fix node # 8 dofs ux uz;
75
76 sigma_z = -3*p0/(1+2*K0);
77 sigma_x = K0*sigma_z;
78 sigma_y = K0*sigma_z;
79
80 //Z-face
81 add load # 1 to node # 1 type linear Fz = sigma_z*m^2/4;
82 add load # 2 to node # 2 type linear Fz = sigma_z*m^2/4;
83 add load # 3 to node # 3 type linear Fz = sigma_z*m^2/4;
84 add load # 4 to node # 4 type linear Fz = sigma_z*m^2/4;
85
86 //X-face
87 add load # 5 to node # 2 type linear Fx = sigma_x*m^2/4;
88 add load # 6 to node # 6 type linear Fx = sigma_x*m^2/4;
89 add load # 7 to node # 7 type linear Fx = sigma_x*m^2/4;
90 add load # 8 to node # 3 type linear Fx = sigma_x*m^2/4;
91
92 add load # 9 to node # 3 type linear Fy = sigma_y*m^2/4;
93 add load # 10 to node # 7 type linear Fy = sigma_y*m^2/4;
94 add load # 11 to node # 8 type linear Fy = sigma_y*m^2/4;
95 add load # 12 to node # 4 type linear Fy = sigma_y*m^2/4;
96
97 Nsteps_static=100;
98 define load factor increment 1/Nsteps_static;
99
100 define solver UMFPack;
101 define convergence test Norm_Displacement_Increment
102   tolerance = 1e-6
103   maximum_iterations = 100

```

```
104     verbose_level = 4;
105 define algorithm Newton ;
106
107 define NDMaterialLT constitutive integration algorithm Euler_One_Step
108     yield_function_relative_tolerance = 0.002
109     stress_relative_tolerance = 0.002
110     maximum_iterations = 1000;
111
112 simulate Nsteps_static steps using static algorithm;
113
114
115 new loading stage "shearing";
116 compute reaction forces;
117 add load # 13 to node # 1 type from_reactions;
118 add load # 14 to node # 4 type from_reactions;
119
120 free node # 1 dofs ux;
121 free node # 4 dofs ux;
122 fix node # 3 dofs uy;
123 fix node # 6 dofs ux;
124 fix node # 7 dofs ux uy;
125 fix node # 8 dofs uy;
126
127 add constraint equal dof with master node # 1 and slave node # 3 dof to constrain ux;
128 add constraint equal dof with master node # 1 and slave node # 4 dof to constrain ux;
129 add constraint equal dof with master node # 1 and slave node # 2 dof to constrain ux;
130 remove constraint equaldof node # 6;
131 remove constraint equaldof node # 7;
132 remove constraint equaldof node # 8;
133
134 n = 1;
135 while(n<=1)
136 {
137     add load # 14+n to node # n type path_time_series
138     Fx = 170.*kN
139     series_file = "path.txt";
140     n+=1;
141 }
142
143 define load factor increment 1/Nsteps;
144
145 define solver UMFPack;
146 define convergence test Norm_Displacement_Increment
147     tolerance = 1e-5
148     maximum_iterations = 100
149     verbose_level = 4;
150 define algorithm Newton ;
151
152 define NDMaterialLT constitutive integration algorithm Euler_One_Step
153     yield_function_relative_tolerance = 0.0002
154     stress_relative_tolerance = 0.002
155     maximum_iterations = 1000;
156
157 simulate Ncyc*Nsteps steps using static algorithm;
```

159 | **bye;**

The ESSI model fei files for this example can be downloaded [here](#).

17 Beam theory

Problem description: Length=6m, Width=1m, Height=1m, F=100N, E=1E8Pa, $\nu = 0.0$. The force direction was shown in Figure (53).

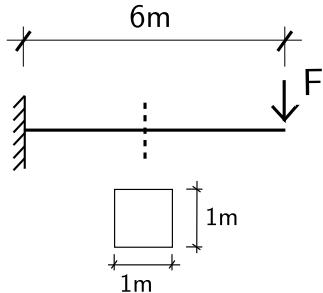


Figure 53: Problem description for cantilever beams

The basic idea to calculate the shear deformation of a beam is

$$\delta = \frac{FL}{GA_v} \quad (1)$$

where A_v is the not the gross cross sectional area of the beam. A_v should be the shear area. Thus,

$$\kappa = \frac{A}{A_v} \quad (2)$$

where κ is the form factor, shear correction factor or shear deformation coefficient, A is the gross sectional area and A_v is the shear area of the section.

The history of κ value is long.

1. Timoshenko (1940)¹ define the form factor for rectangular section is 1.5.
2. Cowper (1970)² gave the formula for the form factor:

$$\kappa = \frac{12 + 11\nu}{10(1 + \nu)} \quad (3)$$

3. Renton (1991)³ provided a closed form solution for shear area of rectangular sections. For a rectangular section of depth $2a$ and breadth $2b$.

$$\kappa = \frac{6}{5} + \left(\frac{\nu}{1 + \nu}\right)^2 \sum_{m=0}^{\infty} \sum_{n=1}^{\infty} \frac{144(b/a)^4}{\pi^6(2m+1)^2 n^2 [(2m+1)^2(b/2a)^2 + n^2]} \quad (4)$$

¹Strength of materials, Timoshenko, Krieger Pub Co, 1940

²Cowper, G. R. "The shear coefficient in Timoshenko's beam theory." Journal of applied mechanics 33.2 (1966): 335-340.

³Renton, J. D. "Generalized beam theory applied to shear stiffness." International Journal of Solids and Structures 27.15 (1991): 1955-1967.

For square cross section, $b = a$, therefore,

$$\kappa = \frac{6}{5} + \left(\frac{\nu}{1+\nu}\right)^2 \sum_{m=0}^{\infty} \sum_{n=1}^{\infty} \frac{144}{\pi^6 (2m+1)^2 n^2 [(2m+1)^2 (1/2)^2 + n^2]} \quad (5)$$

The summation of the series are very hard. *Matlab* and *Mathematica* cannot solve it directly. According to the Renton (1991), the intermediate values are given by

$$\kappa = \frac{6}{5} + C_1 \left(\frac{\nu}{1+\nu}\right)^2 \left(\frac{b}{a}\right)^4 \quad (6)$$

When $b = a$, the equation becomes

$$\kappa = \frac{6}{5} + 0.1392 \left(\frac{\nu}{1+\nu}\right)^2 \quad (7)$$

18 Verification of 8NodeBrick elements

18.1 Verification of 8NodeBrick cantilever beams

Problem description: Length=6m, Width=1m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.0$. Use the shear deformation coefficient $\kappa = 1.2$. The force direction was shown in Figure (54).

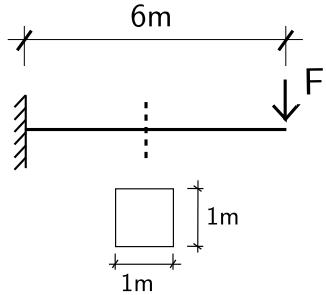


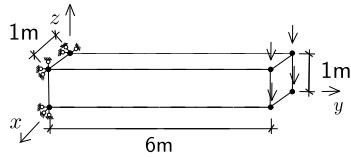
Figure 54: Problem description for cantilever beams

Theoretical displacement (bending and shear deformation):

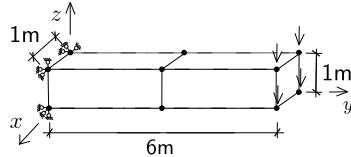
$$\begin{aligned}
 d &= \frac{FL^3}{3EI} + \frac{FL}{GA_v} \\
 &= \frac{FL^3}{3E\frac{bh^3}{12}} + \frac{FL}{\frac{E}{2(1+\nu)}\frac{bh}{\kappa}} \\
 &= \frac{100N \times 6^3 m^3}{3 \times 10^8 N/m^2 \times \frac{1}{12} m^4} + \frac{100N \times 6m}{\frac{10}{2} \times 10^7 N/m^2 \times 1m^2 \times \frac{5}{6}} \\
 &= 8.64 \times 10^{-4} m + 0.144 \times 10^{-4} m \\
 &= 8.784 \times 10^{-4} m
 \end{aligned} \tag{8}$$

Numerical model:

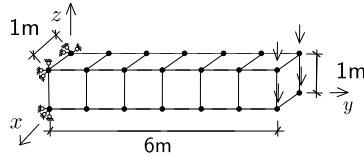
The 8NodeBrick elements were shown in Figure (55).



(a) One 8NodeBrick element



(b) Two 8NodeBrick elements



(c) Six 8NodeBrick elements

Figure 55: 8NodeBrick elements for cantilever beams

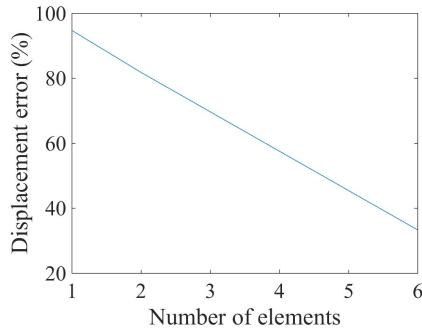
An example ESSI script is shown below.

All the ESSI results were listed in Table (1). The theoretical solution is 8.784E-04 m.

Table 1: Results for 8NodeBrick cantilever beams of different element numbers

Element number	1	2	6
8NodeBrick	4.61E-05 m	1.59E-04 m	5.84E-04 m
Error	94.75%	81.87%	33.52%

The errors were plotted in Figure (56).

Figure 56: 8NodeBrick cantilever beam for different element number
Displacement error versus Number of elements

The ESSI model fei files for the table above are here

18.2 Verification of 8NodeBrick cantilever beam for different Poisson's ratio

Problem description: Length=6m, Width=1m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.0 - 0.49$. The force direction was shown in Figure (57).

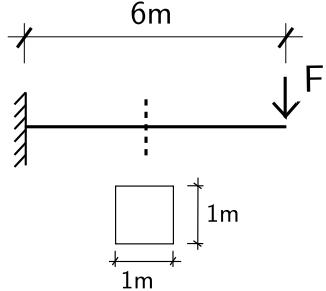


Figure 57: Problem description for cantilever beams of different Poisson's ratios

The theoretical solution for $\nu = 0.0$ was calculated below, while the solution for other Poisson's ratio were calculated by the similar process.

Theoretical displacement (bending and shear deformation):

$$\begin{aligned}
 d &= \frac{FL^3}{3EI} + \frac{FL}{GA_v} \\
 &= \frac{FL^3}{3E\frac{bh^3}{12}} + \frac{FL}{\frac{E}{2(1+\nu)} \frac{bh}{\kappa}} \\
 &= \frac{100N \times 6^3 m^3}{3 \times 10^8 N/m^2 \times \frac{1}{12} m^4} + \frac{100N \times 6m}{\frac{10}{2} \times 10^7 N/m^2 \times 1m^2 \times \frac{5}{6}} \\
 &= 8.64 \times 10^{-4} m + 0.144 \times 10^{-4} m \\
 &= 8.784 \times 10^{-4} m
 \end{aligned} \tag{9}$$

The rotation angle at the end:

$$\theta = \frac{FL^2}{2EI} = \frac{100N \times 6^2 m^2}{2 \times 10^8 N/m^2 \times \frac{1}{12} m^4} = 2.16 \times 10^{-4} rad = 0.0124^\circ \tag{10}$$

The 8NodeBrick elements for cantilever beams of different Poisson's ratios were shown in Figure (58):

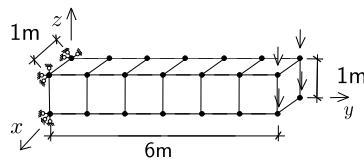


Figure 58: 8NodeBrick elements for cantilever beams of different Poisson's ratios

All the displacement results were listed in Table (2) - (4).

Table 2: ***Displacement*** results for 8NodeBrick cantilever beams
with element side length 1 m

Poisson's ratio	8NodeBrick displacement	Theory displacement (bending)	Theory displacement (shear)	Theory displacement(all)	Error
0.00	5.840E-04 m	8.640E-04 m	1.440E-05 m	8.784E-04 m	33.52%
0.05	5.924E-04 m	8.640E-04 m	1.512E-05 m	8.791E-04 m	32.62%
0.10	5.969E-04 m	8.640E-04 m	1.586E-05 m	8.799E-04 m	32.16%
0.15	5.971E-04 m	8.640E-04 m	1.659E-05 m	8.806E-04 m	32.20%
0.20	5.922E-04 m	8.640E-04 m	1.734E-05 m	8.813E-04 m	32.81%
0.25	5.814E-04 m	8.640E-04 m	1.808E-05 m	8.821E-04 m	34.09%
0.30	5.634E-04 m	8.640E-04 m	1.884E-05 m	8.828E-04 m	36.19%
0.35	5.364E-04 m	8.640E-04 m	1.959E-05 m	8.836E-04 m	39.29%
0.40	4.970E-04 m	8.640E-04 m	2.035E-05 m	8.844E-04 m	43.80%
0.45	4.353E-04 m	8.640E-04 m	2.111E-05 m	8.851E-04 m	50.82%
0.49	3.142E-04 m	8.640E-04 m	2.173E-05 m	8.857E-04 m	64.52%

Then, in the same geometry, the element side length was cut into 0.5m.

Table 3: ***Displacement*** results for 8NodeBrick cantilever beams
with element side length 0.5 m

Poisson's ratio	8NodeBrick displacement	Theory displacement (bending)	Theory displacement (shear)	Theory displacement(all)	Error
0.00	7.787E-04 m	8.640E-04 m	1.440E-05 m	8.784E-04 m	11.35%
0.05	7.824E-04 m	8.640E-04 m	1.512E-05 m	8.791E-04 m	11.00%
0.10	7.839E-04 m	8.640E-04 m	1.586E-05 m	8.799E-04 m	10.91%
0.15	7.829E-04 m	8.640E-04 m	1.659E-05 m	8.806E-04 m	11.09%
0.20	7.790E-04 m	8.640E-04 m	1.734E-05 m	8.813E-04 m	11.61%
0.25	7.717E-04 m	8.640E-04 m	1.808E-05 m	8.821E-04 m	12.51%
0.30	7.597E-04 m	8.640E-04 m	1.884E-05 m	8.828E-04 m	13.95%
0.35	7.406E-04 m	8.640E-04 m	1.959E-05 m	8.836E-04 m	16.18%
0.40	7.089E-04 m	8.640E-04 m	2.035E-05 m	8.844E-04 m	19.84%
0.45	6.466E-04 m	8.640E-04 m	2.111E-05 m	8.851E-04 m	26.95%
0.49	4.990E-04 m	8.640E-04 m	2.173E-05 m	8.857E-04 m	43.66%

Finally, in the same geometry, the element side length was cut into 0.25m.

Table 4: ***Displacement*** results for 8NodeBrick cantilever beams
with element side length 0.25 m

Poisson's ratio	8NodeBrick displacement	Theory displacement (bending)	Theory displacement (shear)	Theory displacement(all)	Error
0.00	8.511E-04 m	8.640E-04 m	1.440E-05 m	8.784E-04 m	3.11%
0.05	8.525E-04 m	8.640E-04 m	1.512E-05 m	8.791E-04 m	3.03%
0.10	8.527E-04 m	8.640E-04 m	1.586E-05 m	8.799E-04 m	3.09%
0.15	8.518E-04 m	8.640E-04 m	1.659E-05 m	8.806E-04 m	3.27%
0.20	8.494E-04 m	8.640E-04 m	1.734E-05 m	8.813E-04 m	3.62%
0.25	8.455E-04 m	8.640E-04 m	1.808E-05 m	8.821E-04 m	4.15%
0.30	8.393E-04 m	8.640E-04 m	1.884E-05 m	8.828E-04 m	4.93%
0.35	8.299E-04 m	8.640E-04 m	1.959E-05 m	8.836E-04 m	6.08%
0.40	8.141E-04 m	8.640E-04 m	2.035E-05 m	8.844E-04 m	7.94%
0.45	7.801E-04 m	8.640E-04 m	2.111E-05 m	8.851E-04 m	11.86%
0.49	6.603E-04 m	8.640E-04 m	2.173E-05 m	8.857E-04 m	25.45%

The errors were plotted in Figure (59).

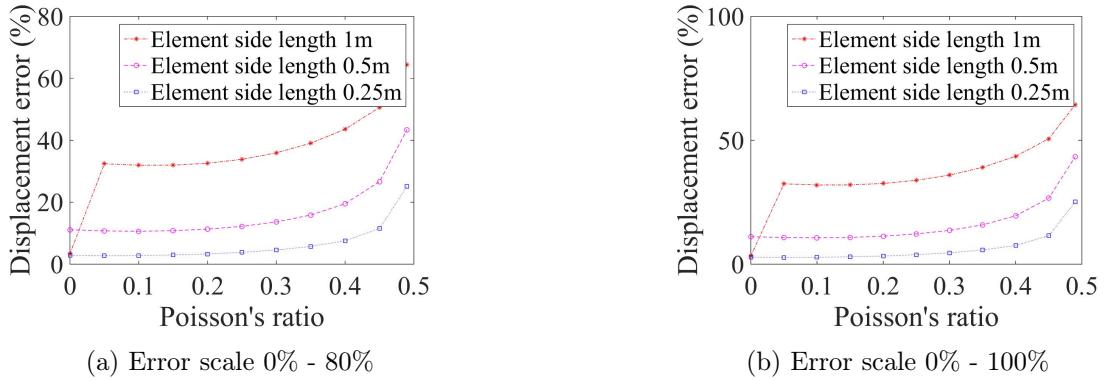


Figure 59: 8NodeBrick cantilever beam for different Poisson's ratio
Displacement error versus Poisson's ratio

The angle results were listed in Table (5).

Table 5: ***Rotation angle*** results for 8NodeBrick cantilever beams
with element side length 1 m

Poisson's ratio	8NodeBrick angle (unit: $^{\circ}$)	Theory angle (unit: $^{\circ}$)	Error
0.00	8.25E-03	1.24E-02	33.46%
0.05	8.36E-03	1.24E-02	32.55%
0.10	8.42E-03	1.24E-02	32.08%
0.15	8.42E-03	1.24E-02	32.10%
0.20	8.35E-03	1.24E-02	32.67%
0.25	8.20E-03	1.24E-02	33.90%
0.30	7.95E-03	1.24E-02	35.89%
0.35	7.59E-03	1.24E-02	38.83%
0.40	7.07E-03	1.24E-02	43.00%
0.45	6.30E-03	1.24E-02	49.21%
0.49	4.93E-03	1.24E-02	60.20%

Then, in the same geometry, element side length was cut into 0.5m. The angle results were listed in Table (6).

Table 6: ***Rotation angle*** results for 8NodeBrick cantilever beams
with element side length 0.5 m

Poisson's ratio	8NodeBrick angle (unit: $^{\circ}$)	Theory angle (unit: $^{\circ}$)	Error
0.00	1.10E-02	1.24E-02	11.28%
0.05	1.10E-02	1.24E-02	10.91%
0.10	1.11E-02	1.24E-02	10.78%
0.15	1.10E-02	1.24E-02	10.90%
0.20	1.10E-02	1.24E-02	11.32%
0.25	1.09E-02	1.24E-02	12.09%
0.30	1.07E-02	1.24E-02	13.33%
0.35	1.05E-02	1.24E-02	15.29%
0.40	1.01E-02	1.24E-02	18.53%
0.45	9.32E-03	1.24E-02	24.87%
0.49	7.52E-03	1.24E-02	39.35%

Finally, in the same geometry, element side length was cut into 0.25m. The angle results were listed in Table (7).

Table 7: ***Rotation angle*** results for 8NodeBrick cantilever beams
with element side length 0.25 m

Poisson's ratio	8NodeBrick angle (unit: $^{\circ}$)	Theory angle (unit: $^{\circ}$)	Error
0.00	1.20E-02	1.24E-02	3.06%
0.05	1.20E-02	1.24E-02	2.97%
0.10	1.20E-02	1.24E-02	2.99%
0.15	1.20E-02	1.24E-02	3.12%
0.20	1.20E-02	1.24E-02	3.38%
0.25	1.19E-02	1.24E-02	3.79%
0.30	1.19E-02	1.24E-02	4.40%
0.35	1.17E-02	1.24E-02	5.33%
0.40	1.15E-02	1.24E-02	6.87%
0.45	1.11E-02	1.24E-02	10.22%
0.49	9.64E-03	1.24E-02	22.23%

The errors were plotted in Figure (60).

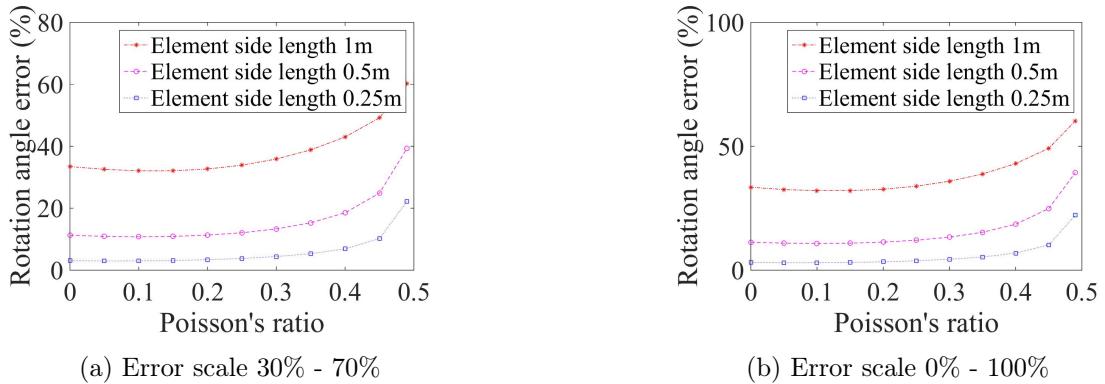


Figure 60: 8NodeBrick cantilever beam for different Poisson's ratio
Rotation angle error versus Poisson's ratio

The ESSI model fei files for the table above are here

18.3 Test of irregular shaped 8NodeBrick cantilever beams

Cantilever model was used as an example. Three different shapes were tested.

In the first test, the upper two nodes of each element were moved one half element size along the $y - axis$, while the lower two nodes were kept at the same location. The element shape was shown in Figure (61).

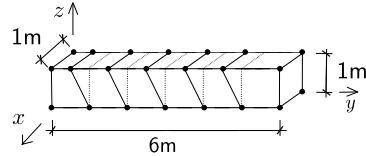


Figure 61: 8NodeBrick cantilever beams for irregular *Shape 1*

In the second test, the upper two nodes of each element were moved 90% element size along the $y - axis$, while the lower two nodes were moved 90% element size in the other direction along the $y - axis$. The element shape was shown in Figure (62).

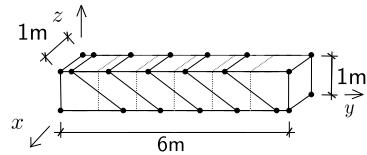


Figure 62: 8NodeBrick cantilever beams for irregular *Shape 2*

In the third test, the upper two nodes of each element were moved one half element size with different directions along the $y - axis$, while the lower two nodes were kept at the same location. The element shape was shown in Figure (63).

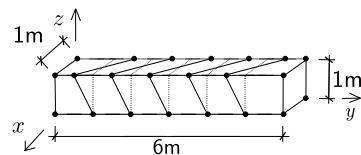
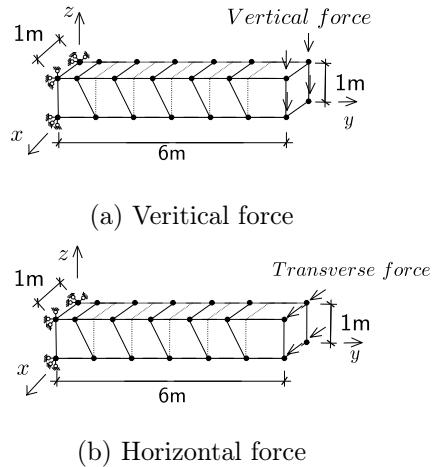
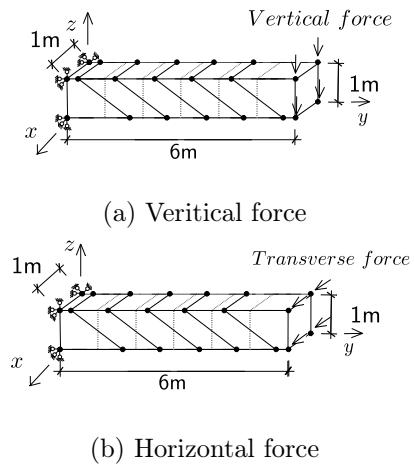
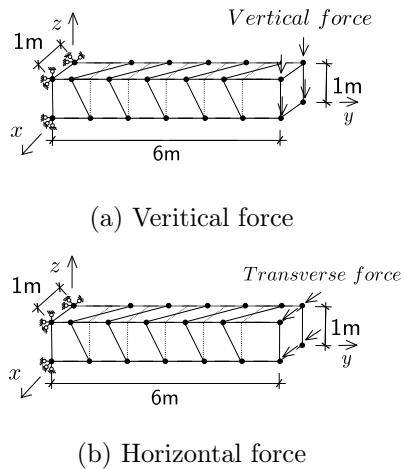


Figure 63: 8NodeBrick cantilever beams for irregular *Shape 3*

The boundary conditions were shown in Figure (64), (65) and (66) .

Figure 64: 8NodeBrick cantilever beam boundary conditions for irregular ***Shape 1***Figure 65: 8NodeBrick cantilever beam boundary conditions for irregular ***Shape 2***Figure 66: 8NodeBrick cantilever beam boundary conditions for irregular ***Shape 3***

The ESSI results were listed in Table (8).

Table 8: Results for 8NodeBrick cantilever beams of irregular shapes

Element Type	Force direction	Normal shape	Shape 1	Shape 2	Shape 3
8NodeBrick	Vertical (z)	5.840E-04 m	5.751E-04 m	2.959E-04 m	3.883E-04 m
8NodeBrick	Transverse (y)	5.840E-04 m	4.529E-04 m	1.390E-04 m	4.744E-04 m
Theoretical	-	8.784E-04 m	8.784E-04 m	8.784E-04 m	8.784E-04 m

The errors were listed in Table (9) and (10).

Table 9: Errors for irregular shaped 8NodeBrick compared to theoretical solution

Element Type	Force direction	Normal shape	Shape 1	Shape 2	Shape 3
8NodeBrick	Vertical (z)	33.52%	34.53%	66.31%	55.79%
8NodeBrick	Transverse (y)	33.52%	48.44%	84.18%	45.99%

Table 10: Errors for irregular shaped 8NodeBrick compared to normal shape

Element Type	Force direction	Normal shape	Shape 1	Shape 2	Shape 3
8NodeBrick	Vertical (z)	0.00%	1.52%	49.33%	33.51%
8NodeBrick	Transverse (y)	0.00%	22.45%	76.20%	18.77%

The ESSI model fei files for the table above are here

Then, the irregular beam was divided into small elements.

Problem description: Length=12m, Width=2m, Height=2m, $q=400\text{N/m}$, $E=1\text{E}8\text{Pa}$, $\nu = 0.0$. Use the shear deformation coefficient $\kappa = 1.2$. The force direction was shown in Figure (67).

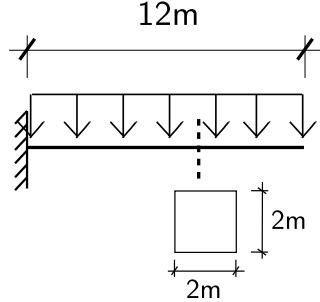


Figure 67: Problem description for cantilever beams under uniform load

Theoretical displacement (bending and shear deformation):

$$\begin{aligned}
 d &= \frac{qL^4}{8EI} + \frac{q\frac{L^2}{2}}{GA_v} \\
 &= \frac{qL^4}{8E\frac{bh^3}{12}} + \frac{q\frac{L^2}{2}}{\frac{E}{2(1+\nu)} \frac{bh}{\kappa}} \\
 &= \frac{400\text{N/m} \times 12^4\text{m}^4}{8 \times 10^8\text{N/m}^2 \times \frac{2^4}{12}\text{m}^4} + \frac{400\text{N/m} \times \frac{12^2}{2}\text{m}^2}{\frac{10^8}{2}\text{N/m}^2 \times 2\text{m} \times 2\text{m} \times \frac{5}{6}} \\
 &= 7.776 \times 10^{-3}\text{m} + 1.728 \times 10^{-4}\text{m} \\
 &= 7.9488 \times 10^{-3}\text{m}
 \end{aligned} \tag{11}$$

The ESSI displacement results were listed in Table (11).

Table 11: Results for 8NodeBrick cantilever beams of irregular shapes with more elements

Element Type	Shape	Force direction	Number of division		
			1	2	4
8NodeBrick	shape1	Vertical (z)	5.37E-03 m	7.08E-03 m	7.71E-03 m
8NodeBrick	shape1	Transverse (y)	4.60E-03 m	6.66E-03 m	7.58E-03 m
8NodeBrick	shape2	Vertical (z)	2.74E-03 m	4.75E-03 m	6.43E-03 m
8NodeBrick	shape2	Transverse (y)	1.46E-03 m	2.72E-03 m	4.63E-03 m
8NodeBrick	shape3	Vertical (z)	9.21E-04 m	6.60E-03 m	7.56E-03 m
8NodeBrick	shape3	Transverse (y)	1.09E-03 m	6.09E-03 m	7.37E-03 m
Theoretical solution			7.95E-03 m	7.95E-03 m	7.95E-03 m

The error were listed in Table (12).

Table 12: Errors for 8NodeBrick cantilever beams of irregular shapes with more elements

Element Type	Shape	Force direction	Number of division		
			1	2	4
8NodeBrick	shape1	Vertical (z)	32.42%	10.95%	3.01%
8NodeBrick	shape1	Transverse (y)	42.16%	16.17%	4.69%
8NodeBrick	shape2	Vertical (z)	65.59%	40.22%	19.05%
8NodeBrick	shape2	Transverse (y)	81.57%	65.76%	41.81%
8NodeBrick	shape3	Vertical (z)	88.42%	16.97%	4.89%
8NodeBrick	shape3	Transverse (y)	86.24%	23.36%	7.28%

The errors were shown in Figure (68), (69) and (70).

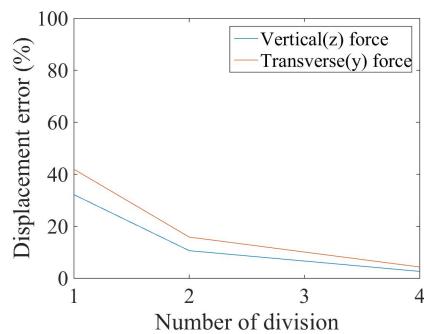


Figure 68: 8NodeBrick cantilever beam for irregular **Shape 1**
Displacement error versus Number of division

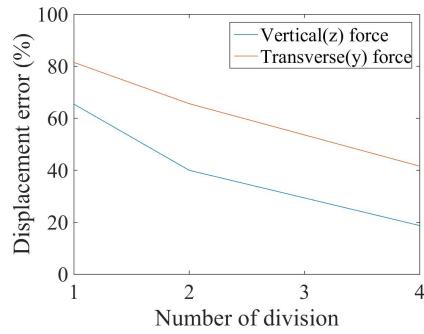


Figure 69: 8NodeBrick cantilever beam for irregular **Shape 2**
Displacement error versus Number of division

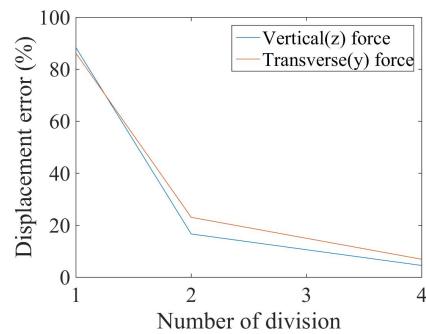


Figure 70: 8NodeBrick cantilever beam for irregular *Shape 3*
Displacement error versus Number of division

The ESSI model fei files for the table above are here

In this section, the beam was cut into smaller elements with element side length 0.5m and 0.25m respectively. And the element side length of the original models is 1.0m. The numerical models were shown in Figure (71), (72) and (73).

Number of division 1:

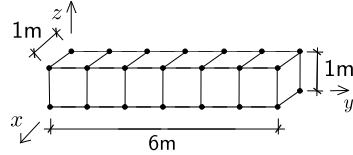


Figure 71: 8NodeBrick clamped beams with element side length 1.0m

Number of division 2:

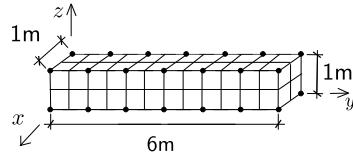


Figure 72: 8NodeBrick clamped beams with element side length 0.5m

Number of division 4:

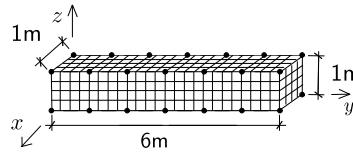


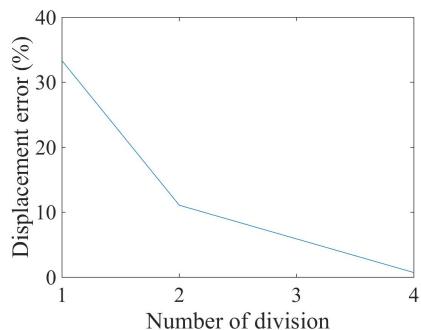
Figure 73: 8NodeBrick clamped beams with element side length 0.25m

The ESSI results were listed in Table (13). The theoretical solution is 1.60E-5 m.

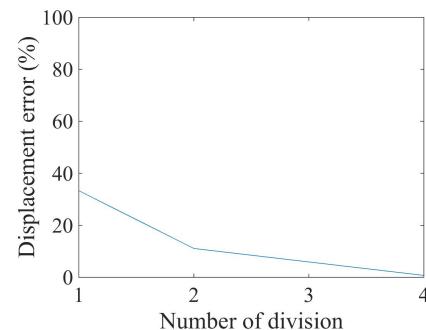
Table 13: Results for 8NodeBrick clamped beams with more elements

Element Type	Element side length		
	1 m	0.5 m	0.25 m
8NodeBrick	1.10E-05 m	1.47E-05 m	1.64E-05 m
Error	33.33%	11.09%	0.73%

The errors were plotted in Figure (74).



(a) Error scale 0% - 40%



(b) Error scale 0% - 100%

Figure 74: 8NodeBrick clamped beam for different element number
Displacement error versus Number of division

The ESSI model fei files for the table above are here

18.4 Verification of 8NodeBrick stress in cantilever beams

Problem description: Length=6m, Width=1m, Height=1m, Force=100N, $E=1E8Pa$, $\nu = 0.0$. Use the shear deformation coefficient $\kappa = 1.2$. The force direction was shown in Figure (75).

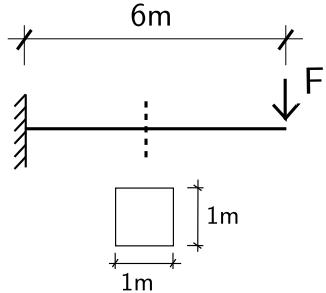


Figure 75: Problem description for cantilever beams of stress verification

The theoretical solution for the stress was calculated below.

The 8NodeBrick elements were shown in Figure (76).

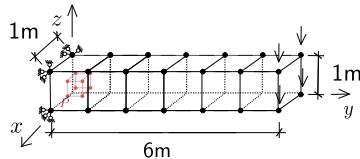


Figure 76: 8NodeBrick for cantilever beams of stress verification

The bending moment at the Gassian Point is

$$M = F(L - P_y) = 100N \times (6 - 0.2113)m = 578.87N \cdot m \quad (12)$$

The bending modulus is

$$I = \frac{bh^3}{12} = \frac{1}{12}m^4 \quad (13)$$

Therefore, the theoretical stress is

$$\sigma = \frac{M \cdot z}{I} = \frac{578.87N \cdot m \times (0.5 - 0.2113)m}{\frac{1}{12}m^4} = 2005Pa \quad (14)$$

To get a better result, the same geometry beam was also cut into small elements. When more elements were used, the theoretical stress was calculated again with the new coordinates. The calculation process is similar to the process above.

The numerical models were shown in Figure (77), (78) and (79).

Number of division 1:

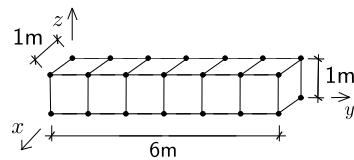


Figure 77: 8NodeBrick stress with element side length 1.0m

Number of division 2:

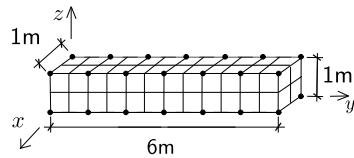


Figure 78: 8NodeBrick stress with element side length 0.5m

Number of division 4:

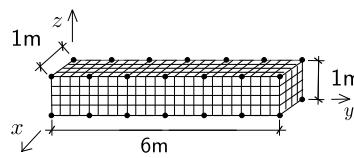


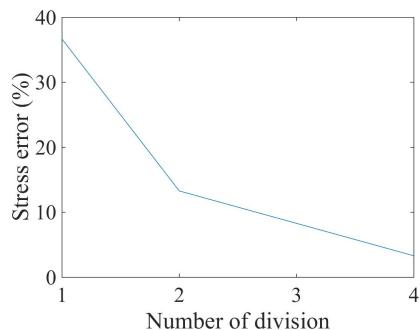
Figure 79: 8NodeBrick stress with element side length 0.25m

All the stress results were listed in Table (14).

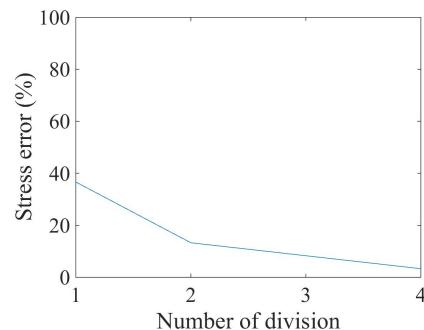
Table 14: Results for 8NodeBrick stress with more elements

Element Type	Element side length		
	1 m	0.5 m	0.25 m
8NodeBrick	1270.17 Pa	2418.60 Pa	3085.48 Pa
Theoretical	2005.26 Pa	2789.23 Pa	3191.27 Pa
Error	36.66%	13.29%	3.31%

The errors were plotted in Figure (80).



(a) Error scale 0% - 40%



(b) Error scale 0% - 100%

Figure 80: 8NodeBrick cantilever beams for stress verification

Stress error versus Number of division

The ESSI model fei files for the table above are here

18.5 Verification of 8NodeBrick square plate with four edges clamped

Problem description: Length=20m, Width=20m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.3$.

The four edges are clamped.

The load is the uniform normal pressure on the whole plate.

The plate flexural rigidity is

$$D = \frac{Eh^3}{12(1 - \nu^2)} = \frac{10^8 N/m^2 \times 1^3 m^3}{12 \times (1 - 0.3^2)} = 9.1575 \times 10^6 N \cdot m \quad (15)$$

The theoretical solution is

$$d = \alpha_c \frac{qa^4}{D} = 0.00406 \times \frac{100 N/m^2 \times 20^4 m^4}{9.1575 \times 10^6 N \cdot m} = 2.2015 \times 10^{-3} m \quad (16)$$

where α_c is a coefficient, which depends on the ratio of plate length to width. In this problem, the coefficient⁴ α_c is 0.00406.

The 8NodeBrick were shown in Figure (81) - (86).

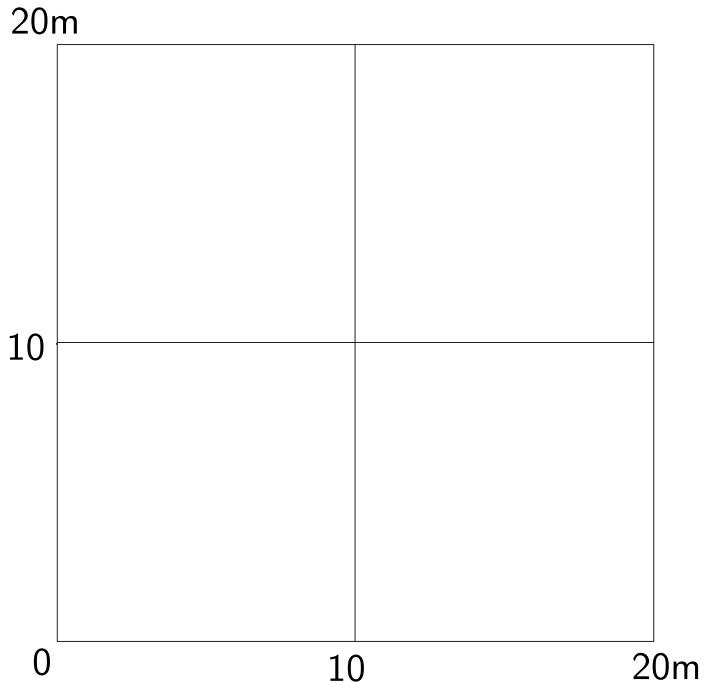


Figure 81: 8NodeBrick edge clamped square plate with element side length 10m

⁴Stephen Timoshenko, Theory of plates and shells (2nd edition). MrGRAW-Hill Inc, page120, 1959.

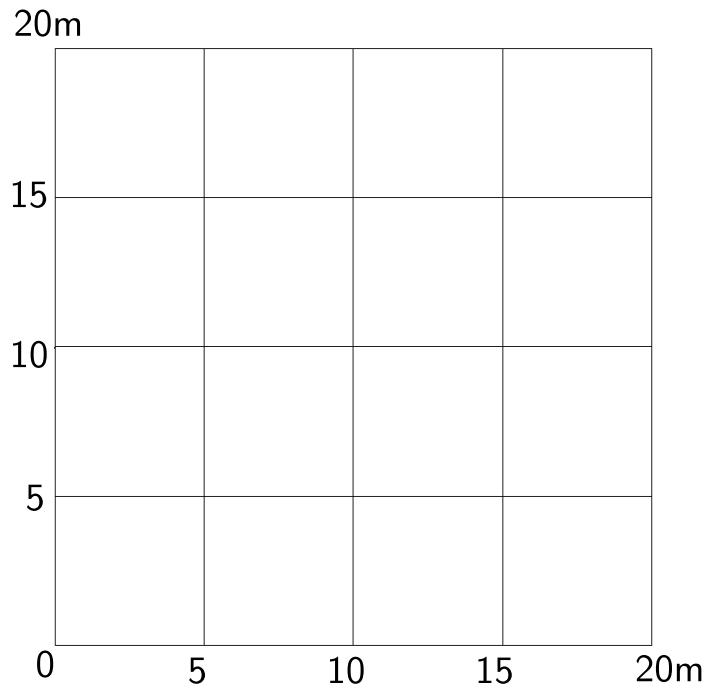


Figure 82: 8NodeBrick edge clamped square plate with element side length 5m

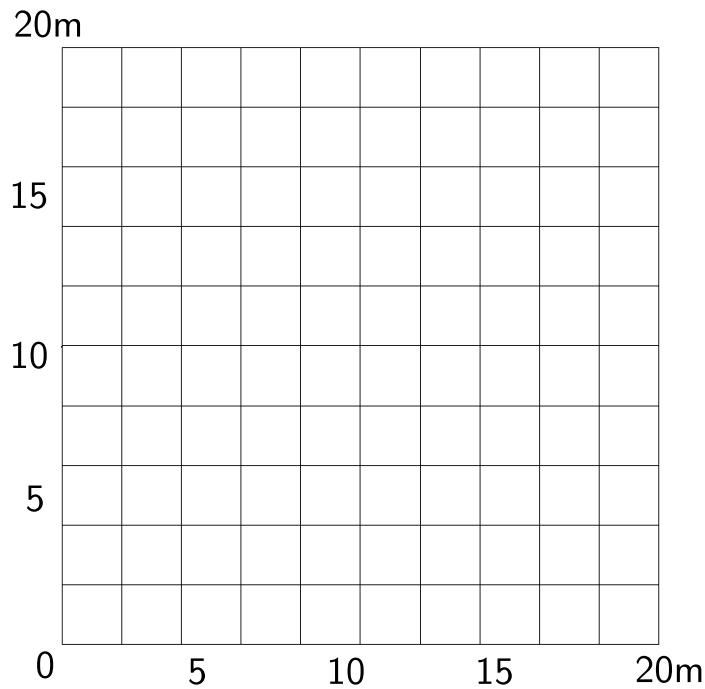


Figure 83: 8NodeBrick edge clamped square plate with element side length 2m

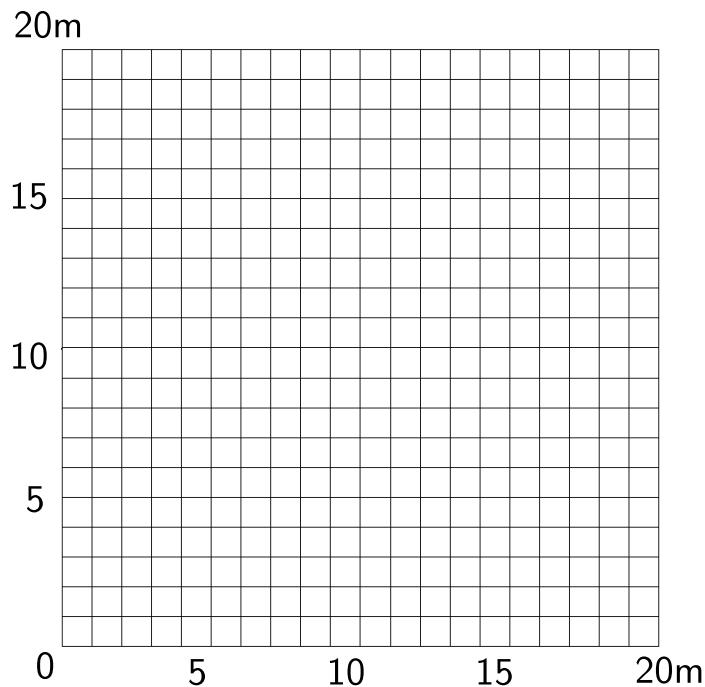


Figure 84: 8NodeBrick edge clamped square plate with element side length 1m

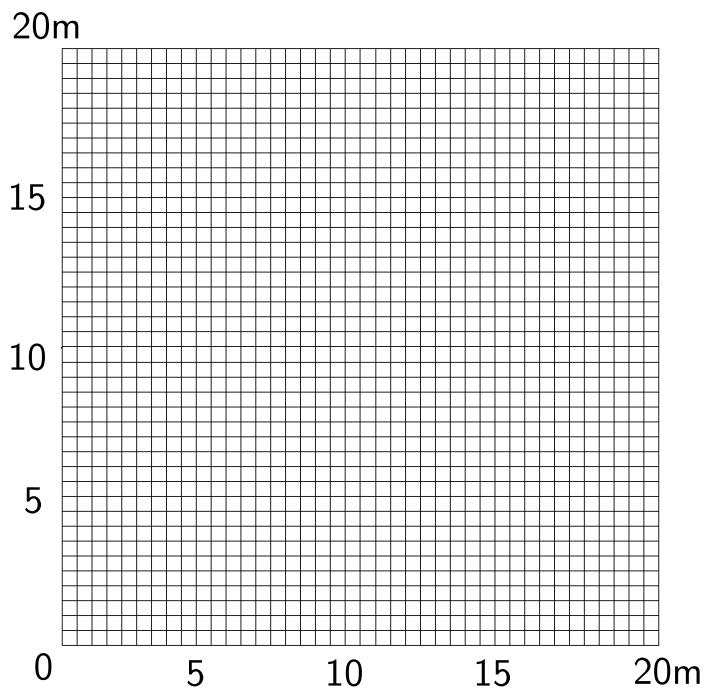


Figure 85: 8NodeBrick edge clamped square plate with element side length 0.5m

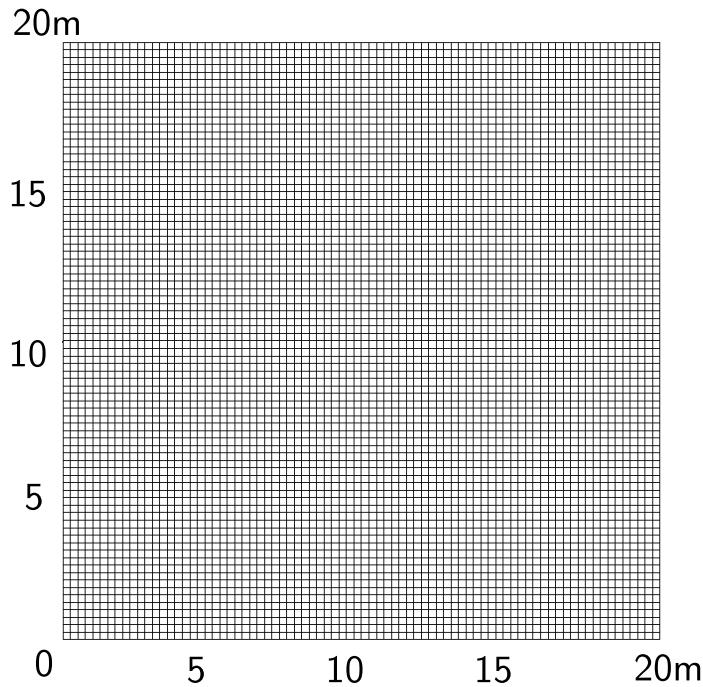


Figure 86: 8NodeBrick edge clamped square plate with element side length 0.25m

The results were listed in Table (15).

Table 15: Results for 8NodeBrick square plate with four edges clamped

Element type	8NodeBrick	8NodeBrick	8NodeBrick	Theoretical displacement
Number of layers	1layer	2layers	4layers	
Element side length	Height:1.00m	Height:0.50m	Height:0.25m	
10m	9.75E-05 m	9.75E-05 m	9.75E-05 m	2.20E-03 m
5m	3.28E-04 m	3.32E-04 m	3.32E-04 m	2.20E-03 m
2m	1.04E-03 m	1.10E-03 m	1.12E-03 m	2.20E-03 m
1m	1.56E-03 m	1.74E-03 m	1.79E-03 m	2.20E-03 m
0.5m	1.80E-03 m	2.30E-03 m	2.12E-03 m	2.20E-03 m
0.25m	1.87E-03 m	2.14E-03 m	2.23E-03 m	2.20E-03 m

The errors were listed in Table (16).

Table 16: Errors for 8NodeBrick square plate with four edges clamped

Element type	8NodeBrick	8NodeBrick	8NodeBrick
Number of layers	1layer	2layers	4layers
Element side length	Height:1.00m	Height:0.50m	Height:0.25m
10m	95.57%	95.57%	95.57%
5m	85.09%	84.94%	84.91%
2m	52.98%	50.09%	49.25%
1m	28.93%	21.17%	18.72%
0.5m	18.26%	4.58%	3.56%
0.25m	15.05%	2.70%	1.37%

The errors were plotted in Figure (87).

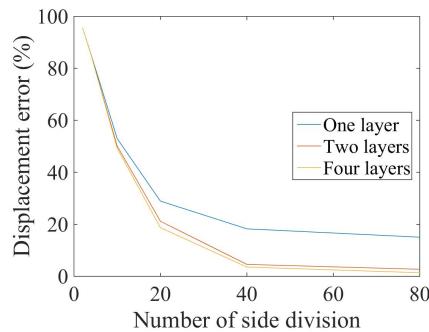


Figure 87: 8NodeBrick square plate with edge clamped
Displacement error versus Number of side division

The ESSI model fei files for the table above are here

18.6 Verification of 8NodeBrick square plate with four edges simply supported

Problem description: Length=20m, Width=20m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.3$.

The four edges are simply supported.

The load is the uniform normal pressure on the whole plate.

The plate flexural rigidity is

$$D = \frac{Eh^3}{12(1 - \nu^2)} = \frac{10^8 N/m^2 \times 1^3 m^3}{12 \times (1 - 0.3^2)} = 9.1575 \times 10^6 N \cdot m \quad (17)$$

The theoretical solution is

$$d = \alpha_s \frac{qa^4}{D} = 0.00126 \times \frac{100 N/m^2 \times 20^4 m^4}{9.1575 \times 10^6 N \cdot m} = 7.0936 \times 10^{-3} m \quad (18)$$

where α_s is a coefficient, which depends on the ratio of plate length to width. In this problem, the coefficient⁵ α_s is 0.00126.

The 8NodeBrick were shown in Figure (88) - (93).

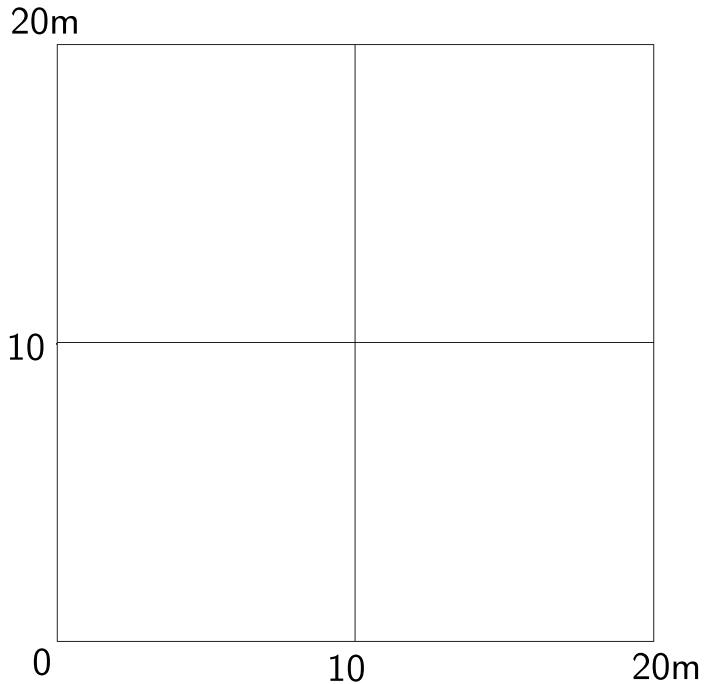


Figure 88: 8NodeBrick edge simply supported square plate with element side length 10m

⁵Stephen Timoshenko, Theory of plates and shells (2nd edition). MrGRAW-Hill Inc, page202, 1959.

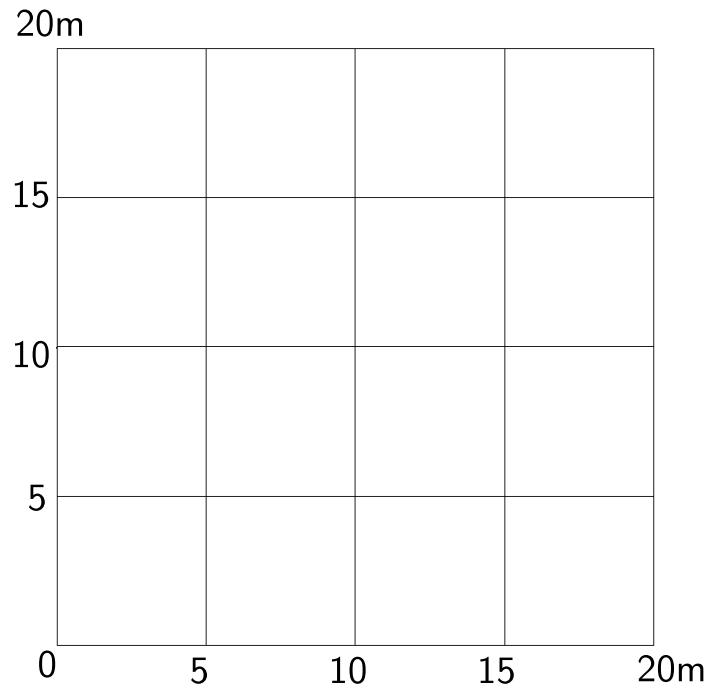


Figure 89: 8NodeBrick edge simply supported square plate with element side length 5m

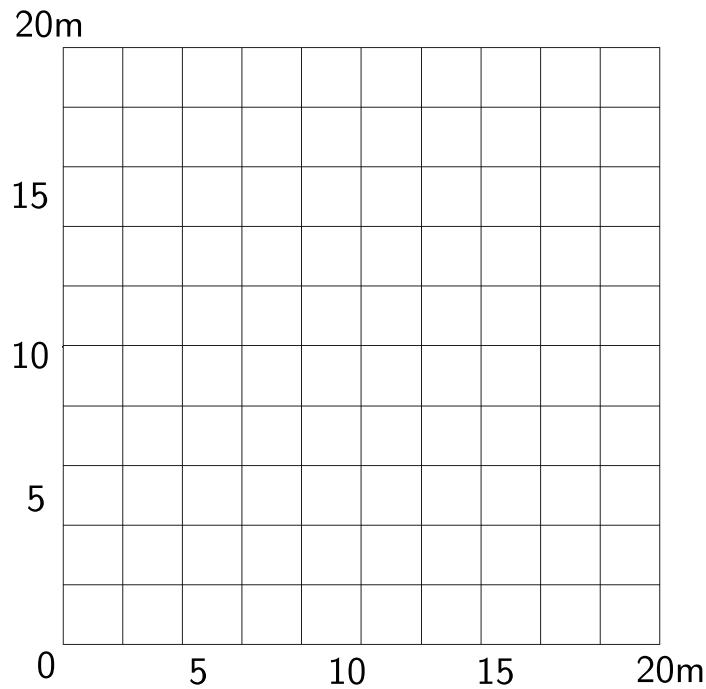


Figure 90: 8NodeBrick edge simply supported square plate with element side length 2m

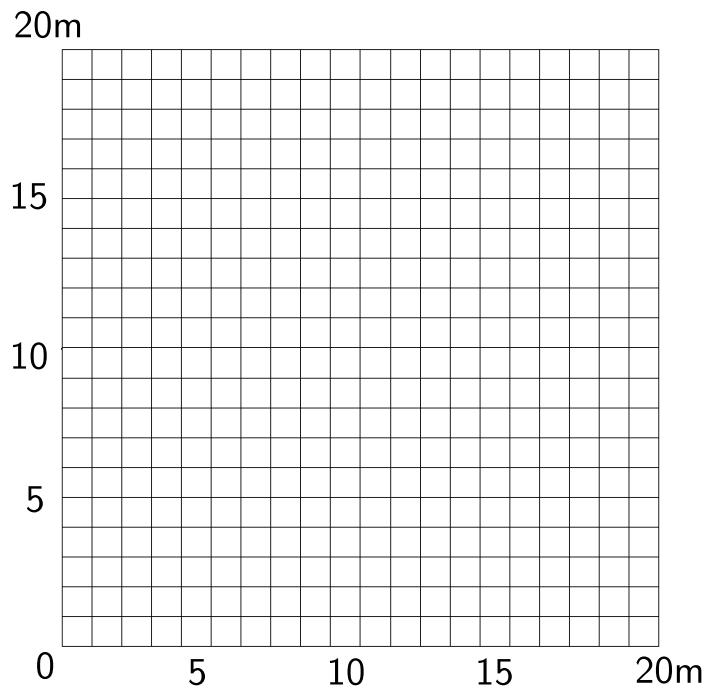


Figure 91: 8NodeBrick edge simply supported square plate with element side length 1m

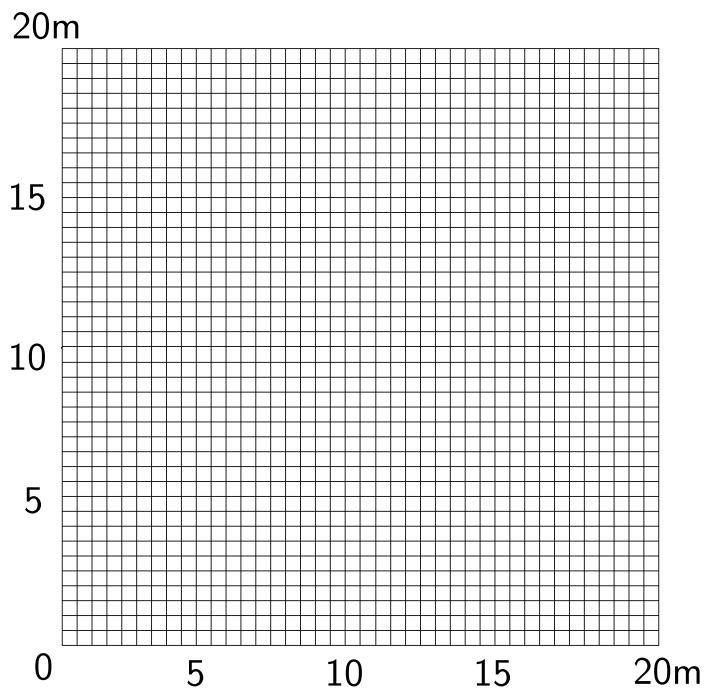


Figure 92: 8NodeBrick edge simply supported square plate with element side length 0.5m

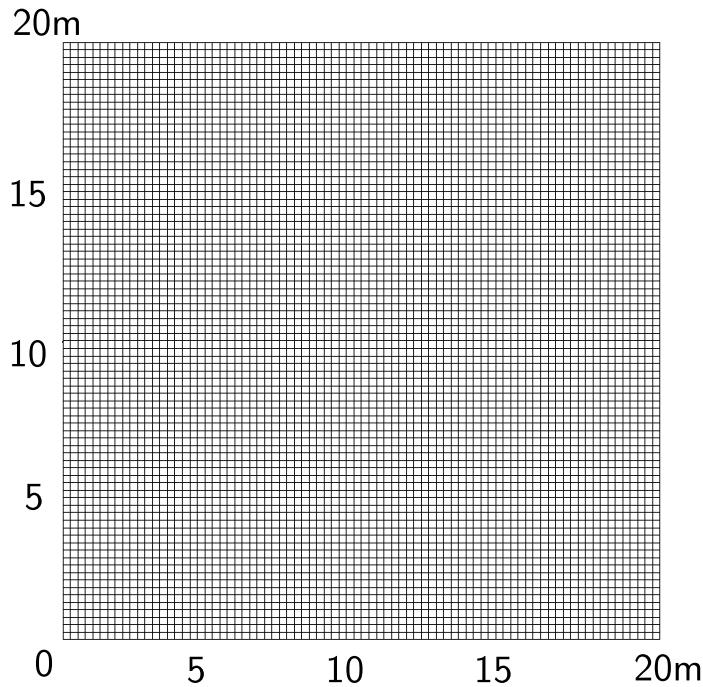


Figure 93: 8NodeBrick edge simply supported square plate with element side length 0.25m

The results were listed in Table (17).

Table 17: Results for 8NodeBrick square plate with four edges simply supported

Element type	8NodeBrick	8NodeBrick	Theoretical displacement
Number of layers	2layers	4layers	
Element side length	Height:0.50m	Height:0.25m	
10m	3.75E-004 m	3.76E-004 m	7.09E-03 m
5m	1.34E-003 m	1.35E-003 m	7.09E-03 m
2m	4.16E-003 m	4.27E-003 m	7.09E-03 m
1m	5.98E-003 m	6.22E-003 m	7.09E-03 m
0.5m	6.75E-003 m	7.04E-003 m	7.09E-03 m
0.25m	8.07E-003 m	7.30E-003 m	7.09E-03 m

The errors were listed in Table (18).

Table 18: Errors for 8NodeBrick square plate with four edges simply supported

Element type	8NodeBrick	8NodeBrick
Number of layers	2layers	4layers
Element side length	Height:0.50m	Height:0.25m
10m	94.72%	94.71%
5m	81.05%	80.91%
2m	41.31%	39.79%
1m	15.64%	12.38%
0.5m	4.88%	0.70%
0.25m	13.74%	2.86%

The errors were plotted in Figure (94).

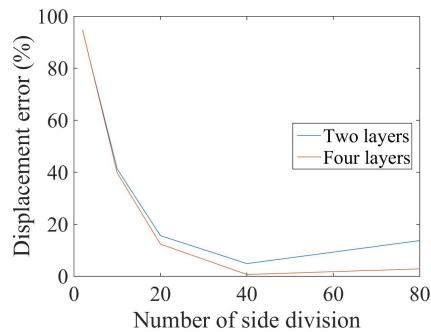


Figure 94: 8NodeBrick square plate with four edges simply supported
Displacement error versus Number of side division

The ESSI model fei files for the table above are here

18.7 Verification of 8NodeBrick circular plate with all edges clamped

Problem description: Diameter=20m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.3$.

The four edges are clamped.

The load is the uniform normal pressure on the whole plate.

The plate flexural rigidity is

$$D = \frac{Eh^3}{12(1-\nu^2)} = \frac{10^8 N/m^2 \times 1^3 m^3}{12 \times (1 - 0.3^2)} = 9.1575 \times 10^6 N \cdot m \quad (19)$$

The theoretical solution⁶ is

$$d = \frac{qa^4}{64D} = \frac{100N/m^2 \times 10^4 m^4}{64 \times 9.1575 \times 10^6 N \cdot m} = 1.7106 \times 10^{-3} m \quad (20)$$

The 8NodeBrick were shown in Figure (95) - (100).

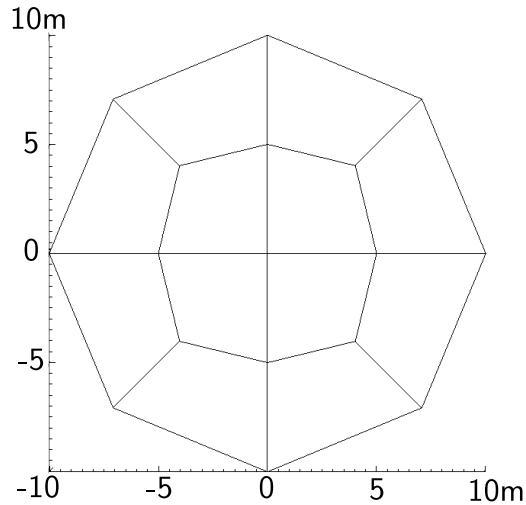


Figure 95: 8NodeBrick edge clamped circular plate with element side length 10m

⁶Stephen Timoshenko, Theory of plates and shells (2nd edition). MrGRAW-Hill Inc, page55, 1959.

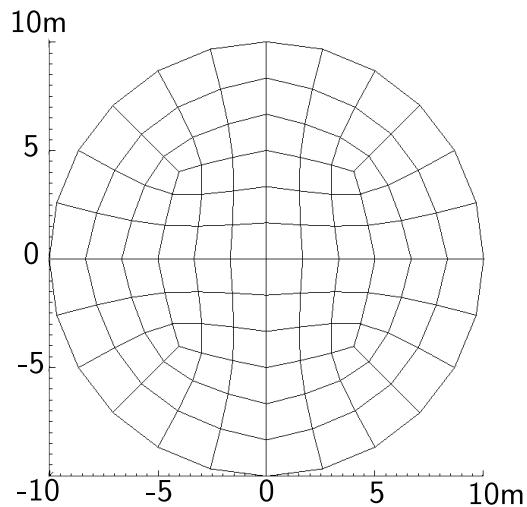


Figure 96: 8NodeBrick edge clamped circular plate with element side length 5m

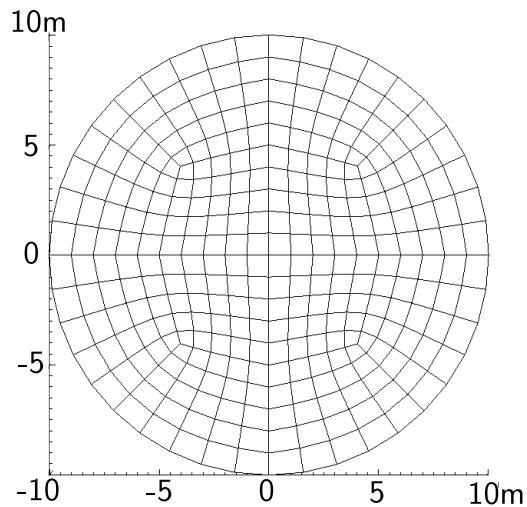


Figure 97: 8NodeBrick edge clamped circular plate with element side length 2m

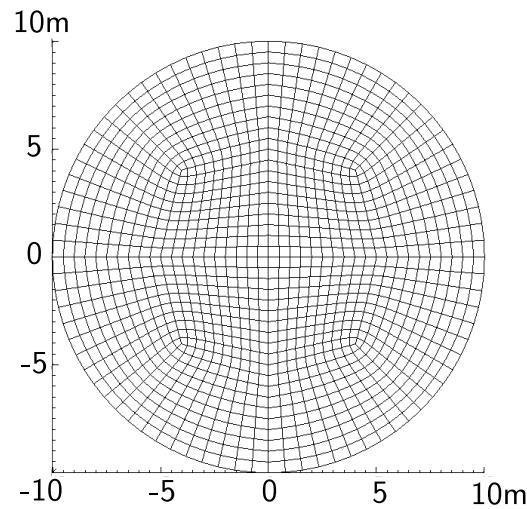


Figure 98: 8NodeBrick edge clamped circular plate with element side length 1m

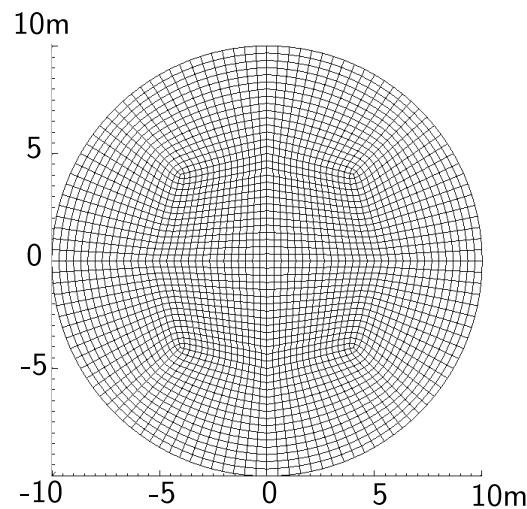


Figure 99: 8NodeBrick edge clamped circular plate with element side length 0.5m

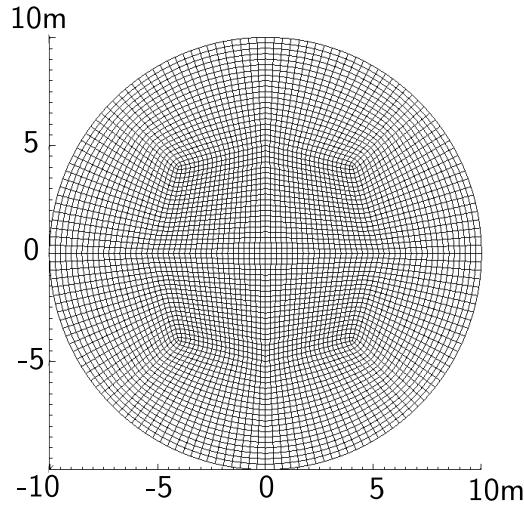


Figure 100: 8NodeBrick edge clamped circular plate with element side length 0.25m

The results were listed in Table (19).

Table 19: Results for 8NodeBrick circular plate with four edges clamped

Element type	8NodeBrick	8NodeBrick	8NodeBrick	Theoretical displacement
Number of layers	1layer	2layers	4layers	
Number of diameter divisions	Height:1.00m	Height:0.50m	Height:0.25m	
4	1.97E-04 m	1.99E-04 m	2.00E-04 m	1.71E-03 m
12	7.95E-04 m	8.47E-04 m	8.62E-04 m	1.71E-03 m
20	1.13E-03 m	1.25E-03 m	1.28E-03 m	1.71E-03 m
40	1.36E-03 m	1.54E-03 m	1.60E-03 m	1.71E-03 m
60	1.41E-03 m	1.62E-03 m	1.68E-03 m	1.71E-03 m
80	1.43E-03 m	1.64E-03 m	1.71E-03 m	1.71E-03 m

The errors were listed in Table (20).

Table 20: Errors for 8NodeBrick circular plate with four edges clamped

Element type	8NodeBrick	8NodeBrick	8NodeBrick
Number of layers	1layer	2layers	4layers
Number of diameter divisions	Height:1.00m	Height:0.50m	Height:0.25m
4	88.43%	88.32%	88.30%
12	53.43%	50.35%	49.47%
20	33.79%	27.00%	24.93%
40	20.14%	9.47%	6.03%
60	17.11%	5.34%	1.51%
80	16.01%	3.80%	0.19%

The errors were shown in Figure (101).

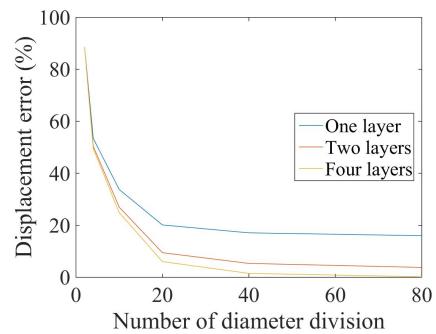


Figure 101: 8NodeBrick circular plate with edge clamped
Displacement error versus Number of side division

The ESSI model fei files for the table above are here

18.8 Verification of 8NodeBrick circular plate with all edges simply supported

Problem description: Diameter=20m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.3$.

The four edges are simply supported.

The load is the uniform normal pressure on the whole plate.

The plate flexural rigidity is

$$D = \frac{Eh^3}{12(1-\nu^2)} = \frac{10^8 N/m^2 \times 1^3 m^3}{12 \times (1 - 0.3^2)} = 9.1575 \times 10^6 N \cdot m \quad (21)$$

The theoretical solution⁷ is

$$d = \frac{(5 + \nu)qa^4}{64(1 + \nu)D} = \frac{(5 + 0.3) \times 100N/m^2 \times 10^4 m^4}{64 \times (1 + 0.3) \times 9.1575 \times 10^6 N \cdot m} = 6.956 \times 10^{-3} m \quad (22)$$

The 8NodeBrick were shown in Figure (102) - (107).

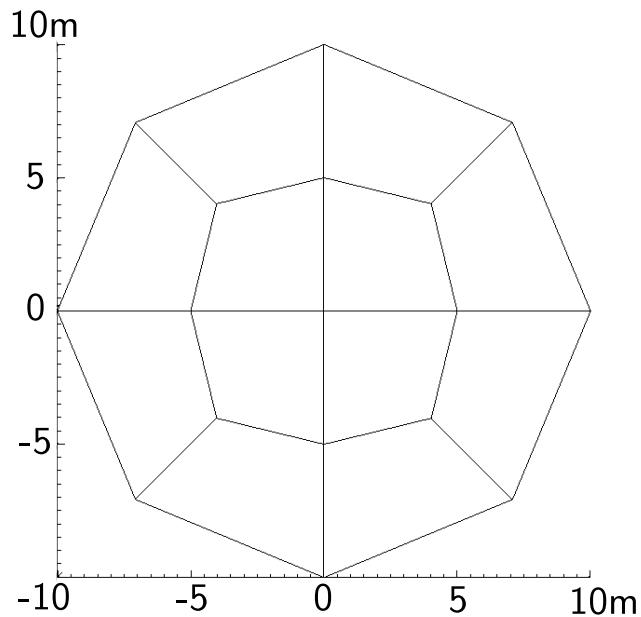


Figure 102: 8NodeBrick edge simply supported circular plate with element side length 10m

⁷Stephen Timoshenko, Theory of plates and shells (2nd edition). MrGRAW-Hill Inc, page55, 1959.

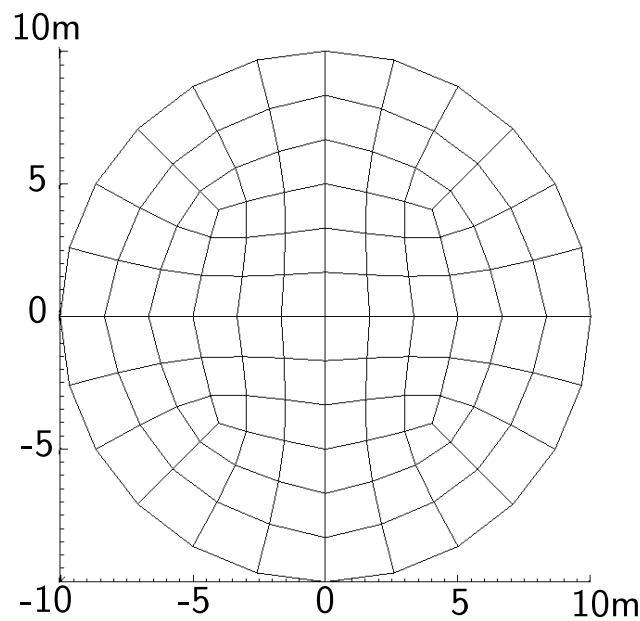


Figure 103: 8NodeBrick edge simply supported circular plate with element side length 5m

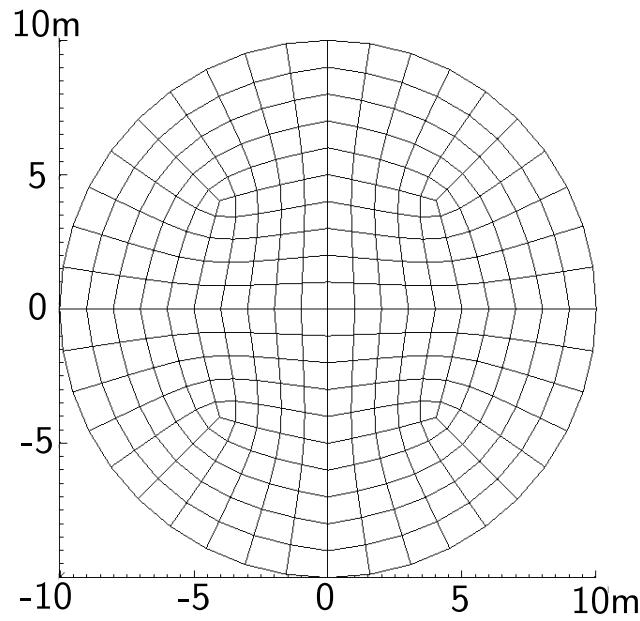


Figure 104: 8NodeBrick edge simply supported circular plate with element side length 2m

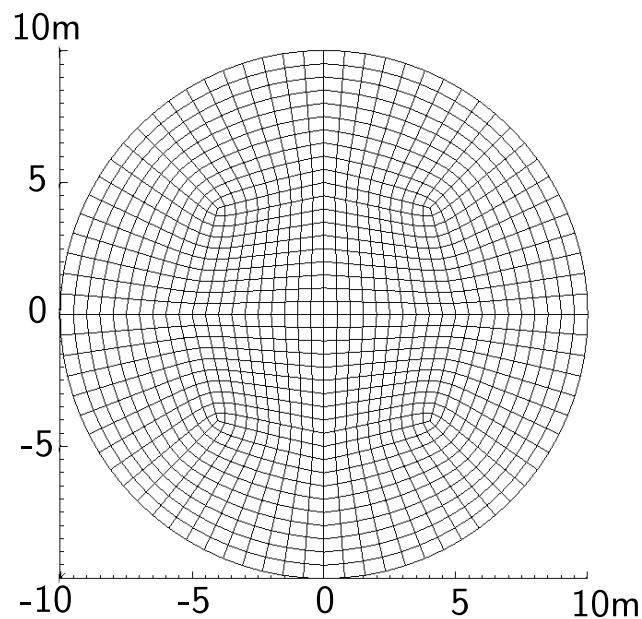


Figure 105: 8NodeBrick edge simply supported circular plate with element side length 1m

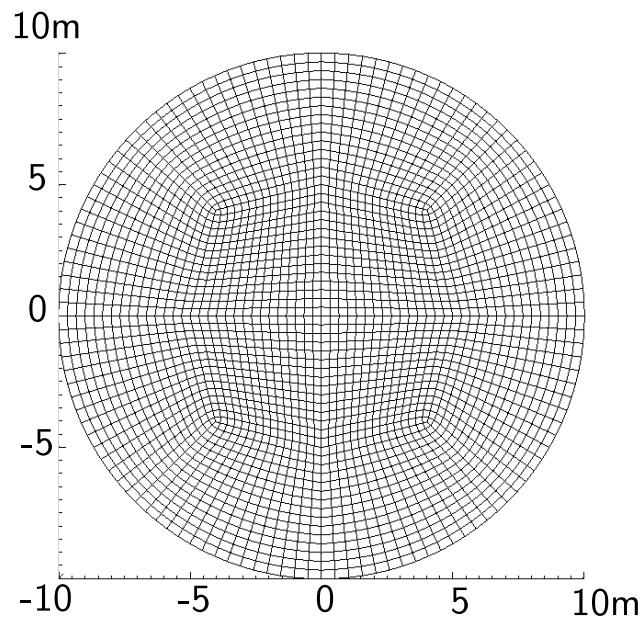


Figure 106: 8NodeBrick edge simply supported circular plate with element side length 0.5m

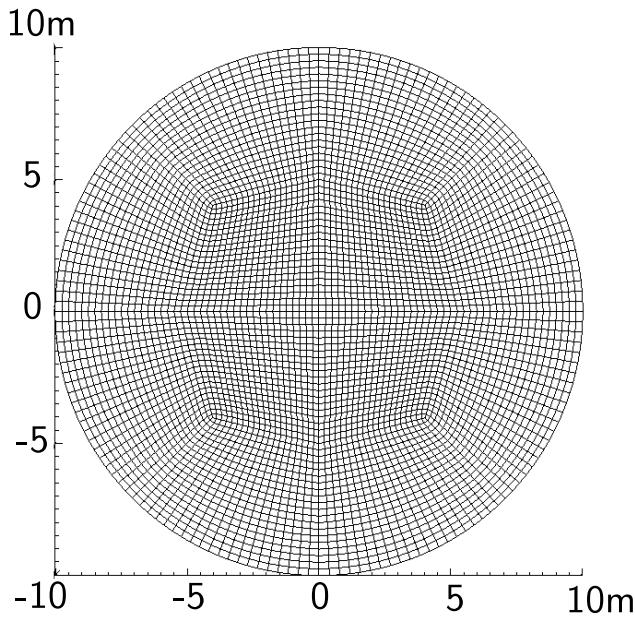


Figure 107: 8NodeBrick edge simply supported circular plate with element side length 0.25m

The results were listed in Table (21).

Table 21: Results for 8NodeBrick cicular plate with four edges simply supported

Element type	8NodeBrick	8NodeBrick	Theoretical displacement
Number of layers	2layers	4layers	Height:0.50m
Number of diameter divisions			
4	6.35E-04 m	6.39E-04 m	6.96E-03 m
12	3.46E-03 m	3.57E-03 m	6.96E-03 m
20	4.96E-03 m	5.18E-03 m	6.96E-03 m
40	6.05E-03 m	6.37E-03 m	6.96E-03 m
60	6.30E-03 m	6.65E-03 m	6.96E-03 m
80	6.39E-03 m	6.76E-03 m	6.96E-03 m

The errors were listed in Table (22).

Table 22: Errors for 8NodeBrick cicular plate with four edges simply supported

Element type	8NodeBrick	8NodeBrick
Number of layers	2layers	4layers
Number of diameter divisions	Height:0.50m	Height:0.25m
4	90.87%	90.82%
12	50.19%	48.65%
20	28.64%	25.47%
40	13.09%	8.40%
60	9.45%	4.36%
80	8.10%	2.85%

The errors were plotted in Figure (108).

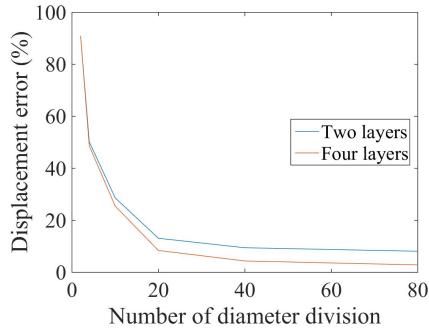


Figure 108: 8NodeBrick circular plate with edge simply supported
Displacement error versus Number of side division

The ESSI model fei files for the table above are here.

19 Verification of 27NodeBrick elements

19.1 Verification of 27NodeBrick cantilever beams

Problem description: Length=6m, Width=1m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.0$. Use the shear deformation coefficient $\kappa = 1.2$. The force direction was shown in Figure (109).

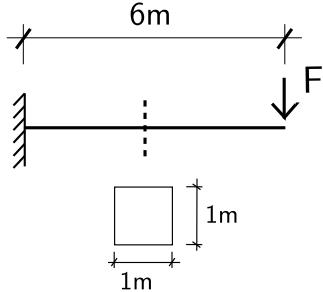


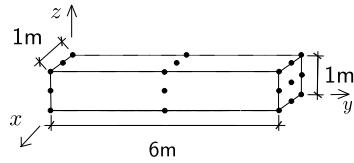
Figure 109: Problem description for cantilever beams

Theoretical displacement (bending and shear deformation):

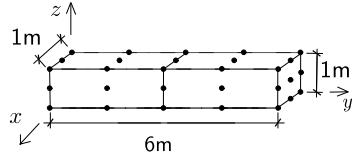
$$\begin{aligned}
 d &= \frac{FL^3}{3EI} + \frac{FL}{GA_v} \\
 &= \frac{FL^3}{3E \frac{bh^3}{12}} + \frac{FL}{\frac{E}{2(1+\nu)} \frac{bh}{\kappa}} \\
 &= \frac{100N \times 6^3 m^3}{3 \times 10^8 N/m^2 \times \frac{1}{12} m^4} + \frac{100N \times 6m}{\frac{10}{2} \times 10^7 N/m^2 \times 1m^2 \times \frac{5}{6}} \\
 &= 8.64 \times 10^{-4} m + 0.144 \times 10^{-4} m \\
 &= 8.784 \times 10^{-4} m
 \end{aligned} \tag{23}$$

Numerical model:

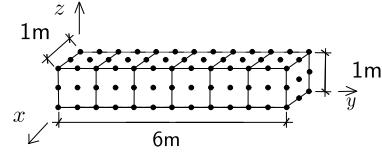
The 27NodeBrick elements were shown in Figure (110).



(a) One 27NodeBrick element



(b) Two 27NodeBrick elements



(c) Six 27NodeBrick elements

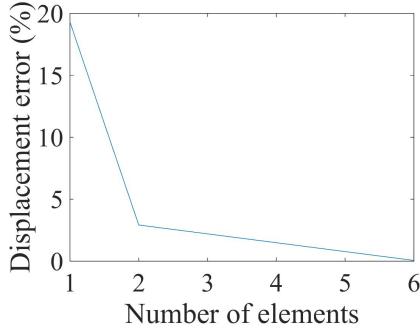
Figure 110: 27NodeBrick elements for cantilever beams

All the ESSI results were listed in Table (23).

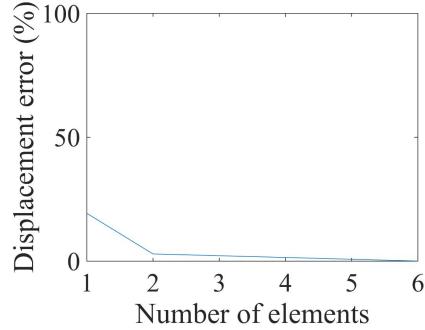
Table 23: Results for 27NodeBrick cantilever beams of different element numbers

Element number	1	2	6
27NodeBrick	7.07E-04 m	8.50E-04 m	8.75E-04 m
Error	19.52%	3.19%	0.34%

The errors were plotted in Figure (111).



(a) Error scale 0% - 20%



(b) Error scale 0% - 100%

Figure 111: 27NodeBrick cantilever beam for different element number
Displacement error versus Number of elements

The ESSI model fei files for the table above are here

19.2 Verification of 27NodeBrick cantilever beam for different Poisson's ratio

Problem description: Length=6m, Width=1m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.0 - 0.49$. The force direction was shown in Figure (112).

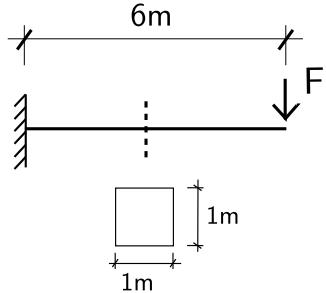


Figure 112: Problem description for cantilever beams of different Poisson's ratios

The theoretical solution for $\nu = 0.0$ was calculated below, while the solution for other Poisson's ratio were calculated by the similar process.

Theoretical displacement (bending and shear deformation):

$$\begin{aligned}
 d &= \frac{FL^3}{3EI} + \frac{FL}{GA_v} \\
 &= \frac{FL^3}{3E\frac{bh^3}{12}} + \frac{FL}{\frac{E}{2(1+\nu)} \frac{bh}{\kappa}} \\
 &= \frac{100N \times 6^3 m^3}{3 \times 10^8 N/m^2 \times \frac{1}{12} m^4} + \frac{100N \times 6m}{\frac{10}{2} \times 10^7 N/m^2 \times 1m^2 \times \frac{5}{6}} \\
 &= 8.64 \times 10^{-4} m + 0.144 \times 10^{-4} m \\
 &= 8.784 \times 10^{-4} m
 \end{aligned} \tag{24}$$

The rotation angle at the end:

$$\theta = \frac{FL^2}{2EI} = \frac{100N \times 6^2 m^2}{2 \times 10^8 N/m^2 \times \frac{1}{12} m^4} = 2.16 \times 10^{-4} rad = 0.0124^\circ \tag{25}$$

The 27NodeBrick elements for cantilever beams of different Poisson's ratios were shown in Figure (113):

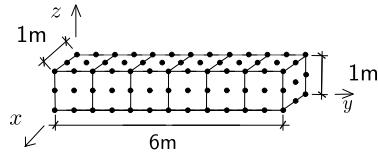


Figure 113: 27NodeBrick elements for cantilever beams of different Poisson's ratios

All the displacement results were listed in Table (24).

Table 24: ***Displacement*** results for 27NodeBrick cantilever beams
with element side length 1 m

Poisson's ratio	27NodeBrick displacement	Theory displacement (bending)	Theory displacement (shear)	Theory displacement(all)	Error
0.00	8.755E-04 m	8.640E-04 m	1.440E-05 m	8.784E-04 m	0.34%
0.05	8.757E-04 m	8.640E-04 m	1.512E-05 m	8.791E-04 m	0.39%
0.10	8.751E-04 m	8.640E-04 m	1.586E-05 m	8.799E-04 m	0.54%
0.15	8.735E-04 m	8.640E-04 m	1.659E-05 m	8.806E-04 m	0.80%
0.20	8.708E-04 m	8.640E-04 m	1.734E-05 m	8.813E-04 m	1.19%
0.25	8.667E-04 m	8.640E-04 m	1.808E-05 m	8.821E-04 m	1.74%
0.30	8.608E-04 m	8.640E-04 m	1.884E-05 m	8.828E-04 m	2.50%
0.35	8.520E-04 m	8.640E-04 m	1.959E-05 m	8.836E-04 m	3.57%
0.40	8.385E-04 m	8.640E-04 m	2.035E-05 m	8.844E-04 m	5.18%
0.45	8.147E-04 m	8.640E-04 m	2.111E-05 m	8.851E-04 m	7.96%
0.49	7.711E-04 m	8.640E-04 m	2.173E-05 m	8.857E-04 m	12.94%

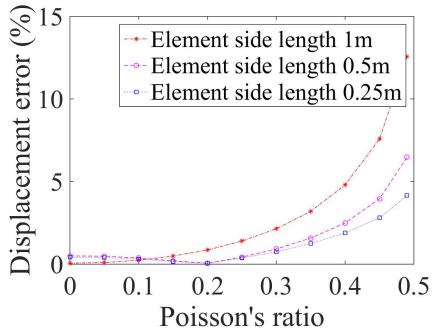
Table 25: ***Displacement*** results for 27NodeBrick cantilever beams
with element side length 0.5 m

Poisson's ratio	27NodeBrick displacement	Theory displacement (bending)	Theory displacement (shear)	Theory displacement(all)	Error
0.00	8.804E-04 m	8.640E-04 m	1.440E-05 m	8.784E-04 m	0.23%
0.05	8.808E-04 m	8.640E-04 m	1.512E-05 m	8.791E-04 m	0.19%
0.10	8.805E-04 m	8.640E-04 m	1.586E-05 m	8.799E-04 m	0.08%
0.15	8.796E-04 m	8.640E-04 m	1.659E-05 m	8.806E-04 m	0.12%
0.20	8.778E-04 m	8.640E-04 m	1.734E-05 m	8.813E-04 m	0.40%
0.25	8.752E-04 m	8.640E-04 m	1.808E-05 m	8.821E-04 m	0.78%
0.30	8.715E-04 m	8.640E-04 m	1.884E-05 m	8.828E-04 m	1.28%
0.35	8.663E-04 m	8.640E-04 m	1.959E-05 m	8.836E-04 m	1.95%
0.40	8.588E-04 m	8.640E-04 m	2.035E-05 m	8.844E-04 m	2.89%
0.45	8.465E-04 m	8.640E-04 m	2.111E-05 m	8.851E-04 m	4.36%
0.49	8.248E-04 m	8.640E-04 m	2.173E-05 m	8.857E-04 m	6.88%

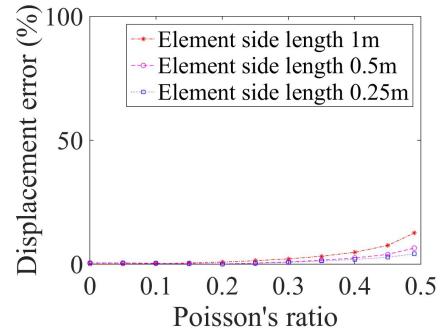
Table 26: ***Displacement*** results for 27NodeBrick cantilever beams
with element side length 0.25 m

Poisson's ratio	27NodeBrick displacement	Theory displacement (bending)	Theory displacement (shear)	Theory displacement(all)	Error
0.00	8.797E-04 m	8.640E-04 m	1.440E-05 m	8.784E-04 m	0.15%
0.05	8.801E-04 m	8.640E-04 m	1.512E-05 m	8.791E-04 m	0.11%
0.10	8.799E-04 m	8.640E-04 m	1.586E-05 m	8.799E-04 m	0.01%
0.15	8.792E-04 m	8.640E-04 m	1.659E-05 m	8.806E-04 m	0.16%
0.20	8.778E-04 m	8.640E-04 m	1.734E-05 m	8.813E-04 m	0.40%
0.25	8.758E-04 m	8.640E-04 m	1.808E-05 m	8.821E-04 m	0.71%
0.30	8.730E-04 m	8.640E-04 m	1.884E-05 m	8.828E-04 m	1.12%
0.35	8.692E-04 m	8.640E-04 m	1.959E-05 m	8.836E-04 m	1.63%
0.40	8.641E-04 m	8.640E-04 m	2.035E-05 m	8.844E-04 m	2.29%
0.45	8.567E-04 m	8.640E-04 m	2.111E-05 m	8.851E-04 m	3.21%
0.49	8.452E-04 m	8.640E-04 m	2.173E-05 m	8.857E-04 m	4.58%

The errors were plotted in Figure (114).



(a) Error scale 0% - 15%



(b) Error scale 0% - 100%

Figure 114: 27NodeBrick cantilever beam for different Poisson's ratio
Displacement error versus Poisson's ratio

The angle results were listed in Table (27).

Table 27: ***Rotation angle*** results for 27NodeBrick cantilever beams
with element side length 1 m

Poisson's ratio	27NodeBrick angle (unit: $^{\circ}$)	Theory angle (unit: $^{\circ}$)	Error
0.00	1.238E-02	1.24E-02	0.19%
0.05	1.237E-02	1.24E-02	0.24%
0.10	1.236E-02	1.24E-02	0.34%
0.15	1.233E-02	1.24E-02	0.53%
0.20	1.230E-02	1.24E-02	0.80%
0.25	1.225E-02	1.24E-02	1.18%
0.30	1.219E-02	1.24E-02	1.70%
0.35	1.210E-02	1.24E-02	2.45%
0.40	1.196E-02	1.24E-02	3.55%
0.45	1.172E-02	1.24E-02	5.47%
0.49	1.130E-02	1.24E-02	8.89%

Table 28: ***Rotation angle*** results for 27NodeBrick cantilever beams
with element side length 0.5 m

Poisson's ratio	27NodeBrick angle (unit: $^{\circ}$)	Theory angle (unit: $^{\circ}$)	Error
0.00	1.242E-02	1.24E-02	0.12%
0.05	1.241E-02	1.24E-02	0.11%
0.10	1.241E-02	1.24E-02	0.06%
0.15	1.239E-02	1.24E-02	0.05%
0.20	1.237E-02	1.24E-02	0.21%
0.25	1.235E-02	1.24E-02	0.44%
0.30	1.231E-02	1.24E-02	0.74%
0.35	1.226E-02	1.24E-02	1.16%
0.40	1.218E-02	1.24E-02	1.76%
0.45	1.206E-02	1.24E-02	2.76%
0.49	1.183E-02	1.24E-02	4.63%

Table 29: ***Rotation angle*** results for 27NodeBrick cantilever beams
with element side length 0.25 m

Poisson's ratio	27NodeBrick angle (unit: $^{\circ}$)	Theory angle (unit: $^{\circ}$)	Error
0.00	1.242E-02	1.24E-02	0.17%
0.05	1.242E-02	1.24E-02	0.15%
0.10	1.241E-02	1.24E-02	0.09%
0.15	1.240E-02	1.24E-02	0.02%
0.20	1.238E-02	1.24E-02	0.17%
0.25	1.235E-02	1.24E-02	0.38%
0.30	1.232E-02	1.24E-02	0.64%
0.35	1.228E-02	1.24E-02	0.98%
0.40	1.222E-02	1.24E-02	1.42%
0.45	1.214E-02	1.24E-02	2.06%
0.49	1.202E-02	1.24E-02	3.08%

The errors were plotted in Figure (115).

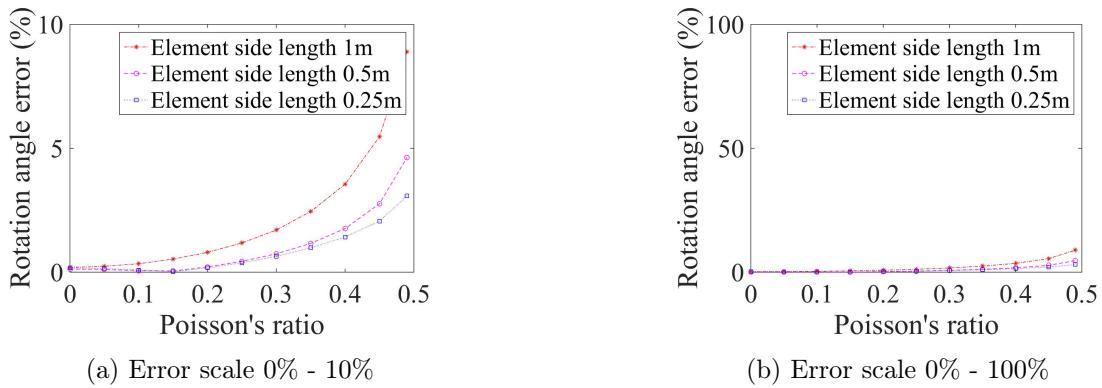


Figure 115: 27NodeBrick cantilever beam for different Poisson's ratio
Rotation angle error versus Poisson's ratio

The ESSI model fei files for the table above are here

Then, different values of elastic modulus were also tried. The errors were plotted below.

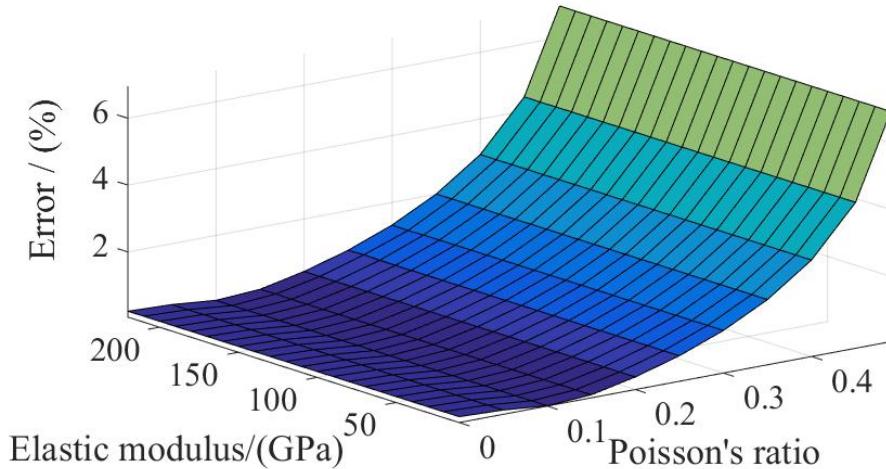


Figure 116: The influence of Poisson's ratio and elastic modulus on the errors

According to Fig.(116)), the different values of elastic modulus will not influence the error.

However, the different Poisson's ratio will influence the error. The error will increase with the Poisson's ratio increase.

19.3 Test of irregular shaped 27NodeBrick cantilever beams

Cantilever model was used as an example. Three different shapes were tested.

In the first test, the upper two nodes of each element were moved one half element size along the $y - axis$, while the lower two nodes were kept at the same location. The element shape was shown in Figure (117).

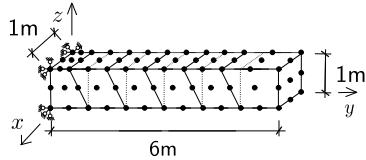


Figure 117: 27NodeBrick cantilever beams for irregular *Shape 1*

In the second test, the upper two nodes of each element were moved 90% element size along the $y - axis$, while the lower two nodes were moved 90% element size in the other direction along the $y - axis$. The element shape was shown in Figure (118).

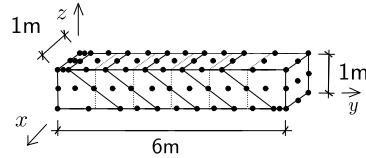


Figure 118: 27NodeBrick cantilever beams for irregular *Shape 2*

In the third test, the upper two nodes of each element were moved one half element size with different directions along the $y - axis$, while the lower two nodes were kept at the same location. The element shape was shown in Figure (119).

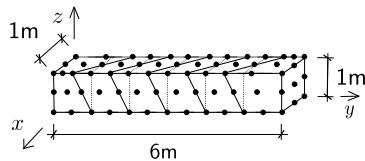
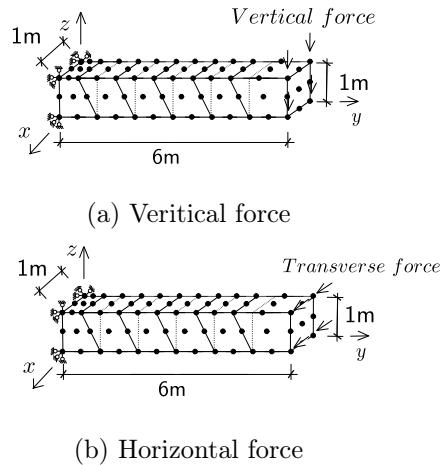
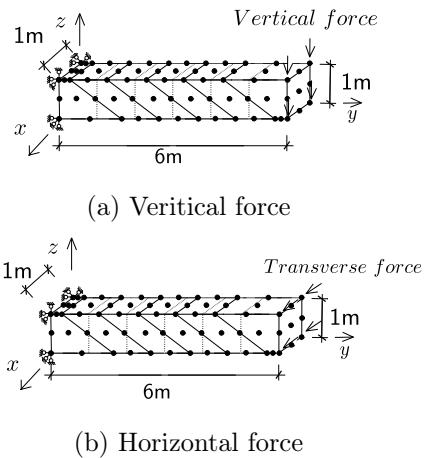
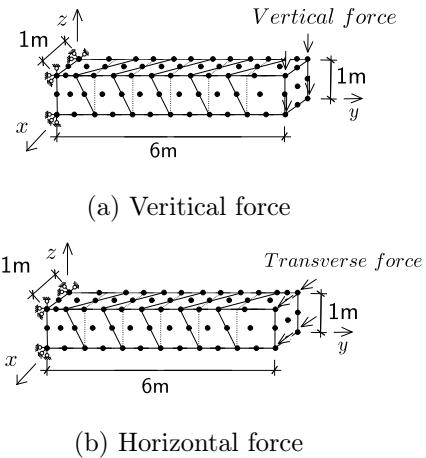


Figure 119: 27NodeBrick cantilever beams for irregular *Shape 3*

The boundary conditions were shown in Figure (120), (121) and (122) .

Figure 120: 27NodeBrick cantilever beam boundary conditions for irregular ***Shape 1***Figure 121: 27NodeBrick cantilever beam boundary conditions for irregular ***Shape 2***Figure 122: 27NodeBrick cantilever beam boundary conditions for irregular ***Shape 3***

The ESSI results were listed in Table (30).

Table 30: Results for 27NodeBrick cantilever beams of irregular shapes

Displacements for irregular shaped element					
Element Type	Force direction	Normal shape	Shape 1	Shape 2	Shape 3
27NodeBrick	Vertical (z)	8.755E-04 m	8.819E-04 m	8.709E-04 m	8.837E-04 m
27NodeBrick	Transverse (y)	8.755E-04 m	8.831E-04 m	8.462E-04 m	8.824E-04 m
Theoretical	-	8.784E-04 m	8.784E-04 m	8.784E-04 m	8.784E-04 m

The errors were listed in Table (31) and (32).

Table 31: Errors for irregular shaped 27NodeBrick compared to theoretical solution

Errors for irregular shaped element, compared to theoretical solutions					
Element Type	Force direction	Normal shape	Shape 1	Shape 2	Shape 3
27NodeBrick	Vertical (z)	0.34%	0.40%	0.85%	0.60%
27NodeBrick	Transverse (y)	0.34%	0.54%	3.67%	0.46%

Table 32: Errors for irregular shaped 27NodeBrick compared to normal shape

Errors for irregular shaped element, compared to normal shape					
Element Type	Force direction	Normal shape	Shape 1	Shape 2	Shape 3
27NodeBrick	Vertical (z)	0.00%	0.74%	0.52%	0.94%
27NodeBrick	Transverse (y)	0.00%	0.87%	3.34%	0.79%

The ESSI model fei files for the table above are here

Then, the beam was divided into small elements.

Problem description: Length=12m, Width=2m, Height=2m, Force=400N/m, E=1E8Pa, $\nu = 0.0$. Use the shear deformation coefficient $\kappa = 1.2$. The force direction was shown in Figure (123).

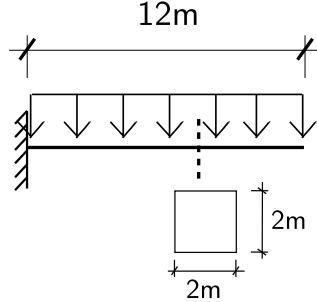


Figure 123: Problem description for cantilever beams under uniform pressure

Theoretical displacement (bending and shear deformation):

$$\begin{aligned}
 d &= \frac{qL^4}{8EI} + \frac{q\frac{L^2}{2}}{GA_v} \\
 &= \frac{qL^4}{8E\frac{bh^3}{12}} + \frac{q\frac{L^2}{2}}{\frac{E}{2(1+\nu)} \frac{bh}{\kappa}} \\
 &= \frac{400N/m \times 12^4 m^4}{8 \times 10^8 N/m^2 \times \frac{24}{12} m^4} + \frac{400N/m \times \frac{12^2}{2} m^2}{\frac{10^8}{2} N/m^2 \times 2m \times 2m \times \frac{5}{6}} \\
 &= 7.776 \times 10^{-3} m + 1.728 \times 10^{-4} m \\
 &= 7.9488 \times 10^{-3} m
 \end{aligned} \tag{26}$$

The ESSI displacement results were listed in Table (33).

Table 33: Results for 27NodeBrick cantilever beams of irregular shapes with more elements

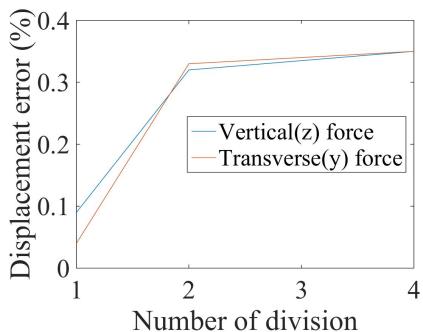
Element Type	Shape	Force direction	Number of division		
			1	2	4
27NodeBrick	shape1	Vertical (z)	7.913E-03 m	7.946E-03 m	7.948E-03 m
27NodeBrick	shape1	Transverse (y)	7.903E-03 m	7.946E-03 m	7.948E-03 m
27NodeBrick	shape2	Vertical (z)	7.741E-03 m	7.930E-03 m	7.947E-03 m
27NodeBrick	shape2	Transverse (y)	7.371E-03 m	7.894E-03 m	7.944E-03 m
27NodeBrick	shape3	Vertical (z)	1.982E-03 m	7.946E-03 m	7.948E-03 m
27NodeBrick	shape3	Transverse (y)	1.979E-03 m	7.947E-03 m	7.948E-03 m
Theoretical solution			7.9488E-03 m	7.9488E-03 m	7.9488E-03 m

The error were listed in Table (34).

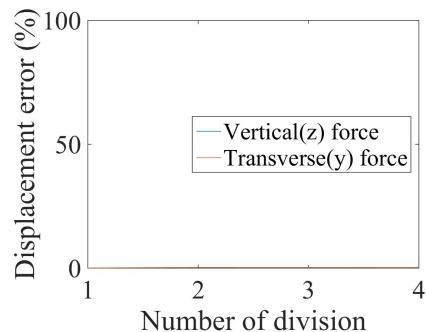
Table 34: Errors for 27NodeBrick cantilever beams of irregular shapes with more elements

Element Type	Shape	Force direction	Number of division		
			1	2	4
27NodeBrick	shape1	Vertical (z)	0.45%	0.04%	0.01%
27NodeBrick	shape1	Transverse (y)	0.32%	0.03%	0.01%
27NodeBrick	shape2	Vertical (z)	2.61%	0.23%	0.03%
27NodeBrick	shape2	Transverse (y)	7.27%	0.69%	0.06%
27NodeBrick	shape3	Vertical (z)	75.06%	0.04%	0.01%
27NodeBrick	shape3	Transverse (y)	75.11%	0.03%	0.01%

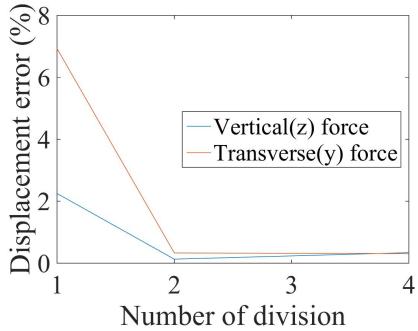
The errors were shown in Figure (124), (125) and (126).



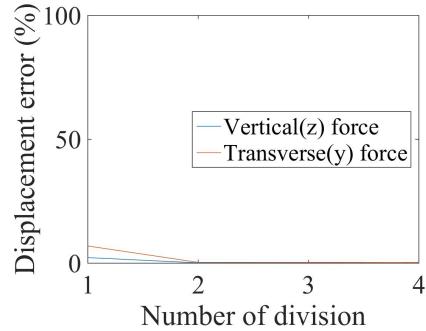
(a) Error scale 0% - 0.4%



(b) Error scale 0% - 100%

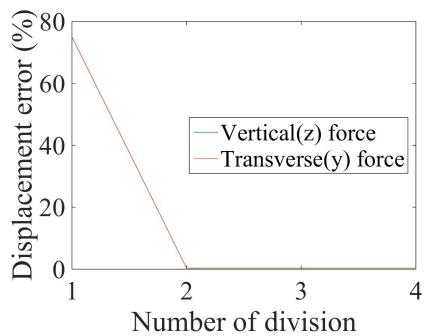
Figure 124: 27NodeBrick cantilever beam for irregular **Shape 1**
Displacement error versus Number of division

(a) Error scale 0% - 8%

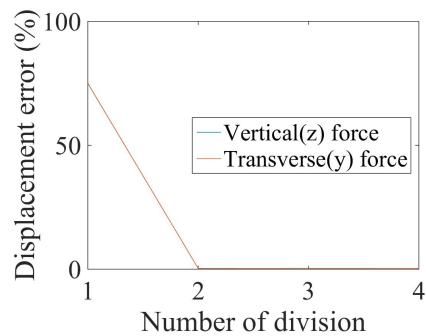


(b) Error scale 0% - 100%

Figure 125: 27NodeBrick cantilever beam for irregular **Shape 2**
Displacement error versus Number of division



(a) Error scale 0% - 80%



(b) Error scale 0% - 100%

Figure 126: 27NodeBrick cantilever beam for irregular ***Shape 3***

Displacement error versus Number of division

The ESSI model fei files for the table above are here

In this section, the beam was cut into smaller elements with element side length 0.5m and 0.25m respectively. And the element side length of the original models is 1.0m. The numerical models were shown in Figure (127), (128) and (129).

Number of division 1:

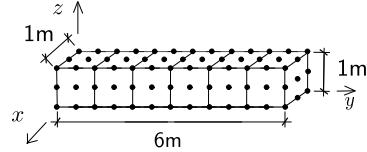


Figure 127: 27NodeBrick clamped beams with element side length 1.0m

Number of division 2:

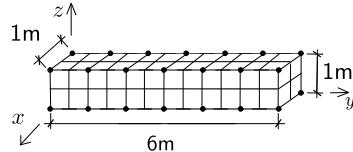


Figure 128: 27NodeBrick clamped beams with element side length 0.5m

Number of division 4:

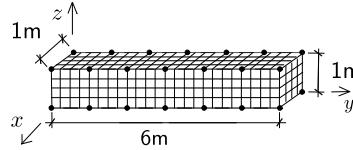


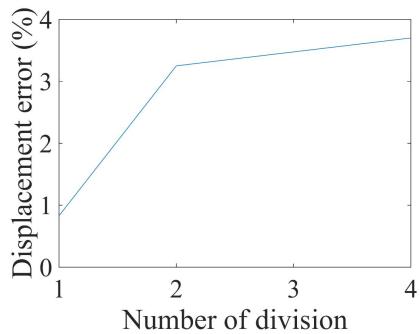
Figure 129: 27NodeBrick clamped beams with element side length 0.25m

The ESSI results were listed in Table (35). The theoretical solution is 1.60E-5 m.

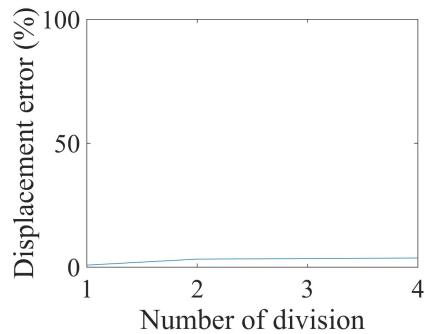
Table 35: Results for 27NodeBrick clamped beams with more elements

Element Type	Element side length		
	1 m	0.5 m	0.25 m
27NodeBrick	1.64E-05 m	1.70E-05 m	1.71E-05 m
Error	0.83%	3.25%	3.70%

The errors were plotted in Figure (130).



(a) Error scale 0% - 4%



(b) Error scale 0% - 100%

Figure 130: 27NodeBrick clamped beam for different element number

Displacement error versus Number of division

The ESSI model fei files for the table above are here

19.4 Verification of 27NodeBrick stress in cantilever beams

Problem description: Length=6m, Width=1m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.0$. Use the shear deformation coefficient $\kappa = 1.2$. The force direction was shown in Figure (131).

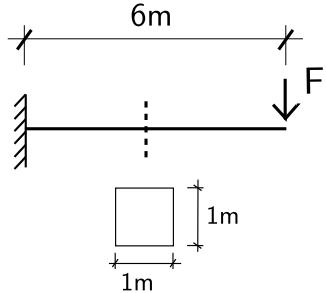


Figure 131: Problem description for cantilever beams of stress verification

The theoretical solution for the stress was calculated below.

The 27NodeBrick elements were shown in Figure (132).

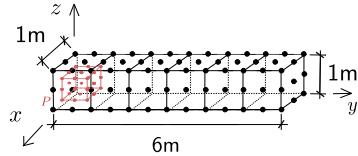


Figure 132: 27NodeBrick for cantilever beams of stress verification

The bending moment at the Gassian Point is

$$M = F(L - P_y) = 100N \times (6 - 0.1127)m = 588.73N \cdot m \quad (27)$$

The bending modulus is

$$I = \frac{bh^3}{12} = \frac{1}{12}m^4 \quad (28)$$

Therefore, the theoretical stress is

$$\sigma = \frac{M \cdot z}{I} = \frac{588.73N \cdot m \times (0.5 - 0.1127)m}{\frac{1}{12}m^4} = 2736Pa \quad (29)$$

To get a better result, the same geometry beam was also cut into small elements. When more elements were used, the theoretical stress was calculated again with the new coordinates. The calculation process is similar to the process above.

The numerical models were shown in Figure (133), (134) and (135).

Number of division 1:

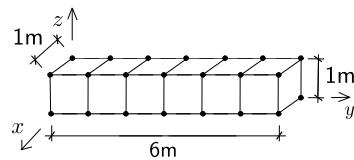


Figure 133: 27NodeBrick stress with element side length 1.0m

Number of division 2:

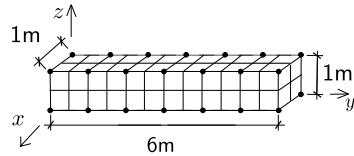


Figure 134: 27NodeBrick stress with element side length 0.5m

Number of division 4:

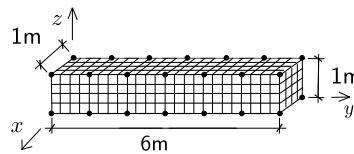
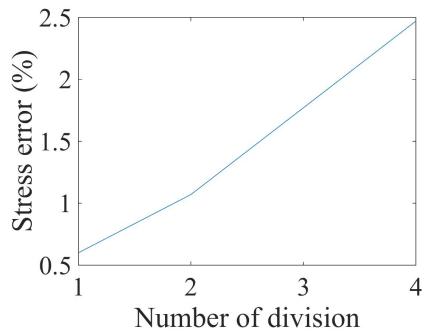


Figure 135: 27NodeBrick stress with element side length 0.25m

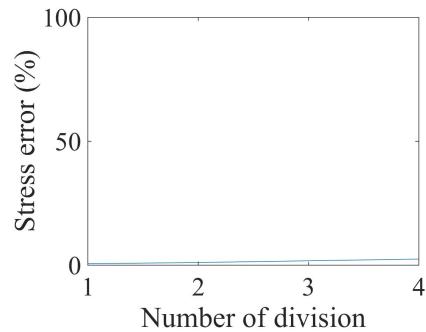
All the stress results were listed in Table (36).

Table 36: Results for 27NodeBrick stress with more elements

Element Type	Element side length		
	1 m	0.5 m	0.25 m
27NodeBrick	2719.81 Pa	3198.19 Pa	3464.76 Pa
Theoretical	2736.17 Pa	3164.27 Pa	3381.18 Pa
Error	0.60%	1.07%	2.47%



(a) Error scale 0% - 2.5%



(b) Error scale 0% - 100%

Figure 136: 27NodeBrick cantilever beams for stress verification

Stress error versus Number of division

The ESSI model fei files for the table above are here

19.5 Verification of 27NodeBrick square plate with four edges clamped

Problem description: Length=20m, Width=20m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.3$.

The four edges are clamped.

The load is the uniform normal pressure on the whole plate.

The plate flexural rigidity is

$$D = \frac{Eh^3}{12(1 - \nu^2)} = \frac{10^8 N/m^2 \times 1^3 m^3}{12 \times (1 - 0.3^2)} = 9.1575 \times 10^6 N \cdot m \quad (30)$$

The theoretical solution is

$$d = \alpha_c \frac{qa^4}{D} = 0.00406 \times \frac{100 N/m^2 \times 20^4 m^4}{9.1575 \times 10^6 N \cdot m} = 2.2015 \times 10^{-3} m \quad (31)$$

where α_c is a coefficient, which depends on the ratio of plate length to width. In this problem, the coefficient⁸ α_c is 0.00406.

The 27NodeBrick were shown in Figure (137) - (142).

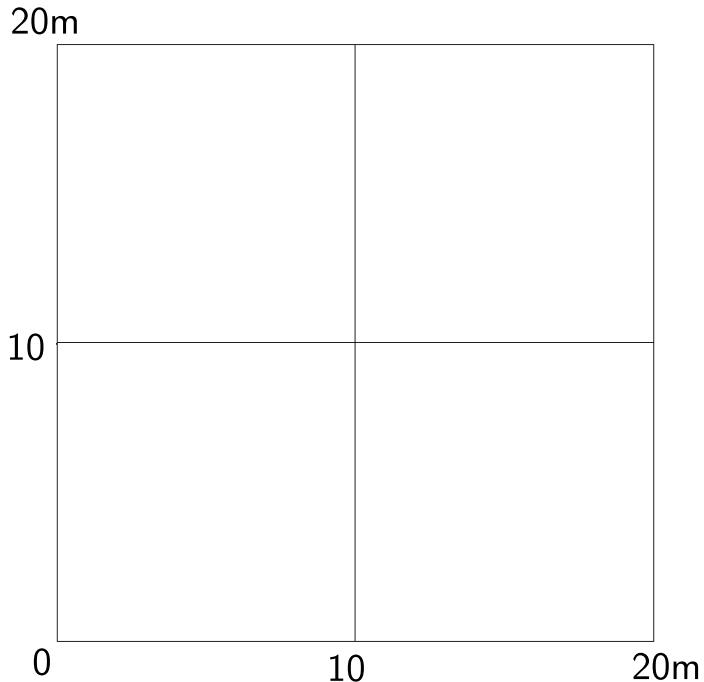


Figure 137: 27NodeBrick edge clamped square plate with element side length 10m

⁸Stephen Timoshenko, Theory of plates and shells (2nd edition). MrGRAW-Hill Inc, page120, 1959.

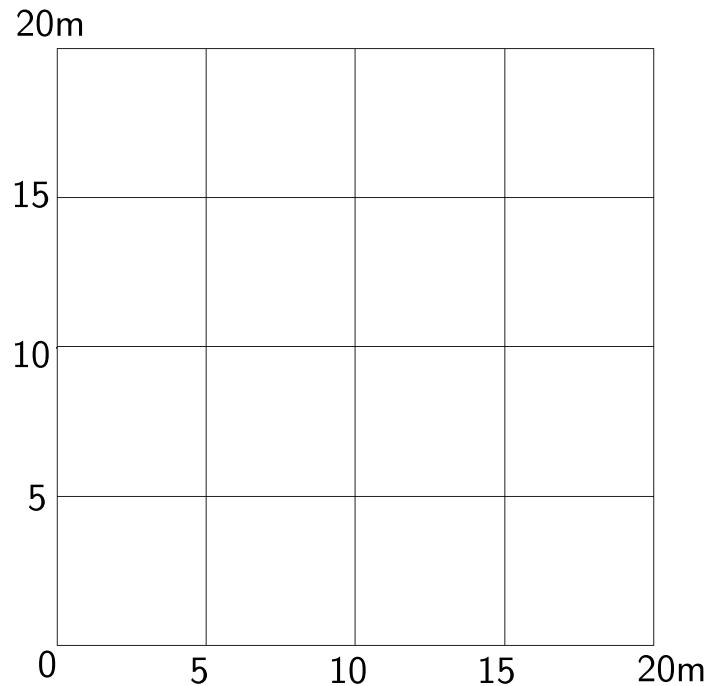


Figure 138: 27NodeBrick edge clamped square plate with element side length 5m

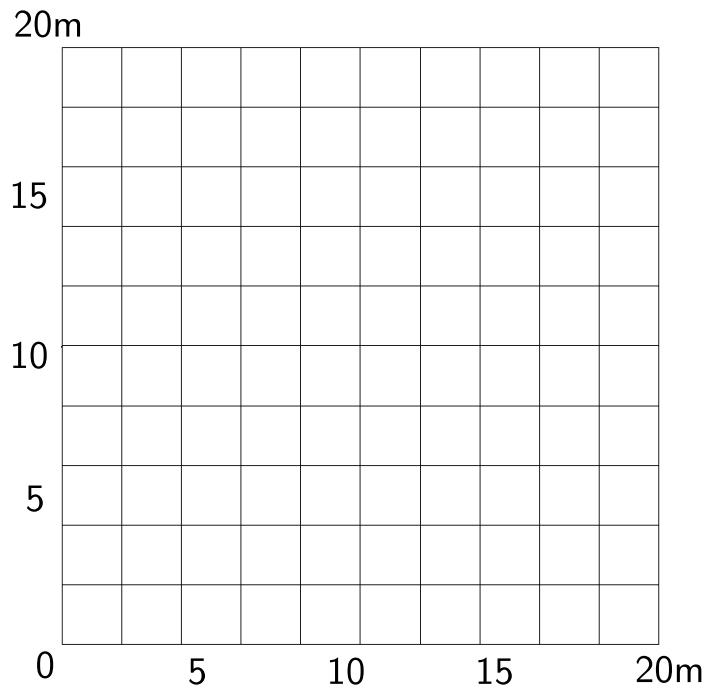


Figure 139: 27NodeBrick edge clamped square plate with element side length 2m

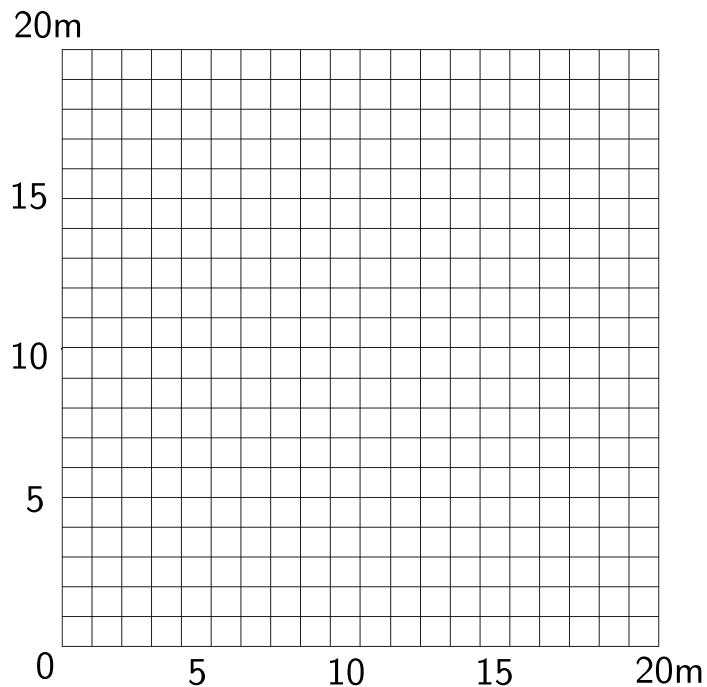


Figure 140: 27NodeBrick edge clamped square plate with element side length 1m

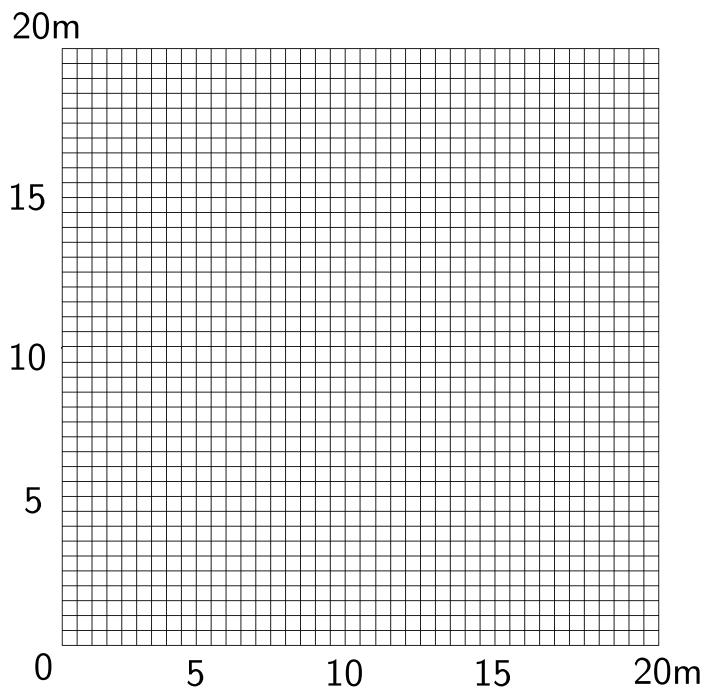


Figure 141: 27NodeBrick edge clamped square plate with element side length 0.5m

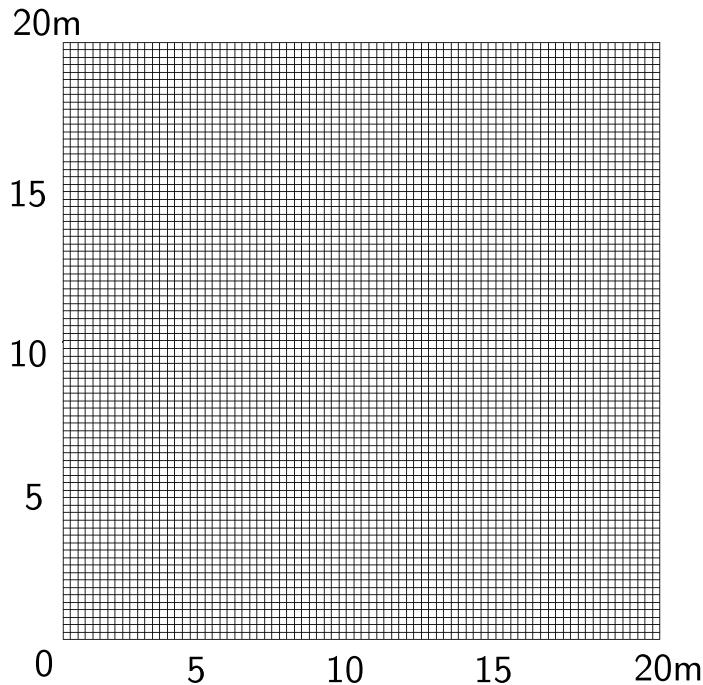


Figure 142: 27NodeBrick edge clamped square plate with element side length 0.25m

The results were listed in Table (37).

Table 37: Results for 27NodeBrick square plate with four edges clamped

Element type	27NodeBrick	27NodeBrick	27NodeBrick	Theoretical displacement
Number of layers	1layer	2layers	4layers	
Element side length	Height:1.00m	Height:0.50m	Height:0.25m	
10m	4.82E-004 m	4.82E-004 m	4.82E-004 m	2.20E-03 m
5m	1.97E-003 m	1.98E-003 m	1.98E-003 m	2.20E-03 m
2m	2.25E-003 m	2.26E-003 m	2.26E-003 m	2.20E-03 m
1m	2.28E-003 m	2.29E-003 m	2.29E-003 m	2.20E-03 m
0.5m	2.29E-003 m	2.30E-003 m	2.30E-003 m	2.20E-03 m
0.25m	2.29E-003 m	2.30E-003 m	- ⁹	2.20E-03 m

The errors were listed in Table (38).

⁹This model run out of memory on machine cml01 (memory: 23.5GB). This model has 233,289 nodes with 3 dofs, which may require 40GB memory.

Table 38: Errors for 27NodeBrick square plate with four edges clamped

Element type	27NodeBrick	27NodeBrick	27NodeBrick
Number of layers	1layer	2layers	4layers
Element side length	Height:1.00m	Height:0.50m	Height:0.25m
10m	78.11%	78.10%	78.10%
5m	10.67%	10.19%	10.16%
2m	2.23%	2.79%	2.83%
1m	3.56%	4.16%	4.22%
0.5m	3.96%	4.58%	4.65%
0.25m	4.08%	4.70%	-

The errors were plotted in Figure (143).

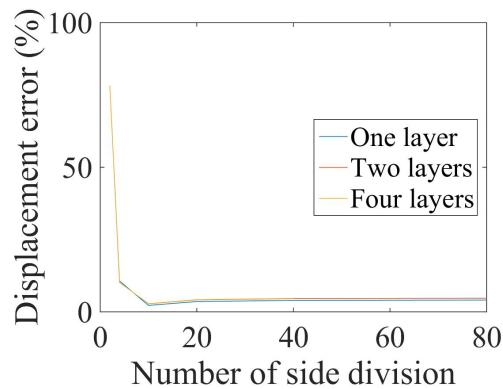


Figure 143: 27NodeBrick square plate with edge clamped
Displacement error versus Number of side division

The ESSI model fei files for the table above are here

19.6 Verification of 27NodeBrick square plate with four edges simply supported

Problem description: Length=20m, Width=20m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.3$.

The four edges are simply supported.

The load is the uniform normal pressure on the whole plate.

The plate flexural rigidity is

$$D = \frac{Eh^3}{12(1 - \nu^2)} = \frac{10^8 N/m^2 \times 1^3 m^3}{12 \times (1 - 0.3^2)} = 9.1575 \times 10^6 N \cdot m \quad (32)$$

The theoretical solution is

$$d = \alpha_s \frac{qa^4}{D} = 0.00126 \times \frac{100 N/m^2 \times 20^4 m^4}{9.1575 \times 10^6 N \cdot m} = 7.0936 \times 10^{-3} m \quad (33)$$

where α_s is a coefficient, which depends on the ratio of plate length to width. In this problem, the coefficient¹⁰ α_s is 0.00126.

The 27NodeBrick were shown in Figure (144) - (149).

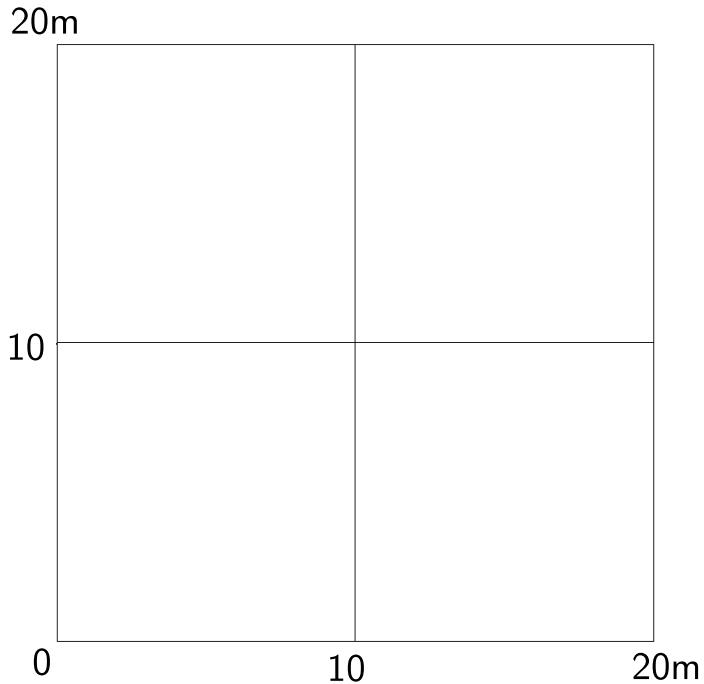


Figure 144: 27NodeBrick edge simply supported square plate with element side length 10m

¹⁰Stephen Timoshenko, Theory of plates and shells (2nd edition). MrGRAW-Hill Inc, page202, 1959.

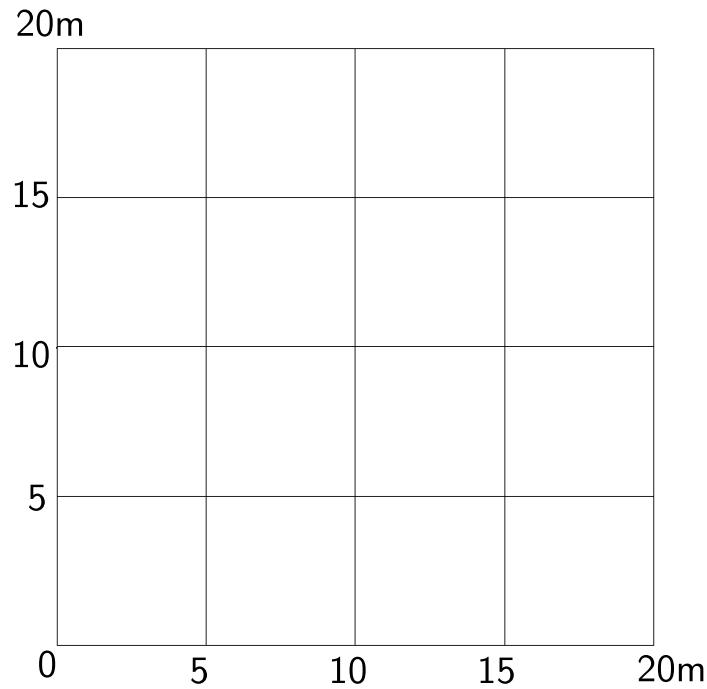


Figure 145: 27NodeBrick edge simply supported square plate with element side length 5m

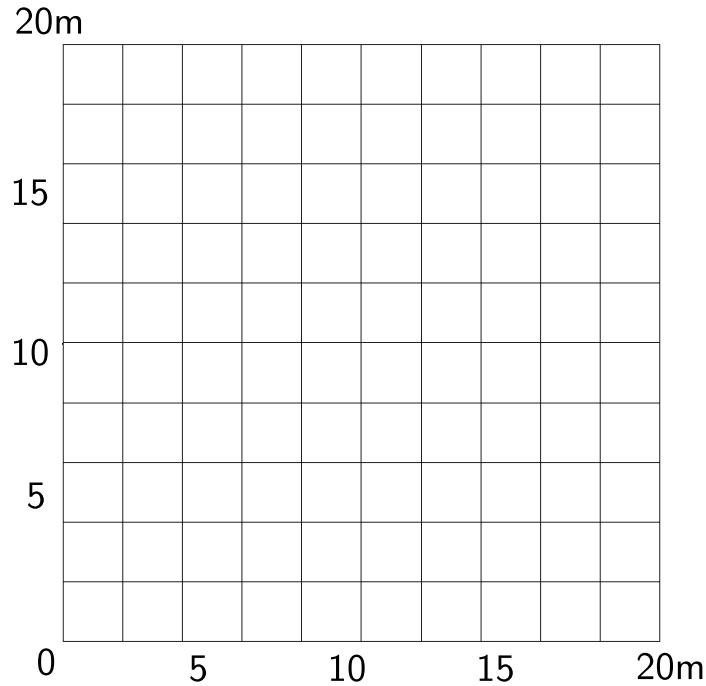


Figure 146: 27NodeBrick edge simply supported square plate with element side length 2m

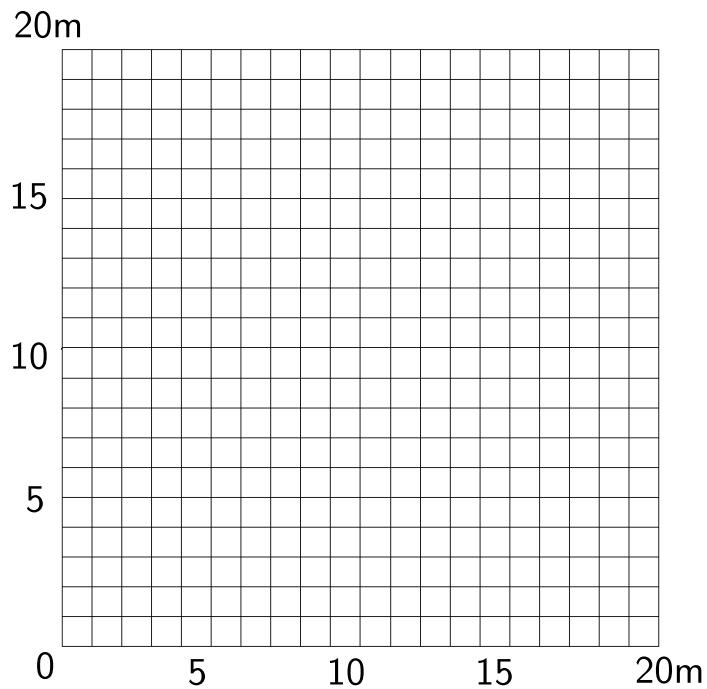


Figure 147: 27NodeBrick edge simply supported square plate with element side length 1m

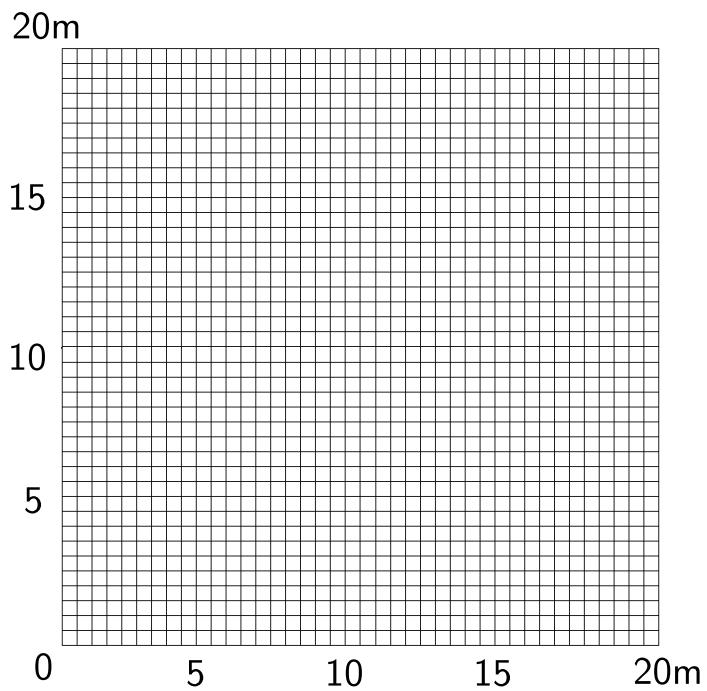


Figure 148: 27NodeBrick edge simply supported square plate with element side length 0.5m

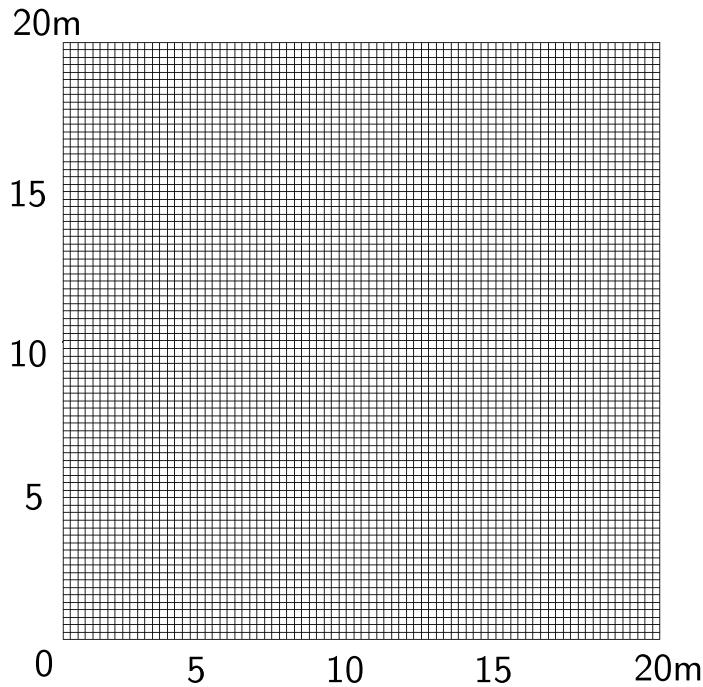


Figure 149: 27NodeBrick edge simply supported square plate with element side length 0.25m

The results were listed in Table (39).

Table 39: Results for 27NodeBrick square plate with four edges simply supported

Element type	27NodeBrick	27NodeBrick	Theoretical displacement
Number of layers	2layers	4layers	
Element side length	Height:0.50m	Height:0.25m	
10m	6.54E-003 m	6.54E-003 m	7.09E-03 m
5m	7.24E-003 m	7.24E-003 m	7.09E-03 m
2m	7.44E-003 m	7.44E-003 m	7.09E-03 m
1m	7.49E-003 m	7.49E-003 m	7.09E-03 m
0.5m	7.50E-003 m	7.50E-003 m	7.09E-03 m
0.25m	7.51E-003 m	- ¹¹	7.09E-03 m

The errors were listed in Table (40).

¹¹This model run out of memory on machine cml01 (memory: 23.5GB). This model has 233,289 nodes with 3 dofs, which may require 40GB memory.

Table 40: Errors for 27NodeBrick square plate with four edges simply supported

Element type	27NodeBrick	27NodeBrick
Number of layers	2layers	4layers
Element side length	Height:0.50m	Height:0.25m
10m	7.87%	7.85%
5m	2.07%	2.10%
2m	4.85%	4.89%
1m	5.54%	5.58%
0.5m	5.74%	5.79%
0.25m	5.80%	-

The errors were plotted in Figure (150).

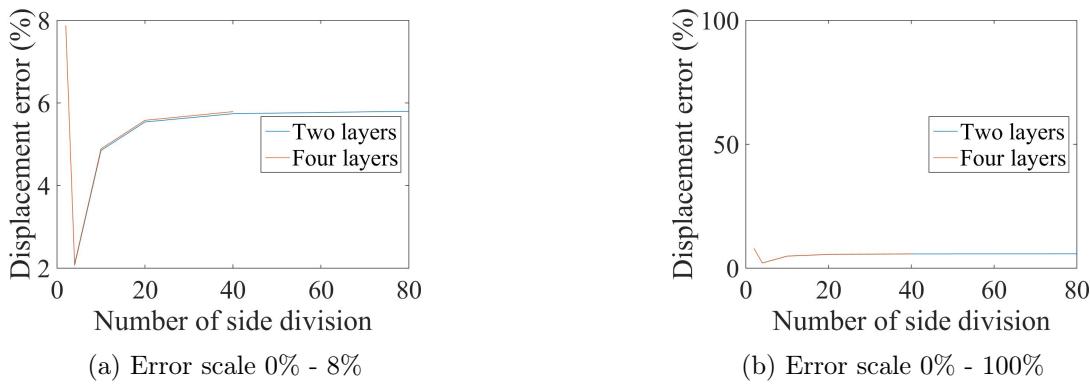


Figure 150: 27NodeBrick square plate with edge simply supported
Displacement error versus Number of side division

The ESSI model fei files for the table above are here

19.7 Verification of 27NodeBrick circular plate with all edges clamped

Problem description: Diameter=20m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.3$.

The four edges are clamped.

The load is the uniform normal pressure on the whole plate.

The plate flexural rigidity is

$$D = \frac{Eh^3}{12(1-\nu^2)} = \frac{10^8 N/m^2 \times 1^3 m^3}{12 \times (1 - 0.3^2)} = 9.1575 \times 10^6 N \cdot m \quad (34)$$

The theoretical solution¹² is

$$d = \frac{qa^4}{64D} = \frac{100N/m^2 \times 10^4 m^4}{64 \times 9.1575 \times 10^6 N \cdot m} = 1.7106 \times 10^{-3} m \quad (35)$$

The 27NodeBrick were shown in Figure (151) - (156).

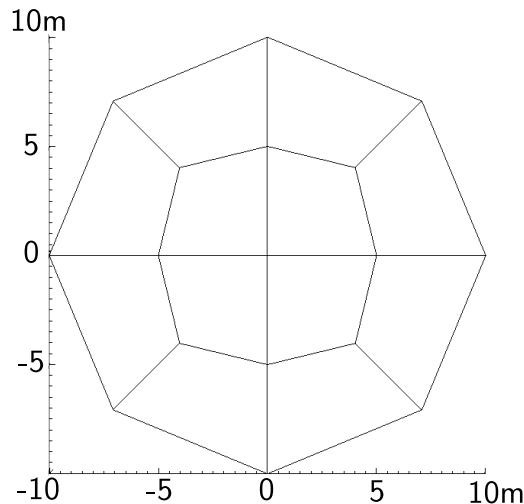


Figure 151: 27NodeBrick edge clamped circular plate with element side length 10m

¹²Stephen Timoshenko, Theory of plates and shells (2nd edition). MrGRAW-Hill Inc, page55, 1959.

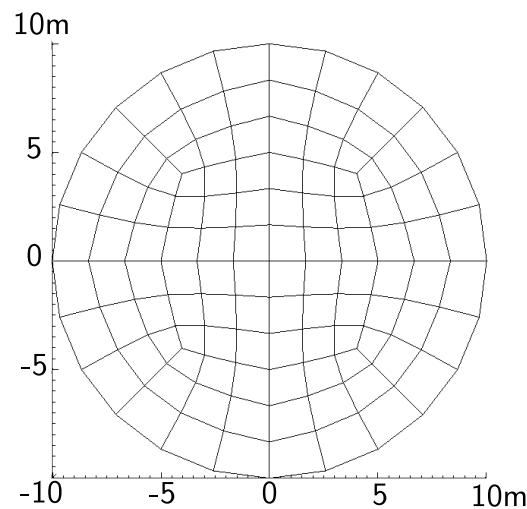


Figure 152: 27NodeBrick edge clamped circular plate with element side length 5m

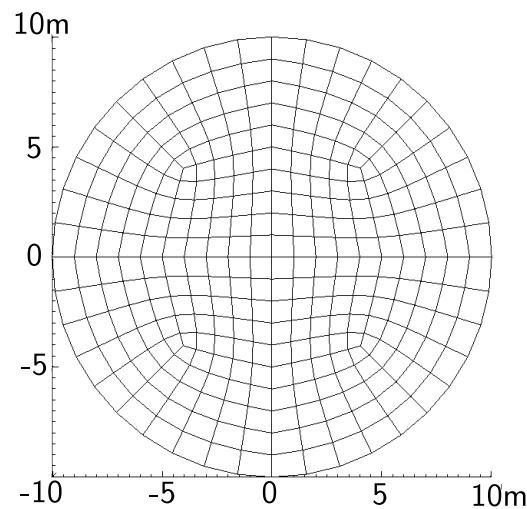


Figure 153: 27NodeBrick edge clamped circular plate with element side length 2m

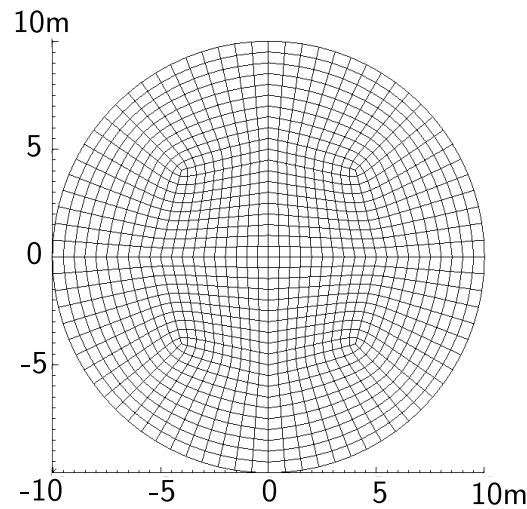


Figure 154: 27NodeBrick edge clamped circular plate with element side length 1m

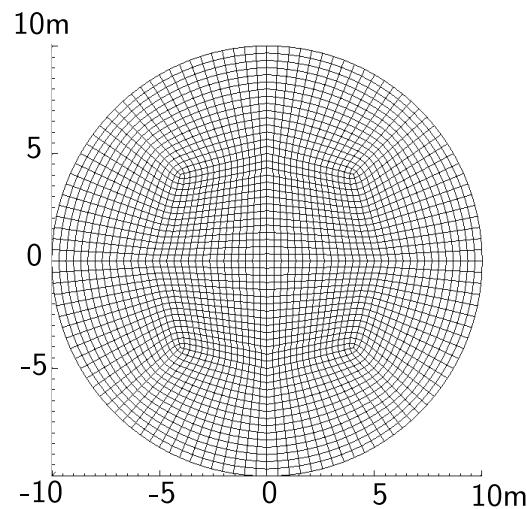


Figure 155: 27NodeBrick edge clamped circular plate with element side length 0.5m

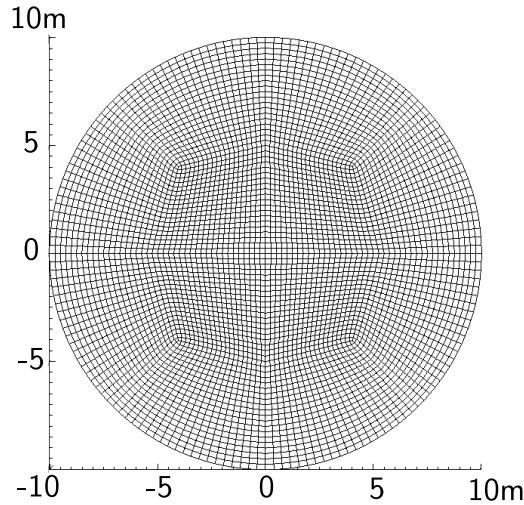


Figure 156: 27NodeBrick edge clamped circular plate with element side length 0.25m

The results were listed in Table (41).

Table 41: Results for 27NodeBrick circular plate with four edges clamped

Element type	27NodeBrick	27NodeBrick	27NodeBrick	Theoretical displacement
Number of layers	1layer	2layers	4layers	
Number of diameter divisions	Height:1.00m	Height:0.50m	Height:0.25m	
4	2.777E-03 m	2.788E-03 m	2.789E-03 m	1.706E-03 m
12	2.772E-03 m	2.786E-03 m	2.787E-03 m	1.706E-03 m
20	2.545E-03 m	2.556E-03 m	2.558E-03 m	1.706E-03 m
40	1.758E-03 m	1.768E-03 m	1.769E-03 m	1.706E-03 m
60	1.762E-03 m	1.772E-03 m	1.773E-03 m	1.706E-03 m
80	1.763E-03 m	1.773E-03 m	1.774E-03 m	1.706E-03 m

The errors were listed in Table (42).

Table 42: Errors for 27NodeBrick circular plate with four edges clamped

Element type	27NodeBrick	27NodeBrick	27NodeBrick
Number of layers	1layer	2layers	4layers
Number of diameter divisions	Height:1.00m	Height:0.50m	Height:0.25m
4	62.75%	63.42%	63.47%
12	62.46%	63.27%	63.34%
20	49.14%	49.82%	49.91%
40	3.03%	3.62%	3.68%
60	3.25%	3.83%	3.91%
80	3.32%	3.91%	3.99%

The errors were shown in Figure (157).

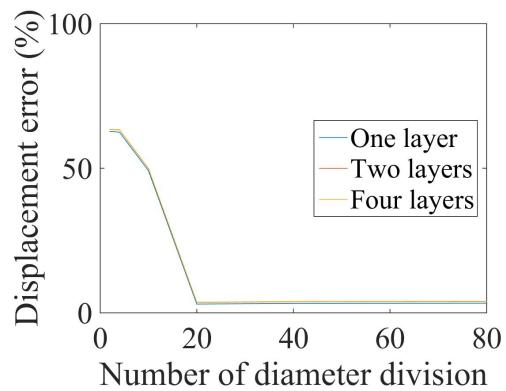


Figure 157: 27NodeBrick circular plate with edge clamped
Displacement error versus Number of side division

The ESSI model fei files for the table above are here

19.8 Verification of 27NodeBrick circular plate with all edges simply supported

Problem description: Diameter=20m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.3$.

The four edges are simply supported.

The load is the uniform normal pressure on the whole plate.

The plate flexural rigidity is

$$D = \frac{Eh^3}{12(1-\nu^2)} = \frac{10^8 N/m^2 \times 1^3 m^3}{12 \times (1 - 0.3^2)} = 9.1575 \times 10^6 N \cdot m \quad (36)$$

The theoretical solution¹³ is

$$d = \frac{(5 + \nu)qa^4}{64(1 + \nu)D} = \frac{(5 + 0.3) \times 100N/m^2 \times 10^4 m^4}{64 \times (1 + 0.3) \times 9.1575 \times 10^6 N \cdot m} = 6.956 \times 10^{-3} m \quad (37)$$

The 27NodeBrick were shown in Figure (158) - (163).

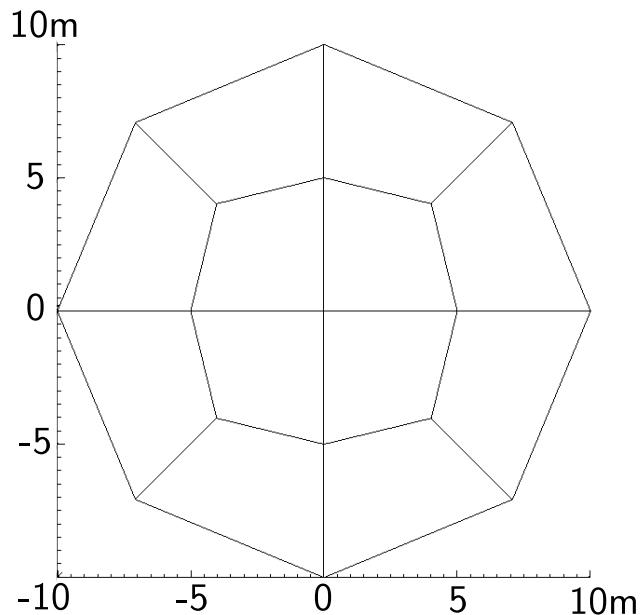


Figure 158: 27NodeBrick edge simply supported circular plate with element side length 10m

¹³Stephen Timoshenko, Theory of plates and shells (2nd edition). MrGRAW-Hill Inc, page55, 1959.

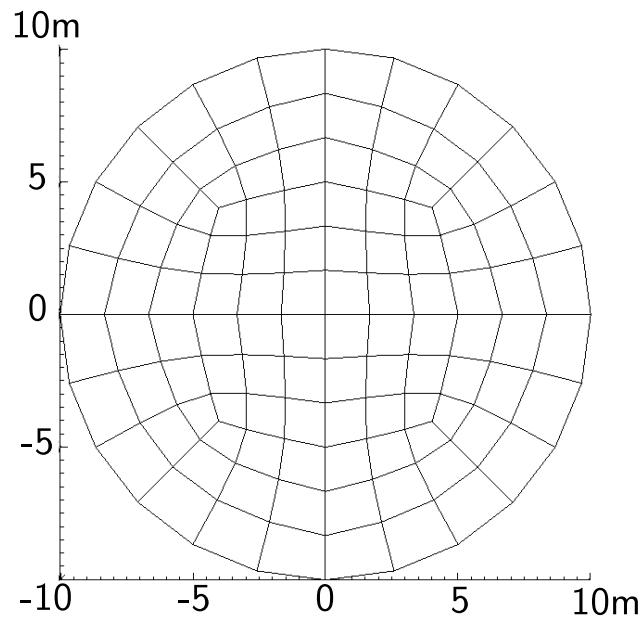


Figure 159: 27NodeBrick edge simply supported circular plate with element side length 5m

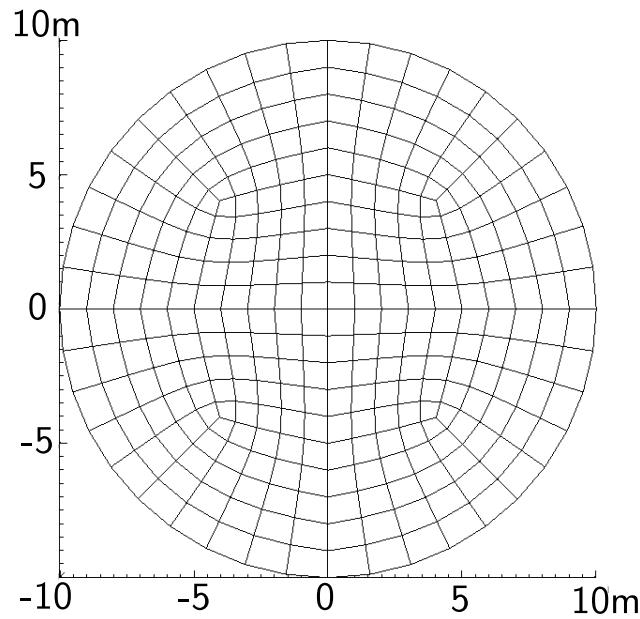


Figure 160: 27NodeBrick edge simply supported circular plate with element side length 2m

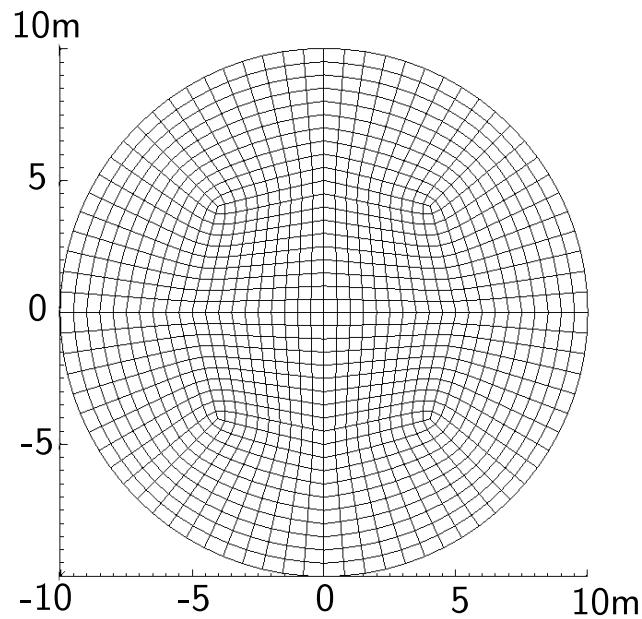


Figure 161: 27NodeBrick edge simply supported circular plate with element side length 1m

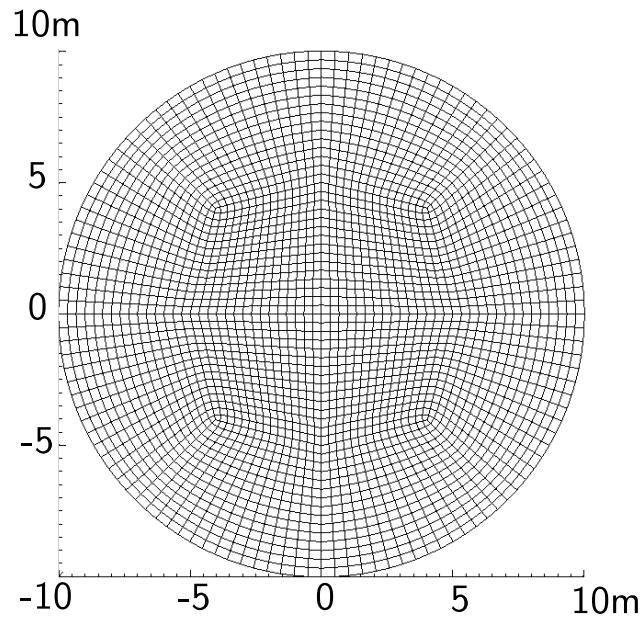


Figure 162: 27NodeBrick edge simply supported circular plate with element side length 0.5m

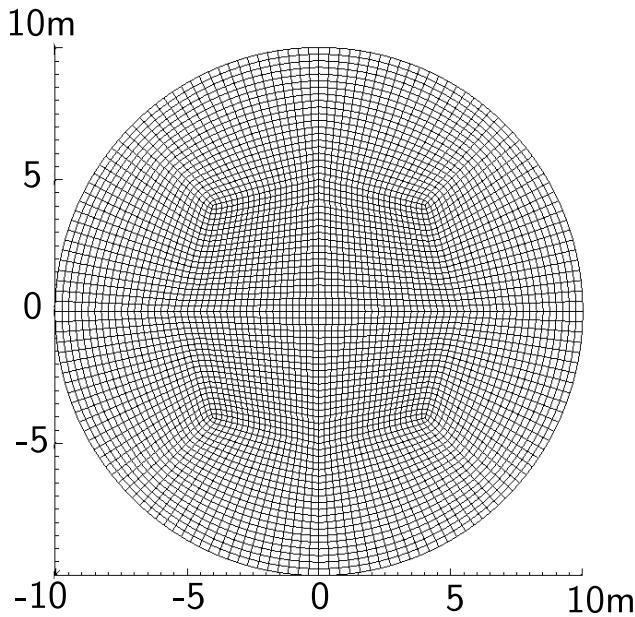


Figure 163: 27NodeBrick edge simply supported circular plate with element side length 0.25m

The results were listed in Table (43).

Table 43: Results for 27NodeBrick cicular plate with four edges simply supported

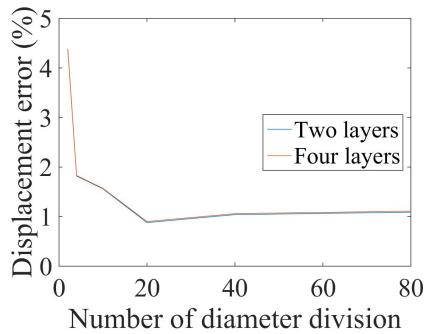
Element type	27NodeBrick	27NodeBrick	Theoretical displacement
Number of layers	2layers	4layers	6.956E-03 m
Number of diameter divisions	Height:0.50m	Height:0.25m	
4	7.259E-03 m	7.261E-03 m	
12	7.083E-03 m	7.084E-03 m	
20	7.064E-03 m	7.065E-03 m	
40	7.018E-03 m	7.019E-03 m	
60	7.029E-03 m	7.030E-03 m	
80	7.032E-03 m	7.034E-03 m	

The errors were listed in Table (44).

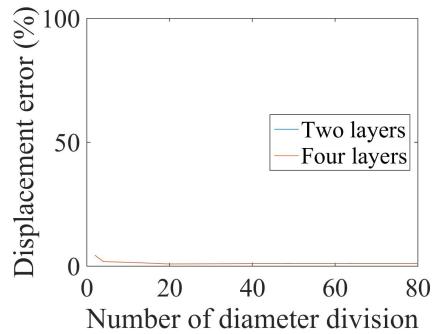
Table 44: Errors for 27NodeBrick cicular plate with four edges simply supported

Element type	27NodeBrick	27NodeBrick
Number of layers	2layers	4layers
Number of diameter divisions	Height:0.50m	Height:0.25m
4	4.36%	4.38%
12	1.82%	1.83%
20	1.56%	1.57%
40	0.88%	0.90%
60	1.04%	1.06%
80	1.09%	1.11%

The errors were plotted in Figure (164).



(a) Error scale 0% - 5%



(b) Error scale 0% - 100%

Figure 164: 27NodeBrick circular plate with edge simply supported
Displacement error versus Number of side division

The ESSI model fei files for the table above are here

Verification for 4NodeANDES

20 Verification of 4NodeANDES elements

20.1 Verification of 4NodeANDES cantilever beams

Problem description: Length=6m, Width=1m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.0$. Use the shear deformation coefficient $\kappa = 1.2$. The force direction was shown in Figure (165).

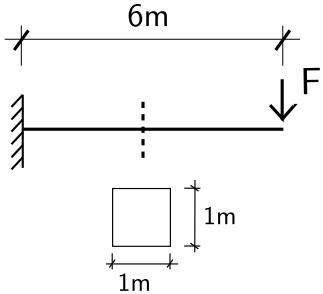


Figure 165: Problem description for cantilever beams

Theoretical displacement (bending and shear deformation):

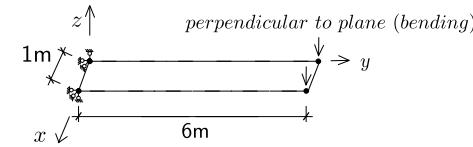
$$\begin{aligned}
 d &= \frac{FL^3}{3EI} + \frac{FL}{GA_v} \\
 &= \frac{FL^3}{3E\frac{bh^3}{12}} + \frac{FL}{\frac{E}{2(1+\nu)} bh} \\
 &= \frac{100N \times 6^3 m^3}{3 \times 10^8 N/m^2 \times \frac{1}{12} m^4} + \frac{100N \times 6m}{\frac{10}{2} \times 10^7 N/m^2 \times 1m^2 \times \frac{5}{6}} \\
 &= 8.64 \times 10^{-4} m + 0.144 \times 10^{-4} m \\
 &= 8.784 \times 10^{-4} m
 \end{aligned} \tag{38}$$

4NodeANDES element model:

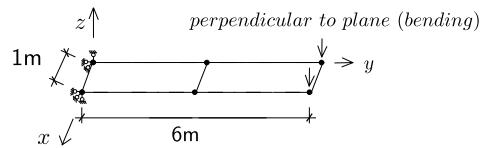
- **Force direction: perpendicular to plane (bending)**

When the force direction is perpendicular to the plane, only the bending deformation is calculated in 4NodeANDES elements.

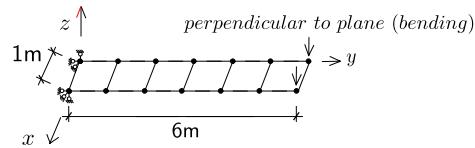
The 4NodeANDES elements were shown in Figure (166).



(a) One 4NodeANDES element



(b) Two 4NodeANDES elements



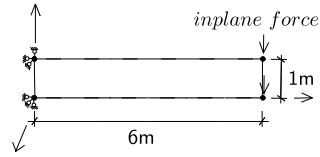
(c) Six 4NodeANDES elements

Figure 166: 4NodeANDES elements for cantilever beams under force perpendicular to plane

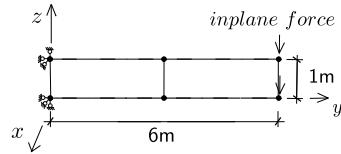
- **Force direction: inplane force**

When the force direction is inplane, both the bending and shear deformation are calculated in 4NodeANDES elements.

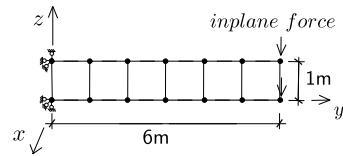
The 4NodeANDES elements under inplane force were shown in Figure (167).



(a) One 4NodeANDES element



(b) Two 4NodeANDES elements



(c) Six 4NodeANDES elements

Figure 167: 4NodeANDES elements for cantilever beams under inplane force

The ESSI results for the force ***perpendicular to plane (bending)*** were listed in Table (45). The theoretical solution is 8.784E-04 m.

Table 45: Results for 4NodeANDES cantilever beams under the force perpendicular to plane (bending)

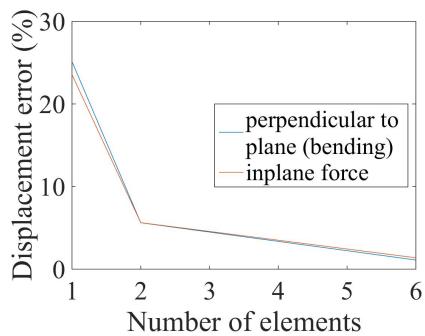
Element number	1	2	6
4NodeANDES	6.56E-04 m	8.27E-04 m	8.86E-04 m
Error	25.34%	5.87%	0.83%

The ESSI results for the ***inplane force*** were listed in Table (46). The theoretical solution is 8.784E-04 m.

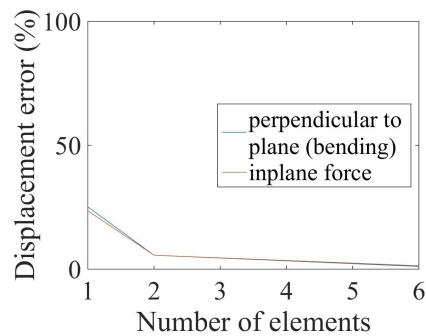
Table 46: Results for 4NodeANDES cantilever beams under the inplane force

Element number	1	2	6
4NodeANDES	6.70E-04 m	8.27E-04 m	8.64E-04 m
Error	23.77%	5.89%	1.65%

The errors were plotted in Figure (168).



(a) Error scale 0% - 30%



(b) Error scale 0% - 100%

Figure 168: 4NodeANDES cantilever beam for different element number
Displacement error versus Number of elements

The ESSI model fei files for the table above are here

20.2 Verification of 4NodeANDES cantilever beam for different Poisson's ratio

Problem description: Length=6m, Width=1m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.0 - 0.49$. The force direction was shown in Figure (169).

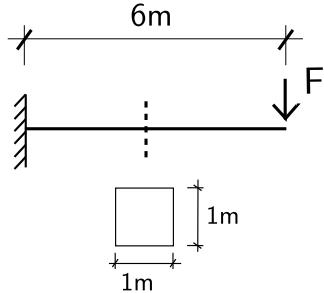


Figure 169: Problem description for cantilever beams of different Poisson's ratios

The theoretical solution for $\nu = 0.0$ was calculated below, while the solution for other Poisson's ratio were calculated by the similar process.

Theoretical displacement (bending and shear deformation):

$$\begin{aligned}
 d &= \frac{FL^3}{3EI} + \frac{FL}{GA_v} \\
 &= \frac{FL^3}{3E\frac{bh^3}{12}} + \frac{FL}{\frac{E}{2(1+\nu)} \frac{bh}{\kappa}} \\
 &= \frac{100N \times 6^3 m^3}{3 \times 10^8 N/m^2 \times \frac{1}{12} m^4} + \frac{100N \times 6m}{\frac{10}{2} \times 10^7 N/m^2 \times 1m^2 \times \frac{5}{6}} \\
 &= 8.64 \times 10^{-4} m + 0.144 \times 10^{-4} m \\
 &= 8.784 \times 10^{-4} m
 \end{aligned} \tag{39}$$

The rotation angle at the end:

$$\theta = \frac{FL^2}{2EI} = \frac{100N \times 6^2 m^2}{2 \times 10^8 N/m^2 \times \frac{1}{12} m^4} = 2.16 \times 10^{-4} rad = 0.0124^\circ \tag{40}$$

The 4NodeANDES elements for cantilever beams of different Poisson's ratios were shown in Figure (170) and (171):

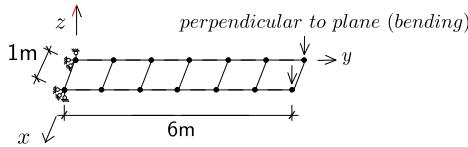


Figure 170: 4NodeANDES elements for different Poisson's ratios under the force perpendicular to plane (bending)

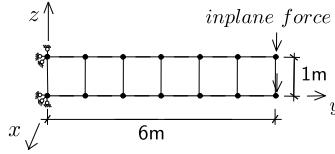


Figure 171: 4NodeANDES elements for different Poisson's ratios under the inplane force

The ESSI results for the force ***perpendicular to plane (bending)*** were listed in Table (47) - (49).

Table 47: ***Displacement error*** results for 4NodeANDES with element side length 1 m under the force perpendicular to plane (bending)

Poisson's ratio	4NodeANDES displacement	Theory displacement (bending)	Theory displacement (shear)	Theory displacement(all)	Error
0.00	8.639E-04 m	8.640E-04 m	1.440E-05 m	8.784E-04 m	1.38%
0.05	8.635E-04 m	8.640E-04 m	1.512E-05 m	8.791E-04 m	1.49%
0.10	8.622E-04 m	8.640E-04 m	1.586E-05 m	8.799E-04 m	1.71%
0.15	8.599E-04 m	8.640E-04 m	1.659E-05 m	8.806E-04 m	2.04%
0.20	8.566E-04 m	8.640E-04 m	1.734E-05 m	8.813E-04 m	2.48%
0.25	8.522E-04 m	8.640E-04 m	1.808E-05 m	8.821E-04 m	3.05%
0.30	8.466E-04 m	8.640E-04 m	1.884E-05 m	8.828E-04 m	3.75%
0.35	8.398E-04 m	8.640E-04 m	1.959E-05 m	8.836E-04 m	4.59%
0.40	8.315E-04 m	8.640E-04 m	2.035E-05 m	8.844E-04 m	5.60%
0.45	8.216E-04 m	8.640E-04 m	2.111E-05 m	8.851E-04 m	6.78%
0.49	8.124E-04 m	8.640E-04 m	2.173E-05 m	8.857E-04 m	7.88%

Table 48: ***Displacement error*** results for 4NodeANDES with element side length 0.5 m under the force perpendicular to plane (bending)

Poisson's ratio	4NodeANDES displacement	Theory displacement (bending)	Theory displacement (shear)	Theory displacement(all)	Error
0.00	8.724E-04 m	8.640E-04 m	1.440E-05 m	8.784E-04 m	0.68%
0.05	8.724E-04 m	8.640E-04 m	1.512E-05 m	8.791E-04 m	0.76%
0.10	8.717E-04 m	8.640E-04 m	1.586E-05 m	8.799E-04 m	0.93%
0.15	8.703E-04 m	8.640E-04 m	1.659E-05 m	8.806E-04 m	1.17%
0.20	8.682E-04 m	8.640E-04 m	1.734E-05 m	8.813E-04 m	1.49%
0.25	8.652E-04 m	8.640E-04 m	1.808E-05 m	8.821E-04 m	1.91%
0.30	8.615E-04 m	8.640E-04 m	1.884E-05 m	8.828E-04 m	2.42%
0.35	8.569E-04 m	8.640E-04 m	1.959E-05 m	8.836E-04 m	3.02%
0.40	8.514E-04 m	8.640E-04 m	2.035E-05 m	8.844E-04 m	3.73%
0.45	8.449E-04 m	8.640E-04 m	2.111E-05 m	8.851E-04 m	4.54%
0.49	8.388E-04 m	8.640E-04 m	2.173E-05 m	8.857E-04 m	5.30%

Table 49: *Displacement error* results for 4NodeANDES with element side length 0.25 m under the force perpendicular to plane (bending)

Poisson's ratio	4NodeANDES displacement	Theory displacement (bending)	Theory displacement (shear)	Theory displacement(all)	Error
0.00	8.640E-04 m	8.640E-04 m	1.440E-05 m	8.784E-04 m	1.64%
0.05	8.637E-04 m	8.640E-04 m	1.512E-05 m	8.791E-04 m	1.75%
0.10	8.627E-04 m	8.640E-04 m	1.586E-05 m	8.799E-04 m	1.95%
0.15	8.611E-04 m	8.640E-04 m	1.659E-05 m	8.806E-04 m	2.21%
0.20	8.588E-04 m	8.640E-04 m	1.734E-05 m	8.813E-04 m	2.56%
0.25	8.559E-04 m	8.640E-04 m	1.808E-05 m	8.821E-04 m	2.97%
0.30	8.523E-04 m	8.640E-04 m	1.884E-05 m	8.828E-04 m	3.46%
0.35	8.480E-04 m	8.640E-04 m	1.959E-05 m	8.836E-04 m	4.03%
0.40	8.429E-04 m	8.640E-04 m	2.035E-05 m	8.844E-04 m	4.69%
0.45	8.370E-04 m	8.640E-04 m	2.111E-05 m	8.851E-04 m	5.44%
0.49	8.316E-04 m	8.640E-04 m	2.173E-05 m	8.857E-04 m	6.11%

The errors were plotted in Figure (172).

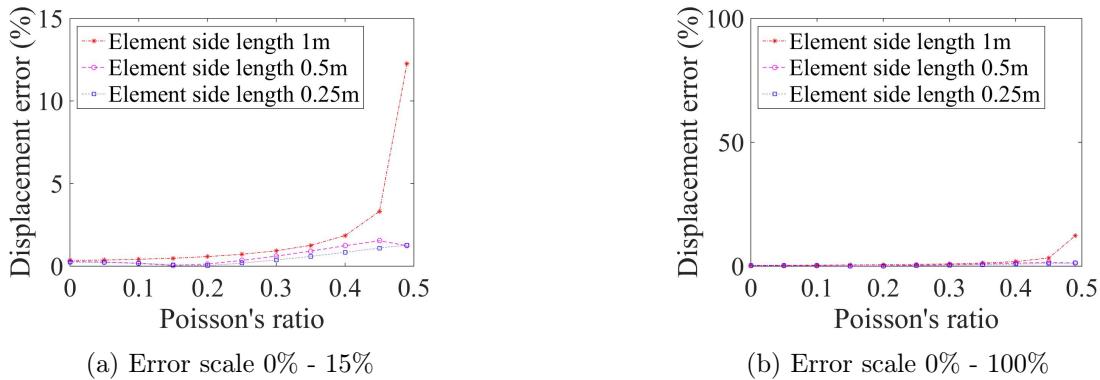


Figure 172: 4NodeANDES cantilever beam for force perpendicular to the plane(bending)

Displacement error versus Poisson's ratio

The ESSI results for the *inplane force* were listed in Table (50) - (52).

Table 50: *Displacement error* results for 4NodeANDES with
element side length 1 m under the inplane force

Poisson's ratio	4NodeANDES displacement	Theory displacement (bending)	Theory displacement (shear)	Theory displacement(all)	Error
0.00	8.790E-04 m	8.640E-04 m	1.440E-05 m	8.784E-04 m	0.07%
0.05	8.799E-04 m	8.640E-04 m	1.512E-05 m	8.791E-04 m	0.09%
0.10	8.809E-04 m	8.640E-04 m	1.586E-05 m	8.799E-04 m	0.12%
0.15	8.821E-04 m	8.640E-04 m	1.659E-05 m	8.806E-04 m	0.17%
0.20	8.835E-04 m	8.640E-04 m	1.734E-05 m	8.813E-04 m	0.25%
0.25	8.853E-04 m	8.640E-04 m	1.808E-05 m	8.821E-04 m	0.37%
0.30	8.878E-04 m	8.640E-04 m	1.884E-05 m	8.828E-04 m	0.56%
0.35	8.913E-04 m	8.640E-04 m	1.959E-05 m	8.836E-04 m	0.87%
0.40	8.971E-04 m	8.640E-04 m	2.035E-05 m	8.844E-04 m	1.44%
0.45	9.107E-04 m	8.640E-04 m	2.111E-05 m	8.851E-04 m	2.89%
0.49	9.901E-04 m	8.640E-04 m	2.173E-05 m	8.857E-04 m	11.79%

Table 51: *Displacement error* results for 4NodeANDES with
element side length 0.5 m under the inplane force

Poisson's ratio	4NodeANDES displacement	Theory displacement (bending)	Theory displacement (shear)	Theory displacement(all)	Error
0.00	8.784E-04 m	8.640E-04 m	1.440E-05 m	8.784E-04 m	0.00%
0.05	8.788E-04 m	8.640E-04 m	1.512E-05 m	8.791E-04 m	0.04%
0.10	8.787E-04 m	8.640E-04 m	1.586E-05 m	8.799E-04 m	0.13%
0.15	8.782E-04 m	8.640E-04 m	1.659E-05 m	8.806E-04 m	0.27%
0.20	8.772E-04 m	8.640E-04 m	1.734E-05 m	8.813E-04 m	0.47%
0.25	8.759E-04 m	8.640E-04 m	1.808E-05 m	8.821E-04 m	0.70%
0.30	8.742E-04 m	8.640E-04 m	1.884E-05 m	8.828E-04 m	0.98%
0.35	8.722E-04 m	8.640E-04 m	1.959E-05 m	8.836E-04 m	1.29%
0.40	8.699E-04 m	8.640E-04 m	2.035E-05 m	8.844E-04 m	1.63%
0.45	8.679E-04 m	8.640E-04 m	2.111E-05 m	8.851E-04 m	1.94%
0.49	8.709E-04 m	8.640E-04 m	2.173E-05 m	8.857E-04 m	1.67%

Table 52: *Displacement error* results for 4NodeANDES with element side length 0.25 m under the inplane force

Poisson's ratio	4NodeANDES displacement	Theory displacement (bending)	Theory displacement (shear)	Theory displacement(all)	Error
0.00	8.782E-04 m	8.640E-04 m	1.440E-05 m	8.784E-04 m	0.02%
0.05	8.786E-04 m	8.640E-04 m	1.512E-05 m	8.791E-04 m	0.06%
0.10	8.788E-04 m	8.640E-04 m	1.586E-05 m	8.799E-04 m	0.12%
0.15	8.786E-04 m	8.640E-04 m	1.659E-05 m	8.806E-04 m	0.23%
0.20	8.781E-04 m	8.640E-04 m	1.734E-05 m	8.813E-04 m	0.37%
0.25	8.774E-04 m	8.640E-04 m	1.808E-05 m	8.821E-04 m	0.53%
0.30	8.763E-04 m	8.640E-04 m	1.884E-05 m	8.828E-04 m	0.74%
0.35	8.750E-04 m	8.640E-04 m	1.959E-05 m	8.836E-04 m	0.97%
0.40	8.734E-04 m	8.640E-04 m	2.035E-05 m	8.844E-04 m	1.24%
0.45	8.717E-04 m	8.640E-04 m	2.111E-05 m	8.851E-04 m	1.52%
0.49	8.706E-04 m	8.640E-04 m	2.173E-05 m	8.857E-04 m	1.71%

The errors were plotted in Figure (172).

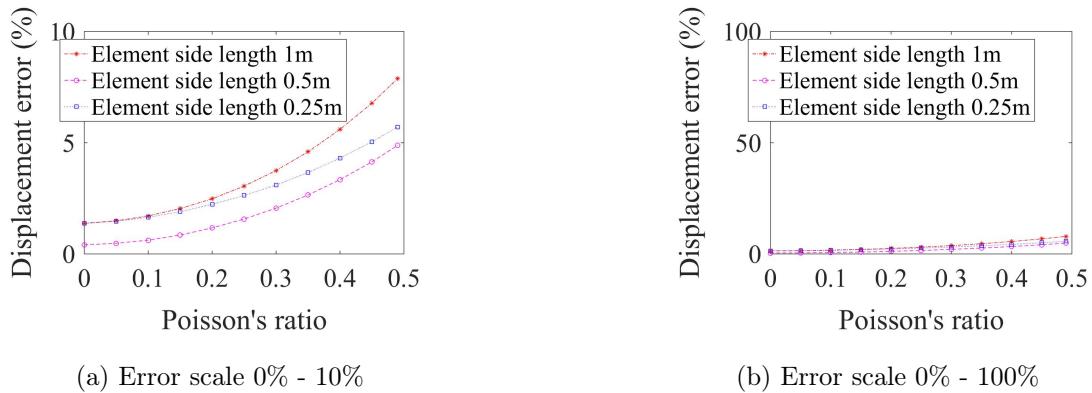


Figure 173: 4NodeANDES cantilever beam for inplane force
Displacement error versus Poisson's ratio

The angle results for the force *perpendicular to plane (bending)* were listed in Table (53).

Table 53: ***Rotation angle*** results for element side length 1 m under the force perpendicular to plane (bending)

Poisson's ratio	4NodeANDES angle (unit: $^{\circ}$)	Theory angle (unit: $^{\circ}$)	Error
0.00	1.238E-02	1.240E-02	0.19%
0.05	1.237E-02	1.240E-02	0.23%
0.10	1.236E-02	1.240E-02	0.34%
0.15	1.234E-02	1.240E-02	0.52%
0.20	1.230E-02	1.240E-02	0.78%
0.25	1.226E-02	1.240E-02	1.12%
0.30	1.221E-02	1.240E-02	1.54%
0.35	1.214E-02	1.240E-02	2.07%
0.40	1.206E-02	1.240E-02	2.70%
0.45	1.197E-02	1.240E-02	3.46%
0.49	1.188E-02	1.240E-02	4.16%

Table 54: ***Rotation angle*** results for element side length 0.5 m the force perpendicular to plane (bending)

Poisson's ratio	4NodeANDES angle (unit: $^{\circ}$)	Theory angle (unit: $^{\circ}$)	Error
0.00	1.239E-02	1.240E-02	0.10%
0.05	1.238E-02	1.240E-02	0.13%
0.10	1.237E-02	1.240E-02	0.22%
0.15	1.236E-02	1.240E-02	0.36%
0.20	1.233E-02	1.240E-02	0.55%
0.25	1.230E-02	1.240E-02	0.81%
0.30	1.226E-02	1.240E-02	1.13%
0.35	1.221E-02	1.240E-02	1.52%
0.40	1.216E-02	1.240E-02	1.97%
0.45	1.209E-02	1.240E-02	2.51%
0.49	1.203E-02	1.240E-02	3.00%

Table 55: ***Rotation angle*** results for element side length 0.25 m under the force perpendicular to plane (bending)

Poisson's ratio	4NodeANDES angle (unit: $^{\circ}$)	Theory angle (unit: $^{\circ}$)	Error
0.00	1.238E-02	1.240E-02	0.19%
0.05	1.237E-02	1.240E-02	0.21%
0.10	1.237E-02	1.240E-02	0.28%
0.15	1.235E-02	1.240E-02	0.39%
0.20	1.233E-02	1.240E-02	0.56%
0.25	1.230E-02	1.240E-02	0.78%
0.30	1.227E-02	1.240E-02	1.05%
0.35	1.223E-02	1.240E-02	1.38%
0.40	1.218E-02	1.240E-02	1.77%
0.45	1.212E-02	1.240E-02	2.23%
0.49	1.207E-02	1.240E-02	2.64%

The errors were plotted in Figure (174).

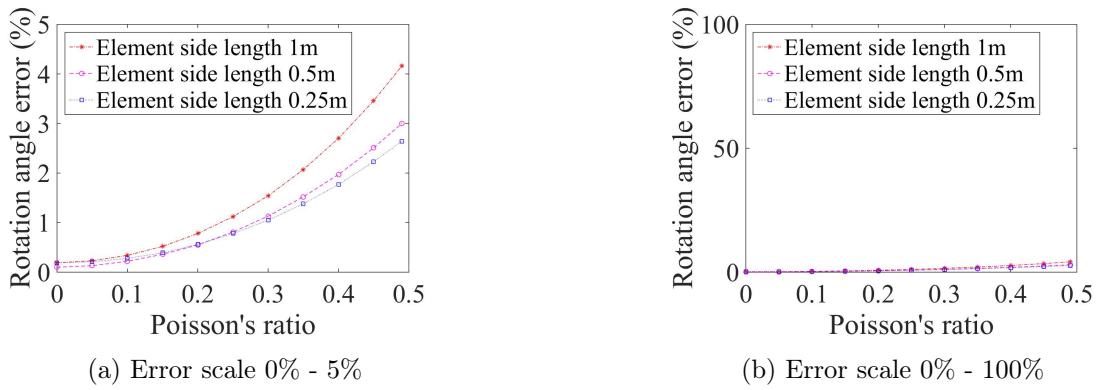


Figure 174: 4NodeANDES cantilever beam for force perpendicular to the plane(bending)
Rotation angle error versus Poisson's ratio

The ESSI results for the *inplane force* were listed in Table (56 - (58).

Table 56: ***Rotation angle*** results for element side length 1 m under the inplane force

Poisson's ratio	4NodeANDES angle (unit: $^{\circ}$)	Theory angle (unit: $^{\circ}$)	Error
0.00	1.254E-02	1.240E-02	1.14%
0.05	1.255E-02	1.240E-02	1.19%
0.10	1.256E-02	1.240E-02	1.26%
0.15	1.257E-02	1.240E-02	1.35%
0.20	1.258E-02	1.240E-02	1.47%
0.25	1.260E-02	1.240E-02	1.64%
0.30	1.263E-02	1.240E-02	1.89%
0.35	1.269E-02	1.240E-02	2.30%
0.40	1.278E-02	1.240E-02	3.08%
0.45	1.305E-02	1.240E-02	5.28%
0.49	1.506E-02	1.240E-02	21.43%

Table 57: ***Rotation angle*** results for element side length 0.5 m under the inplane force

Poisson's ratio	4NodeANDES angle (unit: $^{\circ}$)	Theory angle (unit: $^{\circ}$)	Error
0.00	1.271E-02	1.240E-02	2.51%
0.05	1.272E-02	1.240E-02	2.56%
0.10	1.272E-02	1.240E-02	2.58%
0.15	1.272E-02	1.240E-02	2.60%
0.20	1.273E-02	1.240E-02	2.63%
0.25	1.273E-02	1.240E-02	2.67%
0.30	1.274E-02	1.240E-02	2.77%
0.35	1.277E-02	1.240E-02	2.98%
0.40	1.283E-02	1.240E-02	3.47%
0.45	1.299E-02	1.240E-02	4.79%
0.49	1.361E-02	1.240E-02	9.78%

Table 58: ***Rotation angle*** results for element side length 0.25 m under the inplane force

Poisson's ratio	4NodeANDES angle (unit: $^{\circ}$)	Theory angle (unit: $^{\circ}$)	Error
0.00	1.268E-02	1.240E-02	2.24%
0.05	1.268E-02	1.240E-02	2.27%
0.10	1.268E-02	1.240E-02	2.30%
0.15	1.269E-02	1.240E-02	2.31%
0.20	1.269E-02	1.240E-02	2.33%
0.25	1.269E-02	1.240E-02	2.35%
0.30	1.270E-02	1.240E-02	2.41%
0.35	1.271E-02	1.240E-02	2.53%
0.40	1.275E-02	1.240E-02	2.83%
0.45	1.284E-02	1.240E-02	3.58%
0.49	1.312E-02	1.240E-02	5.77%

The errors were plotted in Figure (174).

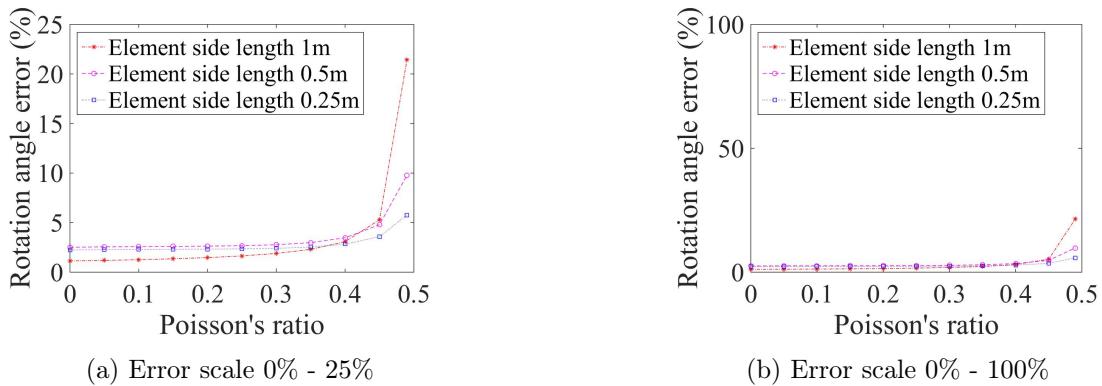


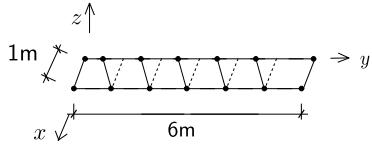
Figure 175: 4NodeANDES cantilever beam for inplane force
Rotation angle error versus Poisson's ratio

The ESSI model fei files for the table above are here

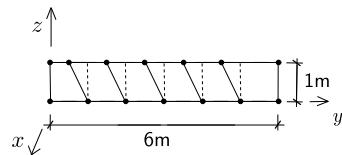
20.3 Test of irregular shaped 4NodeANDES cantilever beams

Cantilever model was used as an example. Three different shapes were tested.

In the ***first*** test, the upper two nodes of each element were moved one half element size along the $y - axis$, while the lower two nodes were kept at the same location. The element shape was shown in Figure (176).



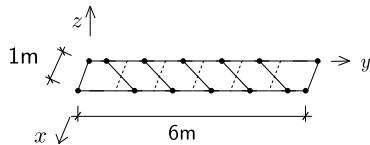
(a) Horizontal plane



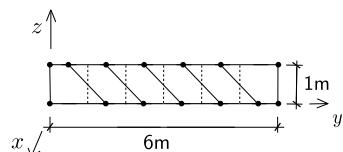
(b) Veritical plane

Figure 176: 4NodeANDES cantilever beam for irregular ***Shape 1***

In the ***second*** test, the upper nodes of each element were moved 50% element size along the $y - axis$, while the lower nodes were moved 50% element size in the other direction along the $y - axis$. The element shape was shown in Figure (177).



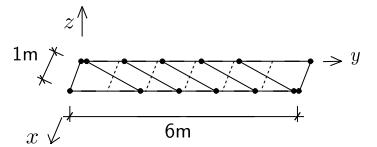
(a) Horizontal plane



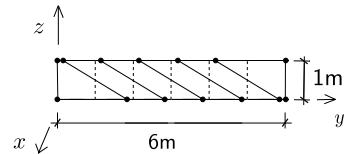
(b) Veritical plane

Figure 177: 4NodeANDES cantilever beam for irregular ***Shape 2***

In the ***third*** test, the upper two nodes of each element were moved 90% element size with different directions along the $y - axis$, while the lower nodes were moved 90% element size in the other direction along the $y - axis$. The element shape was shown in Figure (178).



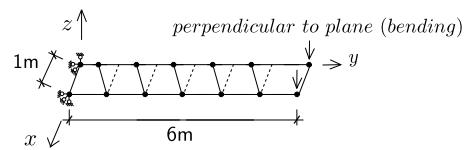
(a) Horizontal plane



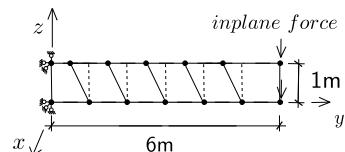
(b) Veritical plane

Figure 178: 4NodeANDES cantilever beam for irregular ***Shape 3***

The boundary conditions were shown in Figure (179), (180) and (181).

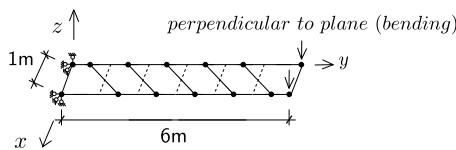


(a) Horizontal plane

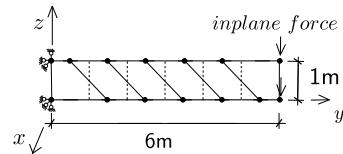


(b) Veritical plane

Figure 179: 4NodeANDES cantilever beam boundary conditions for irregular ***Shape 1***

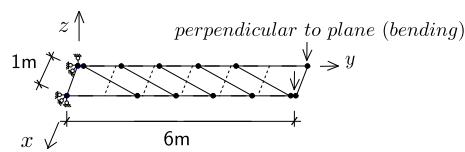


(a) Horizontal plane

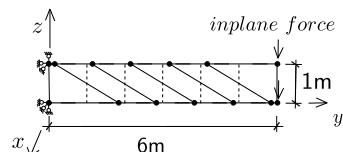


(b) Veritical plane

Figure 180: 4NodeANDES cantilever beam boundary conditions for irregular ***Shape 2***



(a) Horizontal plane



(b) Veritical plane

Figure 181: 4NodeANDES cantilever beam boundary conditions for irregular ***Shape 3***

The ESSI results were listed in Table (59).

Table 59: Results for 4NodeANDES cantilever beams of irregular shapes

Displacements for irregular shaped element					
Element Type	Force direction	Normal shape	Shape 1	Shape 2	Shape 3
4NodeANDES	perpendicular to plane (bending)	8.639E-04 m	8.602E-04 m	8.534E-04 m	7.851E-04 m
4NodeANDES	inplane force	8.857E-04 m	7.036E-04 m	4.263E-04 m	1.909E-04 m
Theoretical	-	8.784E-04 m	8.784E-04 m	8.784E-04 m	8.784E-04 m

The errors were listed in Table (60) and (61).

Table 60: Errors for irregular shaped 4NodeANDES compared to theoretical solution

Errors for irregular shaped element, compared to theoretical solutions					
Element Type	Force direction	Normal shape	Shape 1	Shape 2	Shape 3
4NodeANDES	perpendicular to plane (bending)	1.65%	2.07%	2.85%	10.63%
4NodeANDES	inplane force	0.83%	19.90%	51.47%	78.27%

Table 61: Errors for irregular shaped 4NodeANDES compared to normal shape

Errors for irregular shaped element, compared to normal shape					
Element Type	Force direction	Normal shape	Shape 1	Shape 2	Shape 3
4NodeANDES	perpendicular to plane (bending)	0.00%	0.42%	1.22%	9.12%
4NodeANDES	inplane force	0.00%	20.56%	51.87%	78.45%

The ESSI model fei files for the table above are here

Then, the beam was divided into small elements.

Problem description: Length=6m, Width=1m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.0$. Use the shear deformation coefficient $\kappa = 1.2$. The force direction was shown in Figure (182).

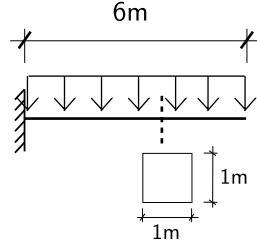


Figure 182: Problem description for cantilever beams under uniform pressure

Theoretical displacement (bending and shear deformation):

$$\begin{aligned}
 d &= \frac{qL^4}{8EI} + \frac{q\frac{L^2}{2}}{GA_v} \\
 &= \frac{qL^4}{8E\frac{bh^3}{12}} + \frac{q\frac{L^2}{2}}{\frac{E}{2(1+\nu)} bh} \\
 &= \frac{400N/m \times 12^4 m^4}{8 \times 10^8 N/m^2 \times \frac{2^4}{12} m^4} + \frac{400N/m \times \frac{12^2}{2} m^2}{\frac{10^8}{2} N/m^2 \times 2m \times 2m \times \frac{5}{6}} \\
 &= 7.776 \times 10^{-3} m + 1.728 \times 10^{-4} m \\
 &= 7.9488 \times 10^{-3} m
 \end{aligned} \tag{41}$$

The ESSI displacement results were listed in Table (62).

Table 62: Results for 4NodeANDES cantilever beams of irregular shapes with more elements

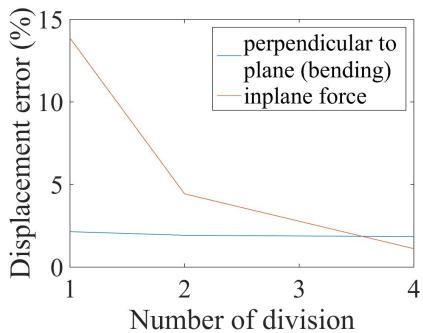
Element Type	Shape	Force direction	Number of division		
			1	2	4
4NodeANDES	shape1	perpendicular to plane (bending)	7.750E-03 m	7.768E-03 m	7.774E-03 m
4NodeANDES	shape1	inplane force	6.822E-03 m	7.569E-03 m	7.832E-03 m
4NodeANDES	shape2	perpendicular to plane (bending)	7.656E-03 m	7.734E-03 m	7.765E-03 m
4NodeANDES	shape2	inplane force	3.875E-03 m	5.855E-03 m	7.074E-03 m
4NodeANDES	shape3	perpendicular to plane (bending)	6.637E-03 m	7.139E-03 m	7.521E-03 m
4NodeANDES	shape3	inplane force	1.555E-03 m	2.424E-03 m	3.896E-03 m
Theoretical solution			7.9488E-03 m	7.9488E-03 m	7.9488E-03 m

The error were listed in Table (63).

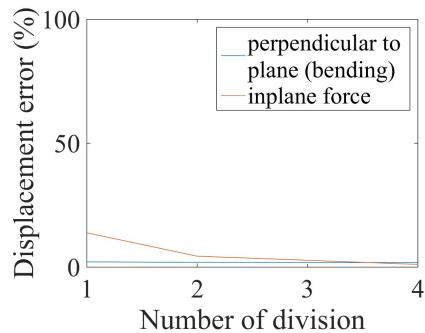
Table 63: Errors for 4NodeANDES cantilever beams of irregular shapes with more elements

Element Type	Shape	Force direction	Number of division		
			1	2	4
4NodeANDES	shape1	perpendicular to plane (bending)	2.51%	2.28%	2.20%
4NodeANDES	shape1	inplane force	14.18%	4.78%	1.48%
4NodeANDES	shape2	perpendicular to plane (bending)	3.68%	2.71%	2.31%
4NodeANDES	shape2	inplane force	51.25%	26.34%	11.00%
4NodeANDES	shape3	perpendicular to plane (bending)	16.51%	10.19%	5.38%
4NodeANDES	shape3	inplane force	80.44%	69.51%	50.98%

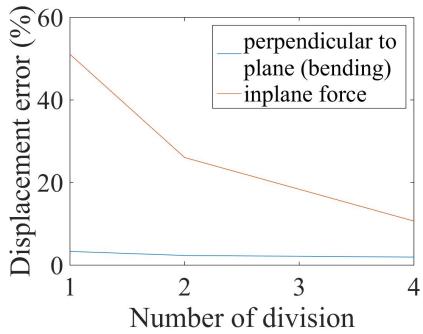
The errors were shown in Figure (183), (184) and (185).



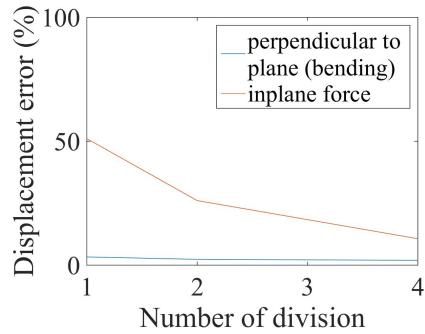
(a) Error scale 0% - 15%



(b) Error scale 0% - 100%

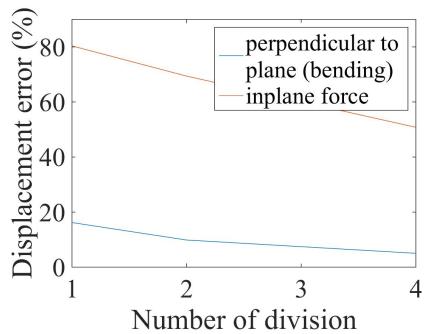
Figure 183: 4NodeANDES cantilever beam for irregular **Shape 1**
Displacement error versus Number of division

(a) Error scale 0% - 60%

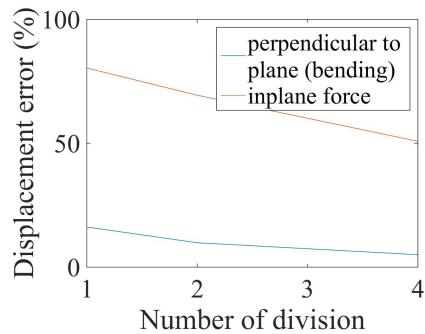


(b) Error scale 0% - 100%

Figure 184: 4NodeANDES cantilever beam for irregular **Shape 2**
Displacement error versus Number of division



(a) Error scale 0% - 80%



(b) Error scale 0% - 100%

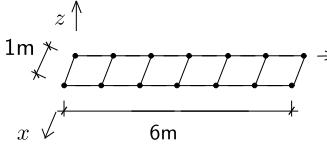
Figure 185: 4NodeANDES cantilever beam for irregular **Shape 3**

Displacement error versus Number of division

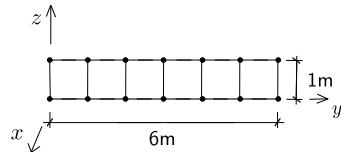
The ESSI model fei files for the table above are here

In this section, the beam was cut into smaller elements with element side length 0.5m and 0.25m respectively. And the element side length of the original models is 1.0m. The numerical models were shown in Figure (186), (187) and (188).

Number of division 1:



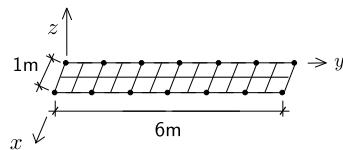
(a) Horizontal plane



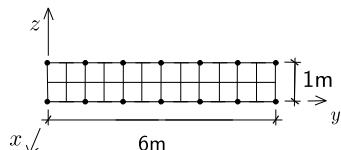
(b) Vertical plane

Figure 186: 4NodeANDES clamped beam with element side length 1.0m

Number of division 2:



(a) Horizontal plane



(b) Vertical plane

Figure 187: 4NodeANDES clamped beam with element side length 0.5m

Number of division 4:

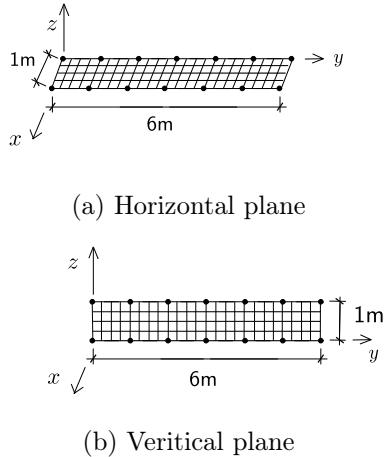


Figure 188: 4NodeANDES clamped beam with element side length 0.25m

The ESSI results for the force ***perpendicular to plane (bending)*** were listed in Table (64). The theoretical solution is 1.60E-5 m.

Table 64: Results for 4NodeANDES clamped beams under the force perpendicular to plane (bending)

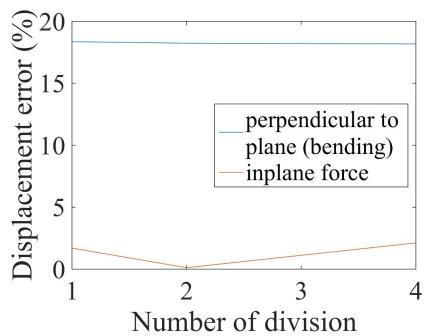
Element Type	Element side length		
	1 m	0.5 m	0.25 m
4NodeANDES	1.347E-05 m	1.35E-05 m	1.35E-05 m
Error	18.36%	18.24%	18.18%

The ESSI results for the ***inplane force*** were listed in Table (65). The theoretical solution is 1.60E-5 m.

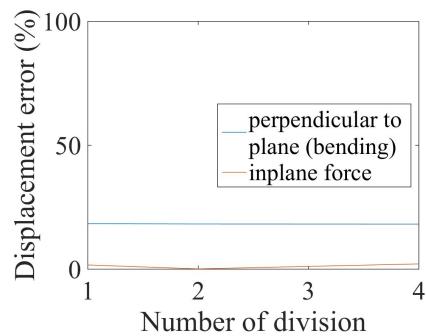
Table 65: Results for 4NodeANDES clamped beams under the inplane force

Element Type	Element side length		
	1 m	0.5 m	0.25 m
4NodeANDES	1.62E-05 m	1.65E-05 m	1.69E-05 m
Error	1.70%	0.12%	2.12%

The errors were plotted in Figure (189).



(a) Error scale 0% - 20%



(b) Error scale 0% - 100%

Figure 189: 4NodeANDES clamped beam for different element number
Displacement error versus Number of division

The ESSI model fei files for the table above are here

20.4 Verification of 4NodeANDES square plate with four edges clamped

Problem description: Length=20m, Width=20m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.3$.

The four edges are clamped.

The load is the uniform normal pressure on the whole plate.

The plate flexural rigidity is

$$D = \frac{Eh^3}{12(1 - \nu^2)} = \frac{10^8 N/m^2 \times 1^3 m^3}{12 \times (1 - 0.3^2)} = 9.1575 \times 10^6 N \cdot m \quad (42)$$

The theoretical solution is

$$d = \alpha_c \frac{qa^4}{D} = 0.00406 \times \frac{100 N/m^2 \times 20^4 m^4}{9.1575 \times 10^6 N \cdot m} = 2.2015 \times 10^{-3} m \quad (43)$$

where α_c is a coefficient, which depends on the ratio of plate length to width. In this problem, the coefficient¹⁴ α_c is 0.00406.

The 4NodeANDES were shown in Figure (190) - (195).

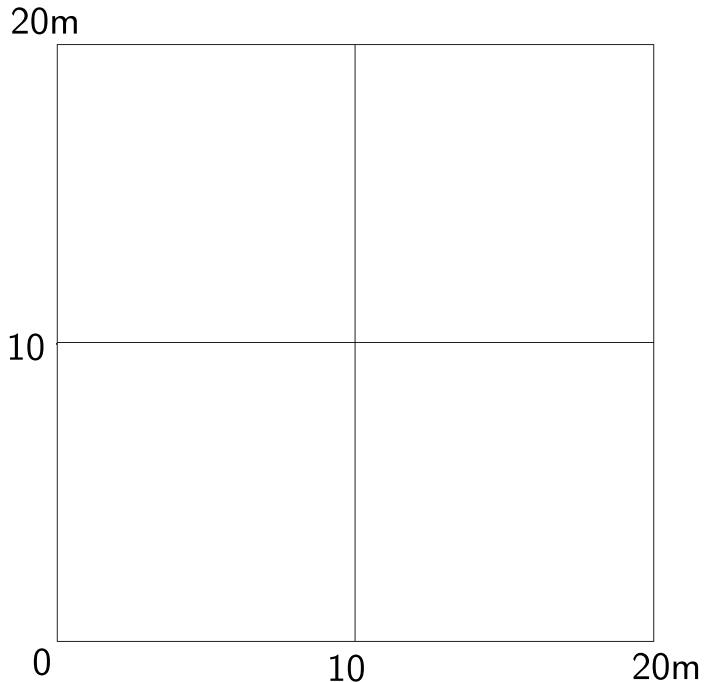


Figure 190: 4NodeANDES edge clamped square plate with element side length 10m

¹⁴Stephen Timoshenko, Theory of plates and shells (2nd edition). MrGRAW-Hill Inc, page120, 1959.

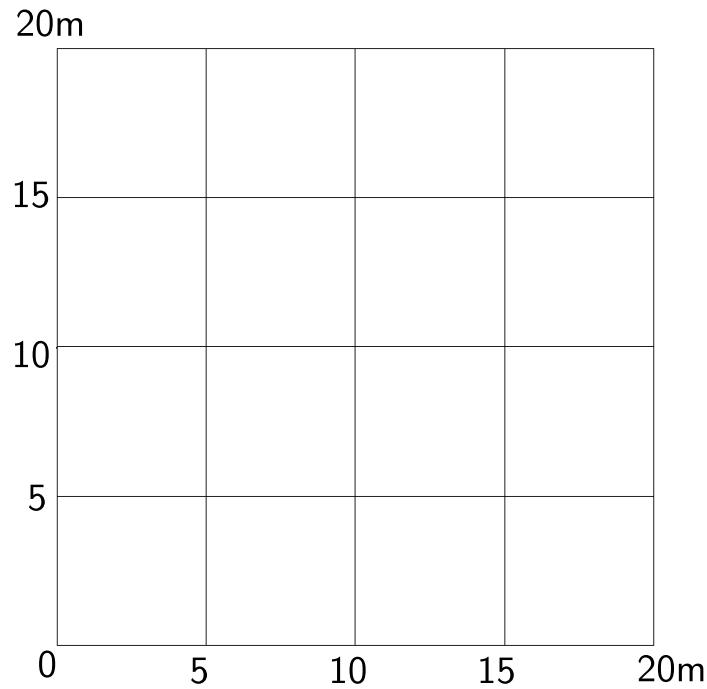


Figure 191: 4NodeANDES edge clamped square plate with element side length 5m

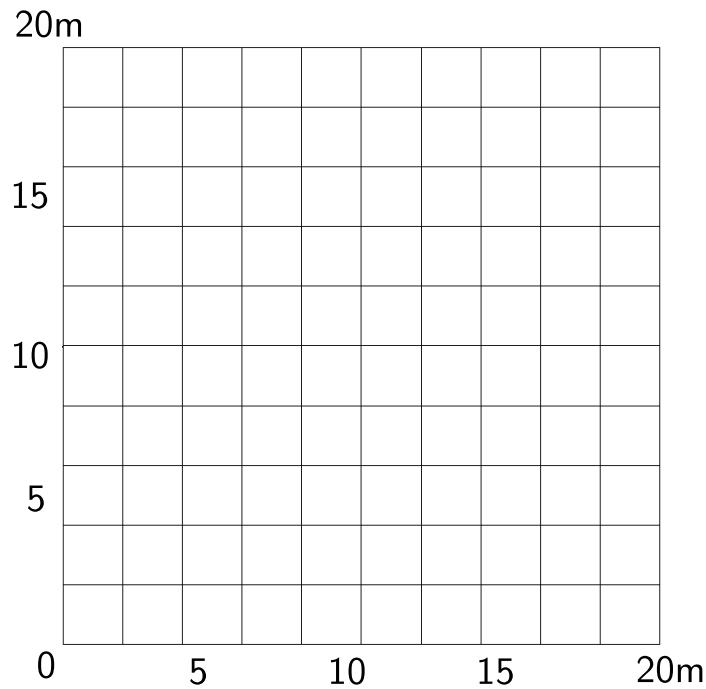


Figure 192: 4NodeANDES edge clamped square plate with element side length 2m

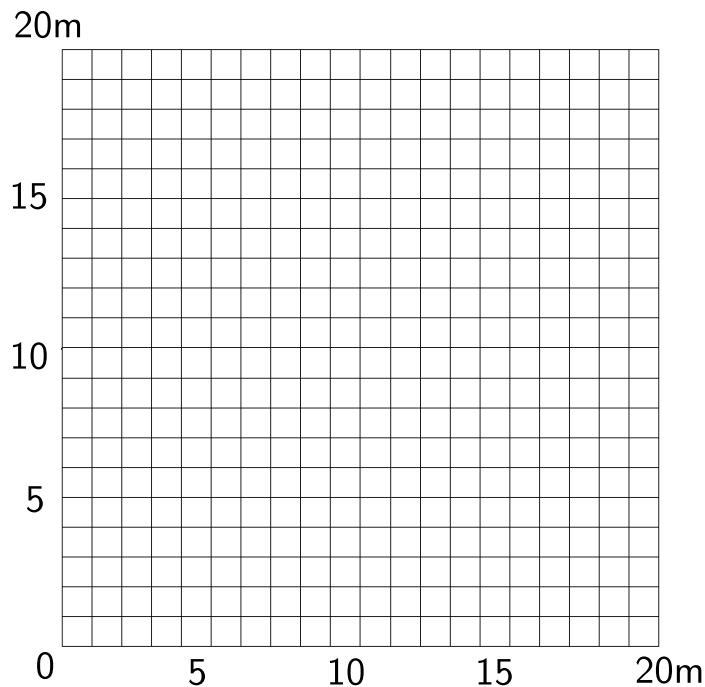


Figure 193: 4NodeANDES edge clamped square plate with element side length 1m

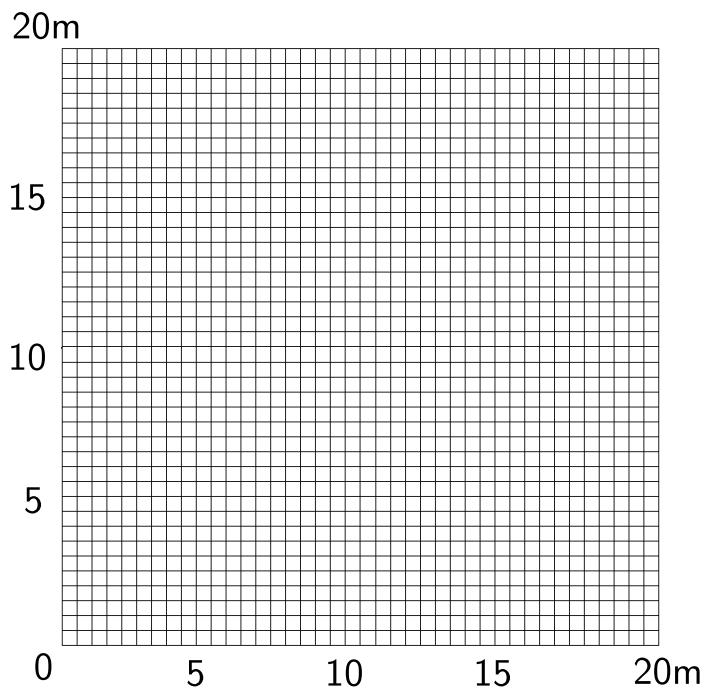


Figure 194: 4NodeANDES edge clamped square plate with element side length 0.5m

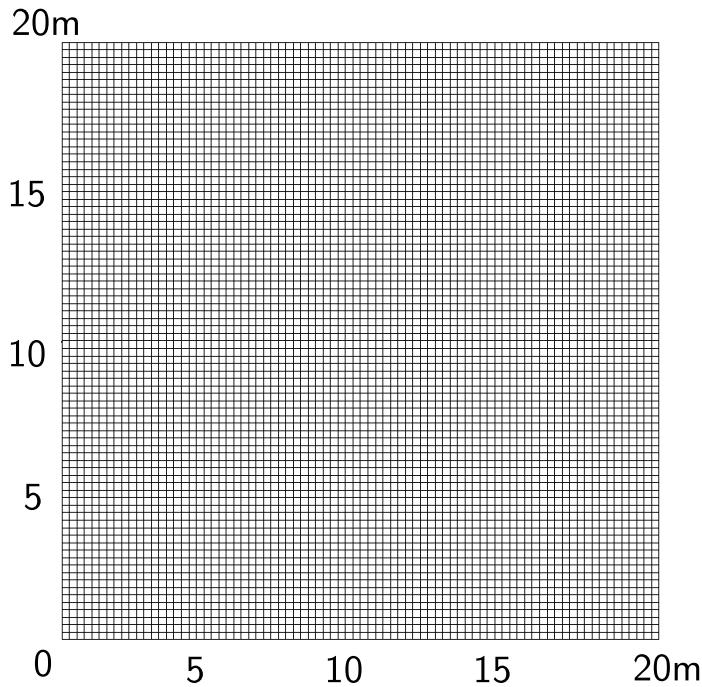


Figure 195: 4NodeANDES edge clamped square plate with element side length 0.25m

The results were listed in Table (66).

Table 66: Results for 4NodeANDES square plate with four edges clamped

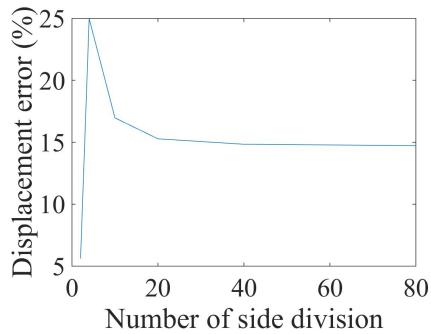
Element type	4NodeANDES	Theoretical displacement
Element side length	Height:1.00m	
10m	2.33E-003 m	2.20E-03 m
5m	2.75E-003 m	2.20E-03 m
2m	2.58E-003 m	2.20E-03 m
1m	2.54E-003 m	2.20E-03 m
0.5m	2.53E-003 m	2.20E-03 m
0.25m	2.53E-003 m	2.20E-03 m

The errors were listed in Table (67).

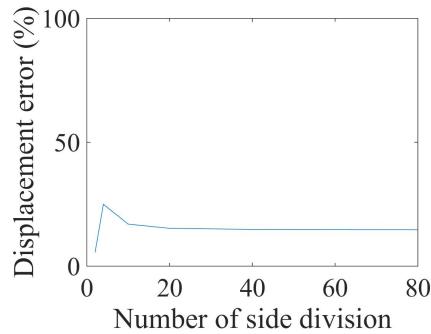
Table 67: Errors for 4NodeANDES square plate with four edges clamped

Element type	4NodeANDES
Element side length	Height:1.00m
10m	5.65%
5m	24.98%
2m	16.97%
1m	15.28%
0.5m	14.84%
0.25m	14.73%

The errors were plotted in Figure (196).



(a) Error scale 0% - 25%



(b) Error scale 0% - 100%

Figure 196: 4NodeANDES square plate with edge clamped
Displacement error versus Number of side division

The ESSI model fei files for the table above are here

20.5 Verification of 4NodeANDES square plate with four edges simply supported

Problem description: Length=20m, Width=20m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.3$.

The four edges are simply supported.

The load is the uniform normal pressure on the whole plate.

The plate flexural rigidity is

$$D = \frac{Eh^3}{12(1 - \nu^2)} = \frac{10^8 N/m^2 \times 1^3 m^3}{12 \times (1 - 0.3^2)} = 9.1575 \times 10^6 N \cdot m \quad (44)$$

The theoretical solution is

$$d = \alpha_s \frac{qa^4}{D} = 0.00126 \times \frac{100 N/m^2 \times 20^4 m^4}{9.1575 \times 10^6 N \cdot m} = 7.0936 \times 10^{-3} m \quad (45)$$

where α_s is a coefficient, which depends on the ratio of plate length to width. In this problem, the coefficient¹⁵ α_s is 0.00126.

The 4NodeANDES were shown in Figure (197) - (202).

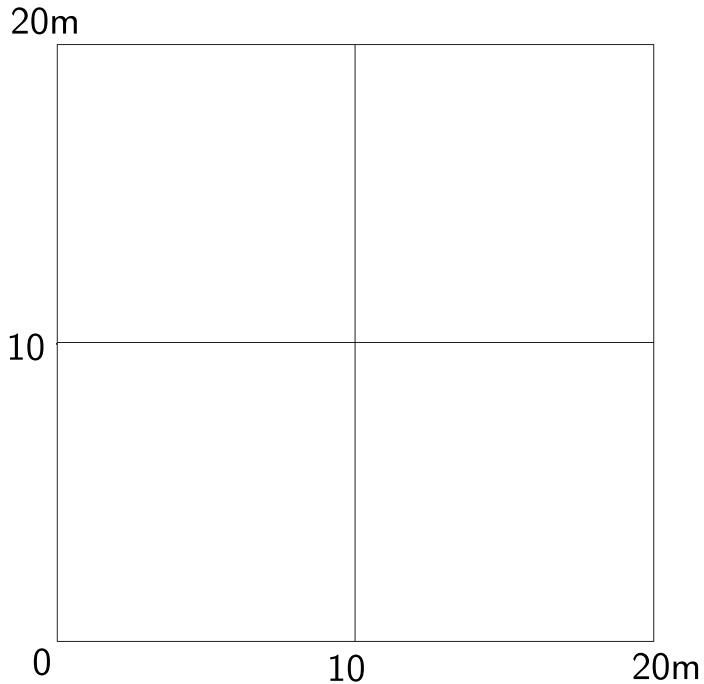


Figure 197: 4NodeANDES edge simply supported square plate with element side length 10m

¹⁵Stephen Timoshenko, Theory of plates and shells (2nd edition). MrGRAW-Hill Inc, page202, 1959.

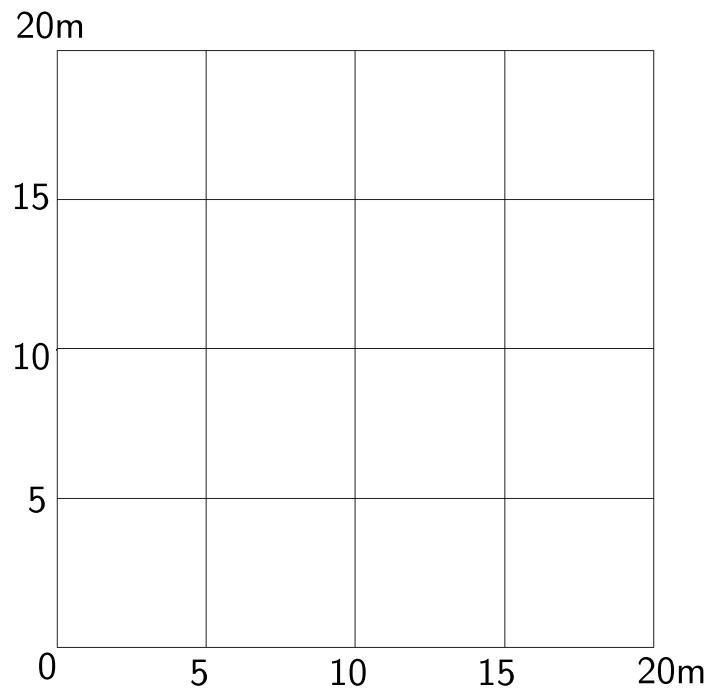


Figure 198: 4NodeANDES edge simply supported square plate with element side length 5m

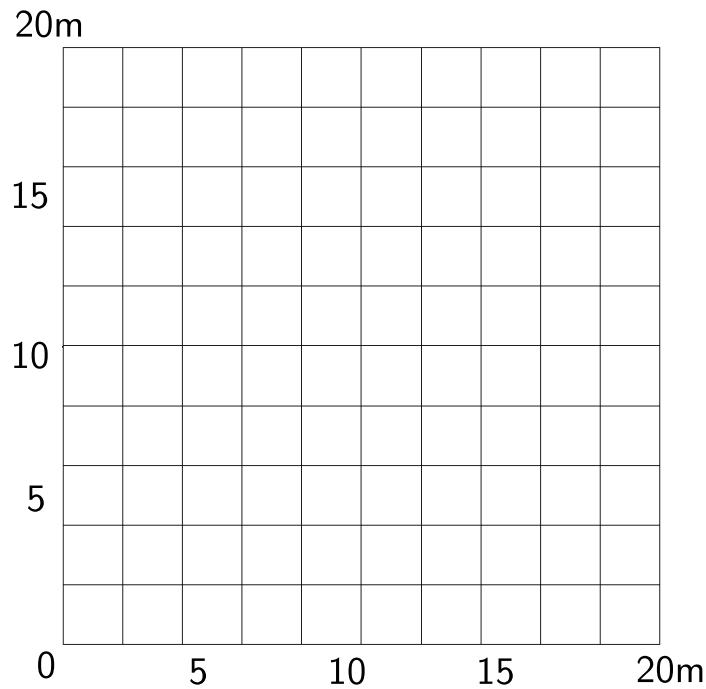


Figure 199: 4NodeANDES edge simply supported square plate with element side length 2m

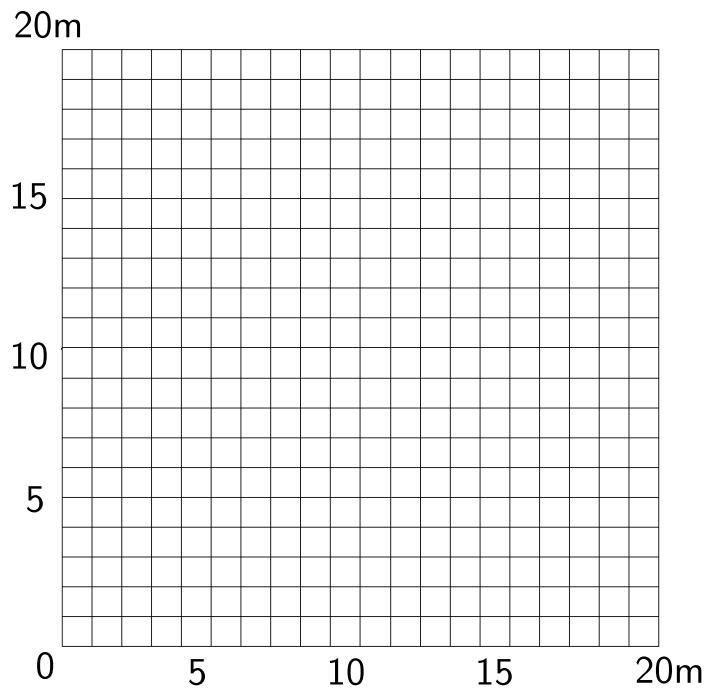


Figure 200: 4NodeANDES edge simply supported square plate with element side length 1m

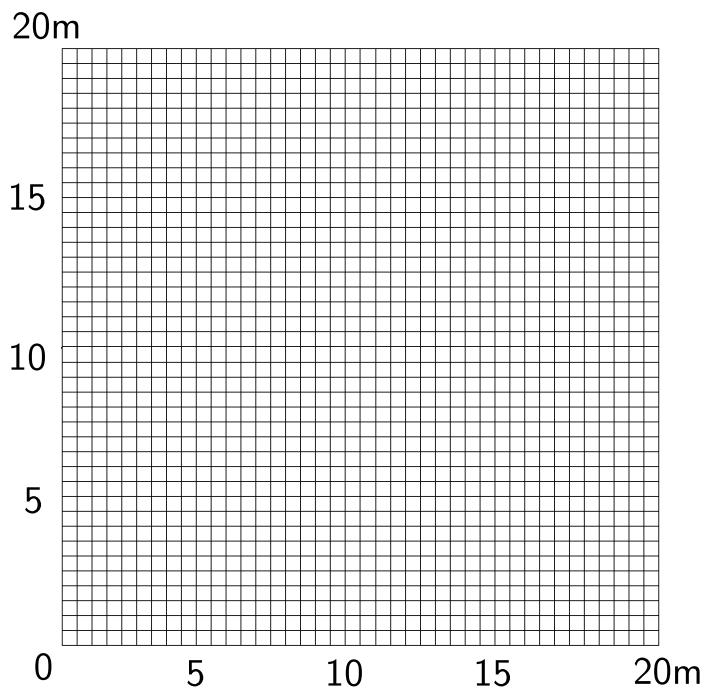


Figure 201: 4NodeANDES edge simply supported square plate with element side length 0.5m

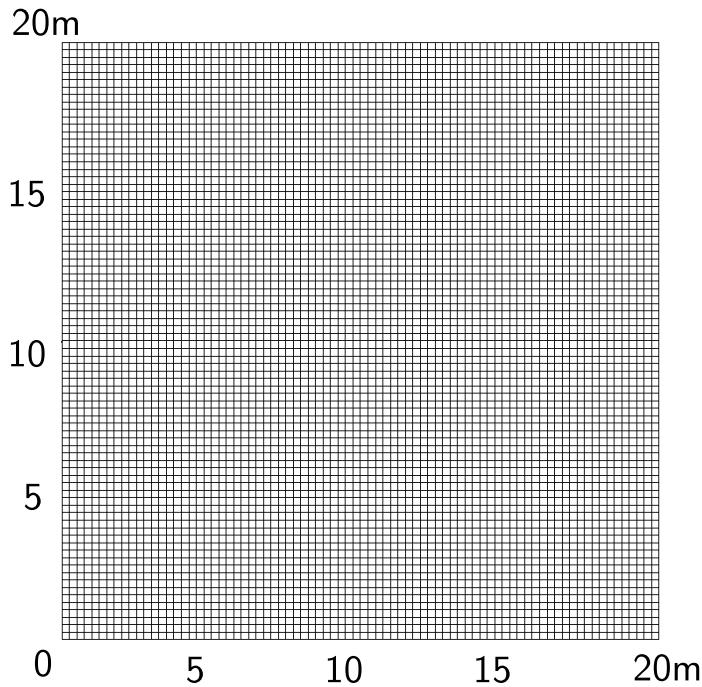


Figure 202: 4NodeANDES edge simply supported square plate with element side length 0.25m

The results were listed in Table (68).

Table 68: Results for 4NodeANDES square plate with four edges simply supported

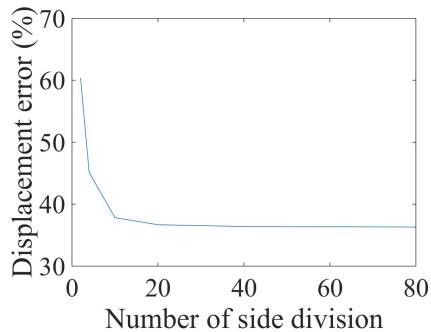
Element type	4NodeANDES	Theoretical displacement
Element side length	Height:1.00m	
10m	1.14E-002 m	7.09E-03 m
5m	1.03E-002 m	7.09E-03 m
2m	9.78E-003 m	7.09E-03 m
1m	9.70E-003 m	7.09E-03 m
0.5m	9.68E-003 m	7.09E-03 m
0.25m	9.67E-003 m	7.09E-03 m

The errors were listed in Table (69).

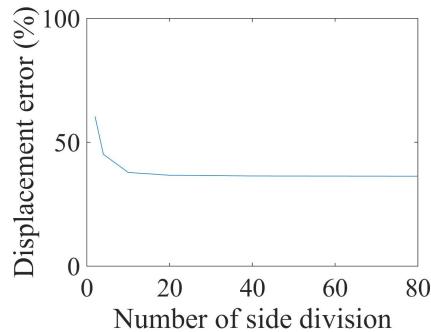
Table 69: Errors for 4NodeANDES square plate with four edges simply supported

Element type	4NodeANDES
Element side length	Height:1.00m
10m	60.34%
5m	45.14%
2m	37.83%
1m	36.69%
0.5m	36.40%
0.25m	36.32%

The errors were plotted in Figure (203).



(a) Error scale 0% - 70%



(b) Error scale 0% - 100%

Figure 203: 4NodeANDES square plate with edge simply supported
Displacement error versus Number of side division

The ESSI model fei files for the table above are here

20.6 Verification of 4NodeANDES circular plate with all edges clamped

Problem description: Diameter=20m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.3$.

The four edges are clamped.

The load is the uniform normal pressure on the whole plate.

The plate flexural rigidity is

$$D = \frac{Eh^3}{12(1-\nu^2)} = \frac{10^8 N/m^2 \times 1^3 m^3}{12 \times (1 - 0.3^2)} = 9.1575 \times 10^6 N \cdot m \quad (46)$$

The theoretical solution¹⁶ is

$$d = \frac{qa^4}{64D} = \frac{100N/m^2 \times 10^4 m^4}{64 \times 9.1575 \times 10^6 N \cdot m} = 1.7106 \times 10^{-3} m \quad (47)$$

The 4NodeANDES were shown in Figure (204) - (209).

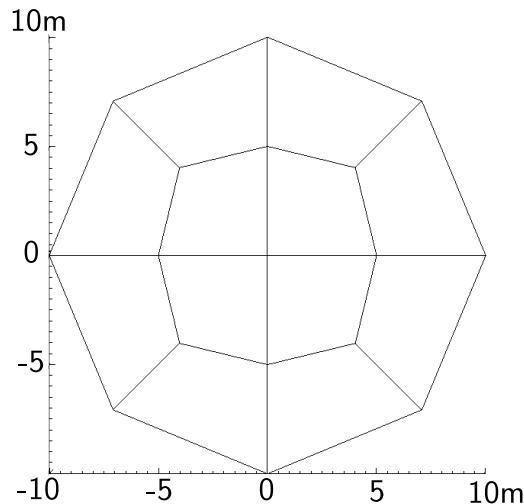


Figure 204: 4NodeANDES edge clamped circular plate with element side length 10m

¹⁶Stephen Timoshenko, Theory of plates and shells (2nd edition). MrGRAW-Hill Inc, page55, 1959.

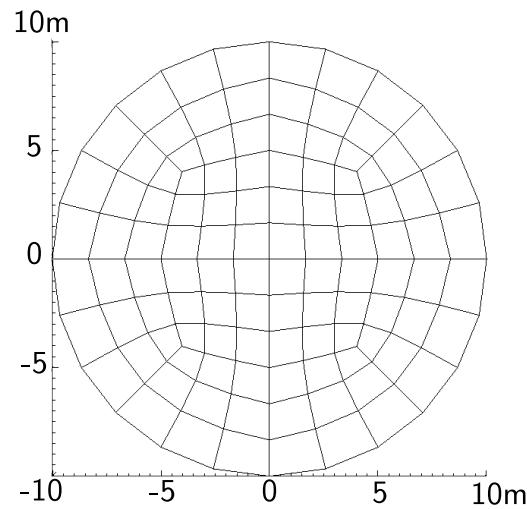


Figure 205: 4NodeANDES edge clamped circular plate with element side length 5m

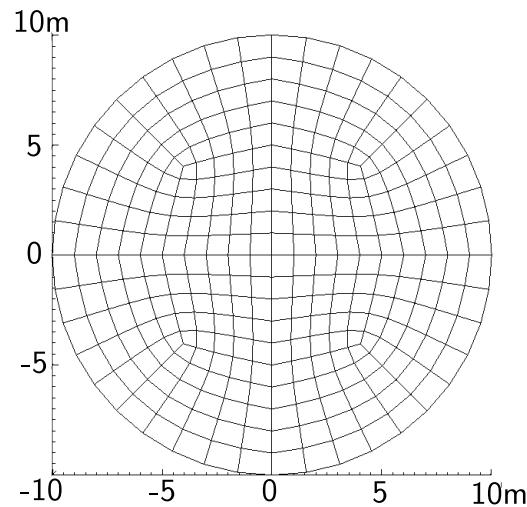


Figure 206: 4NodeANDES edge clamped circular plate with element side length 2m

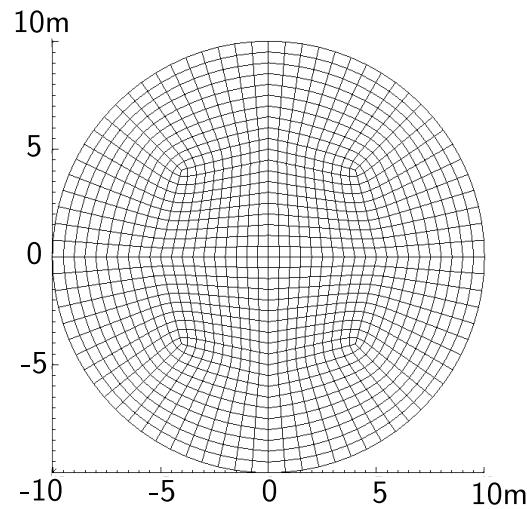


Figure 207: 4NodeANDES edge clamped circular plate with element side length 1m

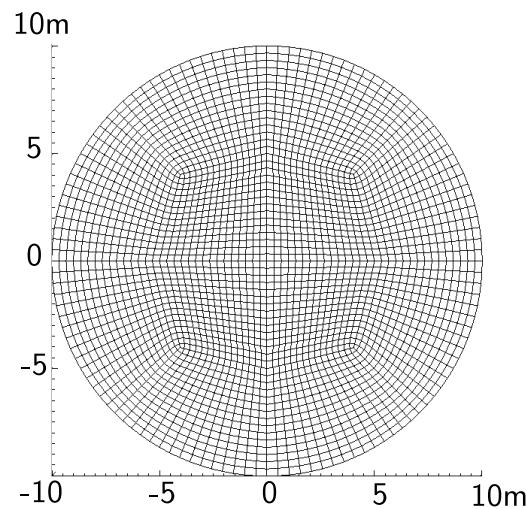


Figure 208: 4NodeANDES edge clamped circular plate with element side length 0.5m

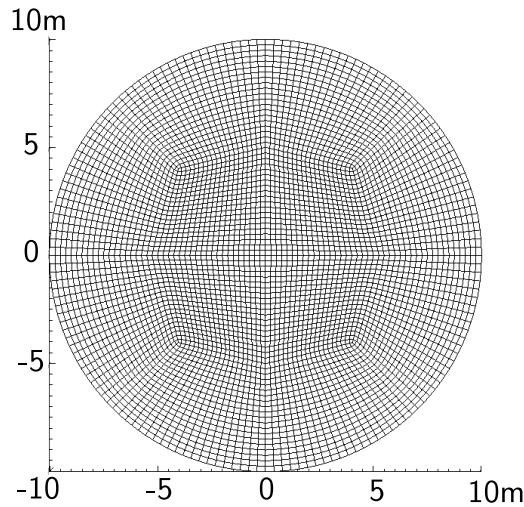


Figure 209: 4NodeANDES edge clamped circular plate with element side length 0.25m

The results were listed in Table (70).

Table 70: Results for 4NodeANDES circular plate with four edges clamped

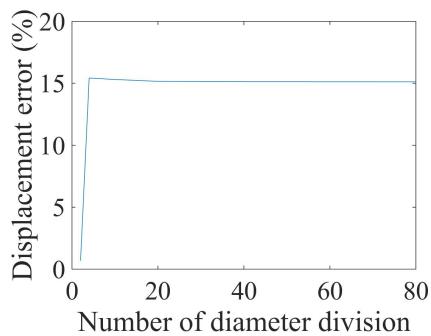
Element type	4NodeANDES	Theoretical displacement
Element side length	Height:1.00m	
10m	1.69E-003 m	1.706E-03 m
5m	1.97E-003 m	1.706E-03 m
2m	1.97E-003 m	1.706E-03 m
1m	1.96E-003 m	1.706E-03 m
0.5m	1.96E-003 m	1.706E-03 m
0.25m	1.96E-003 m	1.706E-03 m

The errors were listed in Table (71).

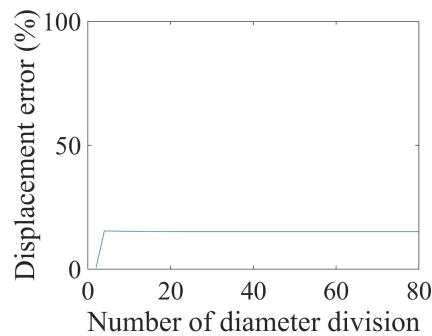
Table 71: Errors for 4NodeANDES circular plate with four edges clamped

Element type	4NodeANDES
Element side length	Height:1.00m
10m	0.71%
5m	15.43%
2m	15.31%
1m	15.16%
0.5m	15.13%
0.25m	15.12%

The errors were shown in Figure (210).



(a) Error scale 0% - 20%



(b) Error scale 0% - 100%

Figure 210: 4NodeANDES circular plate with edge clamped
Displacement error versus Number of side division

The ESSI model fei files for the table above are here

20.7 Verification of 4NodeANDES circular plate with all edges simply supported

Problem description: Diameter=20m, Height=1m, Force=100N, E=1E8Pa, $\nu = 0.3$.

The four edges are simply supported.

The load is the uniform normal pressure on the whole plate.

The plate flexural rigidity is

$$D = \frac{Eh^3}{12(1-\nu^2)} = \frac{10^8 N/m^2 \times 1^3 m^3}{12 \times (1 - 0.3^2)} = 9.1575 \times 10^6 N \cdot m \quad (48)$$

The theoretical solution¹⁷ is

$$d = \frac{(5 + \nu)qa^4}{64(1 + \nu)D} = \frac{(5 + 0.3) \times 100N/m^2 \times 10^4 m^4}{64 \times (1 + 0.3) \times 9.1575 \times 10^6 N \cdot m} = 6.956 \times 10^{-3} m \quad (49)$$

The 4NodeANDES were shown in Figure (211) - (216).

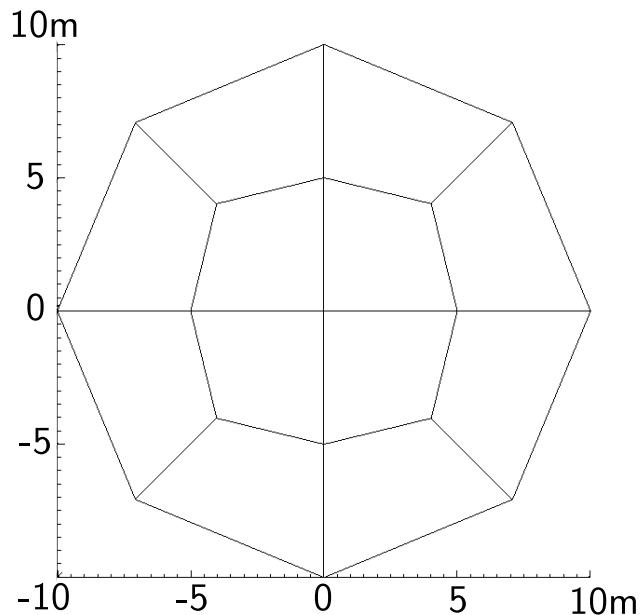


Figure 211: 4NodeANDES edge simply supported circular plate with element side length 10m

¹⁷Stephen Timoshenko, Theory of plates and shells (2nd edition). MrGRAW-Hill Inc, page55, 1959.

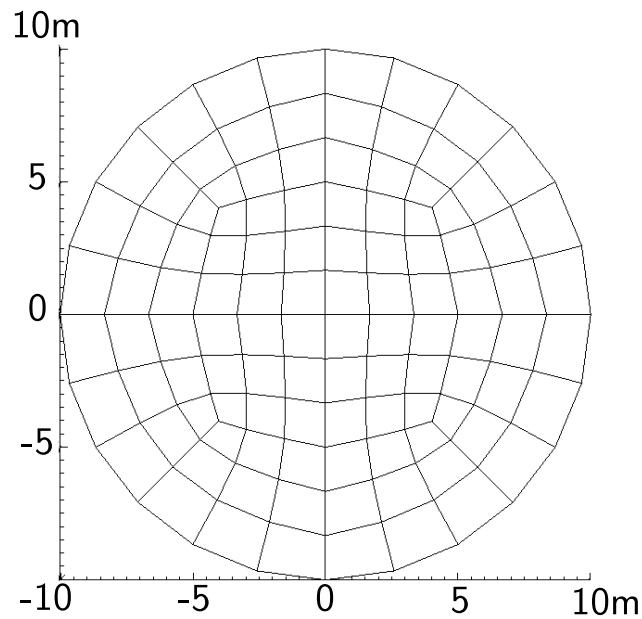


Figure 212: 4NodeANDES edge simply supported circular plate with element side length 5m

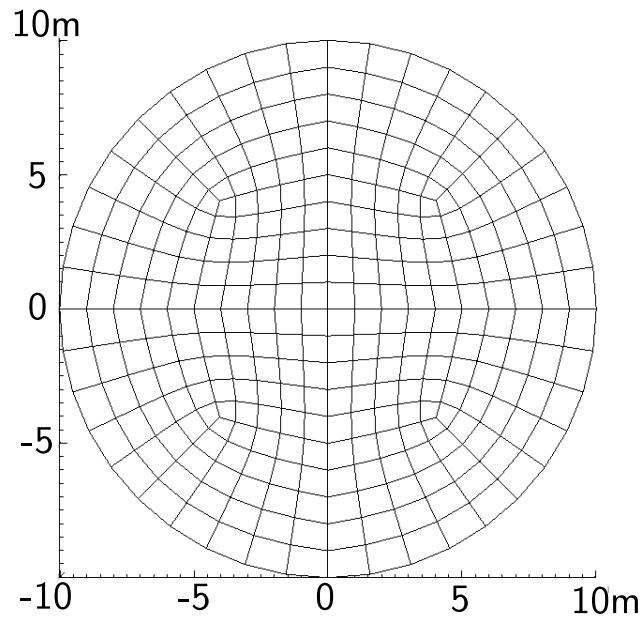


Figure 213: 4NodeANDES edge simply supported circular plate with element side length 2m

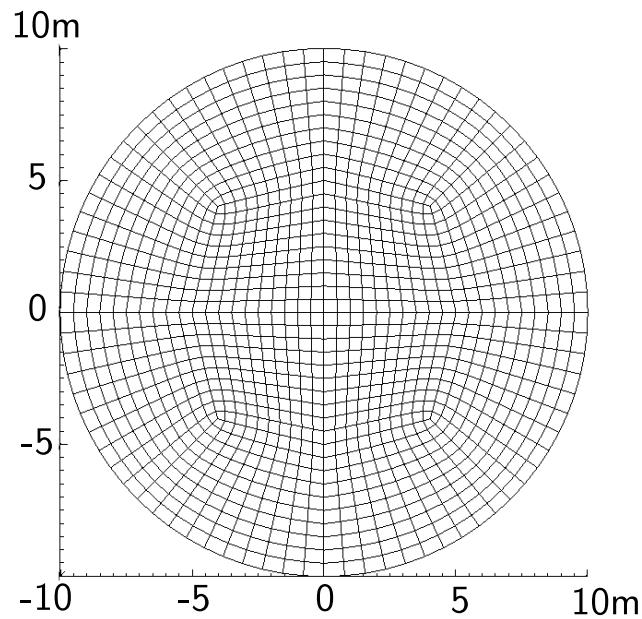


Figure 214: 4NodeANDES edge simply supported circular plate with element side length 1m

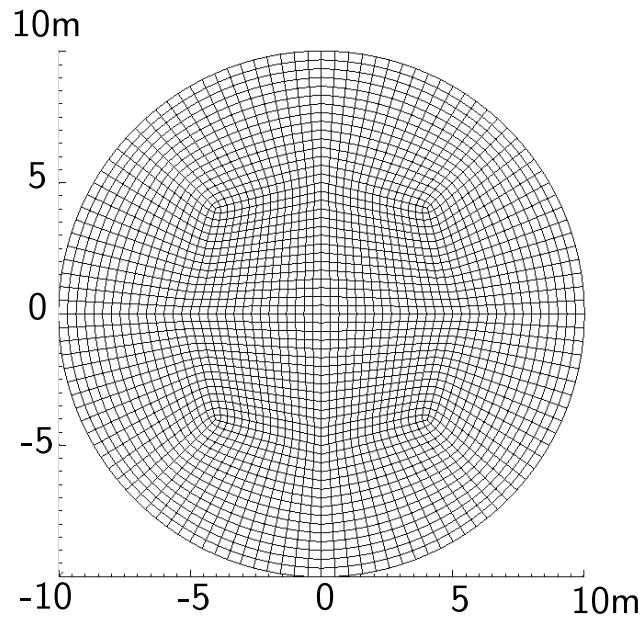


Figure 215: 4NodeANDES edge simply supported circular plate with element side length 0.5m

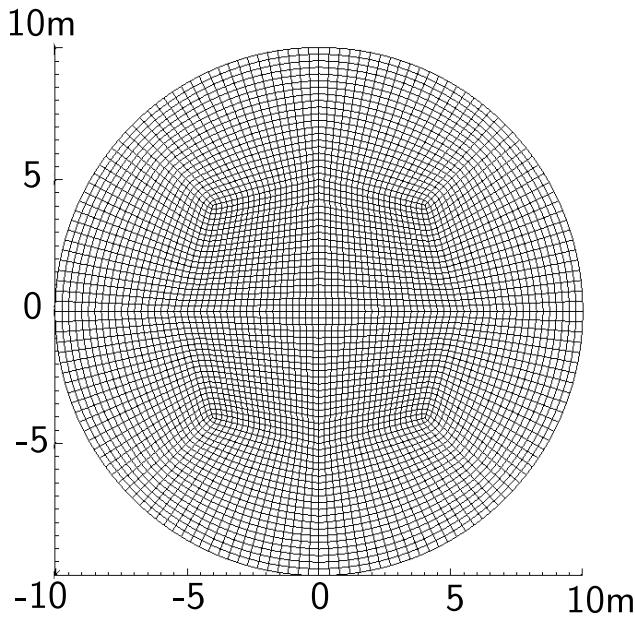


Figure 216: 4NodeANDES edge simply supported circular plate with element side length 0.25m

The results were listed in Table (72).

Table 72: Results for 4NodeANDES cicular plate with four edges simply supported

Element type	4NodeANDES	Theoretical displacement
Element side length	Height:1.00m	
10m	7.50E-003 m	6.956E-03 m
5m	7.29E-003 m	6.956E-03 m
2m	7.25E-003 m	6.956E-03 m
1m	7.23E-003 m	6.956E-03 m
0.5m	7.22E-003 m	6.956E-03 m
0.25m	7.22E-003 m	6.956E-03 m

The errors were listed in Table (73).

Table 73: Errors for 4NodeANDES cicular plate with four edges simply supported

Element type	4NodeANDES
Element side length	Height:1.00m
10m	7.75%
5m	4.73%
2m	4.15%
1m	3.89%
0.5m	3.84%
0.25m	3.82%

The errors were plotted in Figure (217).

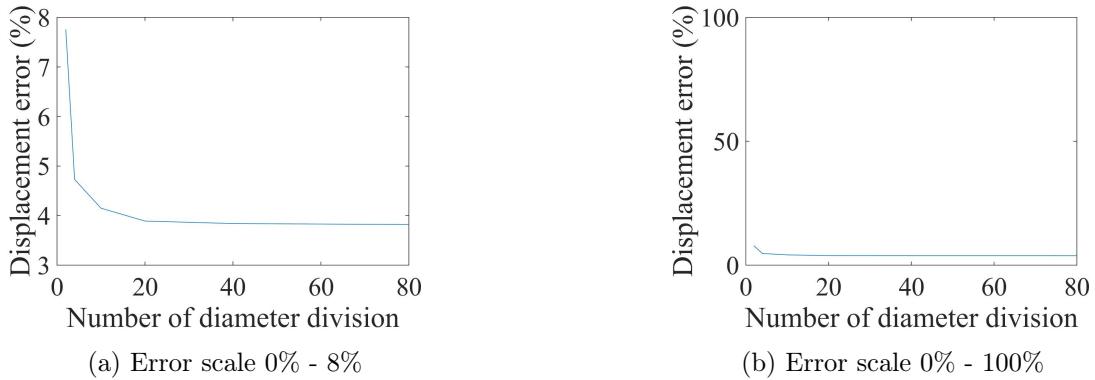


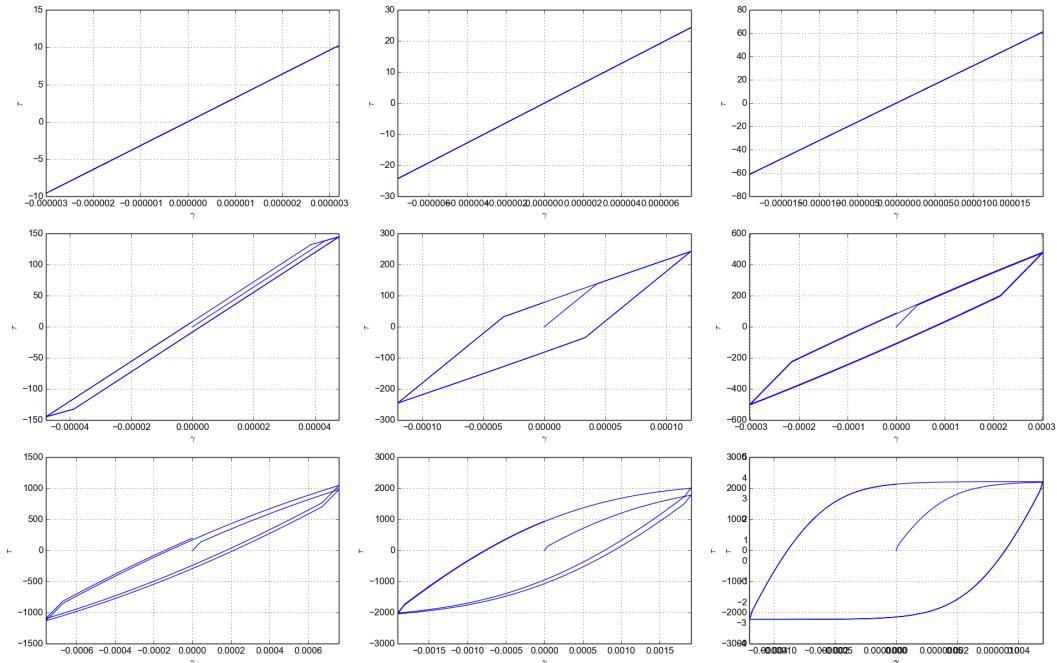
Figure 217: 4NodeANDES circular plate with edge simply supported
Displacement error versus Number of side division

The ESSI model fei files for the table above are here

21 G/Gmax plot by Drucker-Prager Armstrong Frederick

21.1 Plot stress-strain

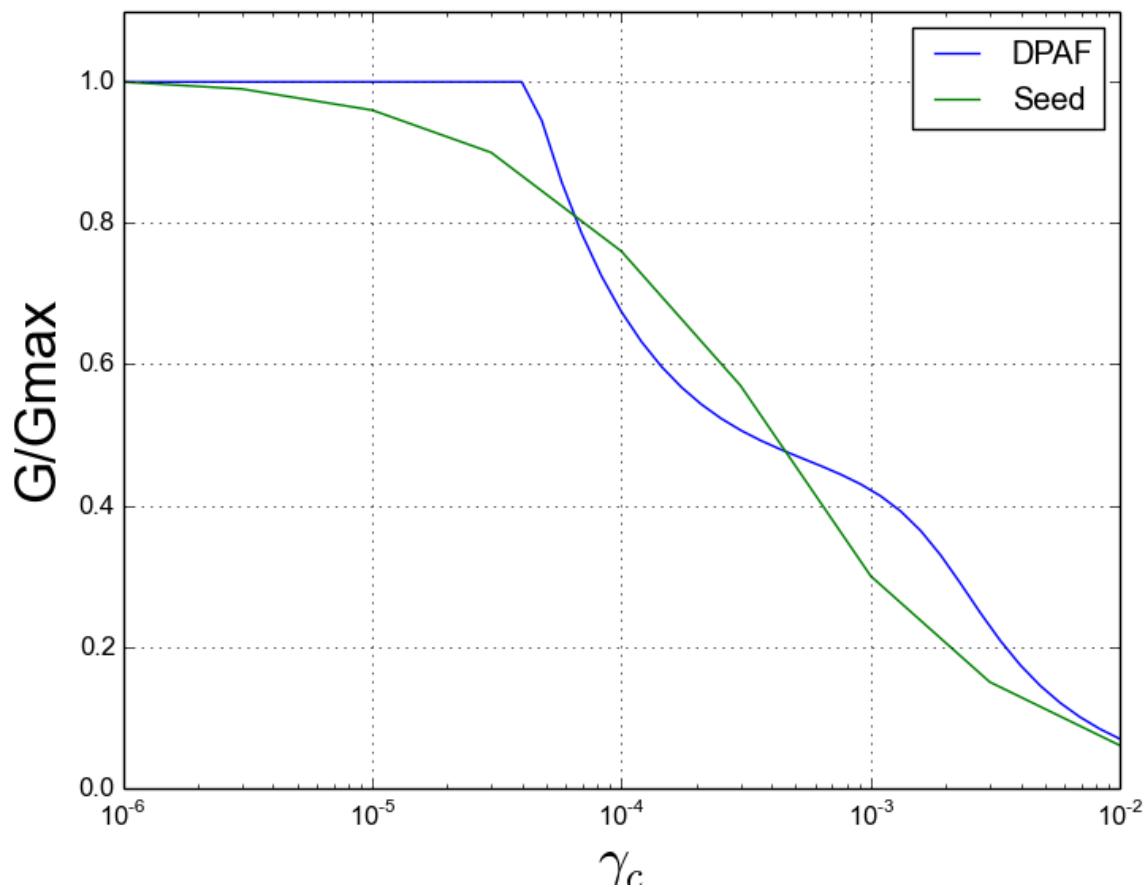
Figure 218: The stress-strain diagram for different γ_c



When γ_c are very small, the materials are in the elastic range.

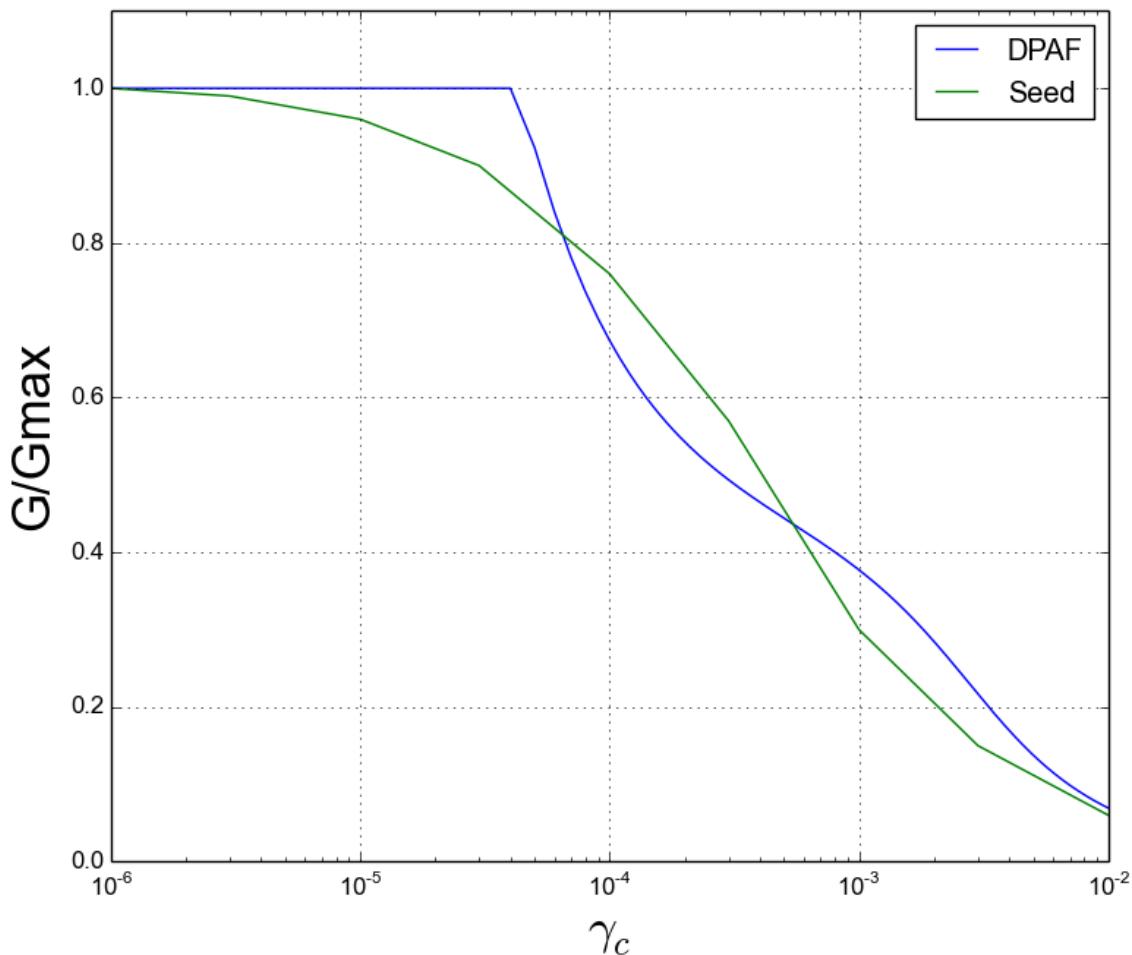
21.2 Plot G/Gmax

Figure 219: The G/Gmax diagram with multiple loops



The $x - axis$ labels are the unitless γ_c , **not** in percent.
So the max shear strain is 1 percent here.

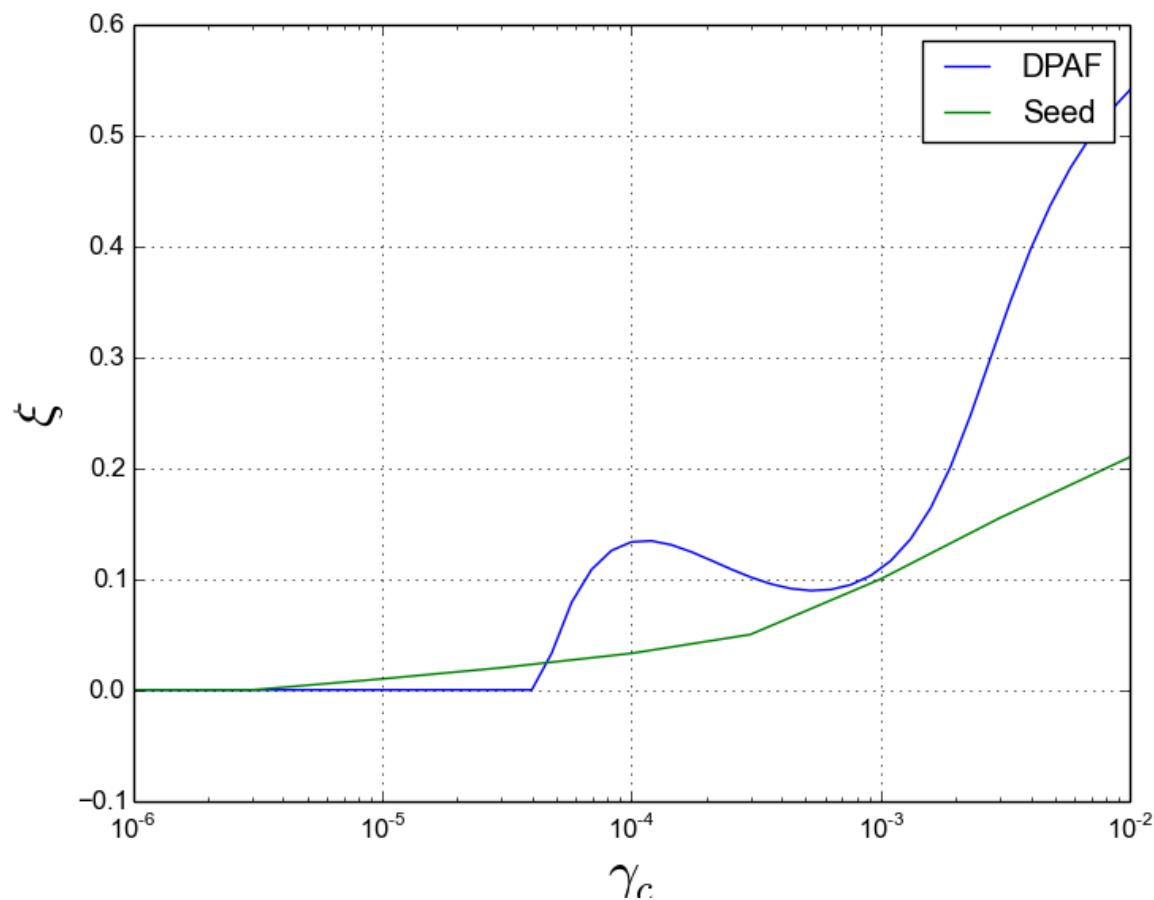
Figure 220: The G/Gmax diagram in one loading



The G/Gmax curve above is plotted by a different method. Only one loading is used to plot G/Gmax. The different plot functions are attached at the end. The curves between different plotting methods are almost the same.

21.3 Plot damping ratio

Figure 221: The damping ratio



The parameters I am using

```
1 // Yuan is testing different Armstrong Frederick Drucker Prager (AFDP) parameters:  
2 // For Real ESSI, Units are processed by other functions.  
3 // Here, all units should be the SI units.  
4 // initial_von_mises_radius is actually the variable 'k' in the function of yield surface  
5 double initial_von_mises_radius =0.0008;  
6 double kinematic_hardening_AF_ha =12;  
7 double kinematic_hardening_AF_cr =1000;  
8 double isotropic_hardening_rate =0.0;  
9 double elastic_modulus      =4e6;    //4MPa  
10 double poisson_ratio       =0.25;  
11 double density             =0.0;  
12 double initial_confining_stress =3e5; //300kPa
```

In the Appendix (21.4), you would see the whole cpp code I am using to define the materials.

21.4 Appendix (Code)

21.4.1 Framework: A cpp main function to test the materials directly

```

1 #include "ClassicElastoplasticMaterial.h"
2 #include "ConstitutiveModels/DruckerPragerArmstrongFrederick.h"
3 #include <iostream>
4 #include <fstream>
5 #include <vector>
6 #include <math.h>
7
8
9 using namespace std;
10
11 void print_this_step(const int &printStep, NDMaterialLT* test_material, ofstream &
12   out_strain,ofstream & out_stress){
13   // Terminal: print the input strain:
14   cout<<"-----step "<<printStep<<"-----start-----"<<endl;
15   auto input_strain = test_material -> getStrainTensor();
16   cout<<"The current strain: "<<endl;
17   cout<<input_strain<<endl;
18
19   // Terminal: Output the calculated stress
20   auto stress_result = test_material -> getStressTensor();
21   cout<<"The output stress: "<<endl;
22   cout<<stress_result<<endl;
23   cout<<"-----step end-----"<<endl;
24
25   // output to file as well
26   out_strain<<input_strain(0,2)<<endl;
27   out_stress<<stress_result(0,2)<<endl;
28 }
29
30 int main(){
31   // The corresponding constructor for reference:
32   //First constructor, creates a material at its "ground state" from its parameters.
33   // DruckerPragerArmstrongFrederick(int tag_in,
34   // double k0_in,
35   // double ha_alpha,
36   // double cr_alpha,
37   // double H_k,
38   // double E,
39   // double nu,
40   // double rho_,
41   // double p0) ;
42
43   // Define and input the material properties.
44   int material_tag          =1;
45
46   // Yuan is testing different Armstrong Frederick Drucker Prager (AFDP) parameters:

```

```
46 // For Real ESSI, units are processed by other functions.  
47 // Here, all units should be the SI units.  
48 // initial_von_mises_radius is actually the variable 'k' in the function of yield surface  
49 double initial_von_mises_radius =0.0008;  
50 double kinematic_hardening_AF_ha =12;  
51 double kinematic_hardening_AF_cr =1000;  
52 double isotropic_hardening_rate =0.0;  
53 double elastic_modulus =4e6; //4MPa  
54 double poisson_ratio =0.25;  
55 double density =0.0;  
56 double initial_confining_stress =3e5; //300kPa  
57  
58 // Set the integration rule and tolerance to the material:  
59 int method = (int) NDMaterialLT_Constitutive_Integration_Method::Euler_One_Step;  
60 double f_relative_tol=0.001;  
61 double stress_relative_tol=0.001;  
62 double n_max_iterations=10;  
63  
64 NDMaterialLT::set_constitutive_integration_method(  
65     method,  
66     f_relative_tol,  
67     stress_relative_tol,  
68     n_max_iterations);  
69  
70  
71 // generate the uniformly distributed values on the logarithmic axis.  
72 double strain_start=0.0001/100.0;  
73 double strain_end =1.0/100.0;  
74 double Lstart=log10(strain_start);  
75 double Lend=log10(strain_end);  
76 int num_gammac=51;  
77 std::vector<double> gammac(num_gammac,0.0);  
78  
79 for (int i = 0; i < num_gammac; ++i)  
{  
80     gammac[i] = pow(10.0, Lstart+(Lend-Lstart)*i/(num_gammac-1) );  
81 }  
82  
83  
84 double steplength_loading=strain_start/10.0;  
85  
86 // the Gammac for the step  
87 double theGammac=0.0;  
88 for (int i_gamma = 0; i_gamma < num_gammac; ++i_gamma)  
{  
89     // gammac is the maximum shear strain for this cyclic loading  
90     // theGammac=steplength_gammac*(i_gamma+1);  
91     theGammac=gammac[i_gamma];  
92  
93  
94 // Declare the new materials  
95 NDMaterialLT* DPrackerAF_material = new DruckerPragerArmstrongFrederick( material_tag,  
96     initial_von_mises_radius,  
97     kinematic_hardening_AF_ha,  
98     kinematic_hardening_AF_cr,  
99     isotropic_hardening_rate,
```

```
101     elastic_modulus,
102     poisson_ratio,
103     density,
104     initial_confining_stress) ;
105
106     // Output the parameters:
107     cout<<"-----Start Testing-----"<<endl;
108     cout<<"The input Drucker Prager Armstrong Frederick material parameters:"<<endl;
109     cout<<"initial_von_mises_radius : "<<initial_von_mises_radius<<endl;
110     cout<<"kinematic_hardening_AF_ha : "<<kinematic_hardening_AF_ha<<endl;
111     cout<<"kinematic_hardening_AF_cr : "<<kinematic_hardening_AF_cr<<endl;
112     cout<<"isotropic_hardening_rate : "<<isotropic_hardening_rate<<endl;
113     cout<<"elastic_modulus      : "<<elastic_modulus<<endl;
114     cout<<"poisson_ratio       : "<<poisson_ratio<<endl;
115     cout<<"density            : "<<density<<endl;
116     cout<<"initial_confining_stress : "<<initial_confining_stress<<endl;
117     cout<<"-----"<<endl;
118
119
120     // Print out the initial state of the materials:
121     cout<<"The initial state:"<<endl;
122     auto initial_strain = DPrackerAF_material -> getStrainTensor();
123     cout<<"The initial strain state: "<<endl;
124     cout<<initial_strain<<endl;
125
126     auto initial_stress = DPrackerAF_material -> getStressTensor();
127     cout<<"The initial stress state: "<<endl;
128     cout<<initial_stress<<endl;
129     cout<<"-----"<<endl;
130
131
132     // create the output filename here:
133     ofstream outfile_strain;
134     ofstream outfile_stress;
135     string outfilename_strain= "shearStrain"+ to_string(i_gamma+1) +".txt" ;
136     string outfilename_stress= "shearStress"+ to_string(i_gamma+1) +".txt" ;
137
138     outfile_strain.open(outfilename_strain);
139     outfile_stress.open(outfilename_stress);
140
141     // write the initial state to the file
142     outfile_strain<<"0.0"<<endl;
143     outfile_stress<<"0.0"<<endl;
144
145
146     cout<<"-----Start loading-----"<<endl;
147
148     int mystep=0;
149     DTensor2 myinputStrain( 3, 3, 0.0 );
150
151     short loop=0;
152     for (int loop = 0; loop < 2; ++loop){
153         // Input positive shear strain increment each step
154         myinputStrain(0,2)=steplength_loading;
155         myinputStrain(2,0)=myinputStrain(0,2);
```

```
156     for (int i = 0; i*steplength_loading < theGammac ; ++i)
157     {
158         DPrackerAF_material->setTrialStrainIncr(myinputStrain);
159         DPrackerAF_material->commitState();
160         print_this_step(++mystep,DPrackerAF_material,outfile_strain,outfile_stress);
161     }
162
163     // Input nagative shear strain increment each step
164     myinputStrain(0,2)=-steplength_loading;
165     myinputStrain(2,0)=myinputStrain(0,2);
166     for (int i = 0; i*steplength_loading < theGammac*2; ++i)
167     {
168         DPrackerAF_material->setTrialStrainIncr(myinputStrain);
169         DPrackerAF_material->commitState();
170         print_this_step(++mystep,DPrackerAF_material,outfile_strain,outfile_stress);
171     }
172
173     // Input positive shear strain increment each step
174     myinputStrain(0,2)=steplength_loading;
175     myinputStrain(2,0)=myinputStrain(0,2);
176     for (int i = 0; i*steplength_loading < theGammac; ++i)
177     {
178         DPrackerAF_material->setTrialStrainIncr(myinputStrain);
179         DPrackerAF_material->commitState();
180         print_this_step(++mystep,DPrackerAF_material,outfile_strain,outfile_stress);
181     }
182 }
183
184 // garbage collection before exit this loop.
185 outfile_strain.close();
186 outfile_stress.close();
187 delete DPrackerAF_material;
188 }
189
190 return 0;
191 }
192 }
193
194 // previous data:
195 // calculate 9 values only. The plotted curve are not smooth but this will be fast.
196 // double Seed_strain[]={0.0001, 0.0003, 0.001, 0.003, 0.01, 0.03, 0.1, 0.3, 1};
197 // int num_gammac=sizeof(Seed_strain)/sizeof(double);
198 // vector<double> gammac (Seed_strain, Seed_strain + num_gammac);
199 // for(auto& g: gammac) g=g/100;
```

21.4.2 The function to plot the results with 51 loops

I did not use LBNL's function to plot the results, since I am not using the HDF5 output. I write my own function plot the results.

```

1 #!/usr/bin/python
2
3 import numpy as np
4 import matplotlib.pyplot as plt
5 pi=np.pi
6
7 # Open and read files
8 # fig1 for stress-strain results:
9 fig1=plt.figure()
10 G_Gmax=[1.0]
11 gamma=[0.0]
12 xi=[0.0]
13 for i in range(1,10,1):
14
15     filename_strain ="shearStrain"+str(i)+".txt"
16     filename_stress ="shearStress"+str(i)+".txt"
17     with open(filename_strain) as fStrain:
18         shearStrain = fStrain.read()
19
20     with open(filename_stress) as fStress:
21         shearStress = fStress.read()
22
23     # split the string to strains.
24     strain_spl=shearStrain.split('\n')
25     stress_spl=shearStress.split('\n')
26     # remove the last blankspace.
27     strain_spl.pop()
28     stress_spl.pop()
29     # convert string to int
30     strain=map(float, strain_spl)
31     stress=map(float, stress_spl)
32
33
34     strain_max=max(strain)
35     stress_max=max(stress)
36     G_sec=stress_max/strain_max
37
38     stress_2nd=stress[1]
39     strain_2nd=strain[1]
40     G_max=stress_2nd/strain_2nd
41
42     thisG_Gmax=G_sec/G_max
43     G_Gmax.append(thisG_Gmax)
44     gamma.append(strain_max)
45
46     # -----
47     # add the subplot to the stress-strain results
48     thisFig = fig1.add_subplot(3,3,i)
49     thisFig.plot(strain,stress)

```

```

50 plt.xlabel(r'$\gamma$')
51 plt.ylabel(r'$\tau$')
52 plt.grid(True)
53 plt.autoscale(enable=True, axis='x', tight=True) #
54 # plt.xlim( (-0.1, 0.1) )
55 # plt.ylim((-1e6,1e6))
56
57
58 # -----
59 # calculate the damping ratio
60 # find the number of steps in the one loop
61 num_zero_strain=0
62 strain_len=len(strain)
63 index=strain_len
64 while True:
65     # the last is zero.
66     index=index-1
67     # find the second last and third last zeros.
68     if abs(strain[index-1]) <1e-15:
69         num_zero_strain=num_zero_strain+1
70     if num_zero_strain==2:
71         break;
72 num_steps_perloop=strain_len-index+1
73 strain_last_loop=strain[-num_steps_perloop:]
74 stress_last_loop=stress[-num_steps_perloop:]
75
76 stress_bottom_last_loop= min(stress_last_loop)
77 stress_top_bottom = [i-stress_bottom_last_loop for i in stress_last_loop]
78
79
80 A_loop=0.0
81 for i in range(num_steps_perloop-1):
82     A_loop=A_loop+(stress_top_bottom[i]+stress_top_bottom[i+1])/2*(strain_last_loop[i+1]-strain_last_loop[i])
83
84 damping_ratio=A_loop/strain_max/strain_max/2.0/pi/G_sec
85 xi.append(damping_ratio)
86 print "-----"
87 print "G_sec:",G_sec
88 print "strain_max",strain_max
89 print "A_loop",A_loop
90 print "G_Gmax", thisG_Gmax
91 print "damping_ratio", damping_ratio
92 print "-----"
93
94 # For fig1 strain-stress plot
95 # reduce spacing between subplots to minimize the overlaps.
96 # plt.tight_layout()
97
98 # -----
99 # plot the G_Gmax results
100 fig2=plt.figure()
101 plt.semilogx(gamma,G_Gmax,label = "DPAF")
102 # plot Seed results for comparison
103 Seed_strain_percent = [0.0001, 0.0003, 0.001, 0.003, 0.01, 0.03, 0.1, 0.3, 1.0];
104 Seed_G = [1, 0.99, 0.96, 0.9, 0.76, 0.57, 0.3, 0.15, 0.06];

```

```
105 | Seed_strain=[x/100.0 for x in Seed_strain_percent];
106 | plt.semilogx(Seed_strain, Seed_G, label="Seed")
107 | plt.legend()
108 | # plt.xlim((0,0.011))
109 | plt.autoscale(enable=True, axis='x', tight=True) #
110 | plt.ylim((0,1.1))
111 | plt.xlabel('$\gamma_c$', fontsize=25)
112 | plt.ylabel('G/Gmax', fontsize=25)
113 | plt.grid(True)
114 | # -----
115 | # plot the damping ratio results
116 | fig3=plt.figure()
117 | plt.semilogx(gamma,xi,label = "DPAF")
118 | sameStrain=Seed_strain
119 | Seed_damping=[0.0, 0.0, 0.01, 0.02, 0.033, 0.05, 0.1, 0.155, 0.21]
120 | plt.semilogx(sameStrain,Seed_damping,label="Seed")
121 | plt.legend()
122 |
123 | # plt.ylim((0,0.4))
124 |
125 | plt.xlabel(r'$\gamma_c$', fontsize=25)
126 | plt.ylabel(r'$\xi$', fontsize=25)
127 | plt.grid(True)
128 | # -----
129 | # disp all the results on screen
130 | plt.show()
```

21.4.3 The function to plot the G/Gmax with one loading only

```
1 #!/usr/bin/python
2
3 import numpy as np
4 import matplotlib.pyplot as plt
5
6 # Open and read files
7 # open one file only
8 # 51 is the final loading file only.
9 i=51
10
11 filename_strain ="shearStrain"+str(i)+".txt"
12 filename_stress ="shearStress"+str(i)+".txt"
13 with open(filename_strain) as fStrain:
14     shearStrain = fStrain.read()
15
16 with open(filename_stress) as fStress:
17     shearStress = fStress.read()
18
19 # split the string to strains.
20 strain_spl=shearStrain.split('\n')
21 stress_spl=shearStress.split('\n')
22 # remove the last blankspace.
23 strain_spl.pop()
24 stress_spl.pop()
25 # convert string to int
26 strain=map(float, strain_spl)
27 stress=map(float, stress_spl)
28
29 index=0
30 while True:
31     index=index+1
32     # find the second zero.
33     if strain[index] > strain[index+1]:
34         break;
35
36 strain_1load=strain[:index]
37 stress_1load=stress[:index]
38
39
40 stress_2nd=stress[1]
41 strain_2nd=strain[1]
42 G_max=stress_2nd/strain_2nd
43
44 G_Gmax=[1.0]
45 gamma=[0.0]
46
47 for i in range(index-1):
48     G_sec=stress_1load[i+1]/strain_1load[i+1]
49     thisG_Gmax=G_sec/G_max
50     G_Gmax.append(thisG_Gmax)
51     gamma.append(strain_1load[i])
```

```
52
53
54
55 # -----
56 # plot the G_Gmax results
57 fig2=plt.figure()
58 plt.semilogx(gamma,G_Gmax,label = "DPAF")
59 # plot Seed results for comparison
60 Seed_strain_percent = [0.0001, 0.0003, 0.001, 0.003, 0.01, 0.03, 0.1, 0.3, 1.0];
61 Seed_G = [1, 0.99, 0.96, 0.9, 0.76, 0.57, 0.3, 0.15, 0.06];
62 Seed_strain=[x/100.0 for x in Seed_strain_percent];
63 plt.semilogx(Seed_strain, Seed_G, label="Seed")
64 plt.legend()
65 # plt.xlim((0,0.011))
66 plt.autoscale(enable=True,axis='x',tight=True) #
67 plt.ylim((0,1.1))
68 plt.xlabel('$\gamma_c$',fontsize=25)
69 plt.ylabel('G/Gmax',fontsize=25)
70 plt.grid(True)
71 # -----
72 plt.show()
```

References

- [1] GR Cowper. The shear coefficient in timoshenko's beam theory. *Journal of applied mechanics*, 33(2):335–340, 1966.
- [2] JD Renton. Generalized beam theory applied to shear stiffness. *International Journal of Solids and Structures*, 27(15):1955–1967, 1991.