EXAMPLE A30

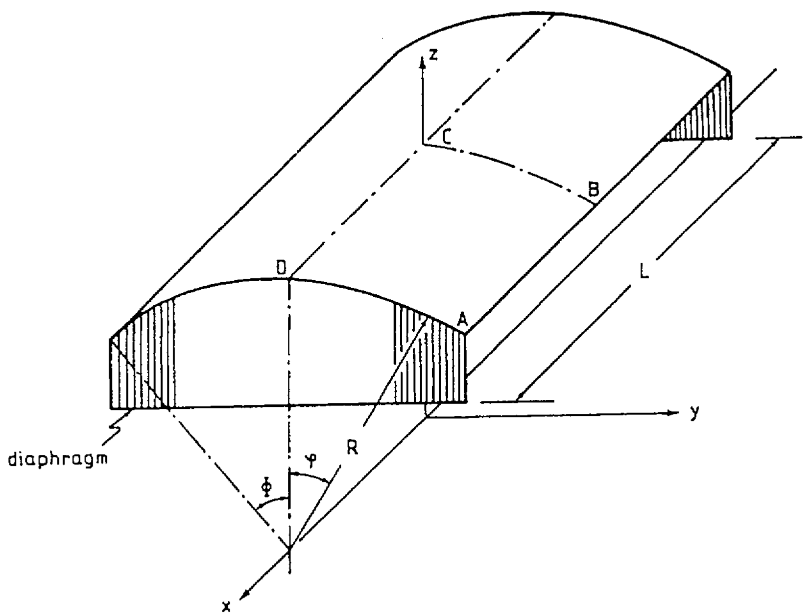
SCORDELIS-LO CYLINDRICAL ROOF, CUBIC SHELL

Objective

To verify the curved isoparametric SHELL element when subjected to gravity loading.

Physical Problem

A cylindrical shell roof subjected to gravity loading is considered, see figure below. The shell roof is supported on diaphragms at the ends and it is free along the longitudinal sides.



$R = 300$ in
$E = 3.0 \cdot 10^6$ psi
$L = 600$ in
$\nu = 0.0$
$h = 3.0$ in (thickness)
$\phi = 40°$
Shell weight = 90.0 lb/sq-ft

Finite Element Model

Due to symmetry, only one quarter of the cylindrical shell roof needs to be considered. The part A-B-C-D in the figure above is modeled using two cubic isoparametric SHELL elements, see figures on page A30.3. Symmetrical boundary conditions are applied along the two sides defined by nodal points between nodes 4 and 1 (line 4-1) and between nodes 3 and 4 (line 3-4). The nodes corresponding to the diaphragm side (line 1-2) are fixed for translation in the Y- and Z-directions.

Solution Results

This example problem has been used extensively as a benchmark problem for shell elements. The analytical shallow shell solution generally quoted for the vertical deflection at the centre of the free edge (point B in the figure above) is -3.703 inches [1] although some authors use -3.696 inches. A deep shell exact analytical solution quoted is -3.53 inches. Input data is shown on page A30.8, for the two element model. The problem has also been analyzed with 8, 32 and 72 SHELL elements.

Number of elements	Displacement Z-direction Point B	Stress-rr Point C	Stress-rr Point B
2 (2×1)	-3.558	1305.4	24.47
8 (4×2)	-3.598	1302.1	-26.26
32 (8×4)	-3.611	1273.5	0.88
72 (12×6)	-3.615	1271.5	2.82

A contour plot and the circumferential stress (stress-rr) and axial stress (stress-ss) along the line CB are shown in the figures on pages A30.4 to A30.7 for the four models.

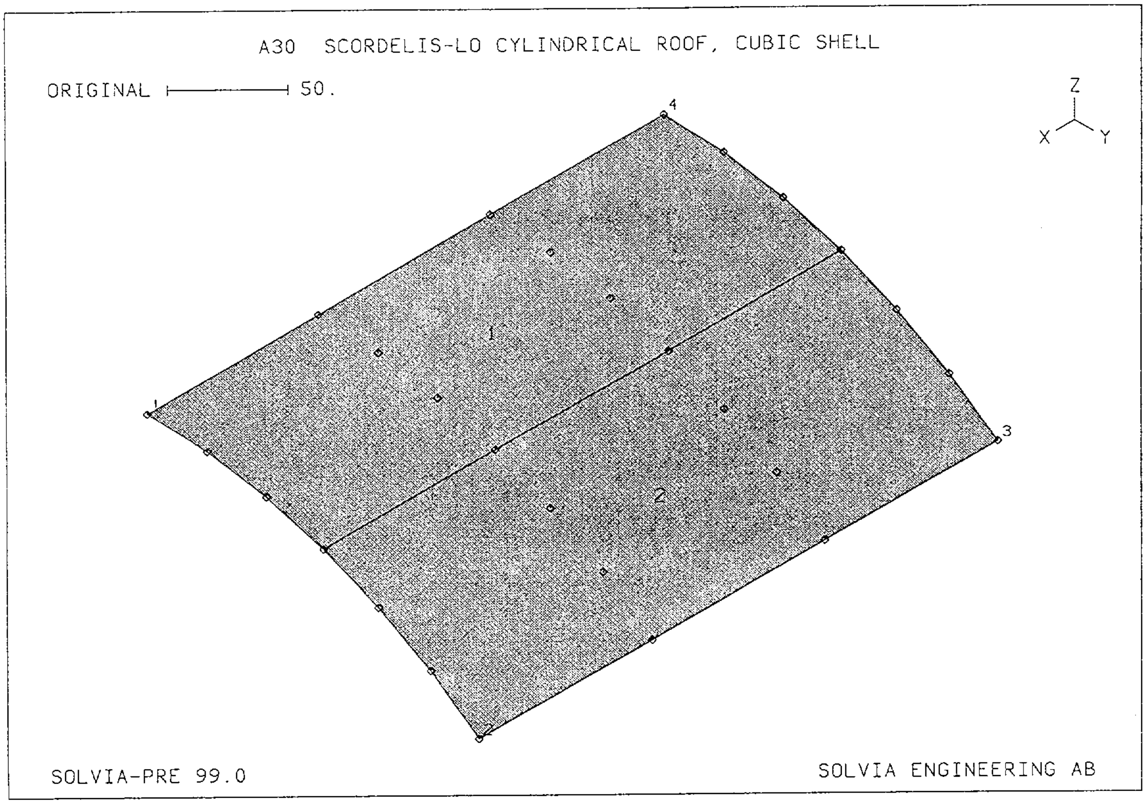
User Hints

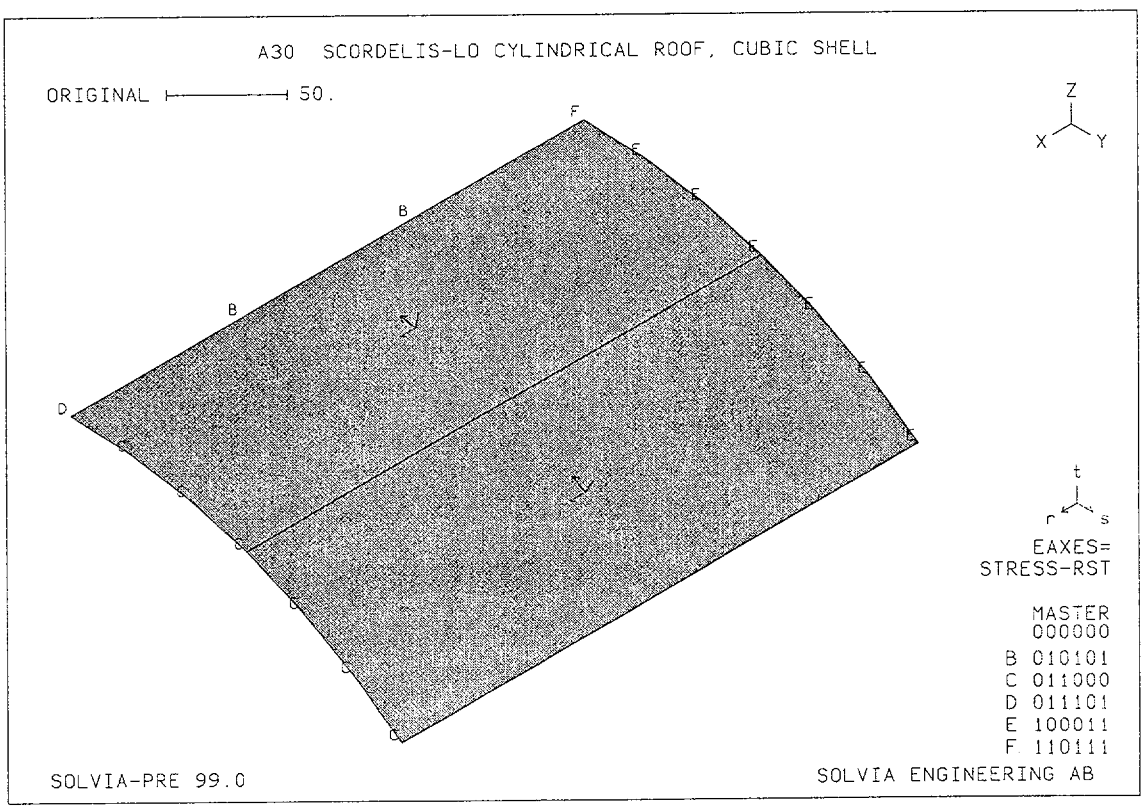
- Note that only two SHELL elements with 4x4x2 Gauss integration points can be used to model this example problem resulting into good agreement with theoretical results for displacements.
- A further description of this example problem can be found in [2].

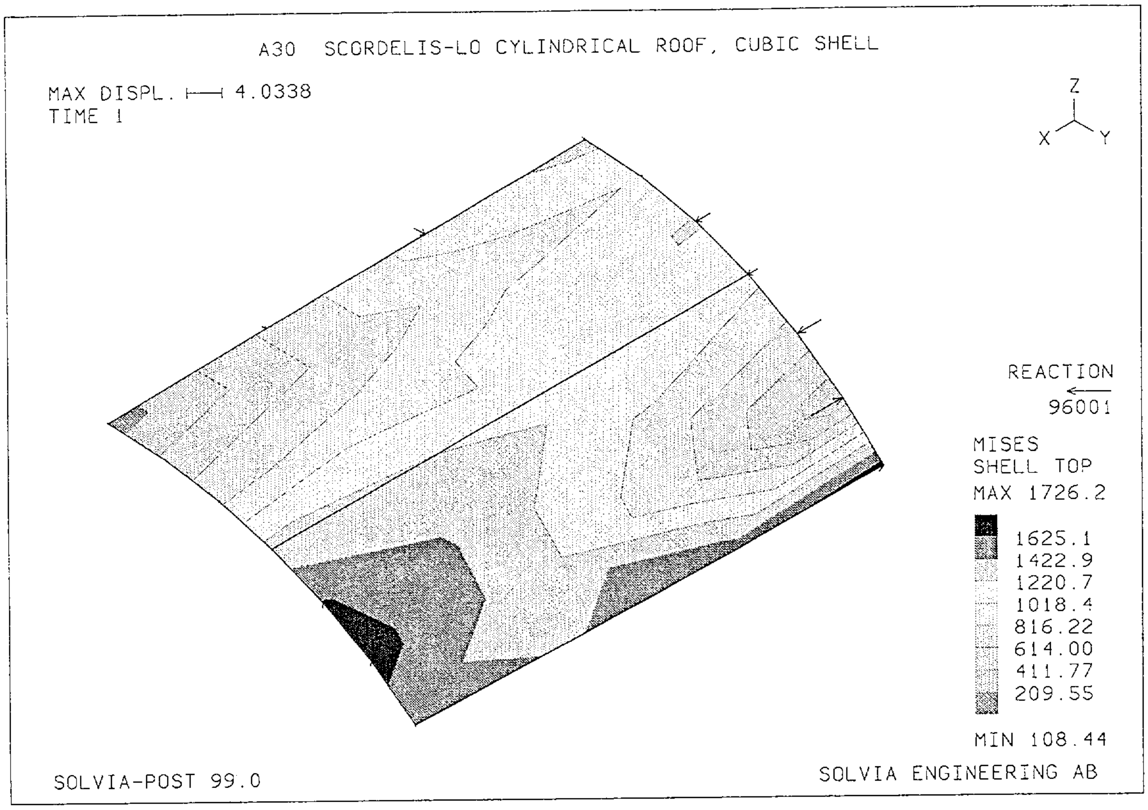
References

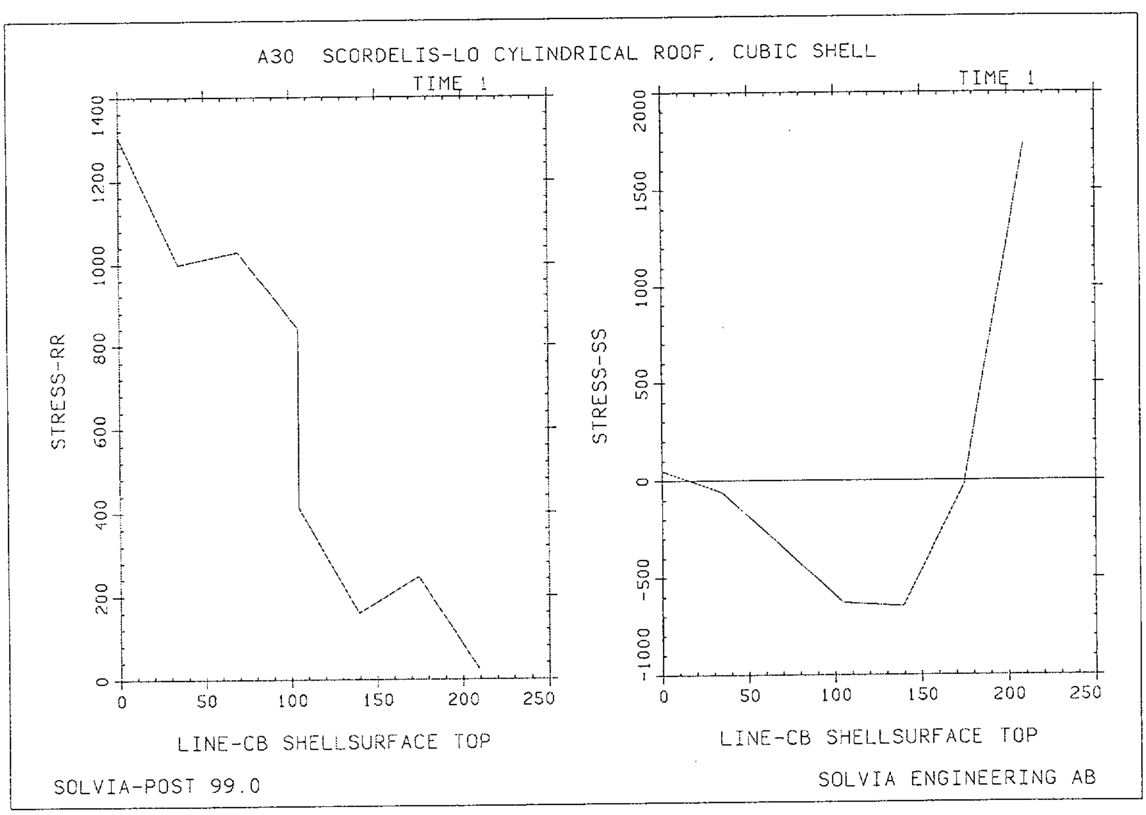
[1] Scordelis, A.C., Lo, K.S., "Computer Analysis of Cylindrical Shells", J. Amer. Concr. Inst., Vol. 61, pp. 539-560, 1964.

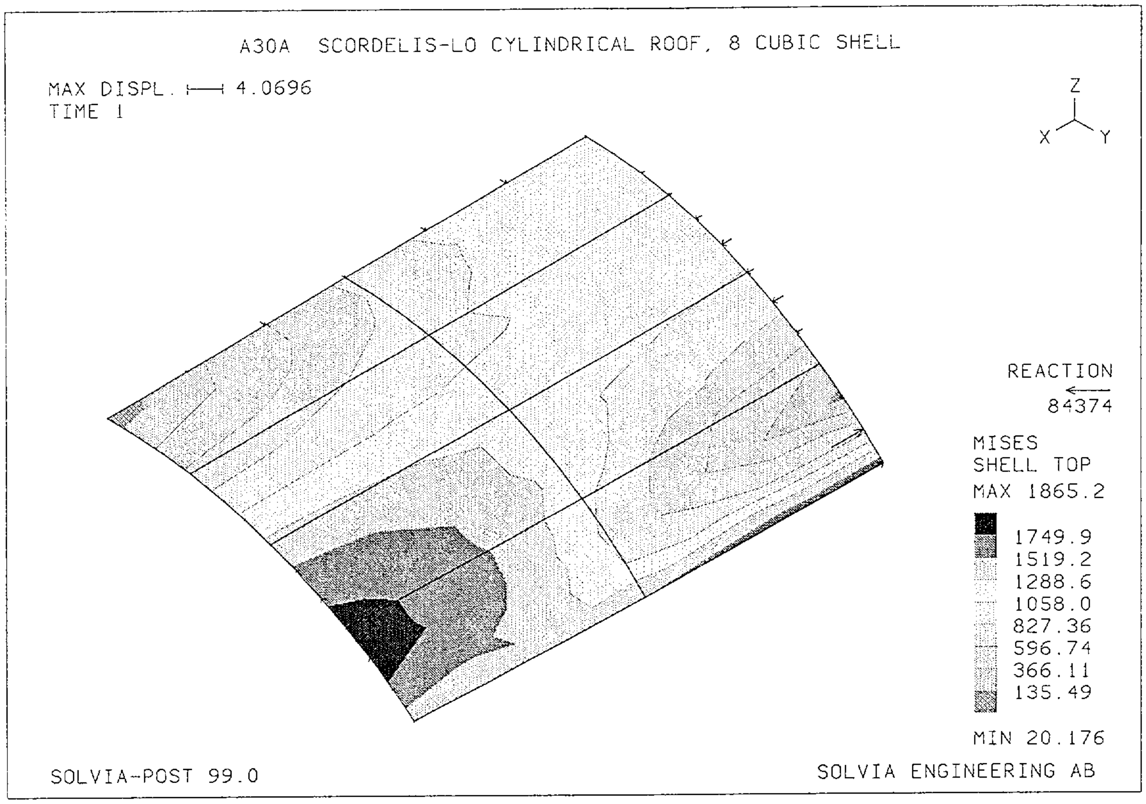
[2] Larsson, G. and Olsson, H., "An Engineering Error Measure for Finite Element Analysis", Finite Element News, April 1988.

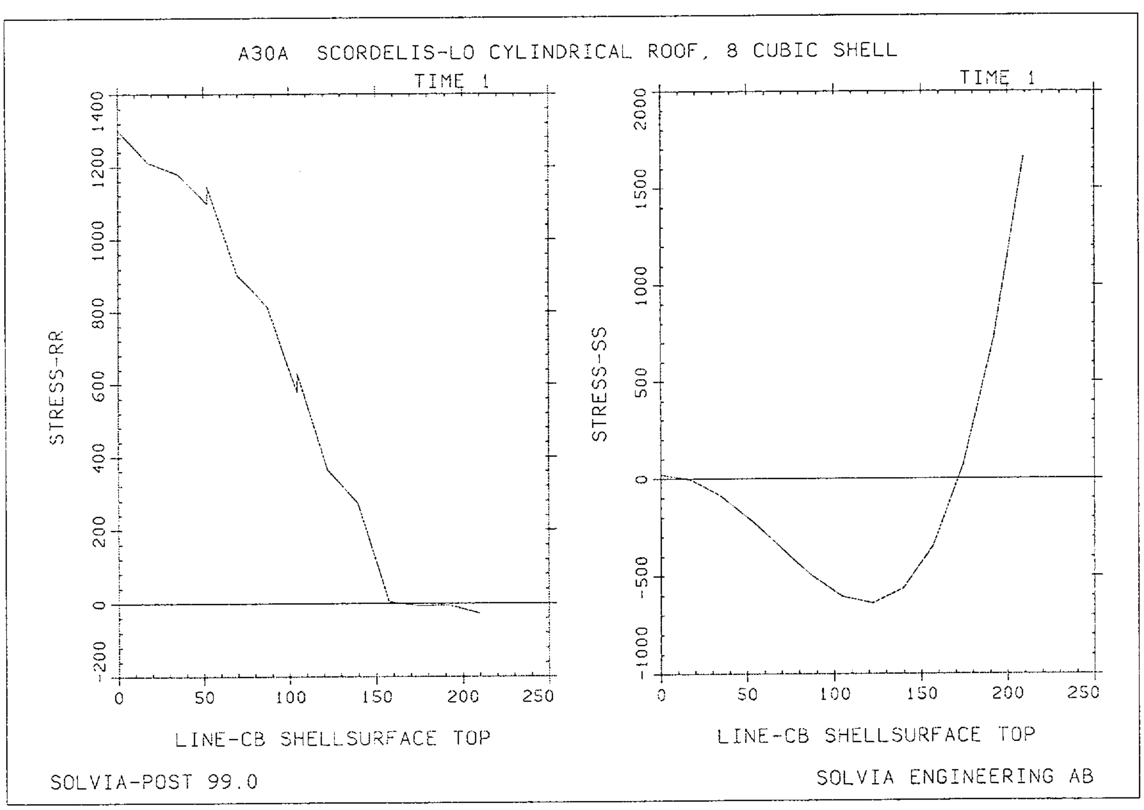

A30 SCORDELIS-LO CYLINDRICAL ROOF, CUBIC SHELL
ORIGINAL 50.
Z
X Y
4
1
3
1
2
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

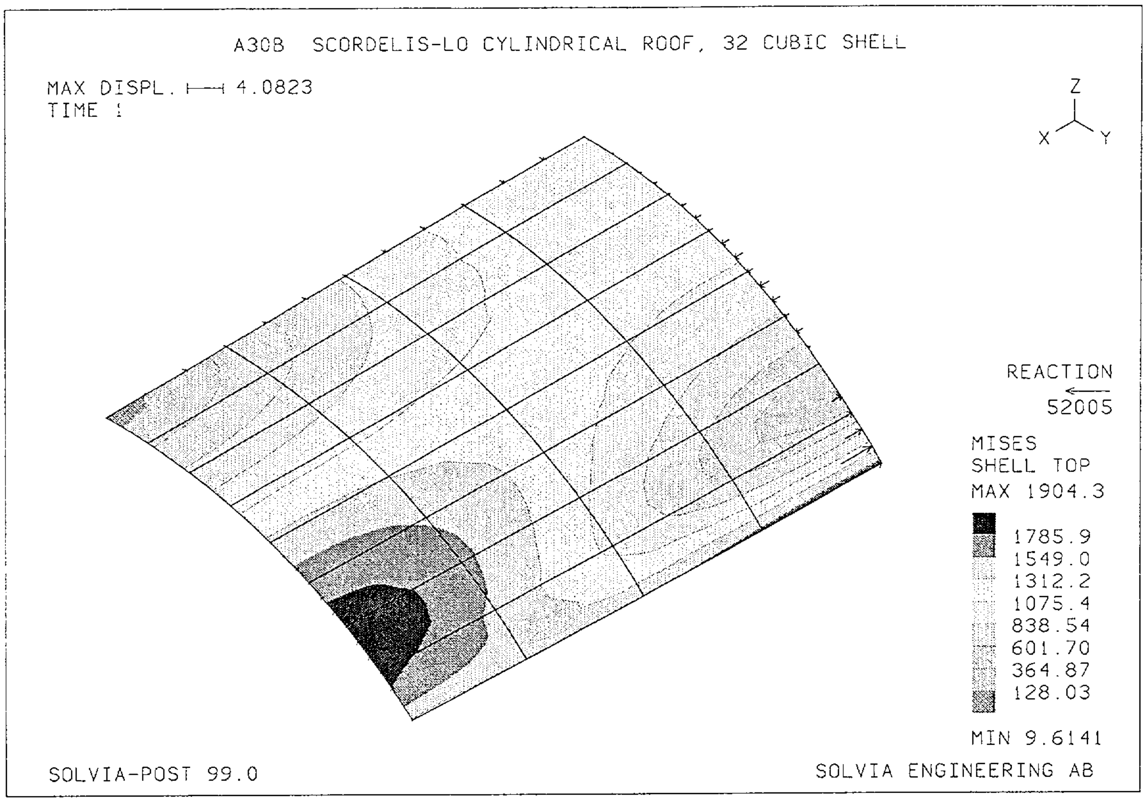

A30 SCORDELIS-LO CYLINDRICAL ROOF, CUBIC SHELL
ORIGINAL 50.
Z
X Y
F
B
B
D
t
r s
EAXES=
STRESS-RST
MASTER
000000
B 010101
C 011000
D 011101
E 100011
F 110111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

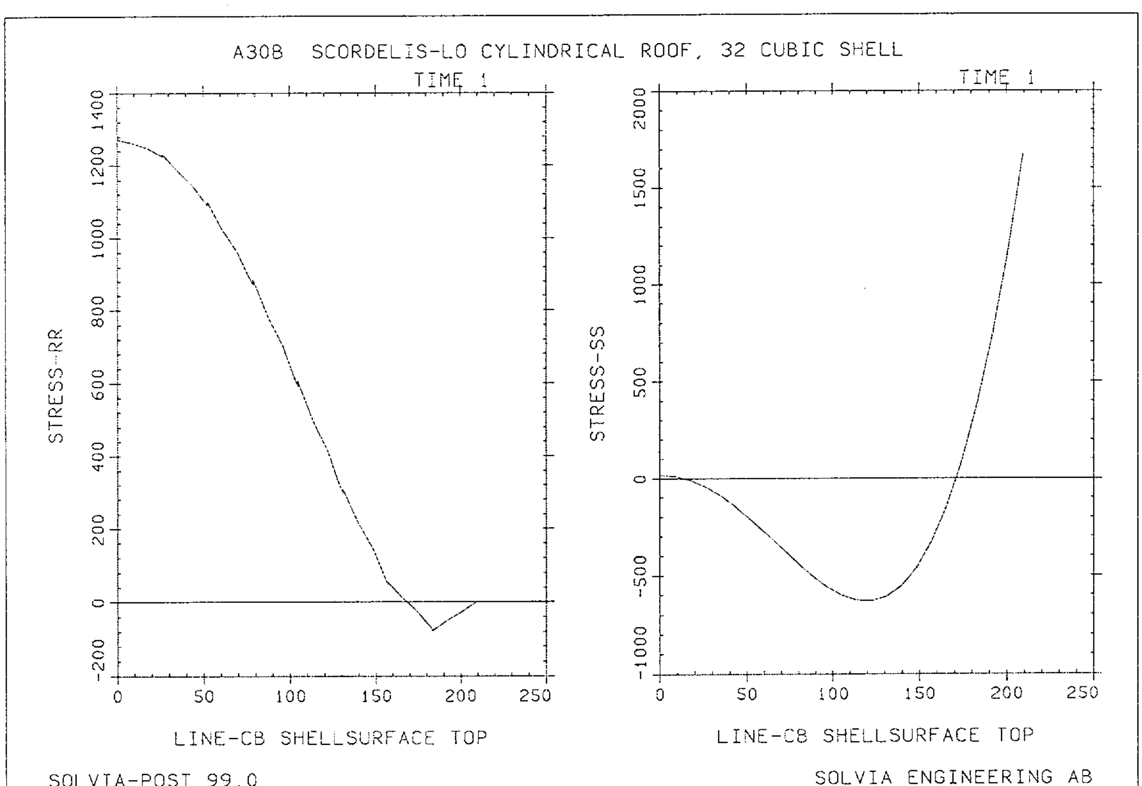

A30 SCORDELIS-LO CYLINDRICAL ROOF, CUBIC SHELL
MAX DISPL. 4.0338
TIME 1
Z
X
Y
REACTION
96001
MISES
SHELL TOP
MAX 1726.2
1625.1
1422.9
1220.7
1018.4
816.22
614.00
411.77
209.55
MIN 108.44
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

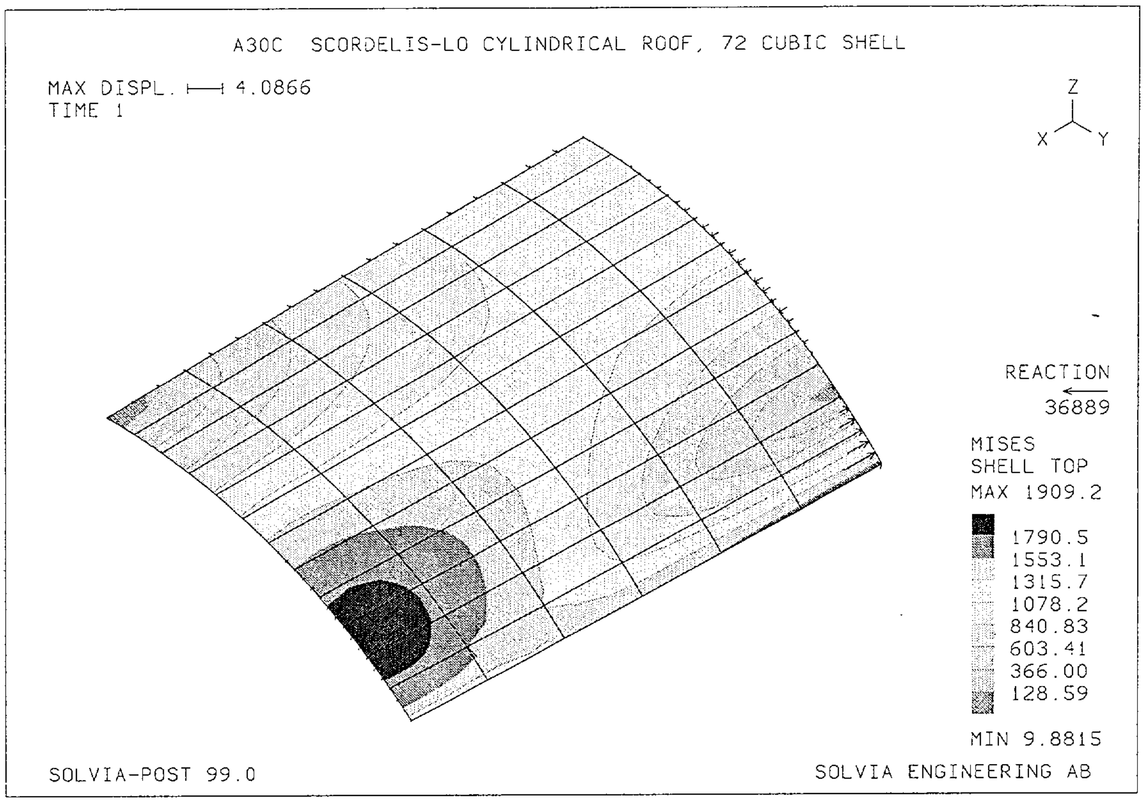

A30 SCORDELIS-LO CYLINDRICAL ROOF, CUBIC SHELL
TIME 1
STRESS-RR
LINE-CB SHELLSURFACE TOP
TIME 1
STRESS-SS
LINE-CB SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

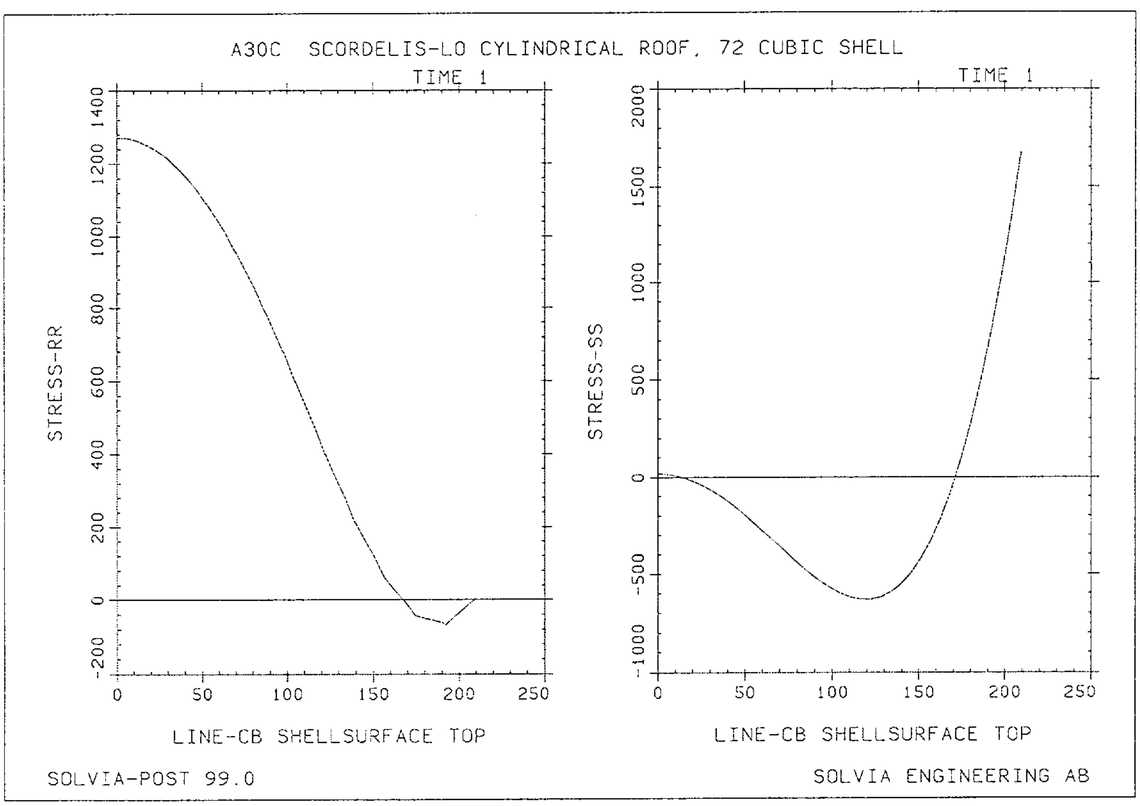

A30A SCORDELIS-LO CYLINDRICAL ROOF, 8 CUBIC SHELL
MAX DISPL. 4.0696
TIME 1
Z
X
Y
REACTION
84374
MISES
SHELL TOP
MAX 1865.2
1749.9
1519.2
1288.6
1058.0
827.36
596.74
366.11
135.49
MIN 20.176
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A30A SCORDELIS-LO CYLINDRICAL ROOF, 8 CUBIC SHELL
TIME 1
STRESS-RR
-200
0
200
400
600
800
1000
1200
1400
0
50
100
150
200
250
LINE-CB SHELLSURFACE TOP
TIME 1
STRESS-SS
-1000
-500
0
500
1000
1500
2000
0
50
100
150
200
250
LINE-CB SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A30B SCORDELIS-LO CYLINDRICAL ROOF, 32 CUBIC SHELL
MAX DISPL. 4.0823
TIME 1
Z
X
Y
REACTION
52005
MISES
SHELL TOP
MAX 1904.3
1785.9
1549.0
1312.2
1075.4
838.54
601.70
364.87
128.03
MIN 9.6141
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A30B SCORDELIS-LO CYLINDRICAL ROOF, 32 CUBIC SHELL
TIME 1
STRESS-RR
LINE-CB SHELLSURFACE TOP
TIME 1
STRESS-SS
LINE-CB SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A30C SCORDELIS-LO CYLINDRICAL ROOF, 72 CUBIC SHELL
MAX DISPL. 4.0866
TIME 1
Z
X
Y
REACTION
36889
MISES
SHELL TOP
MAX 1909.2
1790.5
1553.1
1315.7
1078.2
840.83
603.41
366.00
128.59
MIN 9.8815
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A30C SCORDELIS-LO CYLINDRICAL ROOF, 72 CUBIC SHELL
TIME 1
TIME 1
STRESS-RR
STRESS-SS
LINE-CB SHELLSURFACE TOP
LINE-CB SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A30  SCORDELIS-LO CYLINDRICAL ROOF, CUBIC SHELL'
*
DATABASE  CREATE
*
ANALYSIS  TYPE=STATIC MASSMATRIX=LUMPED
SYSTEM  1 CYLINDRICAL
COORDINATES
 ENTRIES NODE  R  THETA  XL
         1   300  90   300
         2   300  50   300
         3   300  50     0
         4   300  90     0
*
MATERIAL  1  ELASTIC  E=3.E6 NU=0. DEN=0.208333
*
EGROUP  1  SHELL  STRESSREFERENCE=ELEMENT RESULTS=NSTRESSES
THICKNESS  1  3.0
GSURFACE  1 2 3 4  EL1=2 EL2=1 NODES=16 SYSTEM=1
*
FIXBOUNDARIES  246 INPUT=LINE  /  4 1
FIXBOUNDARIES  23  INPUT=LINE  /  1 2
FIXBOUNDARIES  156 INPUT=LINE  /  3 4
*
LOADS  MASSPROPORTIONAL ZFACTOR=1. ACC=-1.
*
MESH  NNUMBERS=MYNODES NSYMBOLS=YES ENUMBER=YES
MESH  BCODE=ALL EAXES=STRESS-RST
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A30  SCORDELIS-LO CYLINDRICAL ROOF, CUBIC SHELL
*
DATABASE  CREATE
*
WRITE  FILENAME='a30.lis'
*
MESH  CONTOUR=MISES VECTOR=REACTION
*
EPLINE  NAME=LINE-CB
 1  4 11 7 3  TO  2  4 11 7 3
*
ELINE  LINENAME=LINE-CB KIND=SRR OUTPUT=ALL SUBFRAME=21
ELINE  LINENAME=LINE-CB KIND=SSS OUTPUT=ALL
*
NLIST  ZONENAME=N3 DIRECTION=23
MASS-PROPERTIES
END
```

EXAMPLE A31

SCORDELIS-LO CYLINDRICAL ROOF, PLATE

Objective

To verify the PLATE element when applied to a curved shell structure subjected to gravity loading.

Physical Problem

Same as for Example A30.

Finite Element Model

As for the previous example only one quarter of the structure needs to be modeled. A 12×12×4 mesh of PLATE elements is used, see figures on page A31.2.

Solution Results

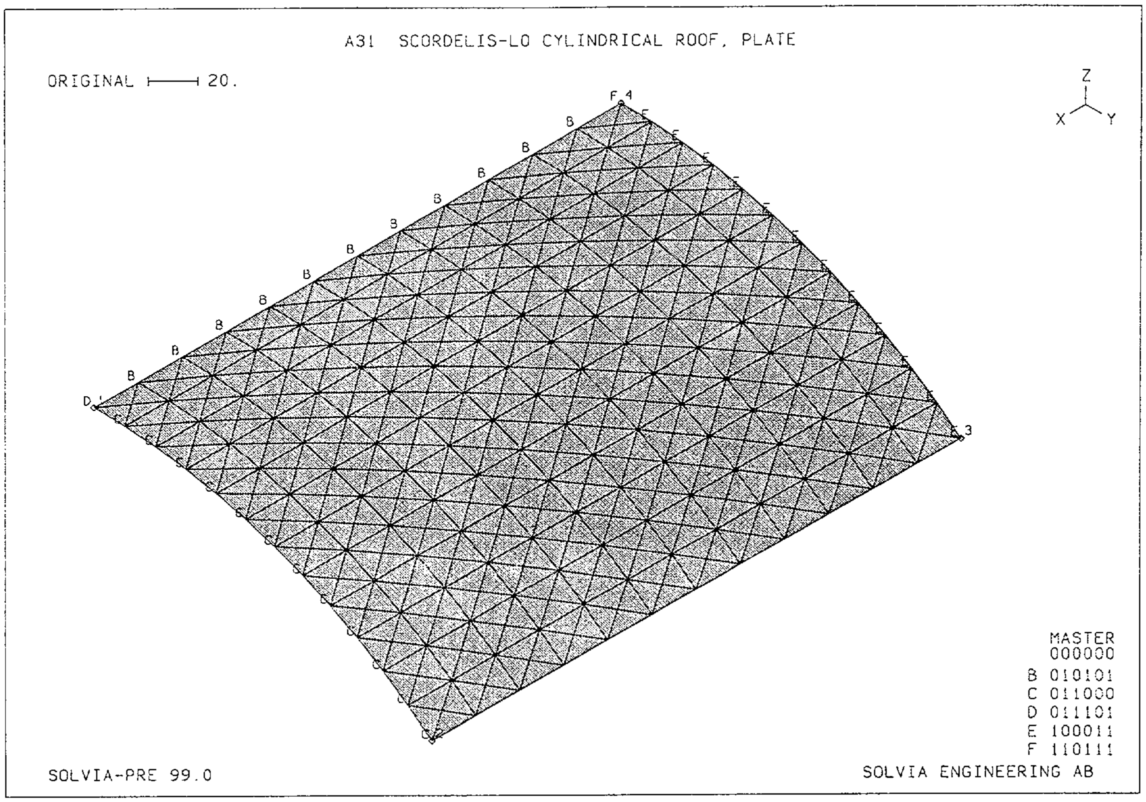
Using the input data shown on pages A31.4 and A31.5 the vertical deflection at point B of figure on page A30-1 is predicted to be -3.521 inches.

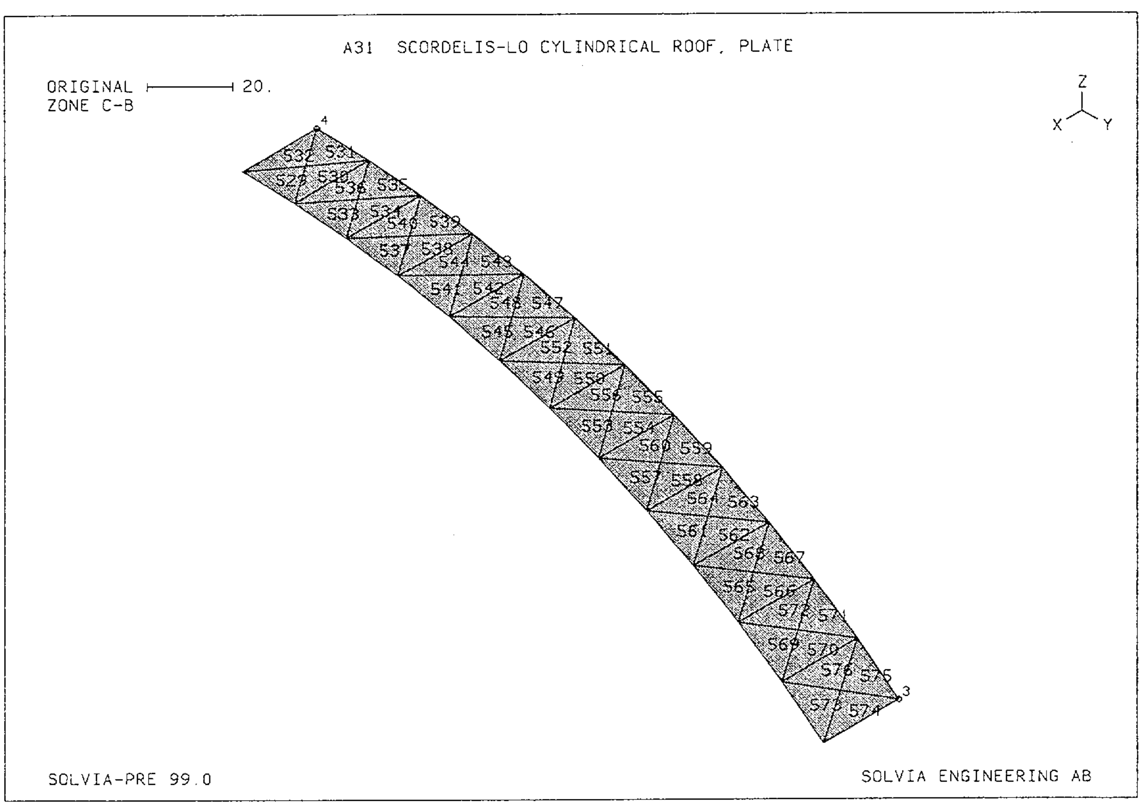
A contour plot of the vertical displacement and the variation of the bending moment about the X-axis (corresponding to bending stresses in the circumferential direction, the x_1-direction in the selected Local Cylindrical System) for two lines are shown in the figures on page A31.3. The line "X-0" is identical to line CD and the line "X-12" is parallel to line CD but goes through the midpoints of the quadrilaterals nearest line CD.

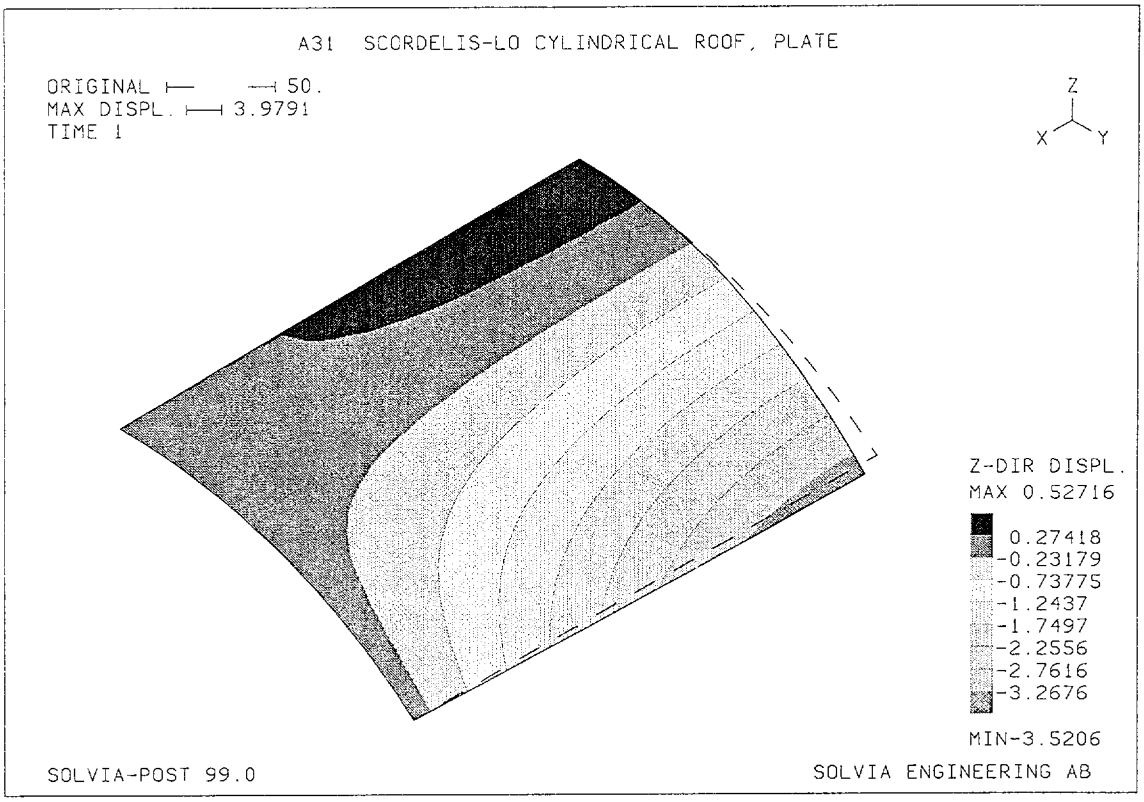
User Hints

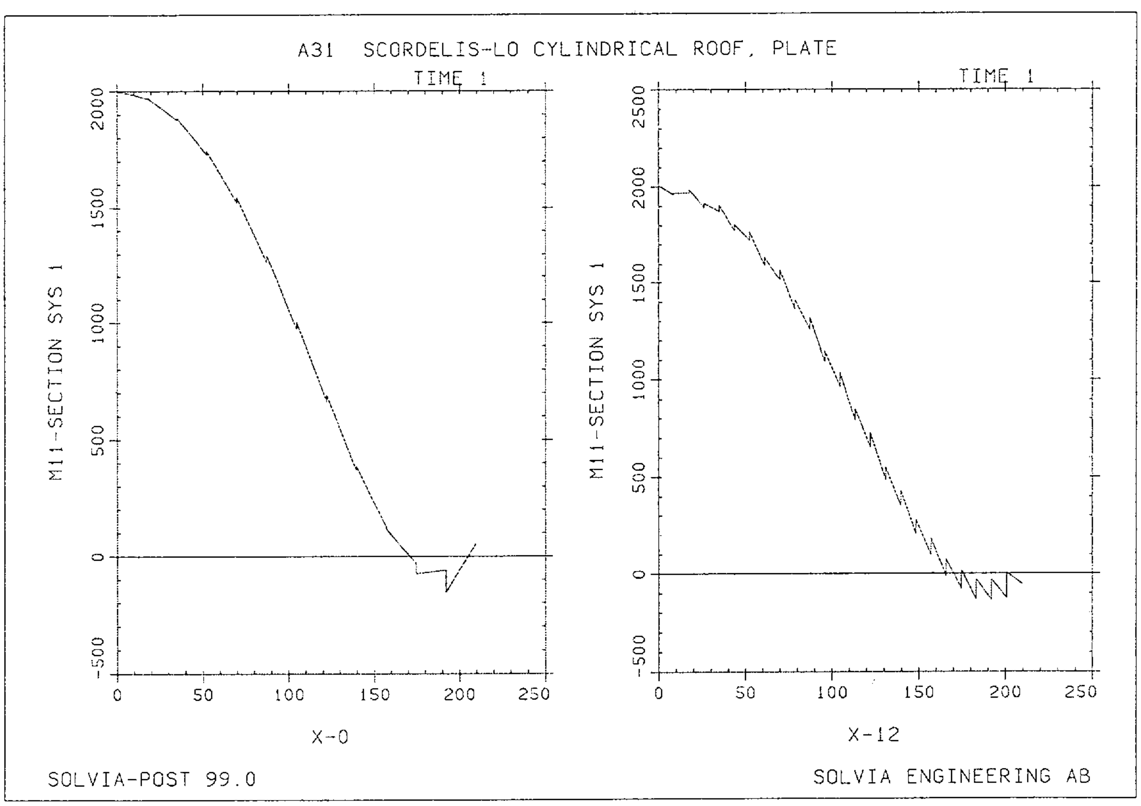
- Note that even though a large number of PLATE elements is used, the displacement solution for the considered point B is still not as good as the solution obtained in Example A30, where only two isoparametric SHELL elements are used. One reason is that the PLATE element is flat and the cylindrical shell roof is, therefore, approximated by straight segments. Another reason is that the membrane action of the PLATE element is the same as for a constant strain triangle. The membrane forces are constant over each element, which limits the capability of the PLATE element to describe structures, where the membrane forces vary significantly.

- The recommended finite element to model this example structure is, therefore, the SHELL element.


A31 SCORDELIS-LO CYLINDRICAL ROOF, PLATE
ORIGINAL 20.
Z
X Y
MASTER
000000
B 010101
C 011000
D 011101
E 100011
F 110111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A31 SCORDELIS-LO CYLINDRICAL ROOF, PLATE
ORIGINAL 20.
ZONE C-B
Z
X Y
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A31 SCORDELIS-LO CYLINDRICAL ROOF, PLATE
ORIGINAL 50.
MAX DISPL. 3.9791
TIME 1
Z
X Y
Z-DIR DISPL.
MAX 0.52716
0.27418
-0.23179
-0.73775
-1.2437
-1.7497
-2.2556
-2.7616
-3.2676
MIN-3.5206
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A31 SCORDELIS-LO CYLINDRICAL ROOF, PLATE
TIME 1
M11-SECTION SYS 1
X-0
TIME 1
M11-SECTION SYS 1
X-12
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A31  SCORDELIS-LO CYLINDRICAL ROOF, PLATE'
*
DATABASE  CREATE
*
ANALYSIS  TYPE=STATIC MASSMATRIX=LUMPED
SYSTEM  1 CYLINDRICAL
COORDINATES
 ENTRIES NODE  R  THETA  XL
           1  300  90   300
           2  300  50   300
           3  300  50     0
           4  300  90     0
*
MATERIAL  1  ELASTIC  E=3.E6 DENSITY=0.208333
*
EGROUP  1  PLATE
STRESSTABLE  1  1 2 3 4 5 6 7
GSURFACE  1 2 3 4  EL1=12 EL2=12 SYSTEM=1
EDATA  /  1 3.0
*
FIXBOUNDARY  246 INPUT=LINE  /  4 1
FIXBOUNDARY  23  INPUT=LINE  /  1 2
FIXBOUNDARY  156 INPUT=LINE  /  3 4
*
LOADS  MASSPROPORTIONAL  ZFACTOR=1. ACCGRA=-1.
*
SET  HEIGHT=0.25 NSYMBOLS=MYNODES NNUMBERS=MYNODES
MESH  BCODE=ALL
ZONE  NAME=C-B INPUT=GLOBAL-LIMITS XMAX=25
MESH  ZONENAME=C-B ENUMBERS=YES
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A31  SCORDELIS-LO CYLINDRICAL ROOF, PLATE
*
DATABASE  CREATE
*
WRITE  FILENAME='a31.lis'
*
MESH  OUTLINE=YES ORIGINAL=DASHED CONTOUR=DZ
*
EPLINE  NAME=X-0
 531  2 4 1  STEP 4 TO  575  2 4 1
EPLINE  NAME=X-12
 532 4 7 3  /  530 3 7 4  /  536 4 7 3  /  534 3 7 4
 540 4 7 3  /  538 3 7 4  /  544 4 7 3  /  542 3 7 4
 548 4 7 3  /  546 3 7 4  /  552 4 7 3  /  550 3 7 4
 556 4 7 3  /  554 3 7 4  /  560 4 7 3  /  558 3 7 4
 564 4 7 3  /  562 3 7 4  /  568 4 7 3  /  566 3 7 4
 572 4 7 3  /  570 3 7 4  /  576 4 7 3  /  574 3 7 4
*
SYSTEM 1 CYLINDRICAL
ELINE  LINENAME=X-0  KIND=M11 OUTPUT=ALL SYSTEM=1 SUBFRAME=21
ELINE  LINENAME=X-12 KIND=M11 OUTPUT=ALL SYSTEM=1
*
NLIST  ZONENAME=N3 DIRECTION=234
MASS-PROPERTIES
END
```

EXAMPLE A32

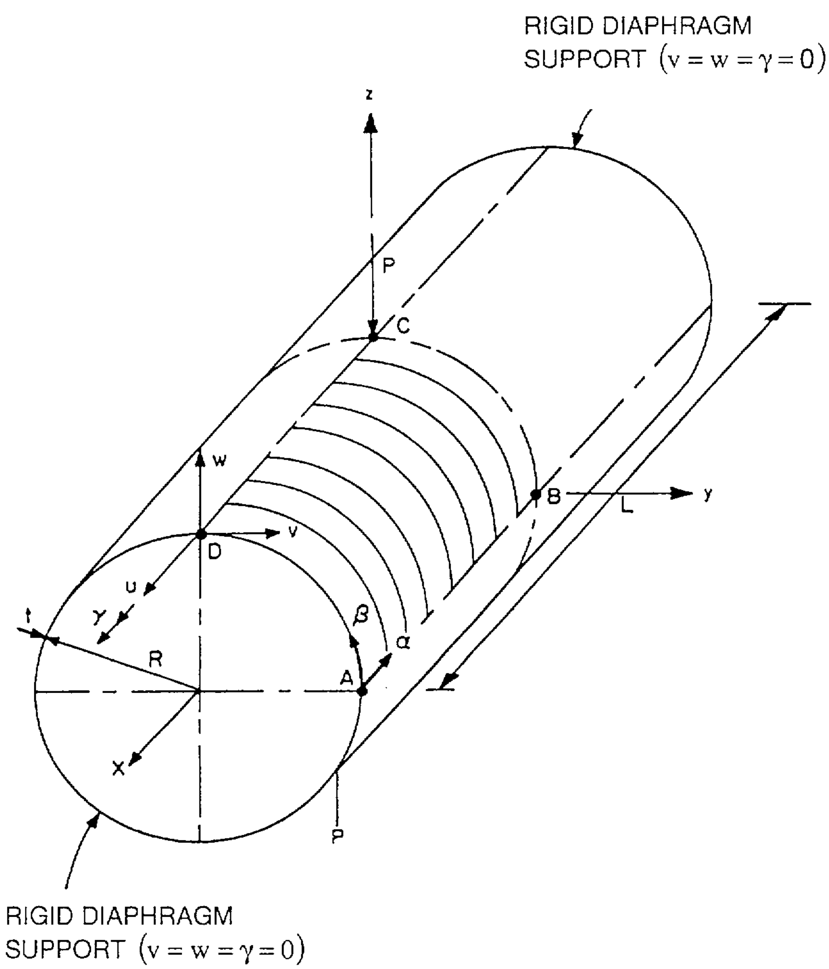
PINCHED CYLINDRICAL SHELL, SHELL ELEMENTS

Objective

To verify the membrane and bending behaviour of the SHELL element when applied to a curved structure.

Physical Problem

The thin cylindrical shell structure shown in the figure below is analyzed for its static response. The cylinder is freely supported at its ends and is loaded by two centrally located and diametrically opposed concentrated forces.



$E = 3.0 \cdot 10^7$ psi

$\nu = 0.3$

$t = 1.0$ in.

$R / t = 100$

$L / R = 2$

$P = 300000$ lbf.

Finite Element Model

Using the three symmetry planes of the structure and the load, only one eighth of the cylinder is analyzed. The 16-node SHELL element is employed using different fineness of the mesh.

Solution Results

The analytical solution for this problem is reported in [1].

The input data used for one of the models (thirty-six 16-node SHELL element, integration 4x4x2) is shown on pages A32.8 and A32.9.

The displacement in Z-direction and the membrane stress-ss (global X-direction) at point C for the different models are shown in the table on the following page.

The graphic results from SOLVIA-POST contain deformed mesh, radial displacement and shell stresses along the line BC. Stress-11 is the stress in the circumferential direction (the x_1-direction) and membrane stress-22 is the membrane stress in axial direction (the x_2-direction).

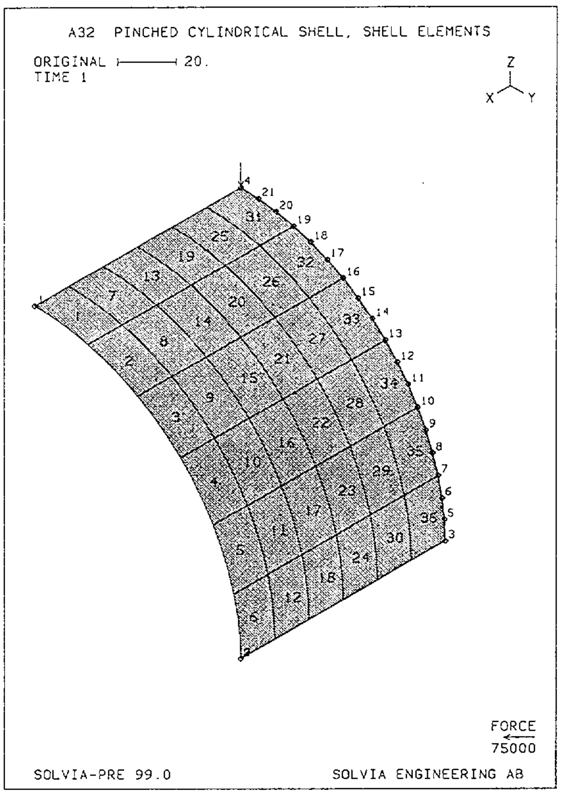
User Hints

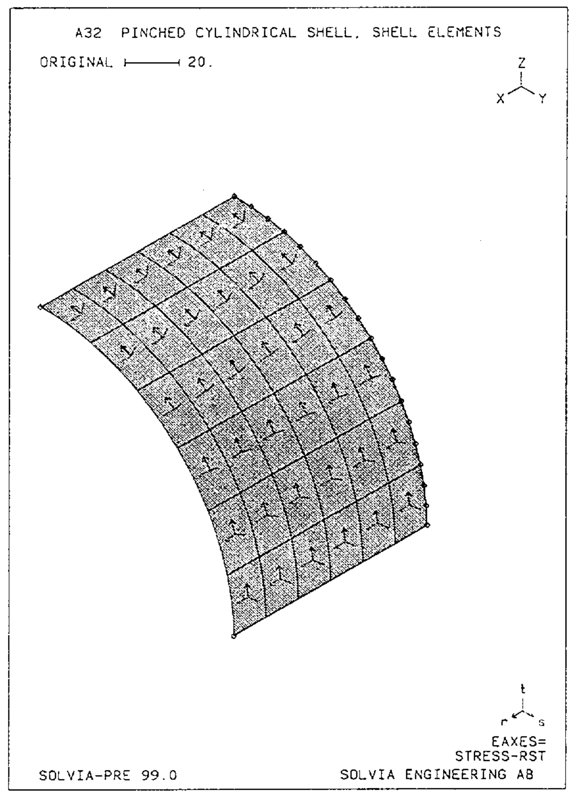
- In SOLVIA-POST the line coordinates are calculated as a line polygon, i.e. the program assumes straight lines between the nodal points. The coordinates on the abscissa of the diagrams in the figures from SOLVIA-POST are, therefore, not equal to the arc length along the circle BC.
- A rather fine mesh around the point of load application is necessary when stress results are desired. If only a displacement solution is of interest a coarser mesh can be used.
- Note that the lower integration order 3x3x2 results in significantly worse stress prediction along the line BC than the 4x4x2 integration order, see figures on page A32.6.

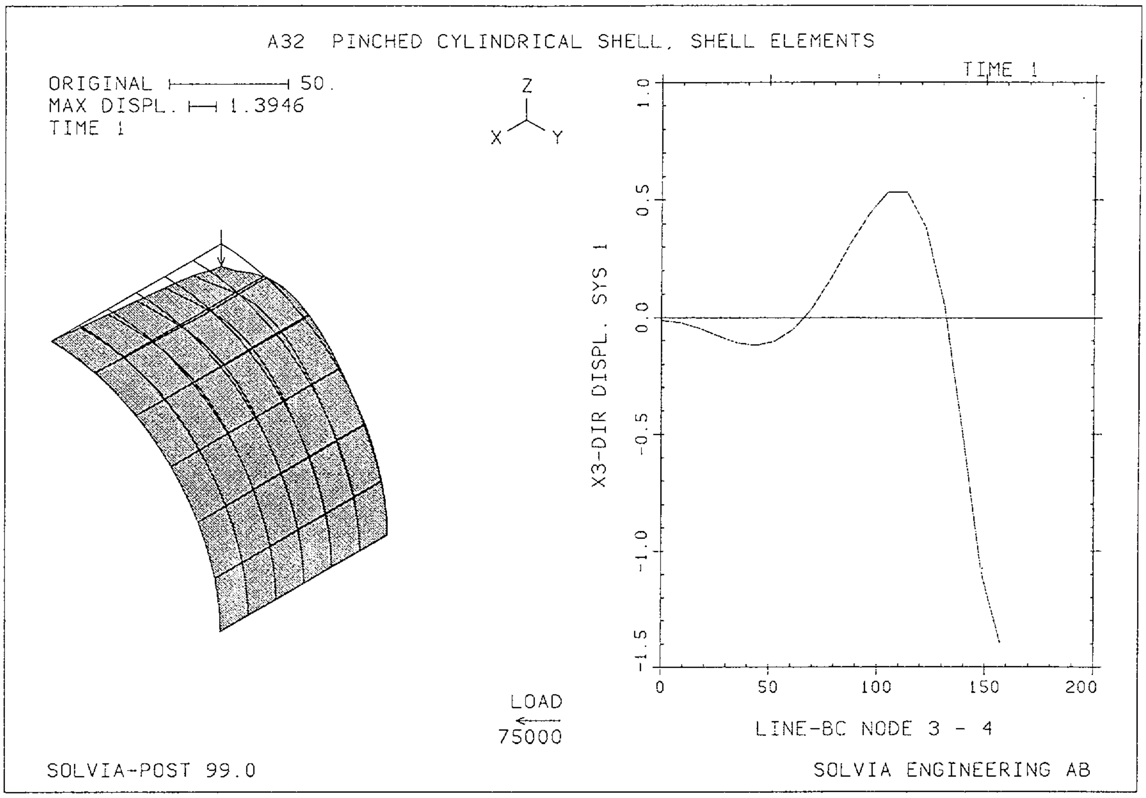
	Z-displacement at point C	Membrane stress-ss at point C	SOLVIA-POST results on page
Analytical solution	-1.642	$-4.72 \cdot 10^4$	
16-node SHELL 36 elements Integration 4×4×2 LINE RATIO=1.0	-1.395	$-6.28 \cdot 10^4$	A32.4
16-node SHELL 36 elements Integration 4×4×2 LINE RATIO=4.0	-1.622	$-5.83 \cdot 10^4$	A32.5
16-node SHELL 36 elements Integration 3×3×2 LINE RATIO=4.0	-1.673	$-11.45 \cdot 10^4$	A32.6
16-node SHELL 144 elements Integration 4×4×2 LINE RATIO=4.0	-1.664	$-5.27 \cdot 10^4$	A32.7

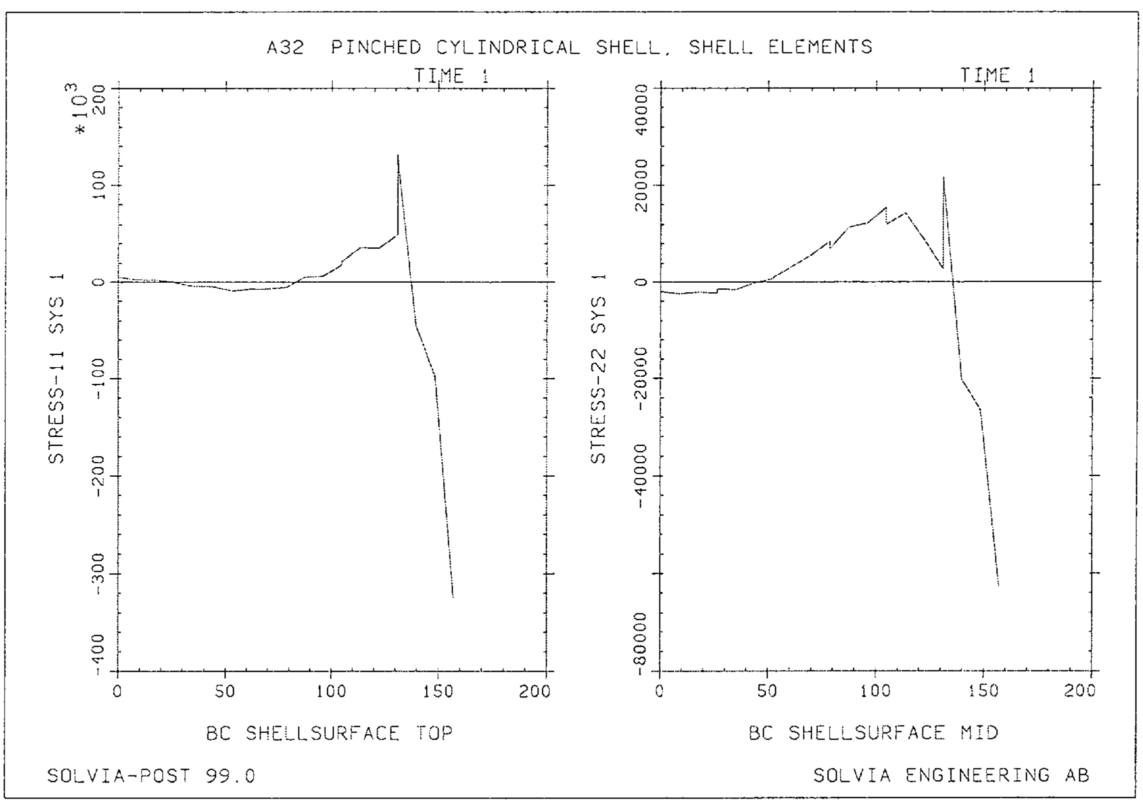
Reference

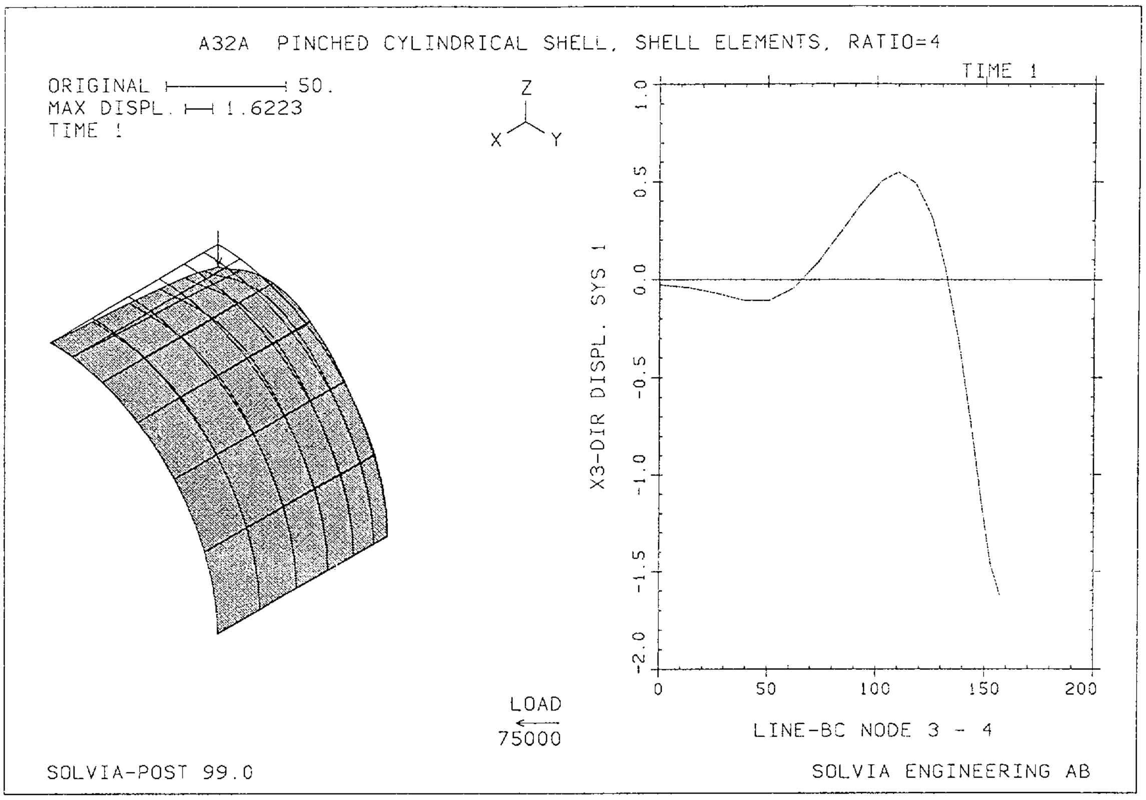
[1] Lindberg, G.M., Olson, M.D. and Cowper, E.R., "New Developments in the Finite Element Analysis of Shells", National Research Council of Canada, Quarterly Bulletin of the Division of Mechanical Engineering and the National Aeronautical Establishment, Vol.4, pp. 1-38, 1969.

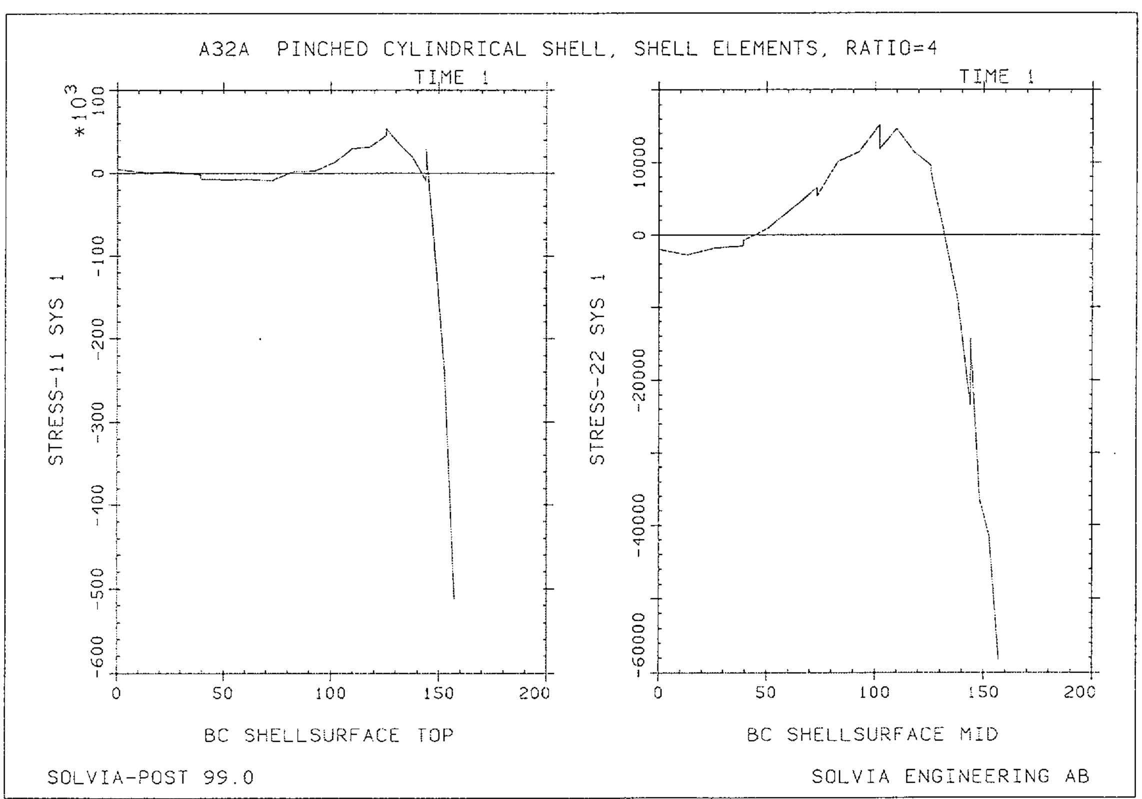


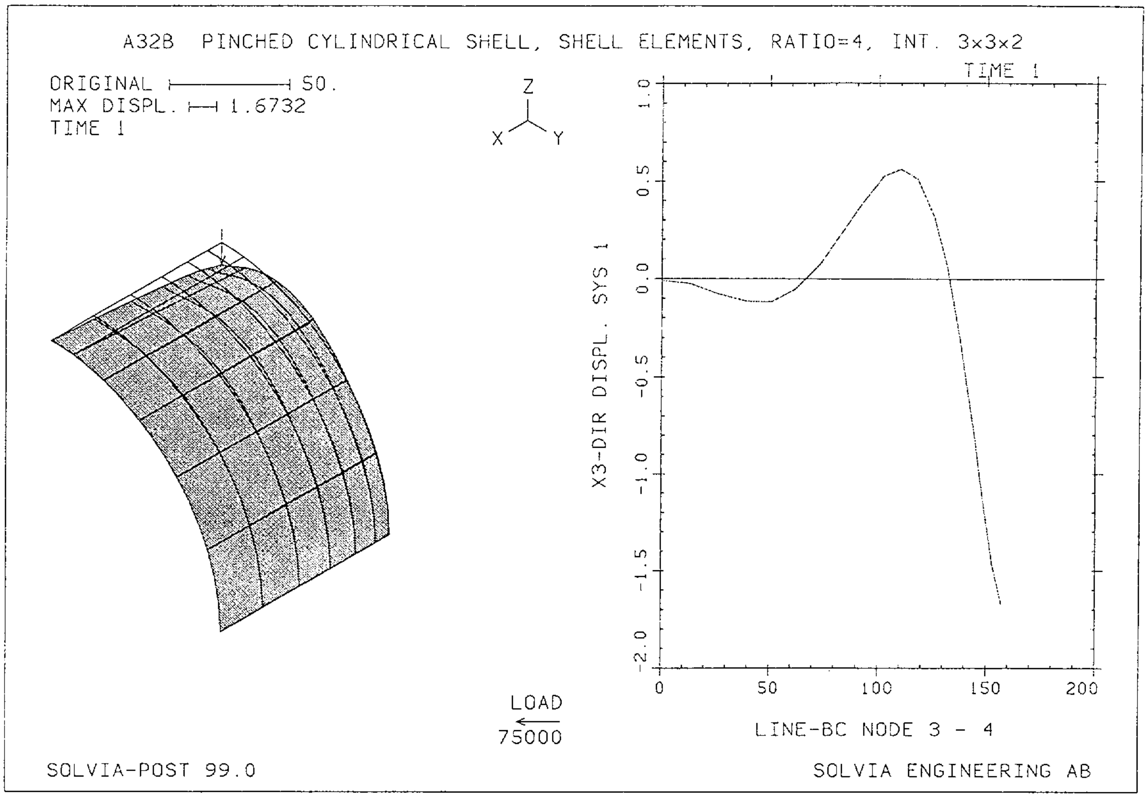


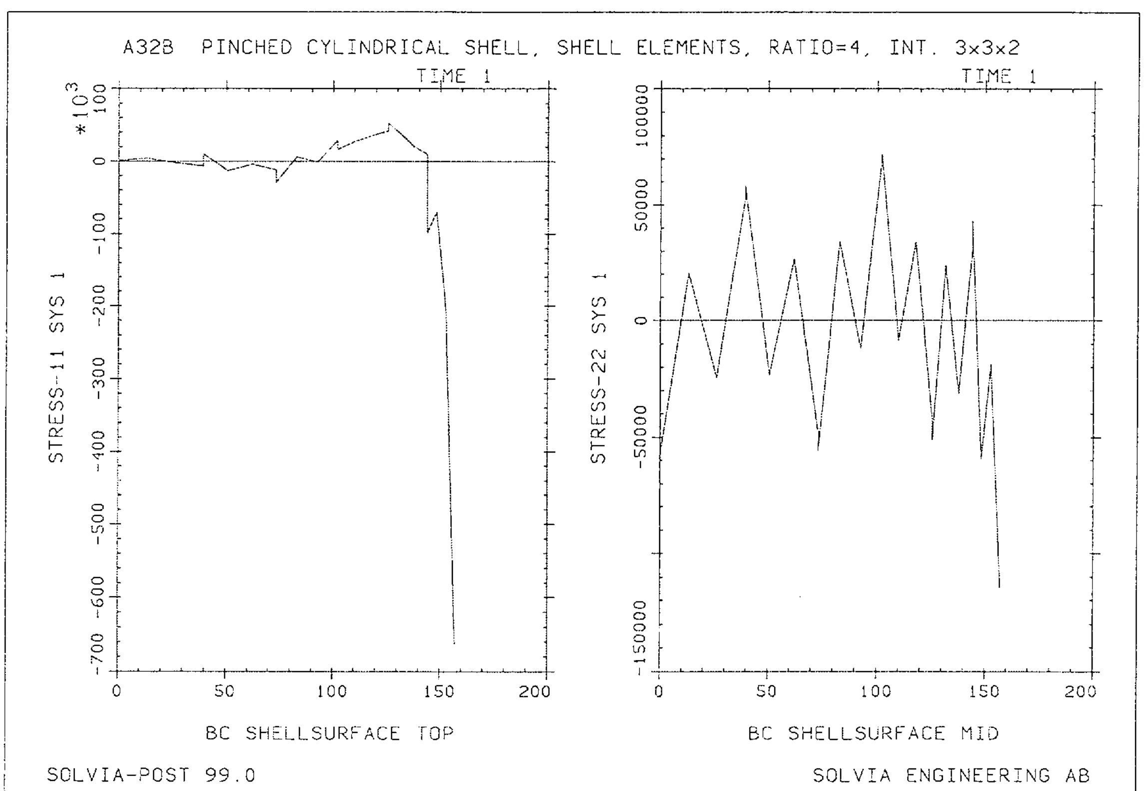

A32 PINCHED CYLINDRICAL SHELL, SHELL ELEMENTS
ORIGINAL 50.
MAX DISPL. 1.3946
TIME 1
Z
X Y
TIME 1
X3-DIR DISPL. SYS 1
1.0
0.5
0.0
-0.5
-1.0
-1.5
0
50
100
150
200
LINE-BC NODE 3 - 4
LOAD
75000
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

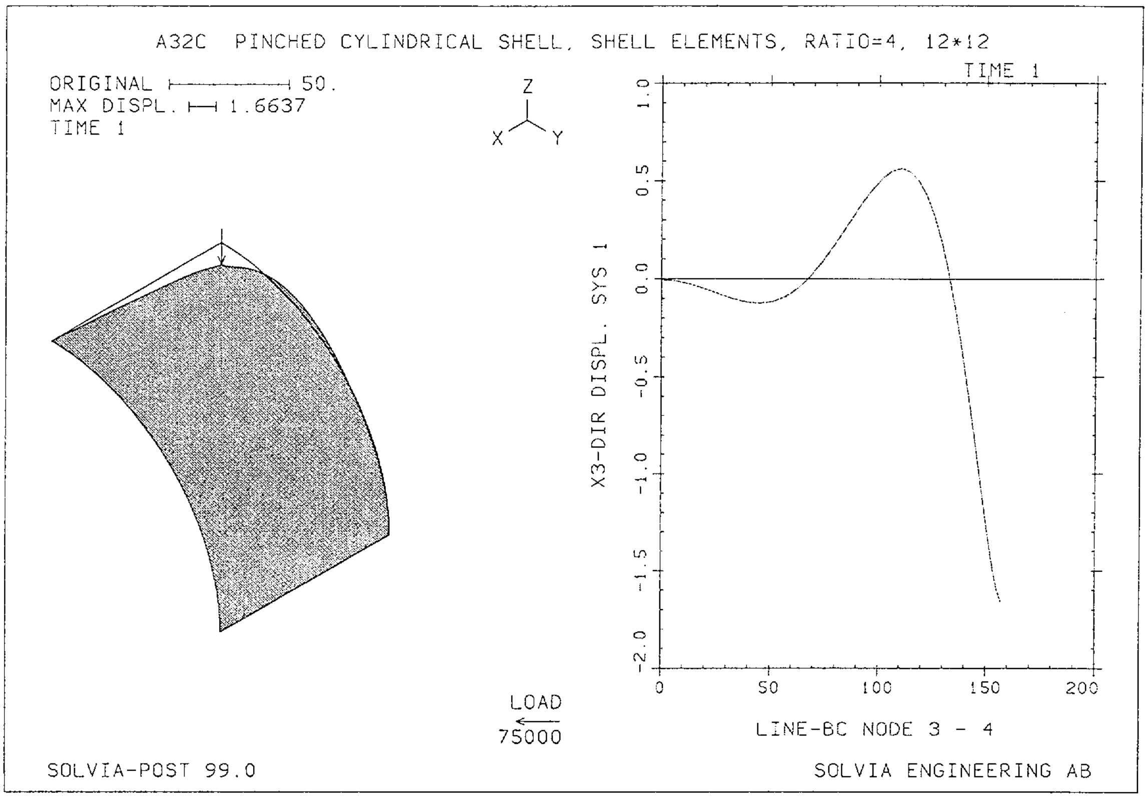

A32 PINCHED CYLINDRICAL SHELL, SHELL ELEMENTS
TIME 1
*10^3
STRESS-11 SYS 1
200
100
0
-100
-200
-300
-400
0
50
100
150
200
BC SHELLSURFACE TOP
TIME 1
STRESS-22 SYS 1
40000
20000
0
-20000
-40000
-80000
0
50
100
150
200
BC SHELLSURFACE MID
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

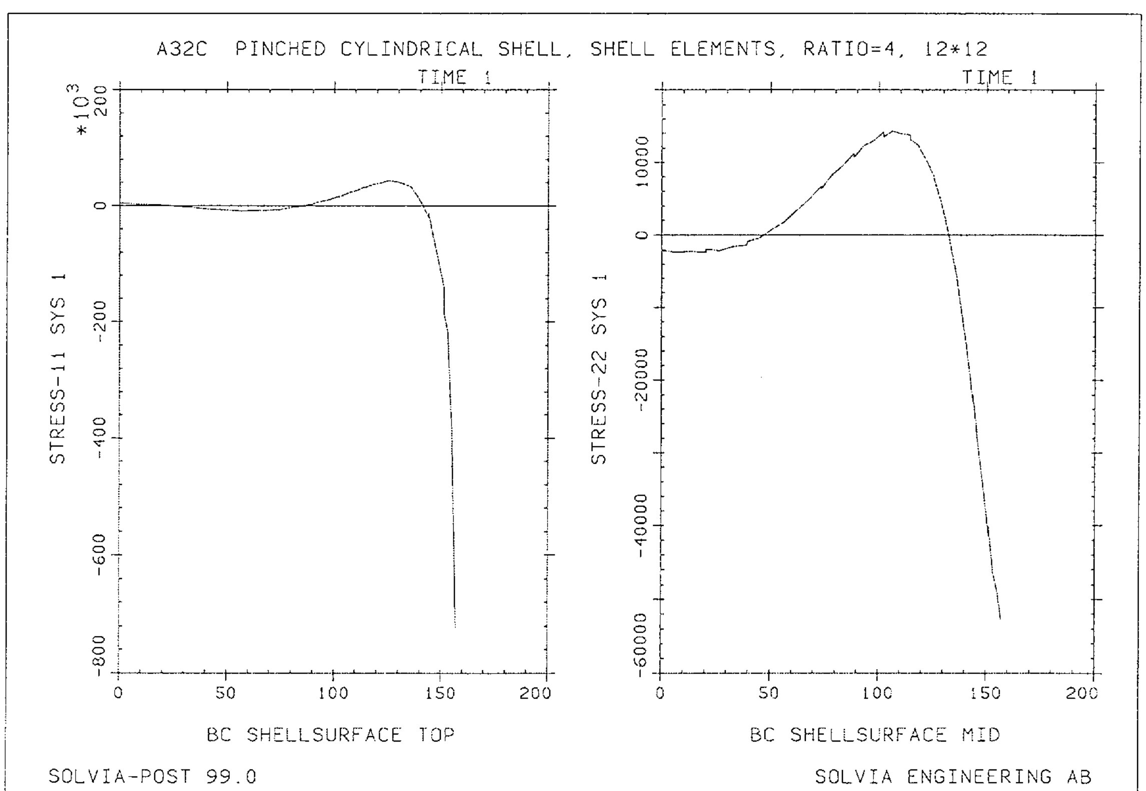

A32A PINCHED CYLINDRICAL SHELL, SHELL ELEMENTS, RATIO=4
ORIGINAL 50.
MAX DISPL. 1.6223
TIME 1
Z
X
Y
TIME 1
X3-DIR DISPL. SYS 1
1.0
0.5
0.0
-0.5
-1.0
-1.5
-2.0
0
50
100
150
200
LOAD
75000
LINE-BC NODE 3 - 4
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A32A PINCHED CYLINDRICAL SHELL, SHELL ELEMENTS, RATIO=4
TIME 1
*10^3
STRESS-11 SYS 1
100
0
-100
-200
-300
-400
-500
-600
0
50
100
150
200
BC SHELLSURFACE TOP
TIME 1
STRESS-22 SYS 1
10000
0
-20000
-40000
-60000
0
50
100
150
200
BC SHELLSURFACE MID
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A32B PINCHED CYLINDRICAL SHELL, SHELL ELEMENTS, RATIO=4, INT. 3x3x2
ORIGINAL 50.
MAX DISPL. 1.6732
TIME 1
Z
X
Y
TIME 1
X3-DIR DISPL. SYS 1
LINE-BC NODE 3 - 4
LOAD
75000
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A32B PINCHED CYLINDRICAL SHELL, SHELL ELEMENTS, RATIO=4, INT. 3x3x2
TIME 1
*10³
STRESS-11 SYS 1
BC SHELLSURFACE TOP
TIME 1
STRESS-22 SYS 1
BC SHELLSURFACE MID
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A32C PINCHED CYLINDRICAL SHELL, SHELL ELEMENTS, RATIO=4, 12*12
ORIGINAL 50.
MAX DISPL. 1.6637
TIME 1
Z
X
Y
TIME 1
X3-DIR DISPL. SYS 1
1.0
0.5
0.0
-0.5
-1.0
-1.5
-2.0
0
50
100
150
200
LINE-BC NODE 3 - 4
LOAD
75000
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A32C PINCHED CYLINDRICAL SHELL, SHELL ELEMENTS, RATIO=4, 12*12
TIME 1
*10^3
STRESS-11 SYS 1
200
0
-200
-400
-600
-800
0
50
100
150
200
BC SHELLSURFACE TOP
TIME 1
STRESS-22 SYS 1
10000
0
-20000
-40000
-60000
0
50
100
150
200
BC SHELLSURFACE MID
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A32  PINCHED CYLINDRICAL SHELL, SHELL ELEMENTS'
*
DATABASE  CREATE
*
SYSTEM  1 CYLINDRICAL
COORDINATES
 ENTRIES  NODE  R   THETA  XL
           1   100  90    100
           2   100   0    100
           3   100   0      0
           4   100  90      0
LINE  CYLINDRICAL  N1=3 N2=4 EL=6 MIDNODES=2 NFIRST=5
*
MATERIAL  1  ELASTIC  E=3.E7 NU=0.3
*
EGROUP  1  SHELL  STRESSREFERENCE=ELEMENT RESULTS=NSTRESSES
GSURFACE  1 2 3 4  EL1=6 EL2=6 NODES=16 SYSTEM=1
THICKNESS  1  1.0
*
FIXBOUNDARIES  23  INPUT=LINES  /  1 2
FIXBOUNDARIES  345 INPUT=LINES  /  2 3
FIXBOUNDARIES  156 INPUT=LINES  /  3 4
FIXBOUNDARIES  246 INPUT=LINES  /  4 1
*
LOADS  CONCENTRATED
 4 3 -75000.
*
SET  NSYMBOLS=MYNODES PLOTORIENTATION=PORTRAIT
MESH  ENUMBER=YES NNUMBERS=MYNODES VECTOR=LOAD
MESH  EAXES=STRESS-RST
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A32  PINCHED CYLINDRICAL SHELL, SHELL ELEMENTS
*
DATABASE  CREATE
*
WRITE  FILENAME='a32.lis'
*
SUBFRAME  21
MESH  ORIGINAL=YES VECTOR=LOAD
*
NPLINE  NAME=LINE-BC
 3  5 TO 21  4
*
SYSTEM 1 CYLINDRICAL
NLINE  LINENAME=LINE-BC DIRECTION=3 OUTPUT=ALL SYSTEM=1
*
EPLINE  NAME=BC
 36  3 7 11 4  TO  31  3 7 11 4
ELINE  LINENAME=BC KIND=S11 OUTPUT=ALL SYSTEM=1 SUBFRAME=21
*
SHELLSURFACE  PLOTRESULTS=MID
ELINE  LINENAME=BC KIND=S22 OUTPUT=ALL SYSTEM=1
END
```

EXAMPLE A33

PINCHED CYLINDRICAL SHELL, PLATE ELEMENTS

Objective

To verify the membrane and bending behaviour of the PLATE element when modeling a cylindrical shell surface.

Physical Problem

The same shell structure as described in Example A32 is considered, see figure on page A32.1.

Finite Element Model

As for Example A32 only one eighth of the cylinder need to be considered. The finite element model consists of 576 PLATE elements as shown in the figures on page A33.2 giving a total number of 1739 equations.

Solution Results

The input data used for the 12×12×4 diamond mesh is shown on pages A33.4 and A33.5.

The resulting deformation of the finite element model and the radial deflection along line BC is shown in the top figure on page A33.3. The membrane force in the global X-direction (x_2 direction of the Local Cylindrical System) and bending moment M_{11} (about global X-direction corresponding to bending stresses in the circumferential direction, the x_1 direction) at the stresstable points on the line BC are displayed in the bottom figure on page A33.3.

The theoretical solution is given in [1].

δ_{zc} = displacement in Z-direction at point C.

σ_{xx} = membrane stress at point C in global X-direction.

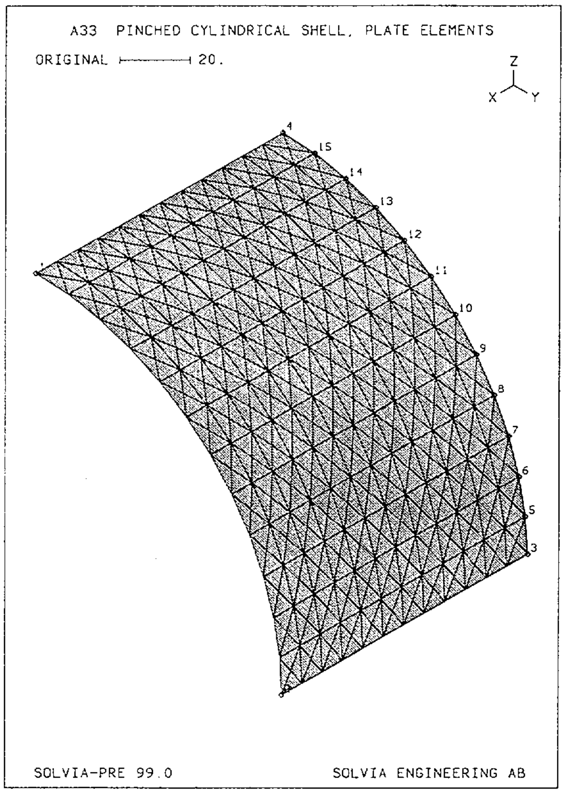
δ_{zc} [in]		σ_{xx} [psi]	
Theory	SOLVIA	Theory	SOLVIA
-1.642	-1.619	$-4.72 \cdot 10^4$	$-3.30 \cdot 10^4$

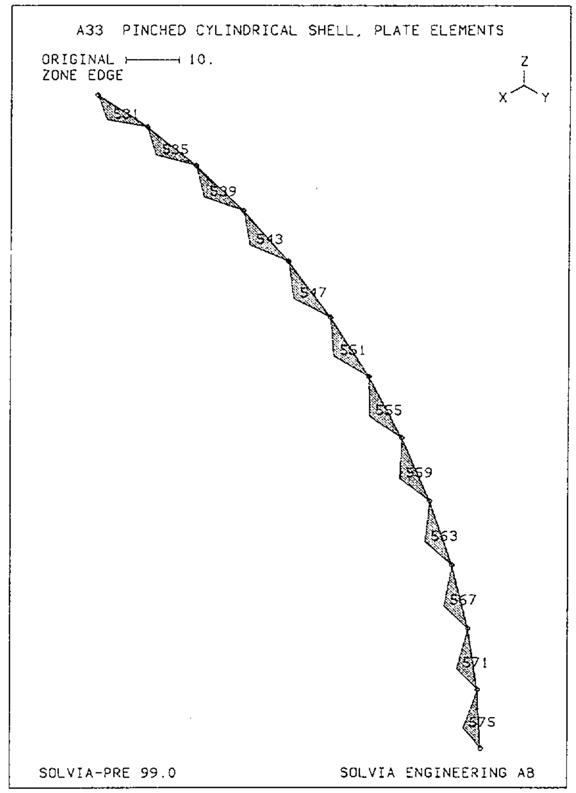
User Hints

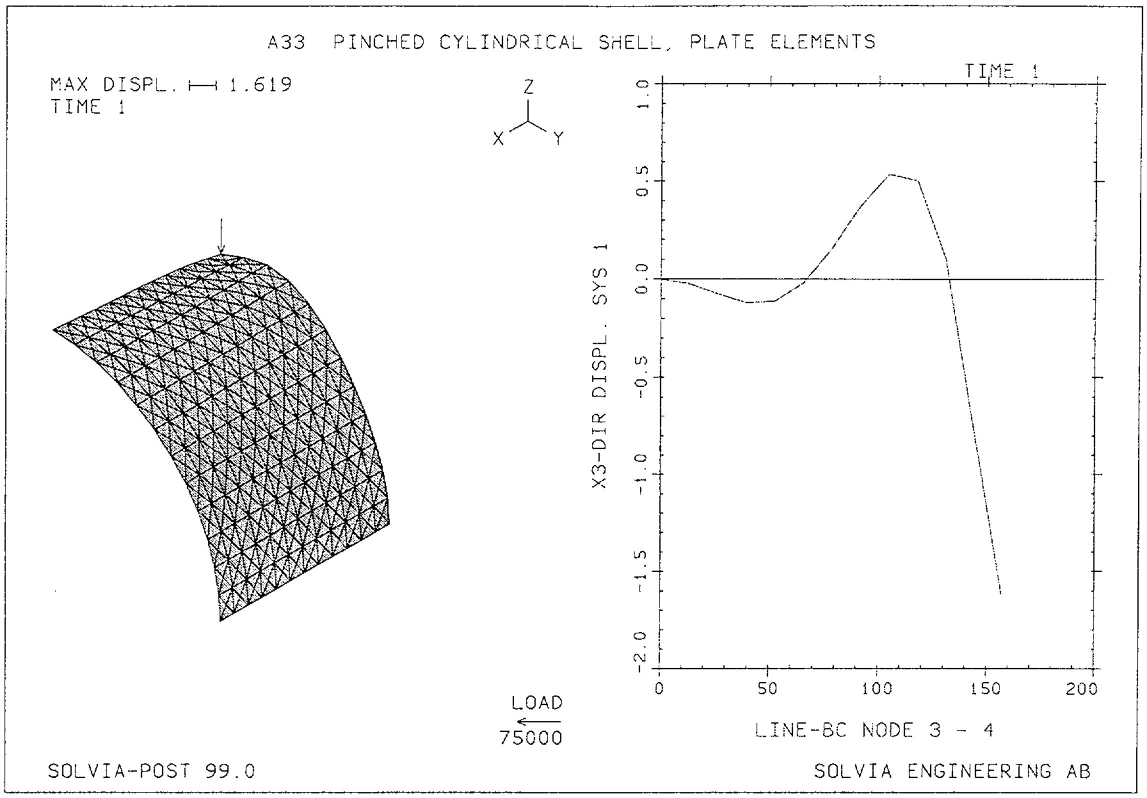
- As discussed already for Examples A30 and A31, a larger number of elements is required in general when modeling a curved shell structure by PLATE elements.

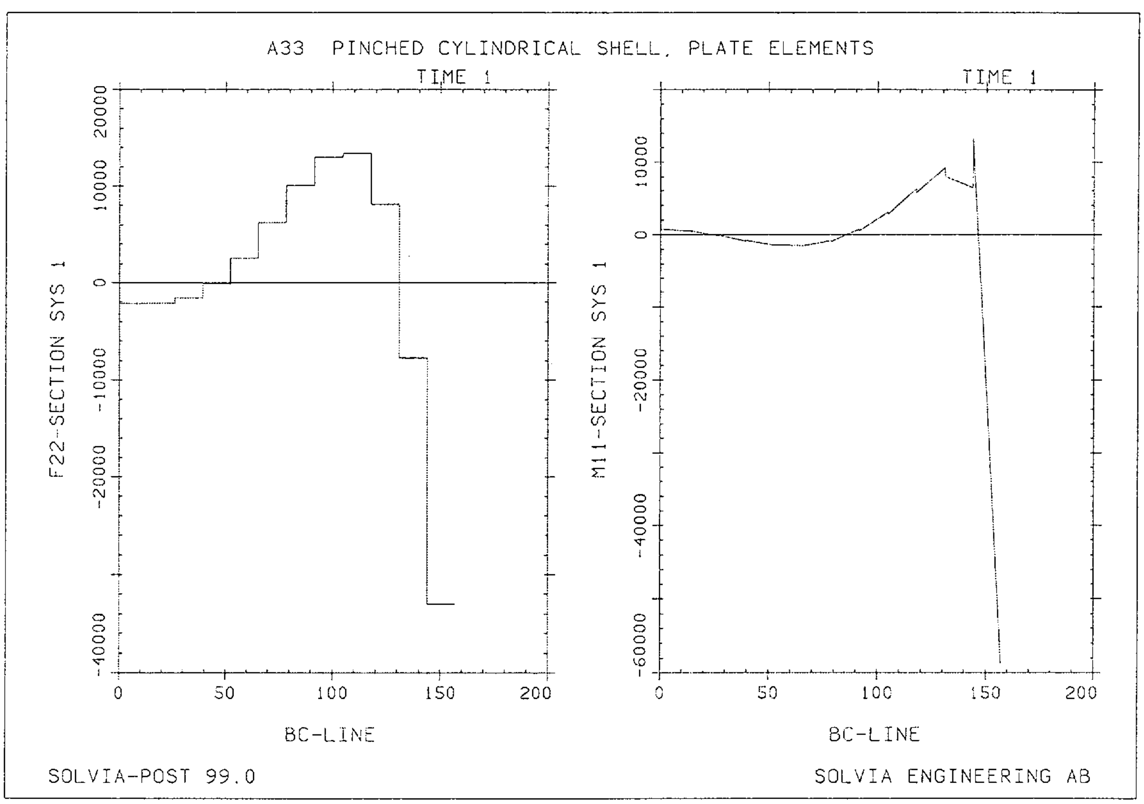
Reference

[1] Lindberg, G.M., Olson, M.D. and Cowper, E.R., "New Developments in the Finite Element Analysis of Shells", National Research Council of Canada, Quarterly Bulletin of the Division of Mechanical Engineering and the National Aeronautical Establishment, Vol. 4, pp. 1-38, 1969.






A33 PINCHED CYLINDRICAL SHELL, PLATE ELEMENTS
MAX DISPL. 1.619
TIME 1
Z
X
Y
TIME 1
X3-DIR DISPL. SYS 1
1.0
0.5
0.0
-0.5
-1.0
-1.5
-2.0
0
50
100
150
200
LINE-BC NODE 3 - 4
LOAD
75000
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A33 PINCHED CYLINDRICAL SHELL, PLATE ELEMENTS
TIME 1
TIME 1
F22-SECTION SYS 1
20000
10000
0
-10000
-20000
-40000
0
50
100
150
200
BC-LINE
M11-SECTION SYS 1
10000
0
-20000
-40000
-60000
0
50
100
150
200
BC-LINE
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A33  PINCHED CYLINDRICAL SHELL, PLATE ELEMENTS'
*
DATABASE  CREATE
*
SYSTEM  1 CYLINDRICAL
COORDINATES
 ENTRIES  NODE  R  THETA  XL
           1   100  90   100
           2   100   0   100
           3   100   0     0
           4   100  90     0
LINE  CYLINDRICAL  N1=3 N2=4 EL=12 NFIRST=5
*
MATERIAL  1  ELASTIC  E=3.E7 NU=0.3
*
EGROUP  1  PLATE
GSURFACE  1 2 3 4  EL1=12 EL2=12 SYSTEM=1
EDATA  /  1 1.0
STRESSTABLE  1  1 2 3 4 5 6 7
*
FIXBOUNDARIES  23  INPUT=LINES  /  1 2
FIXBOUNDARIES  345 INPUT=LINES  /  2 3
FIXBOUNDARIES  156 INPUT=LINES  /  3 4
FIXBOUNDARIES  246 INPUT=LINES  /  4 1
*
LOADS  CONCENTRATED
 4 3 -75000.
*
SET  PLOTORIENTATION=PORTRAIT
MESH  NSYMBOLS=MYNODES NNUMBERS=MYNODES
ZONE  NAME=EDGE INPUT=GLOBAL-LIMITS XMAX=5
MESH  ZONENAME=EDGE ENUMBER=YES NSYMBOLS=MYNODES
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A33  PINCHED CYLINDRICAL SHELL, PLATE ELEMENTS
*
DATABASE  CREATE
*
WRITE  FILENAME='a33.lis'
*
SUBFRAME  21
MESH  VECTOR=LOAD
*
NPLINE  NAME=LINE-BC
 3  5 TO 15  4
*
SYSTEM 1 CYLINDRICAL
NLINE  LINENAME=LINE-BC DIRECTION=3 OUTPUT=ALL SYSTEM=1
*
EPLINE  NAME=BC-LINE
 575  1 4 2  STEP -4 TO  531  1 4 2
*
ELINE  LINENAME=BC-LINE KIND=F22 OUTPUT=ALL SYSTEM=1 SUBFRAME=21
ELINE  LINENAME=BC-LINE KIND=M11 OUTPUT=ALL SYSTEM=1
END
```

EXAMPLE A34

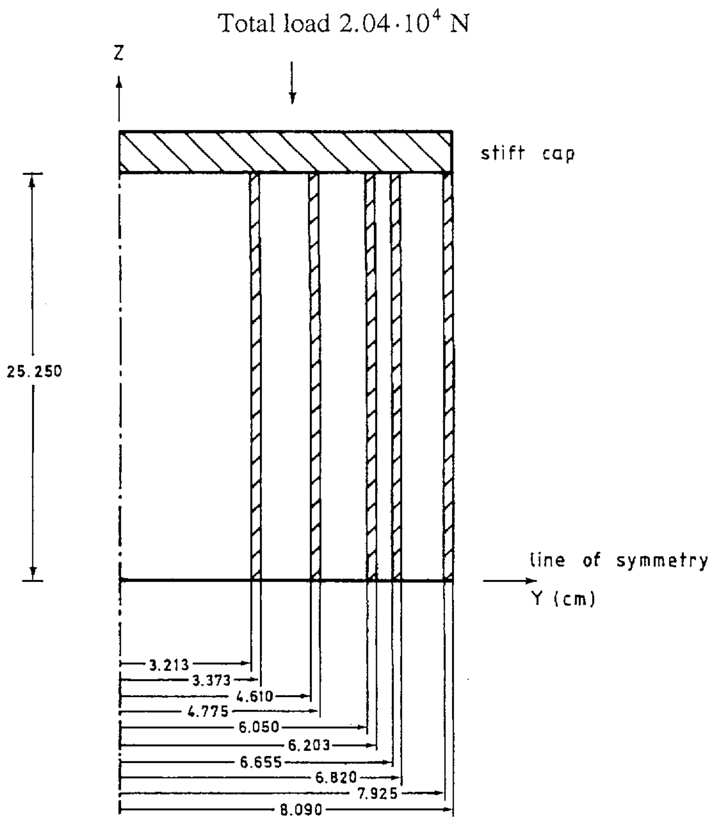
ANALYSIS OF CONCENTRIC FLUID-FILLED CYLINDERS

Objective

To verify the behaviour of the FLUID2 element in a fluid-structure static analysis.

Physical Problem

Five concentric fluid-filled cylinders are analyzed for an axial load applied to the stiff end cap as shown in the figure below.



Cylinders (6351-T6 aluminum)
Young's modulus $E = 6.895 \cdot 10^{10}$ N/m^2
Poisson's ratio $\nu = 0.33$

Fluid (Light hydraulic oil)
Bulk modulus $K = 1.64 \cdot 10^9$ N/m^2

Finite Element Model

Because of symmetry only one half of the cylinders need to be considered in the axisymmetric model shown in the figures on page A34.2. The finite element model consists of 8-node PLANE and FLUID2 AXISYMMETRIC elements. Fluid and structural elements are modeled with separate nodes. The nodal displacement in the Y-direction for adjacent structural and fluid element nodes are constrained. The stiff cap is simulated using rigid links to the cylinder nodes. The top fluid nodes are constrained to move in the Z-direction with the top cylinder nodes. Irrotational displacement conditions are enforced in the fluid elements.

The total load of 20400 N corresponds to the axisymmetric model load of 3246.76 N since the circumferential extension is 1 radian.

Solution Results

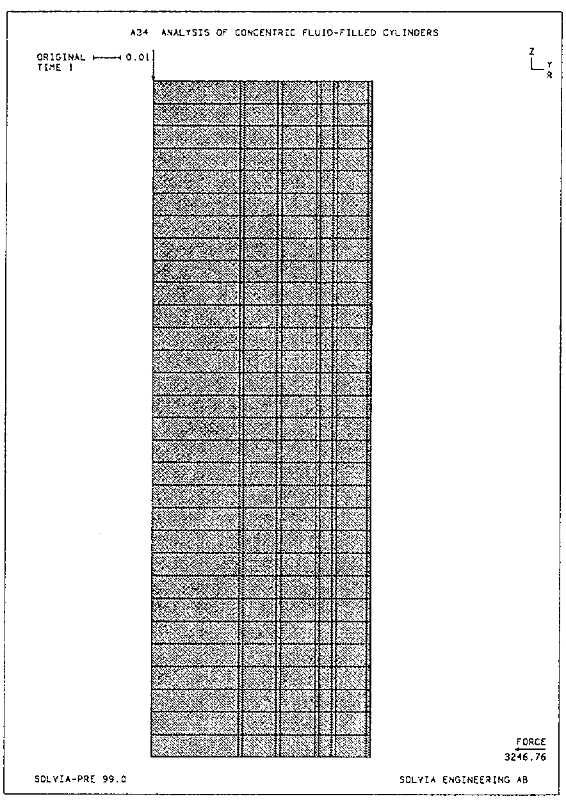
The input data on page A34.3 - A34.5 was used in the finite element analysis. The calculated fluid pressures are compared with the experimental pressures presented in [1].

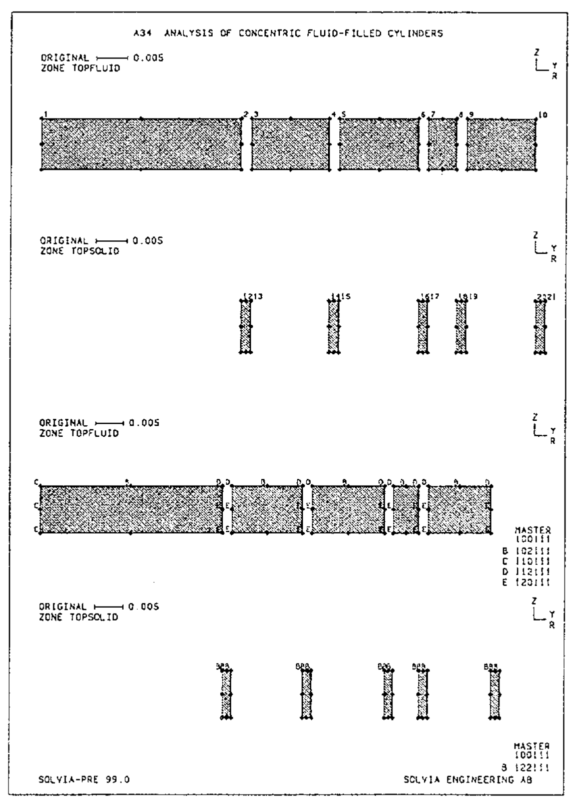
Cylinder	Pressure $[N/m^2]$ Experimental	SOLVIA
1	$5.57 \cdot 10^4$	$5.23 \cdot 10^4$
2	$4.52 \cdot 10^4$	$4.28 \cdot 10^4$
3	$3.40 \cdot 10^4$	$3.32 \cdot 10^4$
4	$2.53 \cdot 10^4$	$2.33 \cdot 10^4$
5	$1.41 \cdot 10^4$	$1.25 \cdot 10^4$

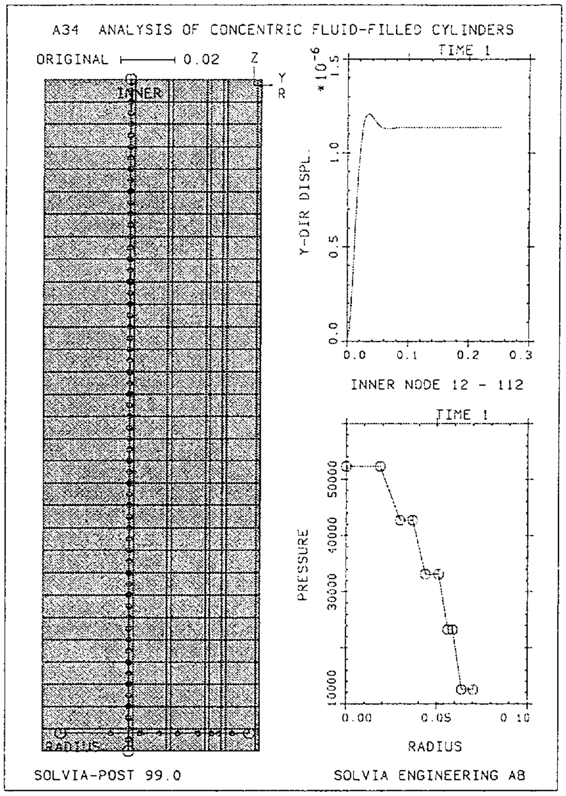
The deformed configuration as well as the radial displacement along the cylinder at radius Y = 3.213 cm (line INNER) and the pressure variation along the line RADIUS are displayed by SOLVIA-POST in the top figures on page A34.3. Note that meter is used in the input data for this example.

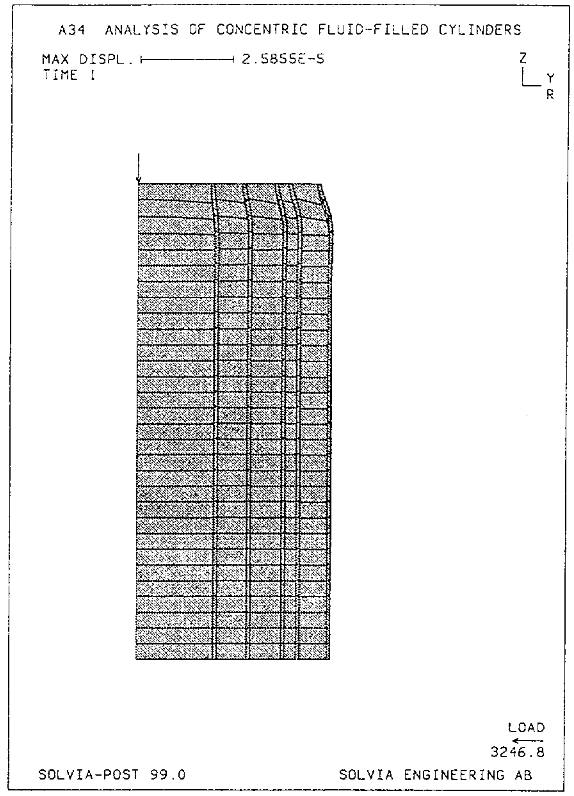
Reference

[1] Munro, M. and Piekarski, K., "Stress Induced Radial Pressure Gradients in Liquid-Filled Multiple Concentric Cylinders", J. of Appl. Mech., Vol. 44, No. 2, pp. 218-221, 1977.









SOLVIA-PRE input

```
HEADING  'A34  ANALYSIS OF CONCENTRIC FLUID-FILLED CYLINDERS'
*
DATABASE  CREATE
MASTER  IDOF=100111
SET NODES=8 MYNODES=200
PARAMETER $EL=30
*
COORDINATES / ENTRIES  NODE    Y
* fluid top nodes  1 - 10:
 1          /  2  .03213  /   3  .03373  /  4  .04610
 5  .04775  /  6  .06050  /   7  .06203  /  8  .06655
 9  .06820  / 10  .07925
* structure top nodes 12 - 20:
12  .03213  / 13  .03373  / 14  .04610  / 15  .04775
16  .06050  / 17  .06203  / 18  .06655  / 19  .06820
20  .07925  / 21  .08090
* fluid and structure bottom nodes:
NGENERATION  ZSTEP=-.25250 NSTEP=100  /   1 TO 21
*
MATERIAL  1  ELASTIC E=6.895E10 NU=0.33
MATERIAL  2  FLUID   K=1.64E9
*
EGROUP  1  PLANE AXISYMMETRIC  MATERIAL=1
GSURFACE  13 12 112 113  EL2=$EL
GSURFACE  15 14 114 115  EL2=$EL
GSURFACE  17 16 116 117  EL2=$EL
GSURFACE  19 18 118 119  EL2=$EL
GSURFACE  21 20 120 121  EL2=$EL
*
```

SOLVIA-PRE input (cont.)

```
EGROUP  2  FLUID2 AXISYMMETRIC  MATERIAL=2 IRROTATIONALITY=YES
GSURFACE  2  1 101 102  EL2=$EL NCOINCIDE=NO
GSURFACE  4  3 103 104  EL2=$EL NCOINCIDE=NO
GSURFACE  6  5 105 106  EL2=$EL NCOINCIDE=NO
GSURFACE  8  7 107 108  EL2=$EL NCOINCIDE=NO
GSURFACE 10  9 109 110  EL2=$EL NCOINCIDE=NO
*
CONSTRAINTS INPUT=LINES M-INPUT=LINES
* sn1 sn2 sdir mn1 mn2 mdir
    2  102  2   12  112  2
    3  103  2   13  113  2
    4  104  2   14  114  2
    5  105  2   15  115  2
    6  106  2   16  116  2
    7  107  2   17  117  2
    8  108  2   18  118  2
    9  109  2   19  119  2
   10  110  2   20  120  2
CONSTRAINTS INPUT=NODES / DELETE 1 TO 10
CONSTRAINTS INPUT=LINES
* n1 n2 dir master mdir
    1  2  3     1     3
    3  4  3     1     3
    5  6  3     1     3
    7  8  3     1     3
    9 10  3     1     3
*
RIGIDLINK INPUT=LINES
* n1  n2  master
   12  13    1
   14  15    1
   16  17    1
   18  19    1
   20  21    1
*
FIXBOUNDARIES  2  INPUT=LINE /  1 101
FIXBOUNDARIES  2  INPUT=NODES / 2 TO 10
ZONE SYM GLOBAL-LIMITS ZMAX=-.25
FIXBOUNDARIES  3  INPUT=ZONE ZONE1=SYM
*
LOADS  CONCENTRATED  /  1  3 -3246.76084
*
SET  PLOTORIENTATION=PORTRAIT
MESH VECTOR=LOAD
ZONE TOP GLOBAL-LIMITS ZMIN=-0.01
ZONE TOPFLUID OPERATION=ADD ZONE1=TOP
ZONE TOPSOLID OPERATION=ADD ZONE1=TOP
ZONE TOPFLUID OPERATION=INTERSECT ZONE1=FLUID2
ZONE TOPSOLID OPERATION=INTERSECT ZONE1=PLANE
SUBFRAME 14
MESH TOPFLUID NNUM=MY NSYM=YES
MESH TOPSOLID NNUM=MY NSYM=YES GSCALE=OLD
MESH TOPFLUID NSYM=YES BCODE=ALL
MESH TOPSOLID NSYM=YES BCODE=ALL GSCALE=OLD
*
LIST LINE 112 12
SOLVIA
END
```

SOLVIA-POST input

```
*  A34  ANALYSIS OF CONCENTRIC FLUID-FILLED CYLINDERS
*
DATABASE  CREATE
WRITE  FILENAME='a34.lis'
*
SET  PLOTORIENTATION=PORTRAIT
*
NPLINE  NAME=INNER
 12 202 TO 260 112
*
EGROUP  2
EPLINE  NAME=RADIUS
 30  2 4 STEP 30 TO  150  2 4
*
MESH PLINES=ALL SUBFRAME=2111
NLINE LINENAME=INNER DIRECTION=2 SUBFRAME=2222
ELINE LINENAME=RADIUS KIND=PRESSURE SYMBOL=1 SUBFRAME=2221
*
MESH DMAX=3 VECTOR=LOAD
*
ZONE  NAME=FLUID INPUT=ELEMENT
 30 STEP 30 TO 150
ELIST  ZONENAME=FLUID
*
END
```

EXAMPLE A35

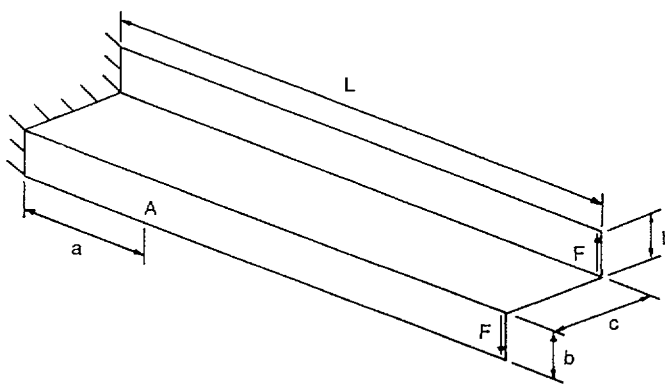
Z-SECTION CANTILEVER UNDER DISTRIBUTED EDGE LOAD

Objective

To verify the membrane behaviour of the SHELL element when subjected to distributed edge loading.

Physical Problem

A Z-section cantilever is subjected to pure torsional loading at its free end as seen in the figure below. At the end X=0, the cantilever is rigidly built-in. This problem is further described in [1] page 35.



Thickness 0.1 m

L = 10.0 m
a = 2.5 m
F = 0.6 MN
b = 1.0 m
c = 2.0 m

Elastic material

$E = 2.1 \cdot 10^{11}\ N/m^2$
$\nu = 0.3$

Finite Element Model

A uniform mesh of 24 cubic SHELL elements are used in the model as seen in the figure on page A35.2. The edge load is defined by SHELL line forces acting in the local SHELL edge direction.

Solution Results

The theoretical solution for this problem is given in [1], pp. 35-41. The stress distribution over the cross-section is defined at a point located 2.5 m along the cantilever from the built-in end. The axial membrane stress varies linearly over the flanges and is constant over the web. The membrane shear stress varies quadratically over the flanges and varies linearly over the web.

Using the input data shown on page A35.5 the following results are obtained:

Line coord.	Cubic SHELL		Analytical [MPa]	
	Stress-rr	Stress-rs	Stress-rr	Stress-rs
0.	-110.3	-0.07	-107.9	0.
1.	36.5	-5.96	36.0	-5.85
3.	36.5	5.96	36.0	5.85
4.	-110.3	0.07	-107.9	0.

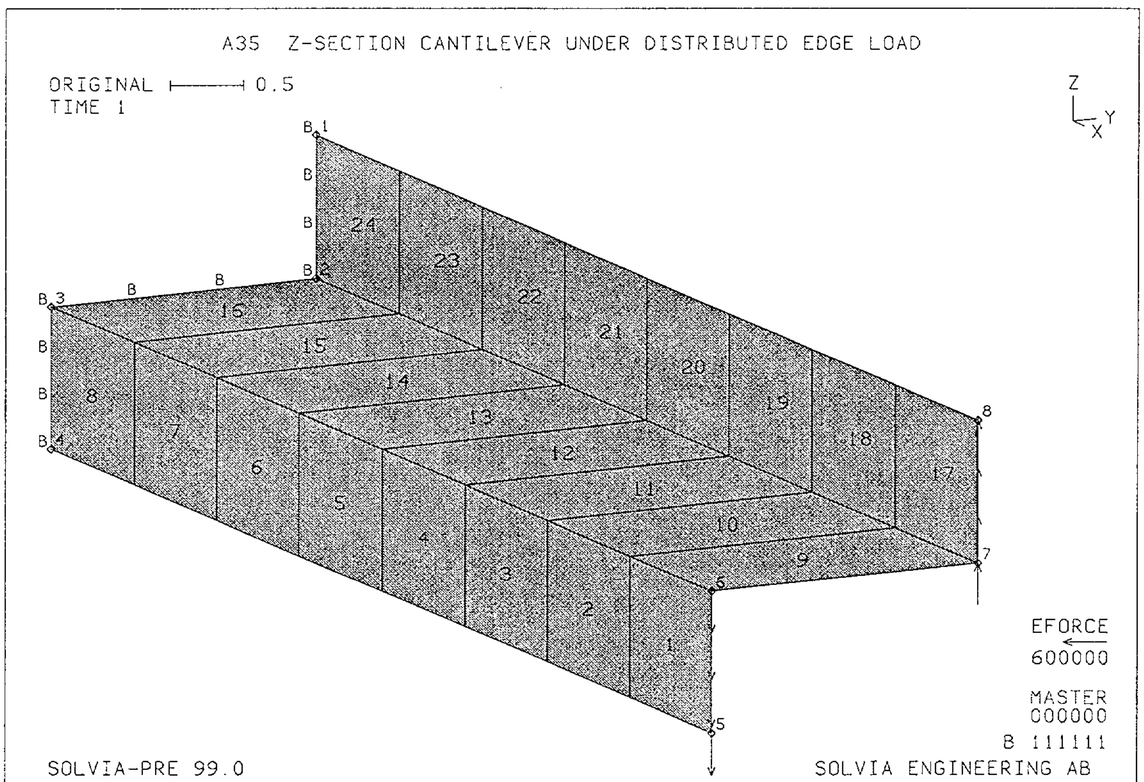
The contour plot of stress-rr and the stress distribution at X = 2.5 m can be seen in figures on page A35.3. The cubic SHELL element performs very well in this example and the quadratic distribution of the shear stresses is in good agreement with the analytical solution.

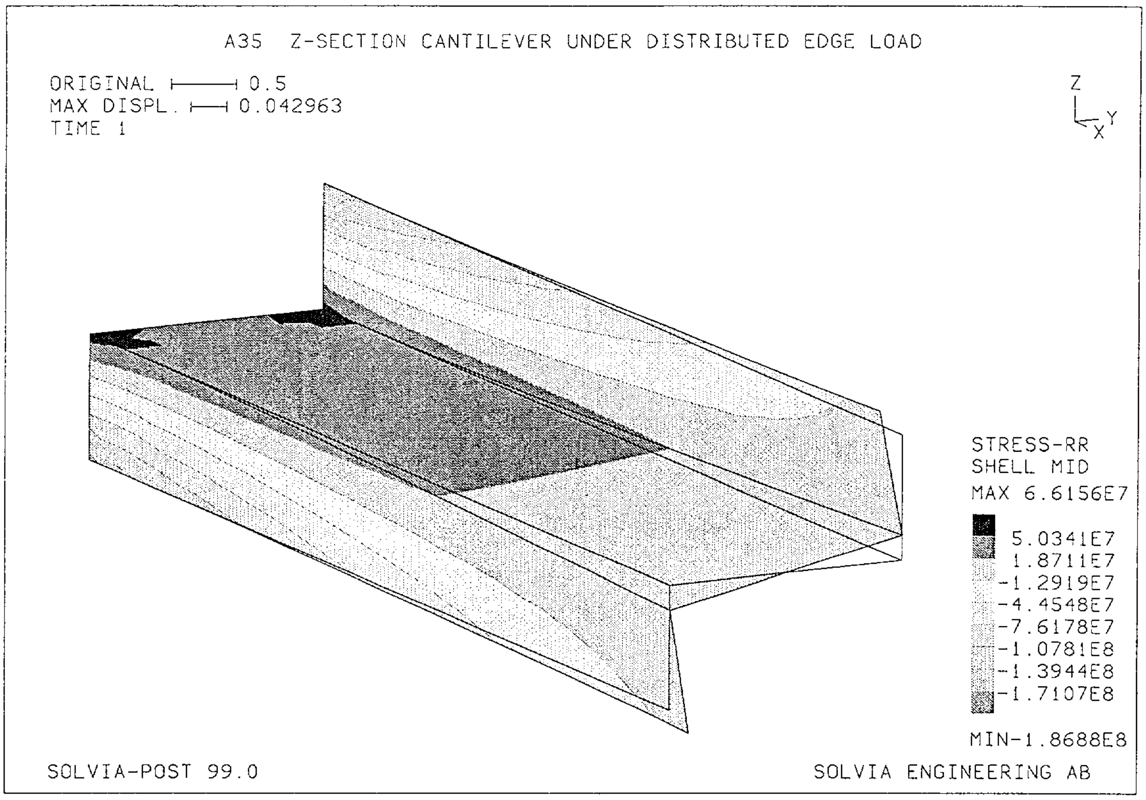
User Hints

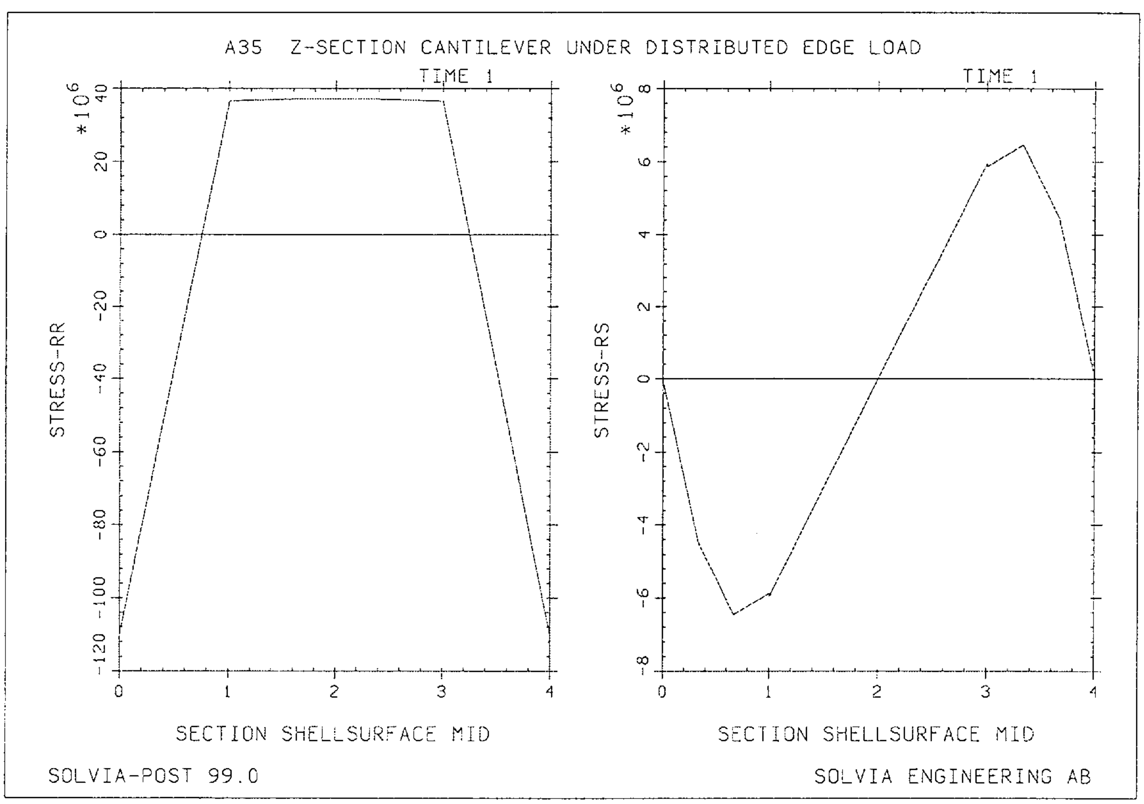
- The results of an analysis using the 9-node SHELL element can be found on page A35.4. The distribution of the direct stress, stress-rr, is in good agreement with the analytical solution, but the shear stress distribution is relatively poor.

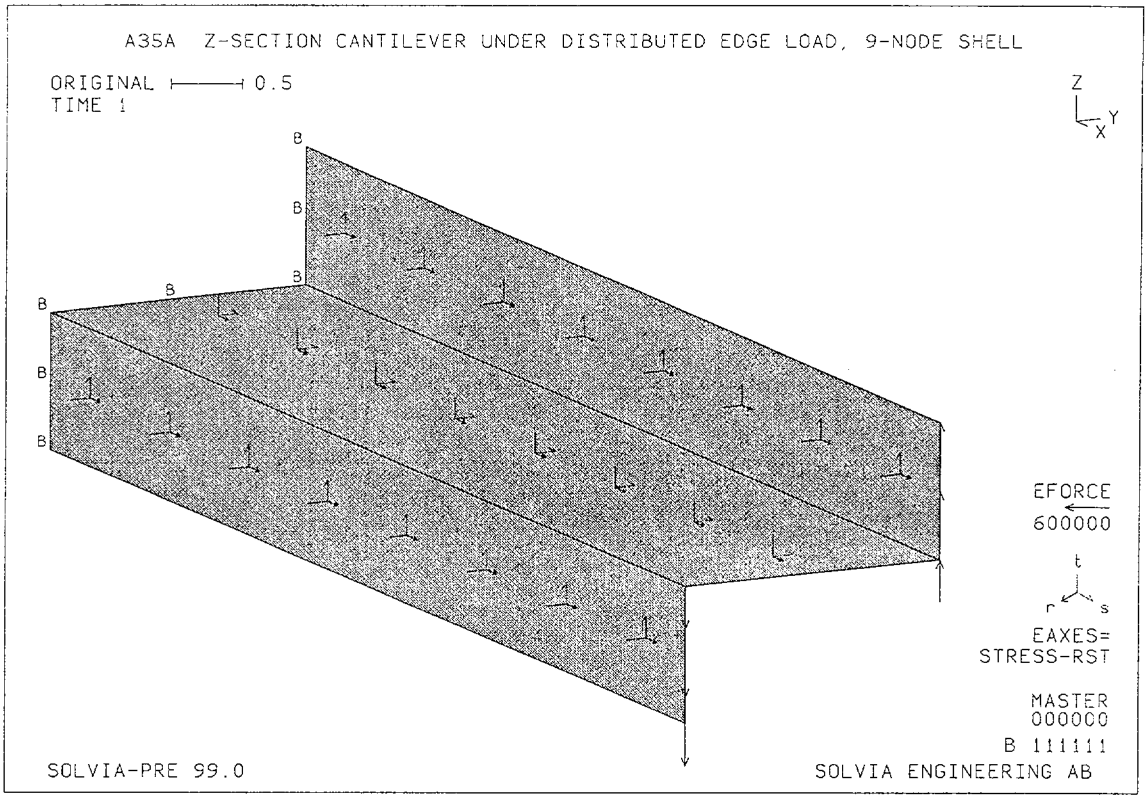
Reference

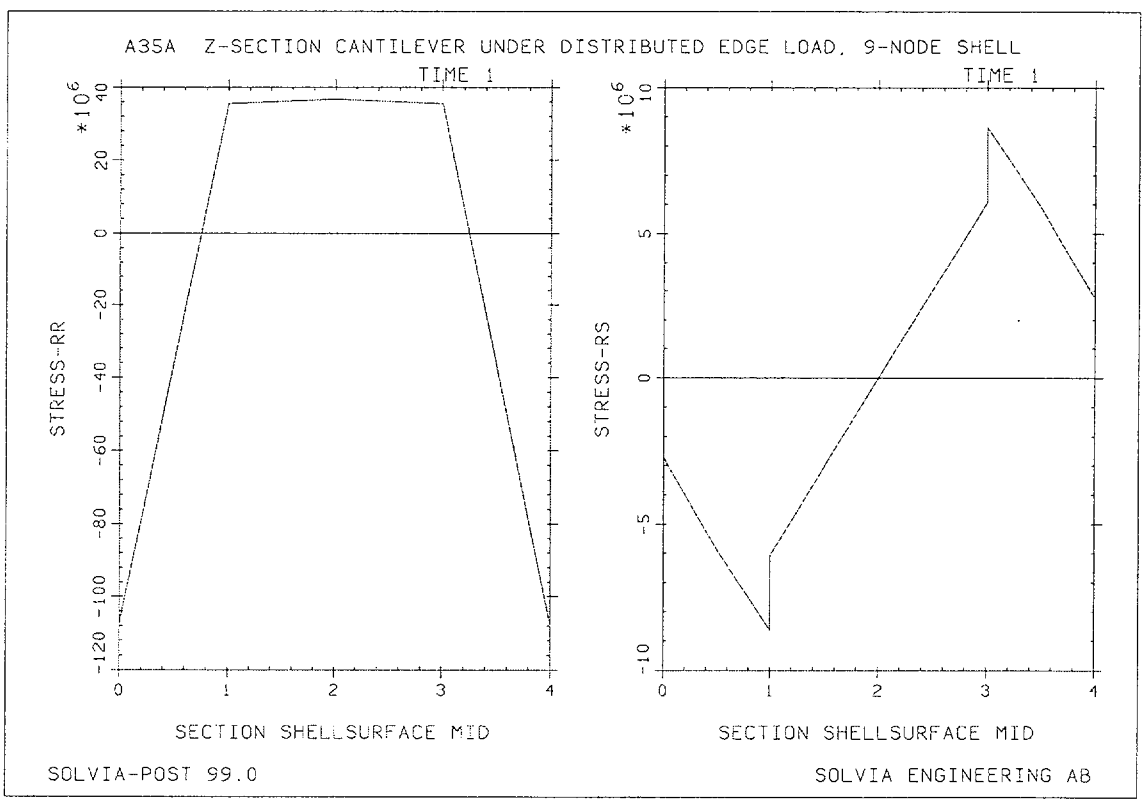
[1] NAFEMS, Background to Benchmarks, 1993.




A35 Z-SECTION CANTILEVER UNDER DISTRIBUTED EDGE LOAD
ORIGINAL 0.5
MAX DISPL. 0.042963
TIME 1
Z
Y
X
STRESS-RR
SHELL MID
MAX 6.6156E7
5.0341E7
1.8711E7
-1.2919E7
-4.4548E7
-7.6178E7
-1.0781E8
-1.3944E8
-1.7107E8
MIN-1.8688E8
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A35 Z-SECTION CANTILEVER UNDER DISTRIBUTED EDGE LOAD
TIME 1
*10^6
STRESS-RR
SECTION SHELLSURFACE MID
TIME 1
*10^6
STRESS-RS
SECTION SHELLSURFACE MID
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A35A Z-SECTION CANTILEVER UNDER DISTRIBUTED EDGE LOAD, 9-NODE SHELL
ORIGINAL 0.5
TIME 1
Z
Y
X
EFORCE
600000
t
r
s
EAXES=
STRESS-RST
MASTER
000000
B 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A35A Z-SECTION CANTILEVER UNDER DISTRIBUTED EDGE LOAD, 9-NODE SHELL
TIME 1
*10^6
STRESS-RR
SECTION SHELLSURFACE MID
TIME 1
*10^6
STRESS-RS
SECTION SHELLSURFACE MID
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A35  Z-SECTION CANTILEVER UNDER DISTRIBUTED EDGE LOAD'
*
DATABASE  CREATE
*
COORDINATES
  1   0.  1.  1.  /  2   0.  1.  0.
  3   0. -1.  0.  /  4   0. -1. -1.
  5  10. -1. -1.  /  6  10. -1.  0.
  7  10.  1.  0.  /  8  10.  1.  1.
*
MATERIAL  1  ELASTIC  E=210.E9 NU=0.3
*
EGROUP  1  SHELL  RESULTS=NSTRESSES STRESSREFERENCE=ELEMENT
THICKNESS  1  0.1
GSURFACE  6 3 4 5  EL1=8 EL2=1 NODES=16
GSURFACE  7 2 3 6  EL1=8 EL2=1 NODES=16
GSURFACE  8 1 2 7  EL1=8 EL2=1 NODES=16
LOADS  ELEMENT  TYPE=FORCE INPUT=LINE
 5 6 edge  0.6E6  0.6E6
 7 8 edge -0.6E6 -0.6E6
*
FIXBOUNDARIES  INPUT=LINE
 1 2  /  2 3  /  3 4
*
VIEW  ID=1  XVIEW=2. YVIEW=-1. ZVIEW=0.5
SET  VIEW=1
MESH  ENUMBERS=YES NNUMBERS=MYNODES NSYMBOLS=MYNODES,
      VECTOR=LOAD  BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A35  Z-SECTION CANTILEVER UNDER DISTRIBUTED EDGE LOAD
*
DATABASE  CREATE
*
WRITE  FILENAME='a35.lis'
*
SHELLSURFACE  PLOTRESULTS=MID
VIEW  ID=1  XVIEW=2. YVIEW=-1. ZVIEW=0.5
MESH  VIEW=1 OUTLINE=SHELL CONTOUR=SRR ORIGINAL=YES
*
EPLINE  NAME=SECTION
 6  3 10 6 2  /  14  3 10 6 2  /  22  3 10 6 2
ELINE  LINENAME=SECTION KIND=SRR OUTPUT=ALL SUBFRAME=21
ELINE  LINENAME=SECTION KIND=SRS OUTPUT=ALL
*
NMAX  DIRECTION=123 NUMBER=2
END
```

EXAMPLE A36

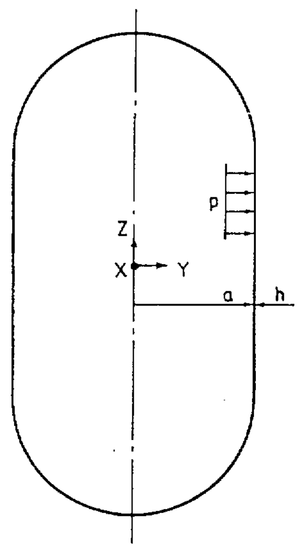
CYLINDRICAL PRESSURE VESSEL WITH HEMISPHERICAL ENDS

Objective

To verify the PLANE AXISYMMETRIC element under distributed loading when used for axisymmetric shell bending problems.

Physical Problem

A pressurized cylinder is capped by hemispheres of the same thickness as shown in the figure below. The radius to thickness ratio is 40 so thin shell theory is applicable. This problem is described in [1].



$E = 2.1 \cdot 10^{11}\ \text{N/m}^2$

$\nu = 0.3$

$a = 1.0\text{m}$ (mean radius)

$h = 0.025\text{m}$ (thickness)

$p = 1 \cdot 10^{6}\ \text{N/m}^2$

Finite Element Model

Due to symmetry only half of the pressure vessel is modeled using 38 PLANE AXISYMMETRIC elements as shown in the bottom figure on page A36.2.

Most of the local bending is concentrated to the joint between the cylinder and the cap and a fine mesh is used for that part. The element meshing of the model is performed as described in [1].

Solution Results

The theoretical solution for this problem can be found in [2] p. 484. Combining the maximum bending stress and the membrane stress at the outer surface we get

$$\text{stress-rr} = \frac{1.293 \cdot a \cdot p_a}{2h} = 25.22 \cdot 10^{6}\ \text{N/m}^2 \qquad \text{where } p_a = \frac{p}{C^2} \text{ and } C = \frac{a}{a - h/2},$$

where p is the internal pressure applied at radius a - h/2. The factor C is introduced to calculate equivalent loads acting at the midsurface, see Example A2.

SOLVIA calculates maximum axial stress at nodal point 7 to be stress-rr = $25.3 \cdot 10^6$ N/m^2 .

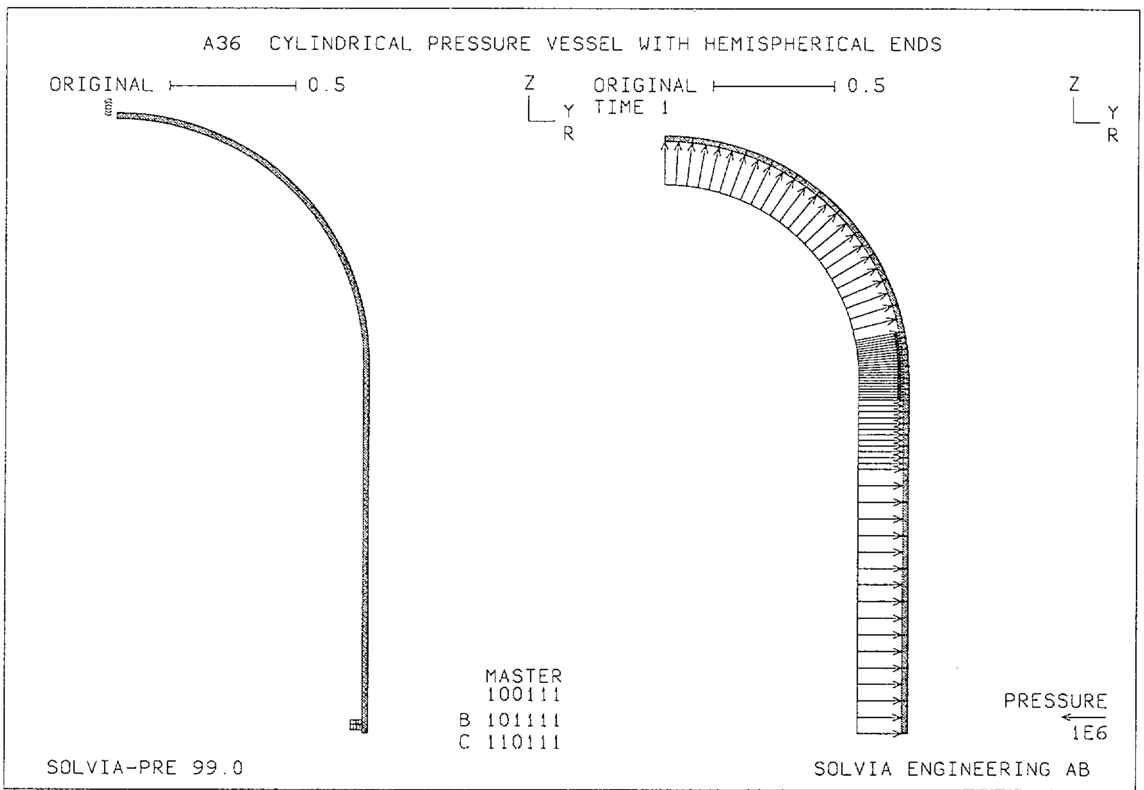
The deformed mesh and the distribution of the axial stress and the hoop stress at the outer surface of the cylinder can be seen in the figures on page A36.3.

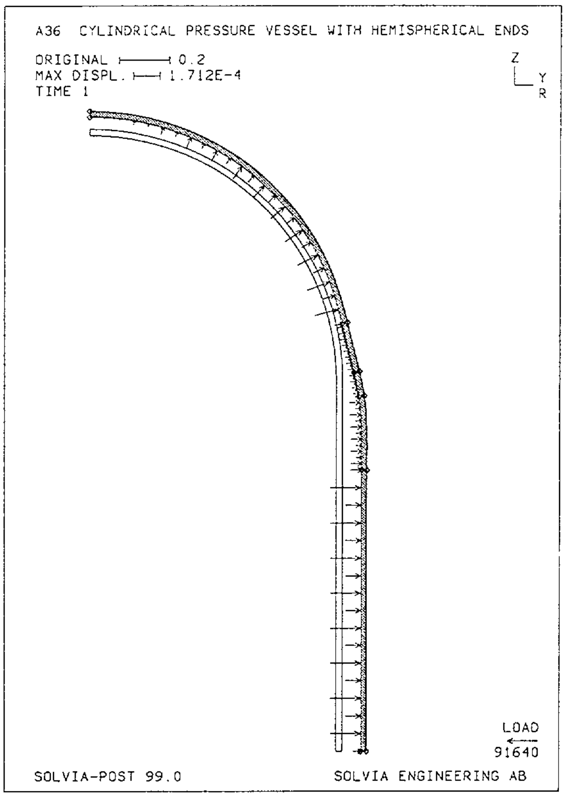
Input data used in the SOLVIA analysis is found on pages A36.4 and A36.5.

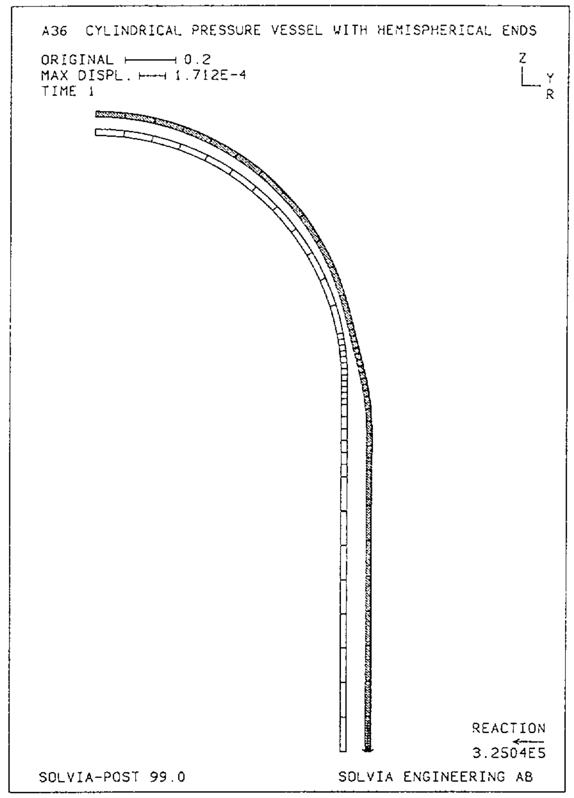
References

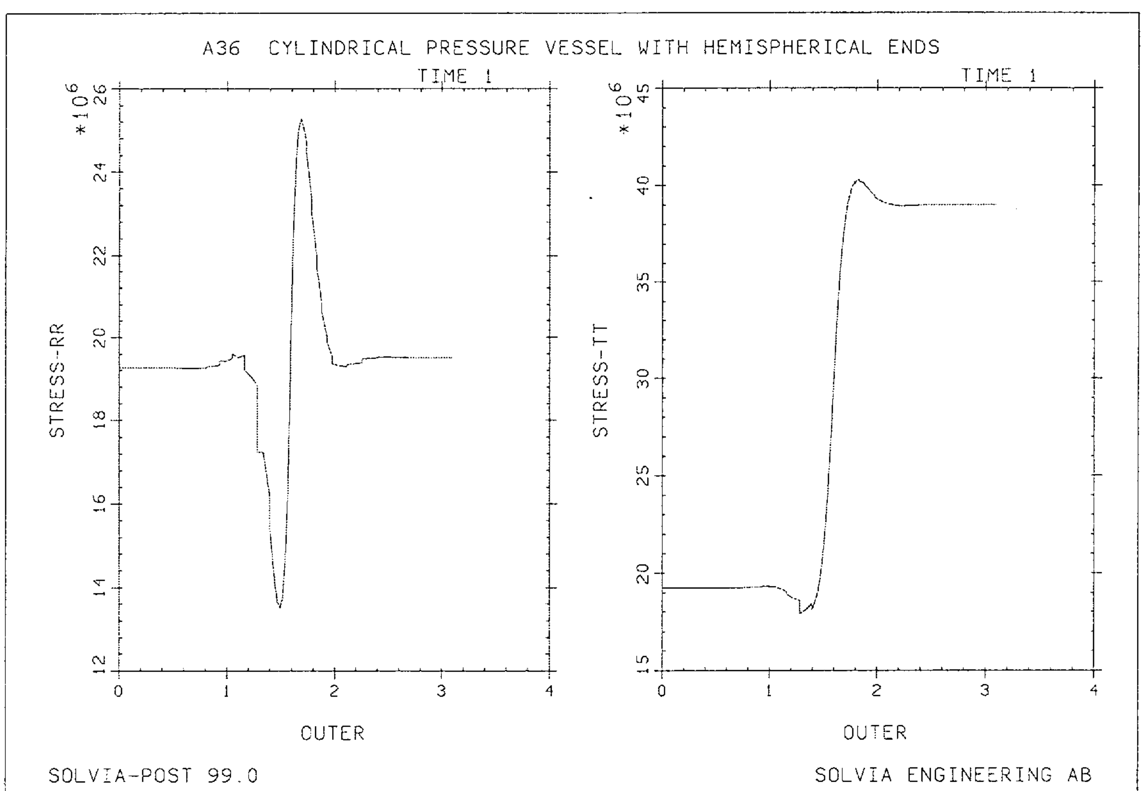
[1] NAFEMS, Background to Benchmarks, 1993.

[2] Timoshenko, S.P. and Woinowsky-Krieger, S., Theory of Plates and Shells, Second Edition, McGraw-Hill, 1959.




A36 CYLINDRICAL PRESSURE VESSEL WITH HEMISPHERICAL ENDS
ORIGINAL 0.2
MAX DISPL. 1.712E-4
TIME 1
Z
Y
R
LOAD
91640
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A36 CYLINDRICAL PRESSURE VESSEL WITH HEMISPHERICAL ENDS
ORIGINAL 0.2
MAX DISPL. 1.712E-4
TIME 1
Z
Y
R
REACTION
3.2504E5
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A36 CYLINDRICAL PRESSURE VESSEL WITH HEMISPHERICAL ENDS
TIME 1
TIME 1
*10^6
STRESS-RR
12
14
16
18
20
22
24
26
STRESS-TT
15
20
25
30
35
40
45
0
1
2
3
4
OUTER
OUTER
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A36  CYLINDRICAL PRESSURE VESSEL WITH HEMISPHERICAL ENDS'
*
DATABASE  CREATE
*
MASTER  IDOF=100111
*
SET  NODES=8
SYSTEM  1  CYLINDRICAL  Z=1.5
COORDINATES
 ENTRIES  NODE  R       THETA
            1   1.0125   90.
            2   0.9875   90.
            3   1.0125   11.068
            4   0.9875   11.068
            5   1.0125    0.
            6   0.9875    0.
SYSTEM  0
COORDINATES
 ENTRIES  NODE  Y       Z
            7   1.0125  1.4034
            8   0.9875  1.4034
            9   1.0125  1.1136
           10   0.9875  1.1136
           11   1.0125
           12   0.9875
*
MATERIAL  1  ELASTIC  E=210.E9 NU=0.3
*
EGROUP  1  PLANE AXISYMMETRIC  RESULTS=NSTRESSES
GSURFACE   2  4  3  1  EL1=12 EL2=1 SYSTEM=1
GSURFACE   4  6  5  3  EL1=8  EL2=1 SYSTEM=1
GSURFACE   6  8  7  5  EL1=4  EL2=1 SYSTEM=0
GSURFACE   8 10  9  7  EL1=6  EL2=1 SYSTEM=0
GSURFACE  10 12 11  9  EL1=8  EL2=1 SYSTEM=0
*
LOADS  ELEMENT  INPUT=LINE
  2  4  1.E6
  4  6  1.E6
  6  8  1.E6
  8 10  1.E6
 10 12  1.E6
*
FIXBOUNDARIES  2  INPUT=LINE  /  1 2
FIXBOUNDARIES  3  INPUT=LINE  /  11 12
*
MESH  BCODE=ALL OUTLINE=YES SUBFRAME=21
MESH  VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A36  CYLINDRICAL PRESSURE VESSEL WITH HEMISPHERICAL ENDS
*
DATABASE  CREATE  STRESSREFERENCE=ELEMENT
WRITE  FILENAME='a36.lis'
*
SET  PLOTORIENTATION=PORTRAIT
MESH  ORIGINAL=YES VECTOR=LOAD OUTLINE=YES NSYMBOLS=MYNODES
MESH  ORIGINAL=YES VECTOR=REACTION
*
SET  PLOTORIENTATION=LANDSCAPE
*
EPLINE  NAME=OUTER
 1  4 7 3  TO  38  4 7 3
ELINE  LINENAME=OUTER KIND=SRR OUTPUT=ALL SUBFRAME=21
ELINE  LINENAME=OUTER KIND=STT
*
SUMMATION  KIND=LOAD
SUMMATION  KIND=REACTION DETAILS=YES
END
```

EXAMPLE A37

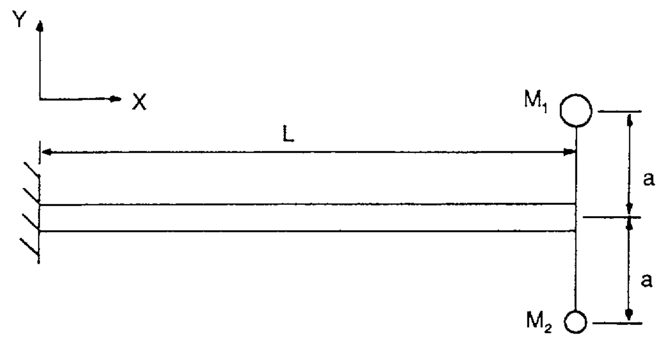
FREQUENCY ANALYSIS OF CANTILEVER WITH OFF-CENTER MASSES

Objective

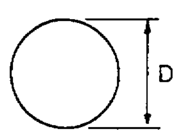
To verify the dynamic behaviour of the BEAM and GENERAL Mass elements and the use of rigid links in frequency analysis.

Physical Problem

A cantilever beam with two off-center point masses as shown in the figure below is considered. The point masses are connected to the tip end of the beam using rigid links. The off-center distance is 2 m. At the other end the beam is rigidly built-in. The six lowest frequencies and eigenmodes of the beam structure are analyzed. This example is described in [1].



Beam cross section



$L = 10$ m
$a = 2$ m
$D = 0.5$ m
$M_1 = 10000$ kg
$M_2 = 1000$ kg

Elastic material
$E = 2 \cdot 10^{11}$ N/m^2
$\nu = 0.3$
$\rho = 8000$ kg/m^3

Finite Element Model

The cantilever beam is modeled using five standard BEAM elements with a circularsolid section. The point masses are modeled using GENERAL Mass element with one node per element. Rigid links are connecting the BEAM elements with the Mass elements. A consistent mass matrix assumption must be used in this example, and the subspace iteration method is employed in the eigenvalue calculations. The calculational model is shown in the figure on page A37.2.

Solution Results

Using the input data shown on pages A37.4 and A37.5 the following results are obtained:

Mode	Reference freq. fr [1]	SOLVIA
1	1.723	1.722
2	1.727	1.725
3	7.413	7.410
4	9.972	9.946
5	18.155	18.044
5	26.957	26.694

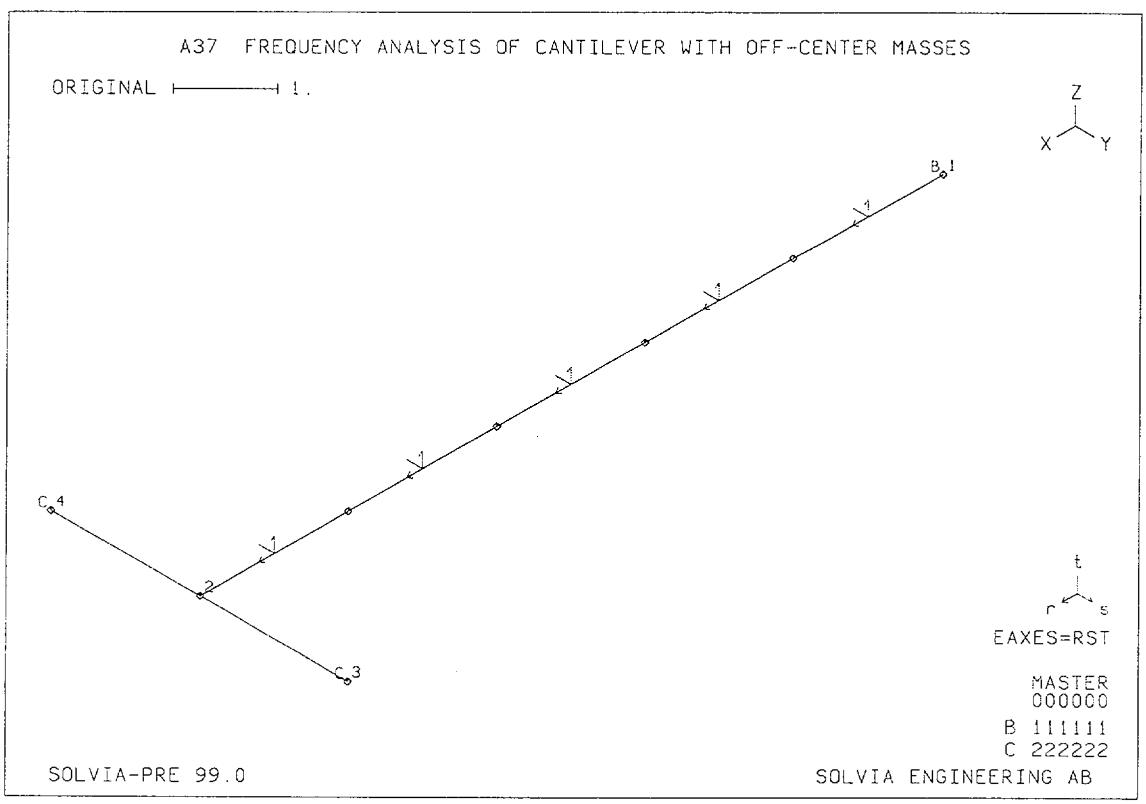
User Hints

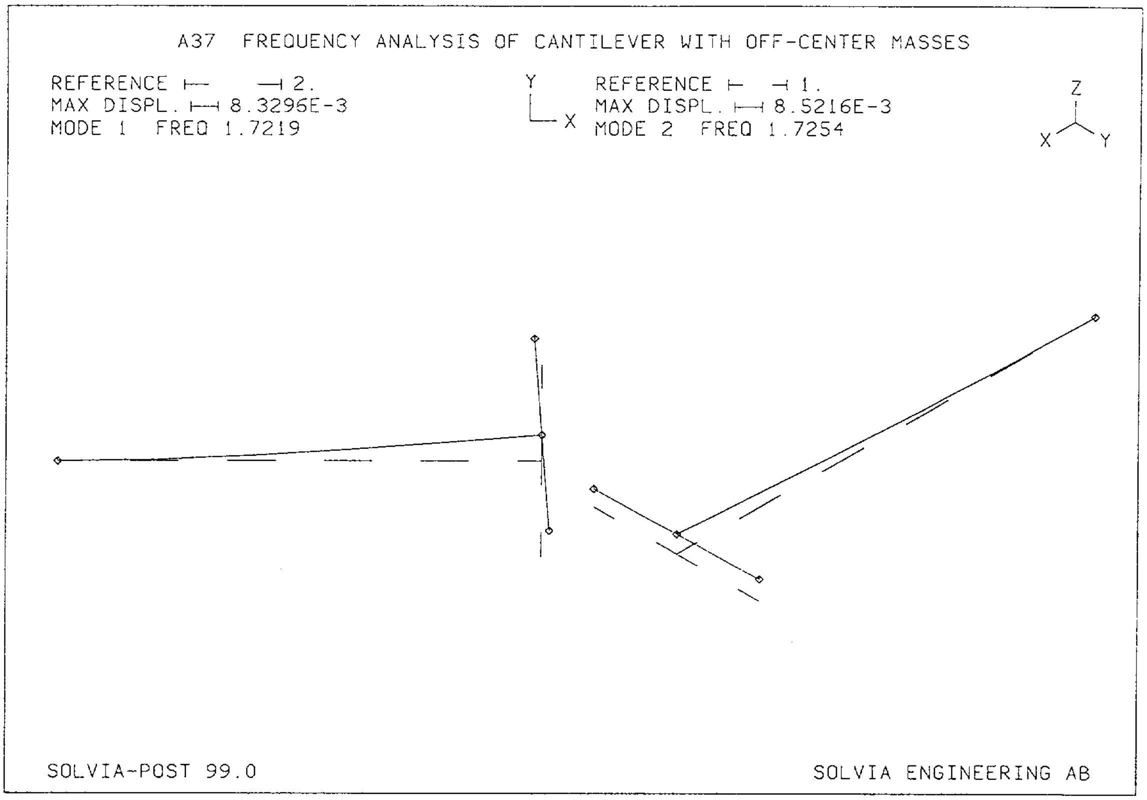
- GENERAL Mass elements and consistent mass matrix assumption **must** be used when concentrated point masses are coupled to a structure via rigid links. If a lumped mass matrix is used or the point masses are defined using concentrated nodal masses the off-diagonal terms from the rigid link transformation are lost and the solution result will be in error.

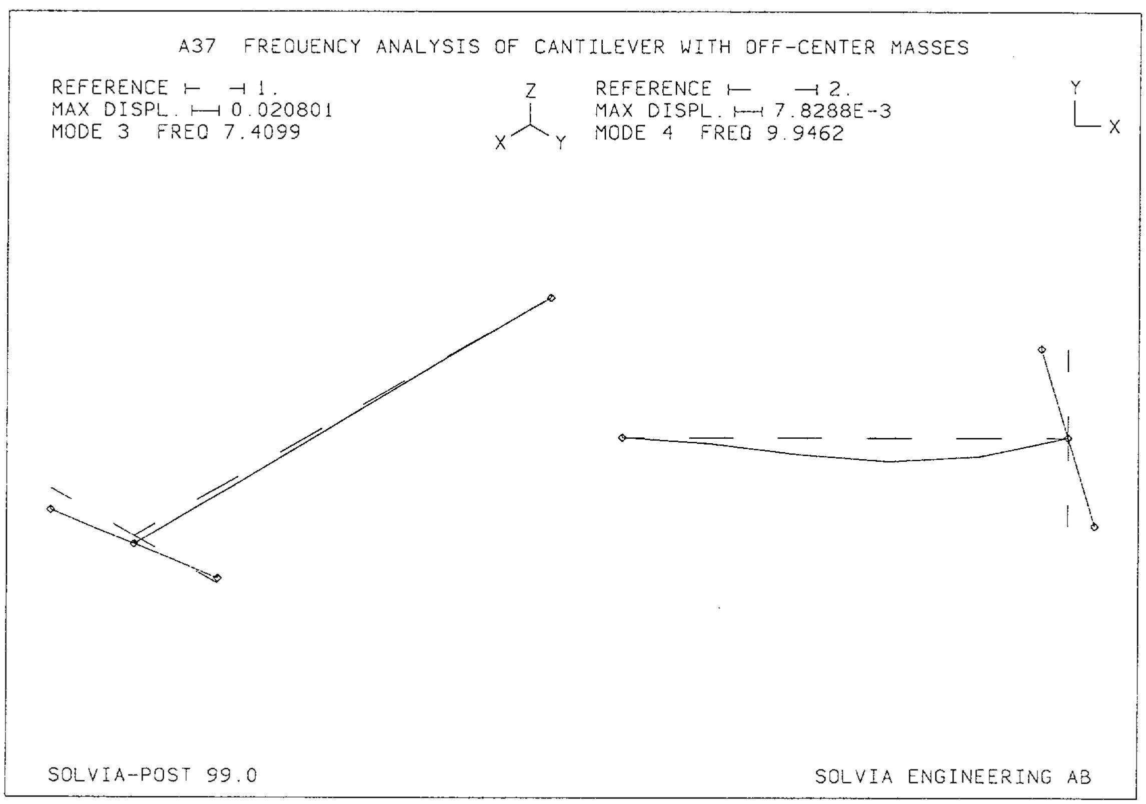
- The standard section BEAM element is formulated including shear deformation. The NAFEMS results refer to an exact 3-D beam element excluding the shear effects. This can be verified by using the general BEAM section and setting the shear areas in s- and t-directions to zero.

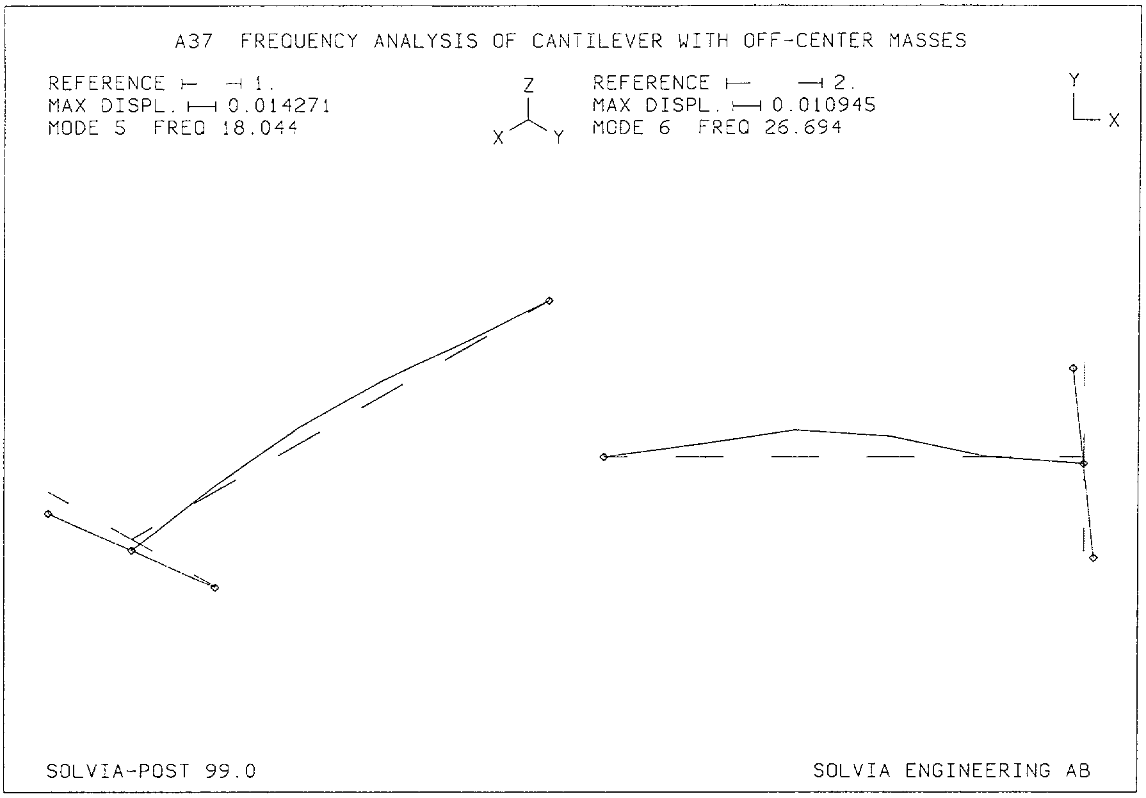
Reference

[1] NAFEMS, The Standard NAFEMS Benchmarks, TSNB, Rev. 3, October 5, 1990.




A37 FREQUENCY ANALYSIS OF CANTILEVER WITH OFF-CENTER MASSES
REFERENCE 2.
MAX DISPL. 8.3296E-3
MODE 1 FREQ 1.7219
Y
X
REFERENCE 1.
MAX DISPL. 8.5216E-3
MODE 2 FREQ 1.7254
Z
X
Y
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A37 FREQUENCY ANALYSIS OF CANTILEVER WITH OFF-CENTER MASSES
REFERENCE 1.
MAX DISPL. 0.020801
MODE 3 FREQ 7.4099
Z
X
Y
REFERENCE 2.
MAX DISPL. 7.8288E-3
MODE 4 FREQ 9.9462
Y
X
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB



SOLVIA-POST input

```
*  A37  FREQUENCY ANALYSIS OF CANTILEVER WITH OFF-CENTER MASSES
*
DATABASE  CREATE
WRITE  FILENAME='a37.lis'
*
FREQUENCIES
MASS-PROPERTIES
*
SET  RESPONSETYPE=VIBRATIONMODE ORIGINAL=DASHED
SET  NSYMBOLS=MYNODES
*
SUBFRAME  21
MESH  VIEW=Z TIME=1
MESH  VIEW=I TIME=2
SUBFRAME  21
MESH  VIEW=I TIME=3
MESH  VIEW=Z TIME=4
SUBFRAME  21
MESH  VIEW=I TIME=5
MESH  VIEW=Z TIME=6
END
```

SOLVIA-PRE input

```
HEAD  'A37  FREQUENCY ANALYSIS OF CANTILEVER WITH OFF-CENTER MASSES'
*
DATABASE  CREATE
*
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=CONSISTENT
FREQUENCIES  SUBSPACE-ITERATION  NEIG=6
COORDINATES
 1
 2  10.
 3  10.  2.
 4  10. -2.
*
MATERIAL  1  ELASTIC  E=200.E9 NU=0.3 DENSITY=8000.
*
EGROUP  1  BEAM
SECTION  1  CIRCULARSOLID D=0.5
BEAMVECTOR
 1  0. 0. 1.
GLINE  N1=1 N2=2 AUX=-1  EL=5
*
EGROUP  2  GENERAL
MATRIXSET  1  MASSMATRIX
 1  1.E4
 2  1.E4
 3  1.E4
 4  0.  /  5  0.  /  6  0.
MATRIXSET  2  MASSMATRIX
 1  1.E3
 2  1.E3
 3  1.E3
 4  0.  /  5  0.  /  6  0.
EDATA  /  1 1  /  2 2
ENODES  /  1 3  /  2 4
*
FIXBOUNDARIES  /  1
RIGIDLINK
 3 2  /  4 2
*
MESH  NSYMBOLS=YES NNUMBERS=MYNODES BCODE=ALL EAXES=RST
*
SOLVIA
END
```

EXAMPLE A38

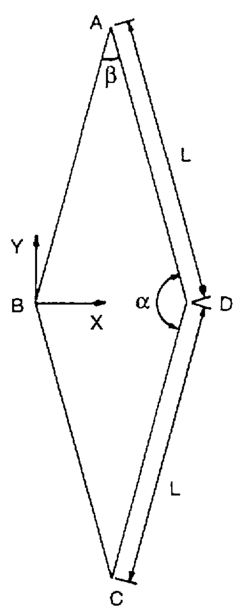
SIMPLY SUPPORTED SKEW PLATE UNDER PRESSURE LOAD

Objective

To verify the SHELL element under distributed load when using distorted elements.

Physical Problem

A simply supported skew plate under pressure load is considered as shown in the figure below. The pressure load acts in the negative Z-direction. The material in the plate is linear elastic isotropic. The problem is described in [1].



Uniform thickness 0.01 m

	Elastic material
$L = 1$ m	$E = 2.1 \cdot 10^{11}$ N/m^2
$\alpha = 150^\circ$	$\nu = 0.3$
$\beta = 30^\circ$	Pressure $p = 700$ N/m^2

Finite Element Model

The skew plate is generated using 4×4 cubic SHELL elements. The Z-displacement is fixed for all edges. The element stresses are evaluated at the element nodal points. The finite element mesh corresponds to the fine mesh described in [1], see figure on page A38.2. However, a good stress solution would require a finer mesh.

Solution Results

The theoretical solution to this problem is discussed in [1] and is based on classical thin plate theory. The primary results are the maximum and minimum principal stresses on the lower surface at the plate centre.

The input data used in the SOLVIA analysis is shown on page A38.5. A finite element model using 8×8 4-node SHELL elements is also used to compare with the NAFEMS results.

Stresses on the bottom surface	NAFEMS	Cubic SHELL	4-node SHELL
Max. principal stress $\left[\mathrm{MN/m^2}\right]$	0.802	0.793	0.757
Mid. principal stress $\left[\mathrm{MN/m^2}\right]$	0.456	0.409	0.421

The distribution of stress-xx along the diagonal BD and the distribution of stress-yy along the diagonal AC can be seen in the top figures on page A38.3. Contour plots of stress-xx and maximum principal stress can also be seen on page A38.3.

The corresponding SOLVIA-POST results for the 4-node SHELL element analysis can be seen on page A38.4. The mesh consists of 8×8 elements. Note the large stress jumps in the stress distribution.

User Hints

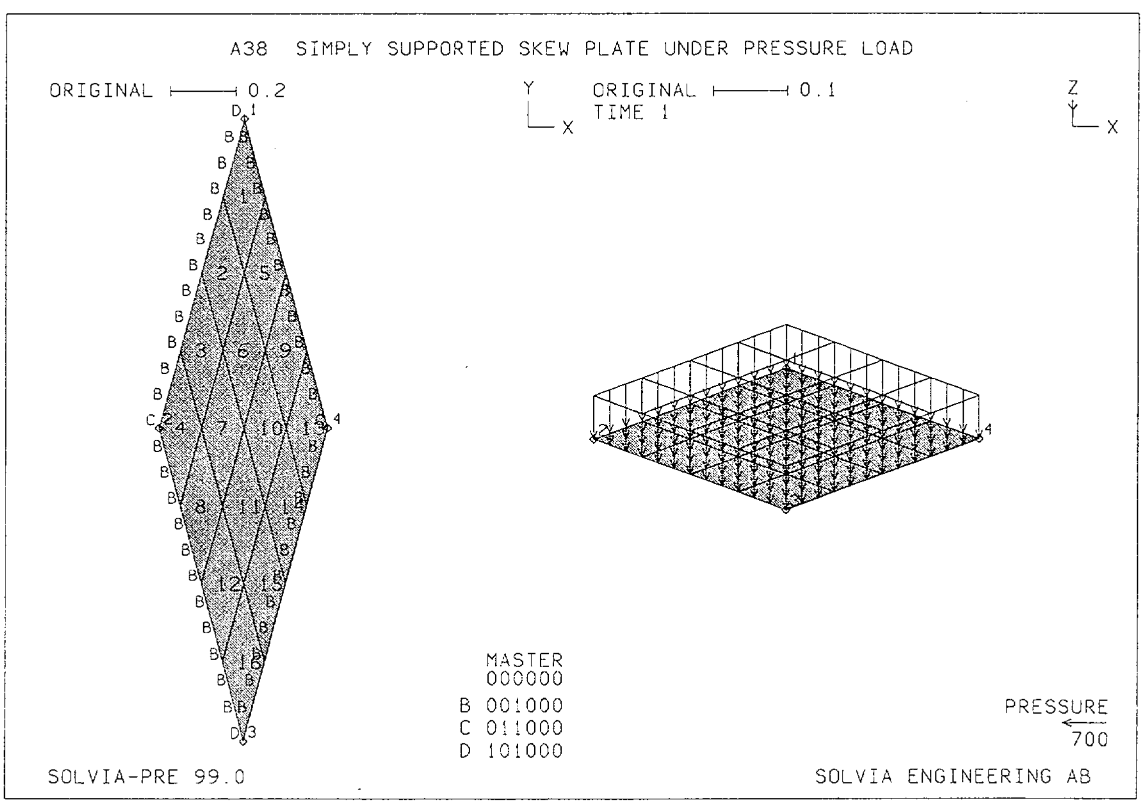
- Note that SOLVIA-PRE gives a warning message regarding the element distortion in the model (skew distortion of 60 degrees).

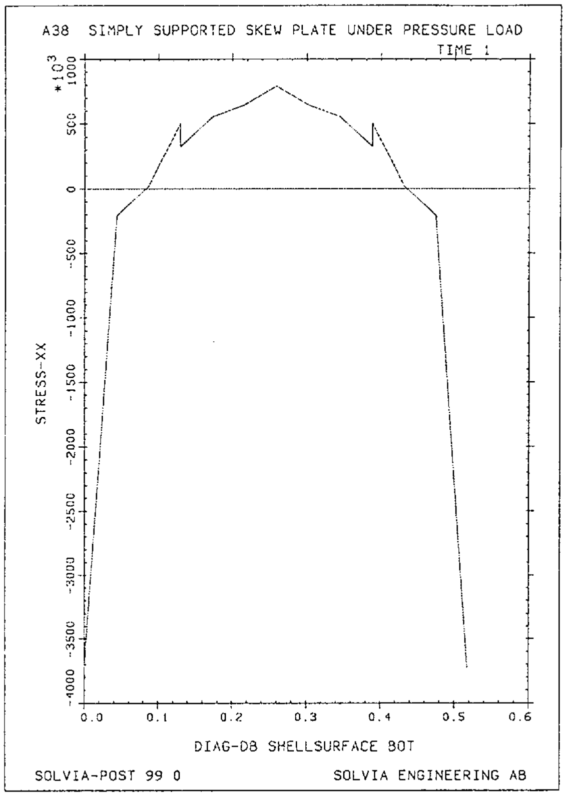
- Note the stress concentration at the corners B and D. A more detailed discussion regarding error sources can be found in [1].

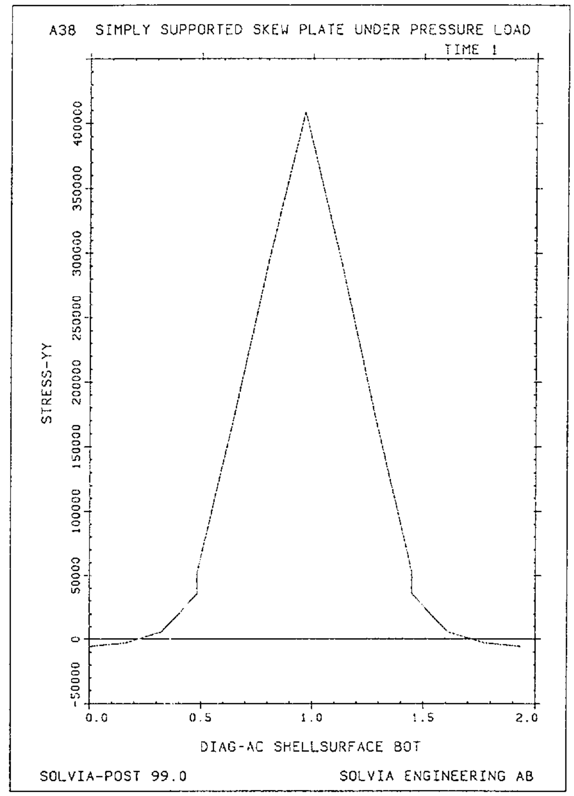
- Simply supported boundary conditions for a square plate are discussed in Example A27.

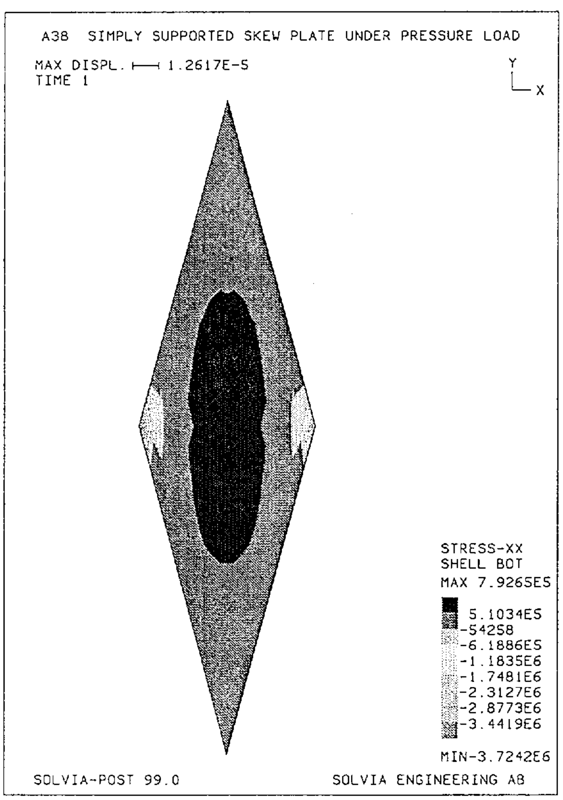
Reference

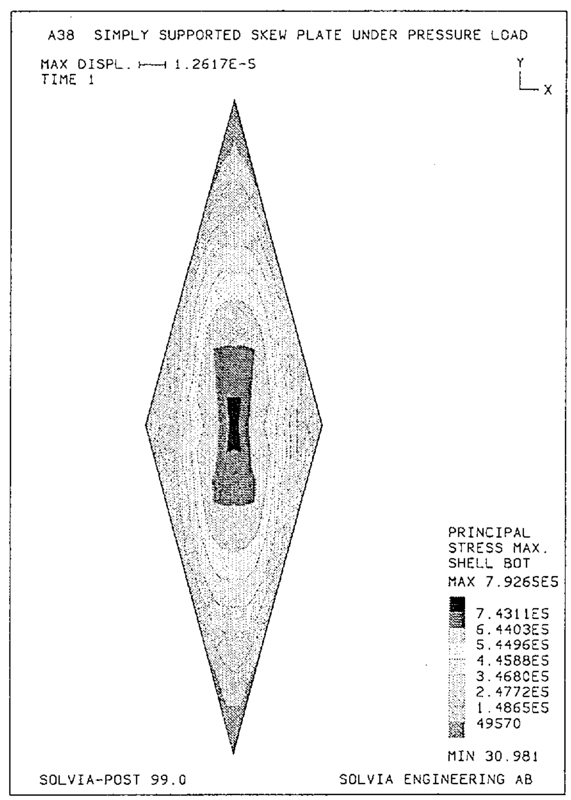
[1] NAFEMS, Background to Benchmarks, 1993.

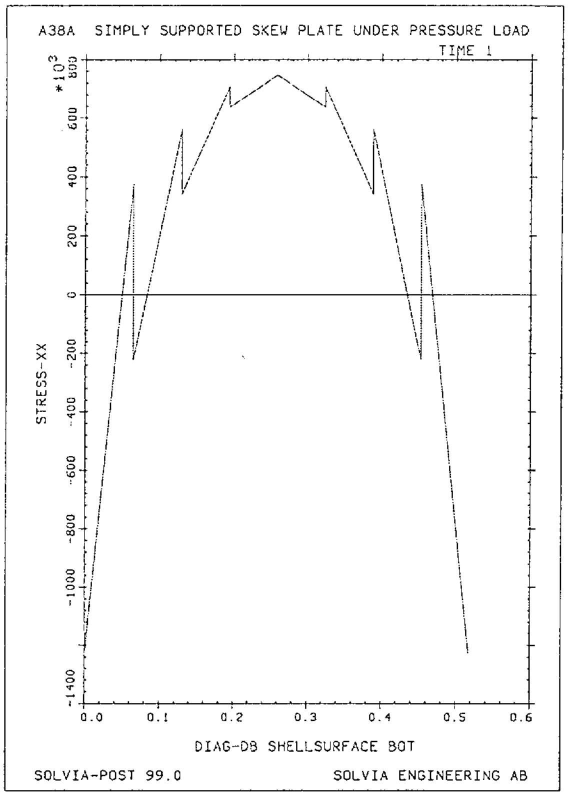


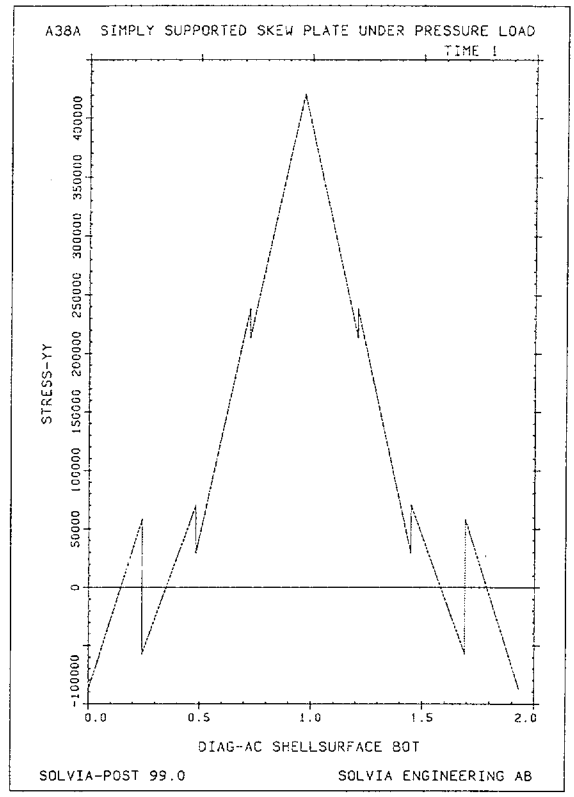

A38 SIMPLY SUPPORTED SKEW PLATE UNDER PRESSURE LOAD
TIME 1
STRESS-XX
DIAG-DB SHELLSURFACE BOT
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

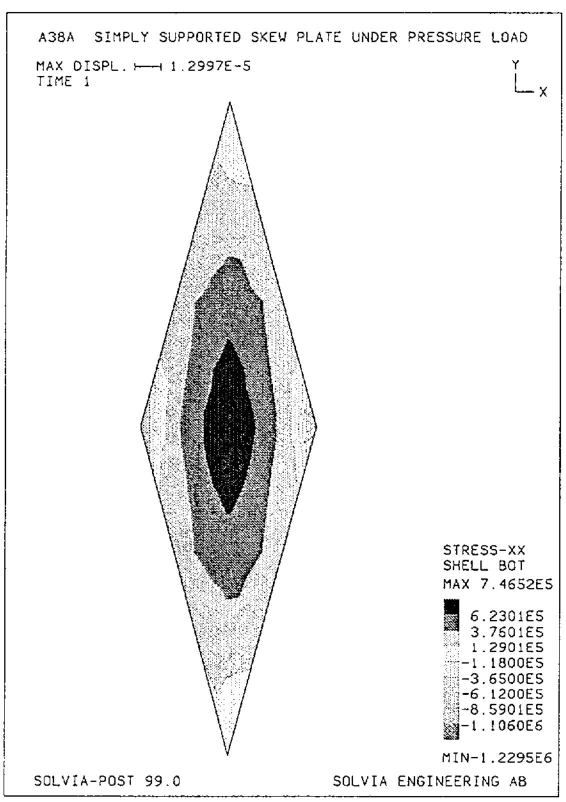

A38 SIMPLY SUPPORTED SKEW PLATE UNDER PRESSURE LOAD
TIME 1
STRESS-YY
DIAG-AC SHELLSURFACE BOT
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

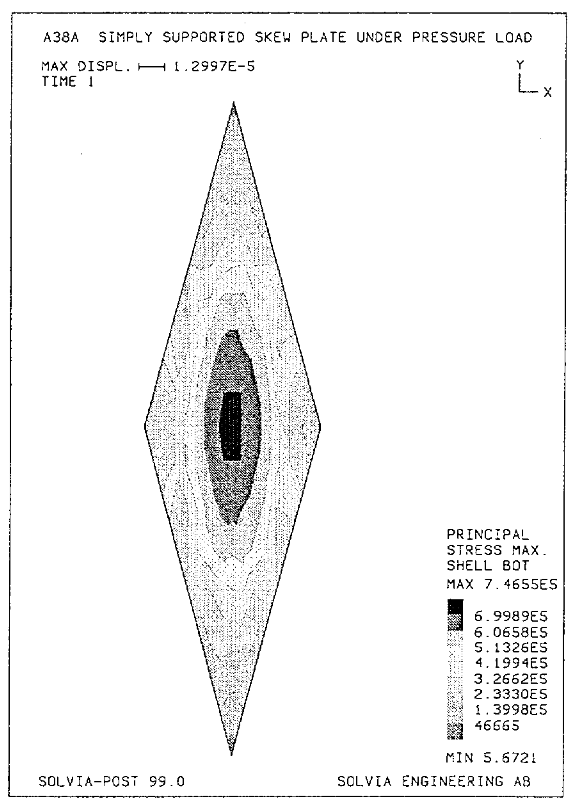

A38 SIMPLY SUPPORTED SKEW PLATE UNDER PRESSURE LOAD
MAX DISPL. 1.2617E-5
TIME 1
STRESS-XX
SHELL BOT
MAX 7.9265E5
5.1034E5
-54258
-6.1886E5
-1.1835E6
-1.7481E6
-2.3127E6
-2.8773E6
-3.4419E6
MIN-3.7242E6
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A38 SIMPLY SUPPORTED SKEW PLATE UNDER PRESSURE LOAD
MAX DISPL. 1.2617E-5
TIME 1
PRINCIPAL
STRESS MAX.
SHELL BOT
MAX 7.9265E5
7.4311E5
6.4403E5
5.4496E5
4.4588E5
3.4680E5
2.4772E5
1.4865E5
49570
MIN 30.981
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A38A SIMPLY SUPPORTED SKEW PLATE UNDER PRESSURE LOAD
TIME 1
STRESS-XX
*10³
DIAG-DB SHELLSURFACE BOT
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A38A SIMPLY SUPPORTED SKEW PLATE UNDER PRESSURE LOAD
TIME 1
STRESS-YY
DIAG-AC SHELLSURFACE BOT
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A38A SIMPLY SUPPORTED SKEW PLATE UNDER PRESSURE LOAD
MAX DISPL. 1.2997E-5
TIME 1
STRESS-XX
SHELL BOT
MAX 7.4652E5
6.2301E5
3.7601E5
1.2901E5
-1.1800E5
-3.6500E5
-6.1200E5
-8.5901E5
-1.1060E6
MIN-1.2295E6
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A38A SIMPLY SUPPORTED SKEW PLATE UNDER PRESSURE LOAD
MAX DISPL. 1.2997E-5
TIME 1
PRINCIPAL
STRESS MAX.
SHELL BOT
MAX 7.4655E5
6.9989E5
6.0658E5
5.1326E5
4.1994E5
3.2662E5
2.3330E5
1.3998E5
46665
MIN 5.6721
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

S

SOLVIA-PRE input

```
HEADING  'A38  SIMPLY SUPPORTED SKEW PLATE UNDER PRESSURE LOAD'
*
DATABASE  CREATE
*
COORDINATES
 1  0.258819045  0.965925826  /  2  0. 0.
 3  0.258819045 -0.965925826  /  4  0.51763809 0.
*
MATERIAL  1  ELASTIC  E=2.1E11 NU=0.3
*
EGROUP  1  SHELL RESULTS=NSTRESSES
THICKNESS  1  0.01
GSURFACE  1 2 3 4  EL1=4 EL2=4 NODES=16
LOADS  ELEMENT  TYPE=PRESSURE INPUT=SURFACE
 1 2 3 4  T  700.
*
FIXBOUNDARIES  3 INPUT=LINES  /  1 2  /  2 3  /  3 4  /  4 1
FIXBOUNDARIES  1 INPUT=NODES  /  1 3
FIXBOUNDARIES  2 INPUT=NODES  /  2 4
*
VIEW  ID=1 XVIEW=0. YVIEW=-1. ZVIEW=0.1
SET  NSYMBOLS=MYNODES NNUMBERS=MYNODES
MESH  VIEW=Z ENUMBER=YES BCODE=ALL SUBFRAME=21
MESH  VIEW=1 VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A38  SIMPLY SUPPORTED SKEW PLATE UNDER PRESSURE LOAD
*
DATABASE  CREATE
WRITE  FILENAME='a38.lis'
*
SET  VIEW=Z PLOTORIENTATION=PORTRAIT
SHELLSURFACE  PLOTRESULTS=BOTTOM LISTRESULTS=BOTTOM
*
EPLINE  NAME=DIAG-DB
  4  2 14 16 4  STEP  3 TO 13  2 14 16 4
EPLINE  NAME=DIAG-AC
 16  3 15 13 1  STEP -5 TO  1  3 15 13 1
ELINE  LINENAME=DIAG-DB KIND=SXX OUTPUT=ALL
ELINE  LINENAME=DIAG-AC KIND=SYY OUTPUT=ALL
*
MESH  CONTOUR=SXX OUTLINE=YES
MESH  CONTOUR=SPMAX OUTLINE=YES
EMAX  SELECT=S-EFFECTIVE NUMBER=3 TYPE=MAXIMUM
END
```

EXAMPLE A39

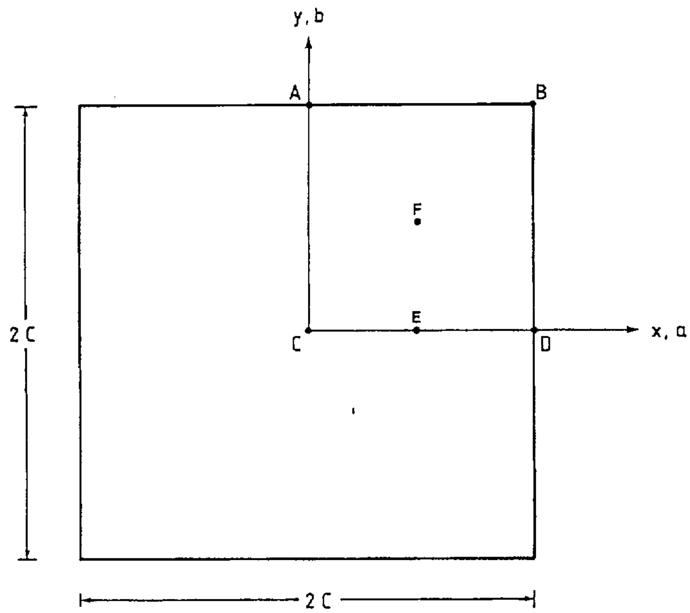
ORTHOTROPIC PLATE UNDER PRESSURE LOAD

Objective

To verify the bending and twisting behaviour of the SHELL element when the material is orthotropic and the SHELL element is subjected to a uniform pressure load.

Physical Problem

A simply supported square plate under uniform loading, as shown in the figure below, is considered.



$C = 0.30$ m
$h = 0.015$ m (thickness)
$p = 1.0 \cdot 10^3$ N/m^2 (pressure)
$E_a = 1.35548 \cdot 10^{10}$ N/m^2
$E_b = 1.12950 \cdot 10^9$ N/m^2
$\nu_{ab} = 3.847826 \cdot 10^{-2}$
$G_{ab} = 1.17 \cdot 10^9$ N/m^2

Finite Element Model

Because of symmetry only one quarter of the plate need to be considered, A-B-C-D. An orthotropic linear elastic material model is used with the principal material axes a and b coinciding with the global coordinate axes X and Y, respectively. Two finite element models have been used, one model consists of 16 cubic SHELL elements as shown in the figure on page A39.3 and the other model consists of 144 4-node SHELL elements.

Solution Results

The theoretical solution for this problem (without transverse shear deformation) is discussed in [1], chapt.11. The expression for the deflection w in the Z-direction takes the form of a double trigonometrical series,

$$w = \sum_{m=1,3,5..} \sum_{n=1,3,5..} a_{mn} \cdot \sin\frac{m\pi x}{a} \sin\frac{n\pi y}{b}$$

and the expression for the coefficients a_{mn} is given on page 371, [1].

From this expression for the deflection in the Z-direction, the bending and twisting moments in the plate are easily derived using eq. 212, [1].

The input data for the model with 16-node SHELL elements is shown on pages A39.6 and A39.7.

Vertical deflection, w(m):

Location	Theory*	16-node SHELL 16 elements	4-node SHELL 144 elements
C	$3.543 \cdot 10^{-4}$	$3.602 \cdot 10^{-4}$	$3.591 \cdot 10^{-4}$
E	$2.529 \cdot 10^{-4}$	$2.573 \cdot 10^{-4}$	$2.564 \cdot 10^{-4}$
F	$1.935 \cdot 10^{-4}$	$1.975 \cdot 10^{-4}$	$1.966 \cdot 10^{-4}$

Stress (N/m^2):

Location	Stress comp.	Theory*	16-node SHELL 16 elements	4-node SHELL 144 elements
C	σ_x:	$-9.997 \cdot 10^5$	$-1.008 \cdot 10^6$	$-1.004 \cdot 10^6$
	σ_y:	$-9.827 \cdot 10^4$	$-9.816 \cdot 10^4$	$-9.884 \cdot 10^4$
E	σ_x:	$-7.584 \cdot 10^5$	$-7.664 \cdot 10^5$ (el. 14,	$-7.217 \cdot 10^5$ (el. 138,
	σ_y:	$-7.147 \cdot 10^4$	$-7.145 \cdot 10^4$ point 3)	$-7.032 \cdot 10^4$ point 3)
E	σ_x:		$-7.664 \cdot 10^5$ (el. 15,	$-8.010 \cdot 10^5$ (el. 139,
	σ_y:		$-7.145 \cdot 10^4$ point 4)	$-7.337 \cdot 10^4$ point 4)

* The theoretical results include terms up to m = n = 29.

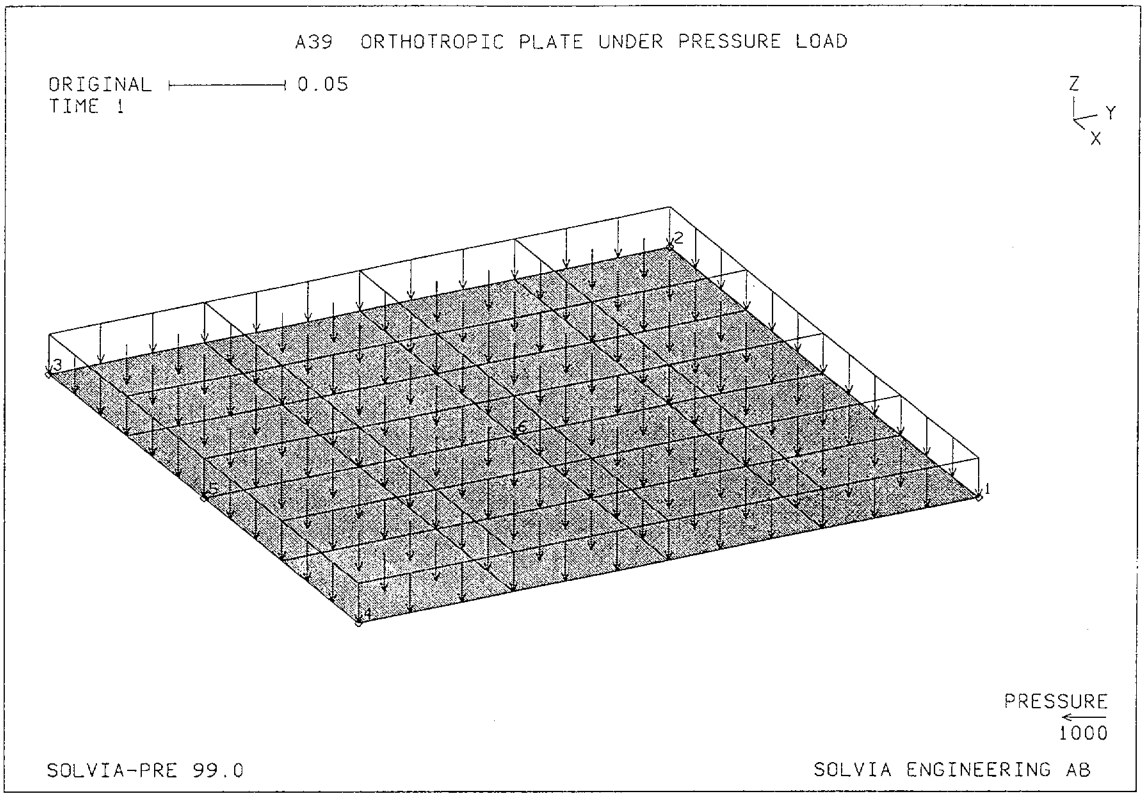
The deformed models are shown on pages A39.4 and A39.5. The distribution of stress-xx along side D-C and stress-yy along side A-C of the plate are also shown on pages A39.4 and A39.5.

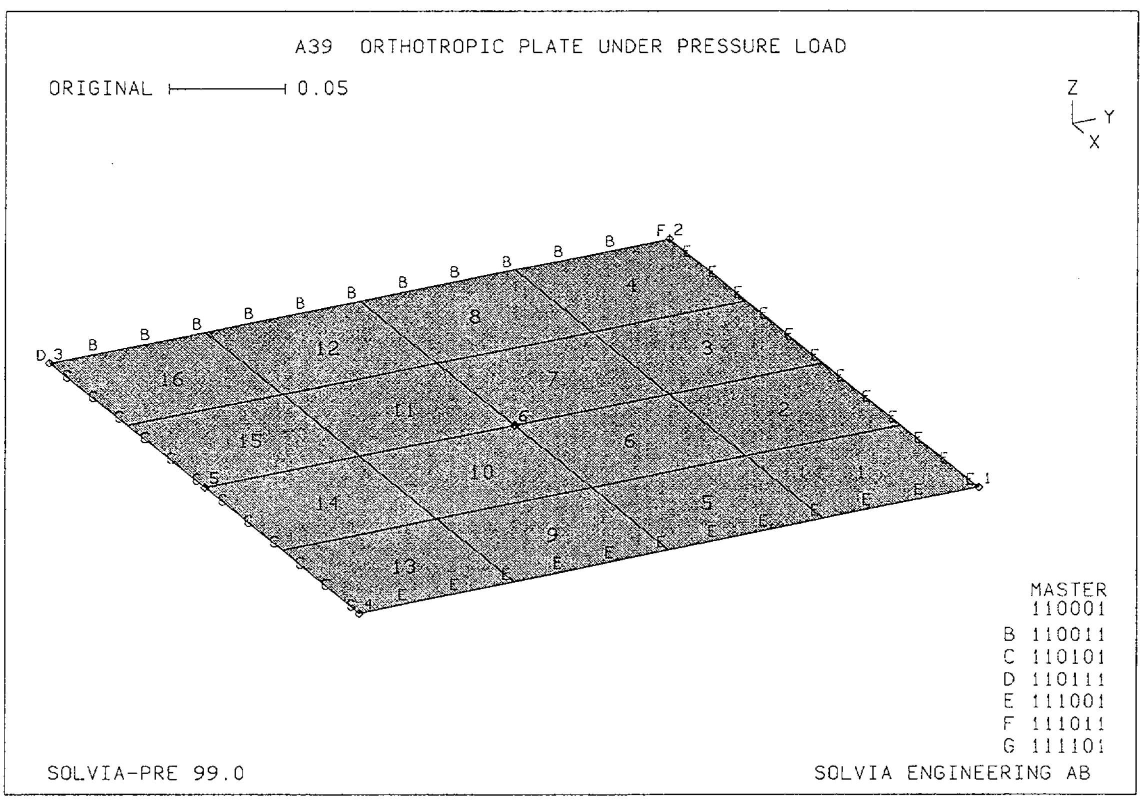
User Hints

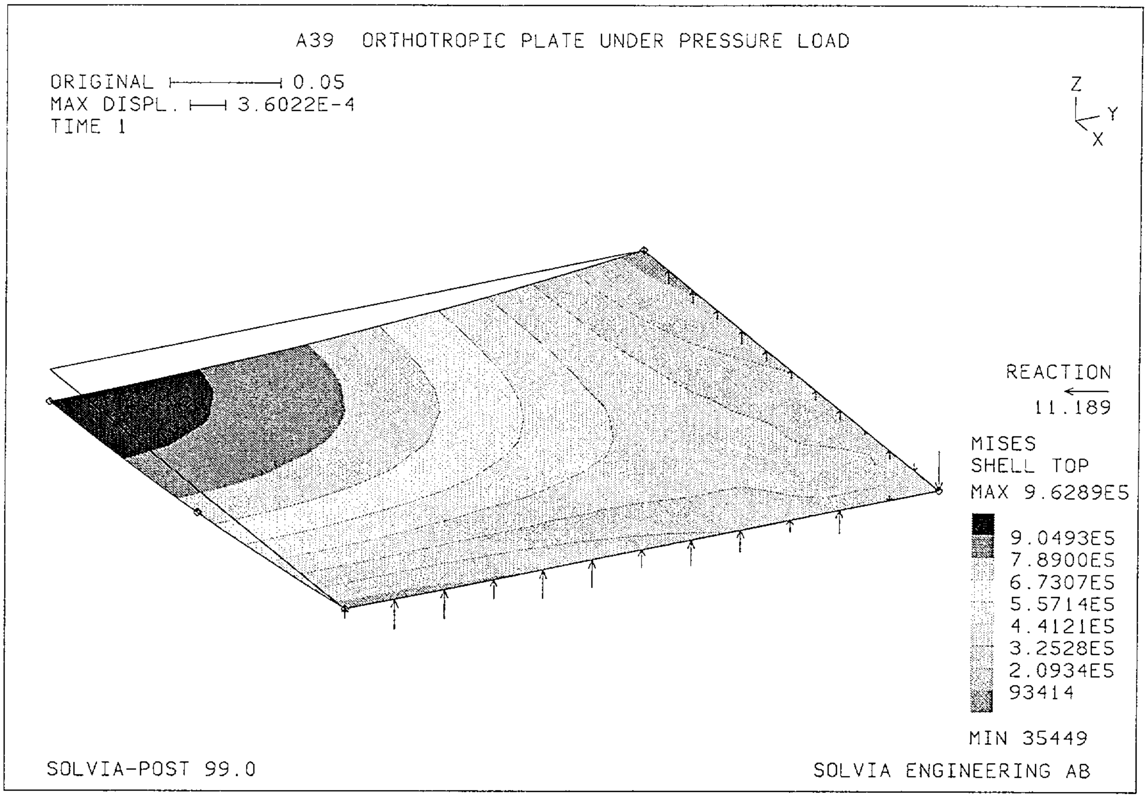
- Note that a finer mesh is necessary when modeling this simply supported plate when orthotropic material is used than when isotropic material is used. This is evident from the large gradient of the σ_y-stress occuring along the line A-C, which in turn is due to the very small E-modulus in the Y-direction. Detailed shear stresses and reactions would require a finer mesh than used here, see Example A27.

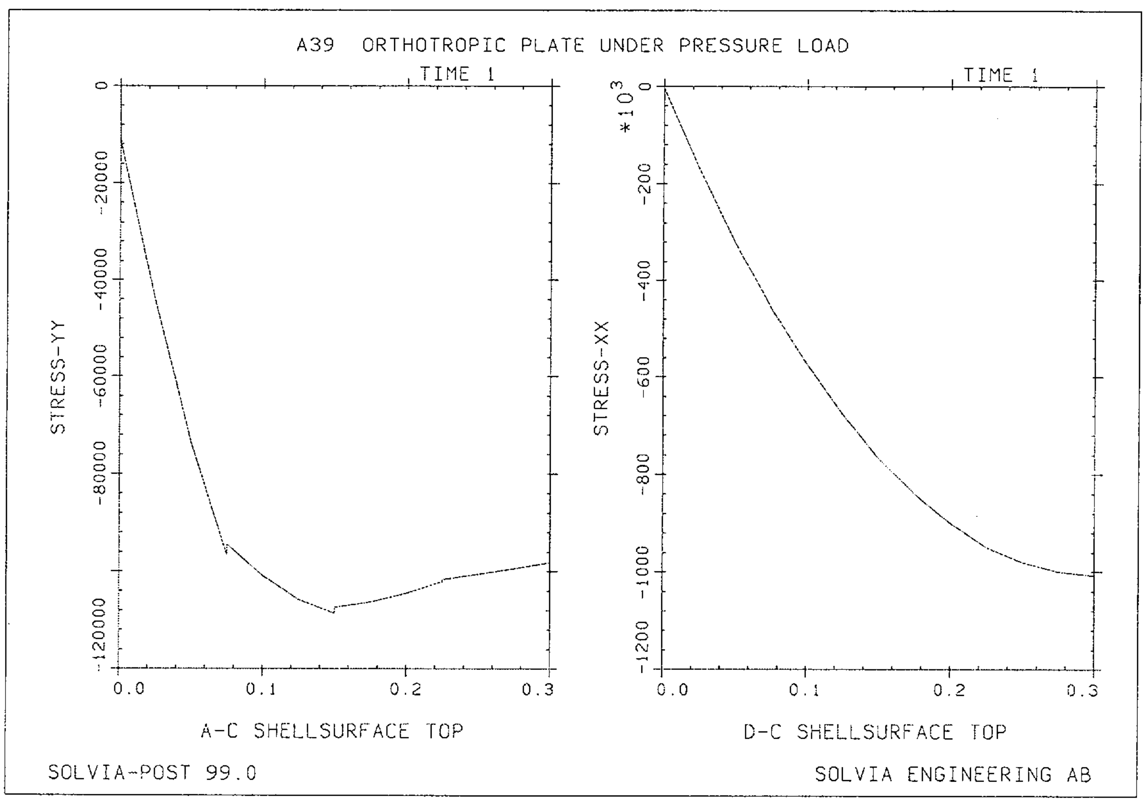
Reference

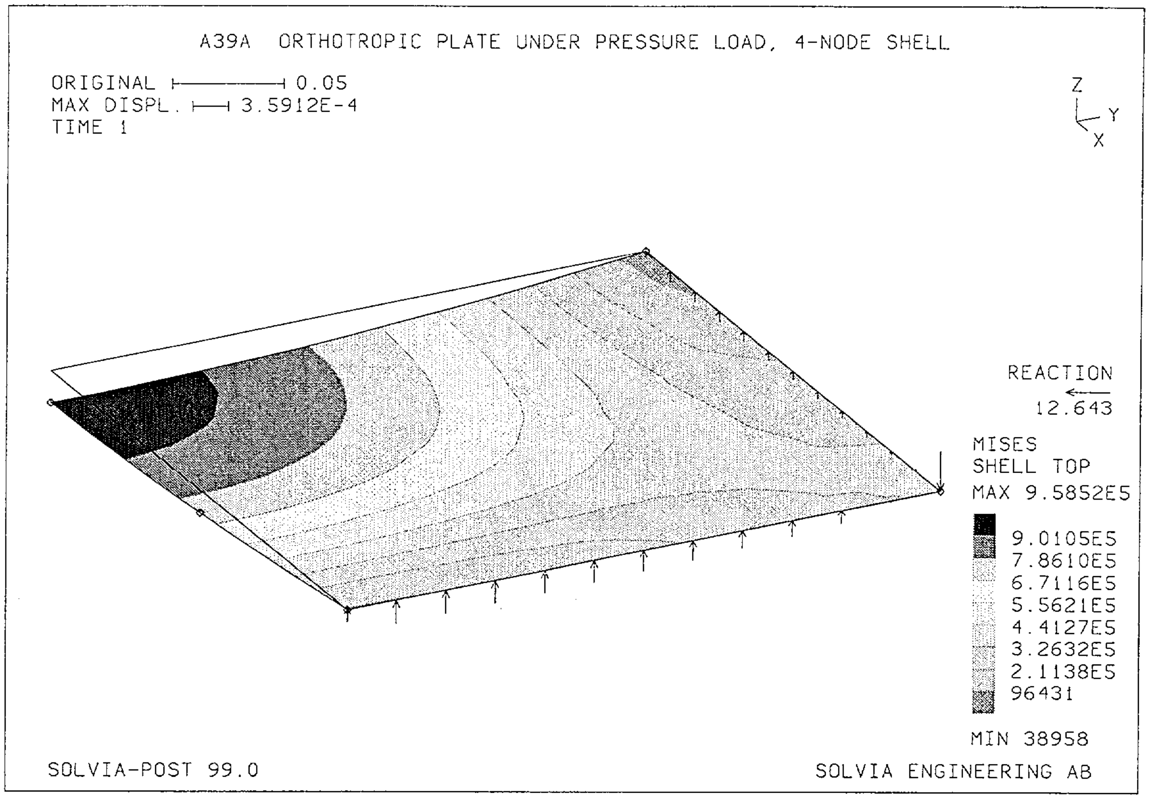
[1] Timoshenko, S.P. and Woinowsky-Krieger, S., Theory of Plates and Shells, Second Edition, McGraw-Hill, 1959.

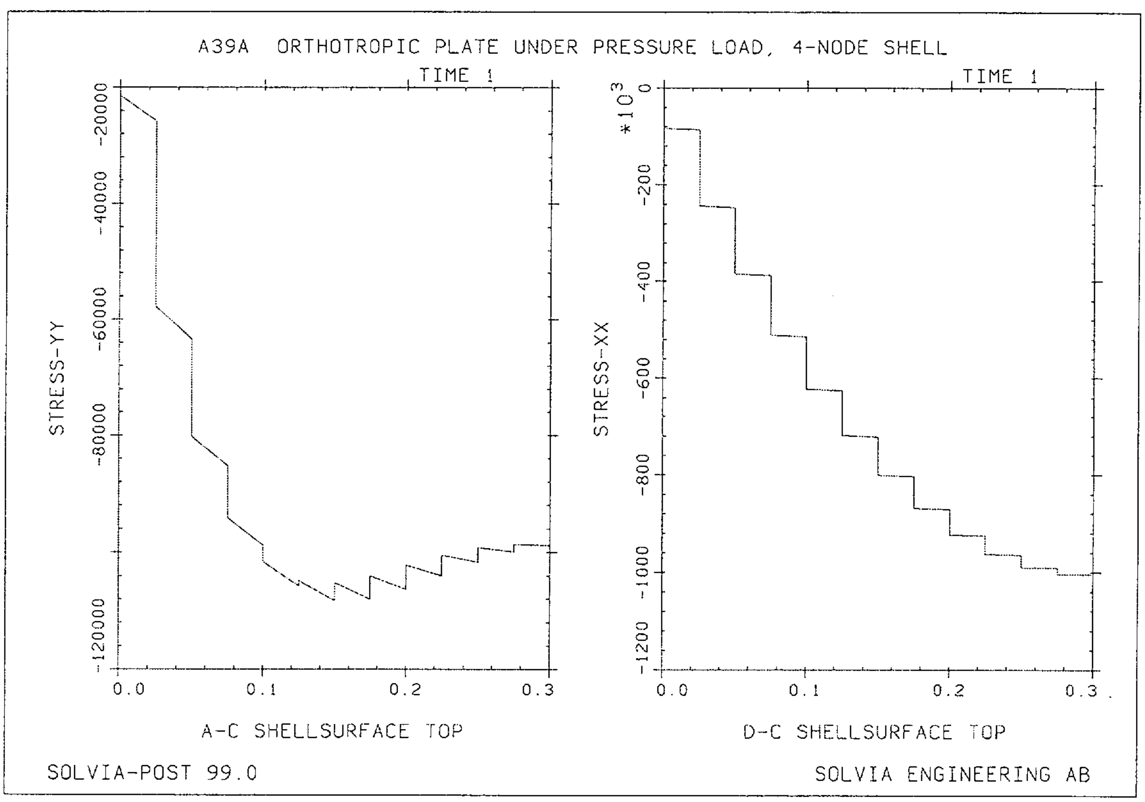

A39 ORTHOTROPIC PLATE UNDER PRESSURE LOAD
ORIGINAL 0.05
TIME 1
Z
Y
X
PRESSURE
1000
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A39 ORTHOTROPIC PLATE UNDER PRESSURE LOAD
ORIGINAL 0.05
Z
Y
X
MASTER
110001
B 110011
C 110101
D 110111
E 111001
F 111011
G 111101
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A39 ORTHOTROPIC PLATE UNDER PRESSURE LOAD
ORIGINAL 0.05
MAX DISPL. 3.6022E-4
TIME 1
Z
Y
X
REACTION
11.189
MISES
SHELL TOP
MAX 9.6289E5
9.0493E5
7.8900E5
6.7307E5
5.5714E5
4.4121E5
3.2528E5
2.0934E5
93414
MIN 35449
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A39 ORTHOTROPIC PLATE UNDER PRESSURE LOAD
TIME 1
STRESS-YY
0
-20000
-40000
-60000
-80000
-120000
0.0
0.1
0.2
0.3
A-C SHELLSURFACE TOP
TIME 1
STRESS-XX
*10^3
0
-200
-400
-600
-800
-1000
-1200
0.0
0.1
0.2
0.3
D-C SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A39A ORTHOTROPIC PLATE UNDER PRESSURE LOAD, 4-NODE SHELL
ORIGINAL 0.05
MAX DISPL. 3.5912E-4
TIME 1
Z
Y
X
REACTION
12.643
MISES
SHELL TOP
MAX 9.5852E5
9.0105E5
7.8610E5
6.7116E5
5.5621E5
4.4127E5
3.2632E5
2.1138E5
96431
MIN 38958
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A39A ORTHOTROPIC PLATE UNDER PRESSURE LOAD, 4-NODE SHELL
TIME 1
STRESS-YY
-20000
-40000
-60000
-80000
-120000
0.0
0.1
0.2
0.3
A-C SHELLSURFACE TOP
TIME 1
STRESS-XX
*10³
0
-200
-400
-600
-800
-1000
-1200
0.0
0.1
0.2
0.3
D-C SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A39  ORTHOTROPIC PLATE UNDER PRESSURE LOAD'
*
DATABASE  CREATE
MASTER  IDOF=110001
COORDINATES
 1  0.3  0.3  /  2  0.   0.3  /  3  /  4  0.3
 5  0.15 0.   /  6  0.15 0.15
*
MATERIAL  1  ORTHOTROPIC EA=1.35548E10 EB=1.1295E9,
             NUAB=3.847826E-2 GAB=1.17E9 GAC=1.17E9
*
EGROUP  1  SHELL RESULTS=NSTRESSES
GSURFACE   1 2 3 4  EL1=4 EL2=4 NODES=16
THICKNESS  1  0.015
EDATA  /  ENTRIES  EL  BETA
 1  0.  TO  16  0.
LOADS  ELEMENT  INPUT=SURFACE
 1 2 3 4  T 1000
*
FIXBOUNDARY  4  INPUT=LINE  /  3 4
FIXBOUNDARY  5  INPUT=LINE  /  2 3
FIXBOUNDARY  3  INPUT=LINE  /  1 2  /  4 1
*
VIEW  ID=1 XVIEW=1. YVIEW=-0.5 ZVIEW=0.5
SET  NSYMBOLS=MYNODES NNUMBERS=MYNODES VIEW=1
MESH  VECTOR=LOAD
MESH  ENUMBERS=YES BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A.39  ORTHOTROPIC PLATE UNDER PRESSURE LOAD
*
DATABASE  CREATE
*
WRITE  FILENAME='a39.lis'
*
VIEW  ID=1 XVIEW=1. YVIEW=-0.5 ZVIEW=0.5
SET  VIEW=1 OUTLINE=YES NSYMBOLS=MYNODES
MESH  ORIGINAL=YES CONTOUR=MISES VECTOR=REACTION
*
EPLINE  NAME=A-C
 4  2 6 10 3  STEP 4 TO  16  2 6 10 3
EPLINE  NAME=D-C
 13  4 11 7 3  TO  16  4 11 7 3
*
ELINE  LINENAME=A-C KIND=SYY OUTPUT=ALL SUBFRAME=21
ELINE  LINENAME=D-C KIND=SXX OUTPUT=ALL
*
ZONE  NAME=RESULT INPUT=ELEMENTS
 14 15
SHELLSURFACE  LISTRESULTS=TOP
ELIST  ZONENAME=RESULT
*
ZONE  NAME=CEF INPUT=NODES  /  3 5 6
NLIST  ZONENAME=CEF
END
```

EXAMPLE A40

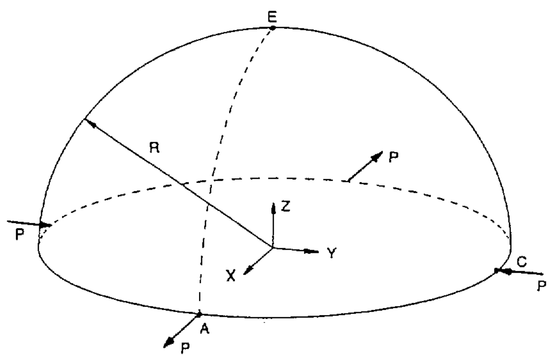
HEMISPHERICAL SHELL UNDER POINT LOADS

Objective

To verify the 4-node SHELL element bending behaviour when applied to a curved structure.

Physical Problem

A hemispherical shell is subjected to concentrated radial loads at its free edge as shown in the figure below. One pair of loads is directed inwards towards the centre of the hemisphere and the other pair of loads is directed outwards away from the centre. This problem is described in [1].



Thickness = 0.04 m

R = 10 m

$E = 68.25 \cdot 10^9$ N/m^2

$\nu = 0.3$

P = 4000 N

Finite Element Model

Due to symmetry only one quarter of the hemisphere need to be considered in the analysis, region A-C-E in the figure above. The finite element mesh is generated in a spherical coordinate system using 4-node SHELL elements. SKEW degree-of-freedom Systems are defined for the symmetry planes X = 0 and Y = 0. Along edge A-E there is symmetry about the XZ-plane. Along edge C-E there is symmetry about the YZ-plane. Edge A-C is free to move. At point E the Z-displacement is fixed. The finite element model is shown in the bottom figure on page A40.2.

Solution Results

The analytical solution for this problem is discussed in [1]. The input data used in the SOLVIA analysis is shown on pages A40.4 and A40.5.

The reported target value from NAFEMS is the outward displacement in the X-direction at point A,

$$u_x = 0.185 \text{ m}$$

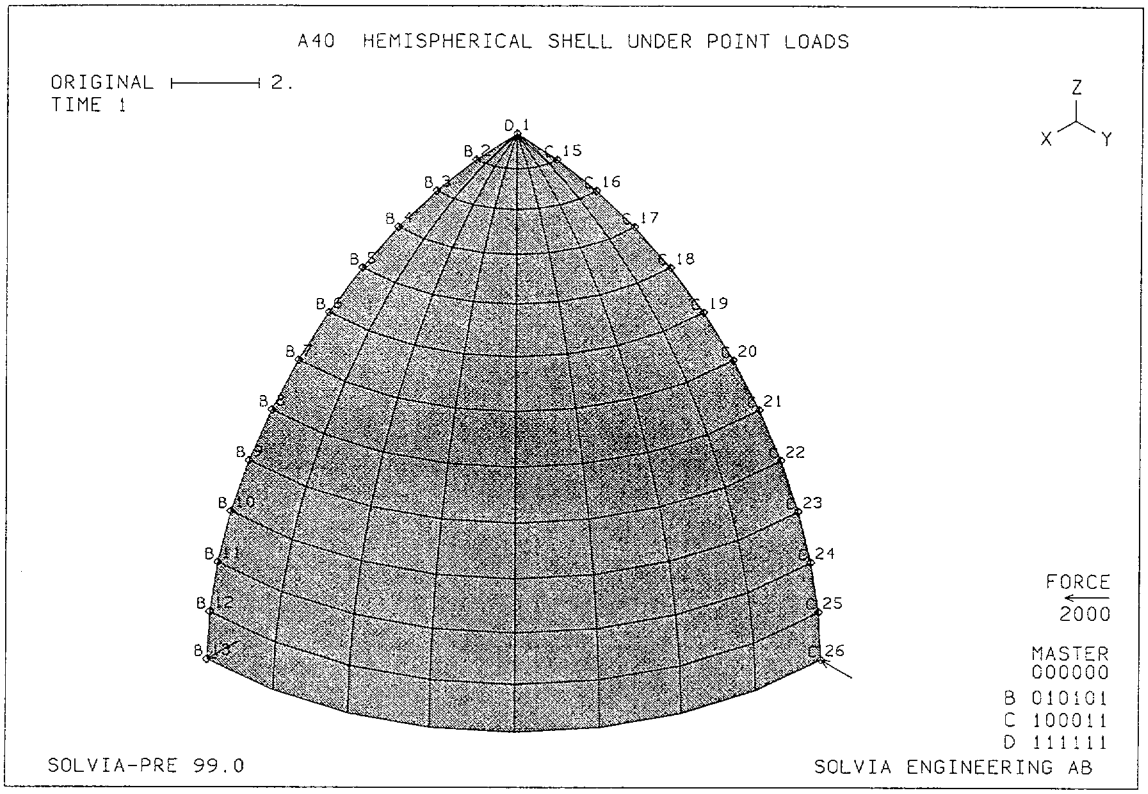
In the SOLVIA analysis the calculated displacement in the X-direction at point A (node 13, radial direction) is

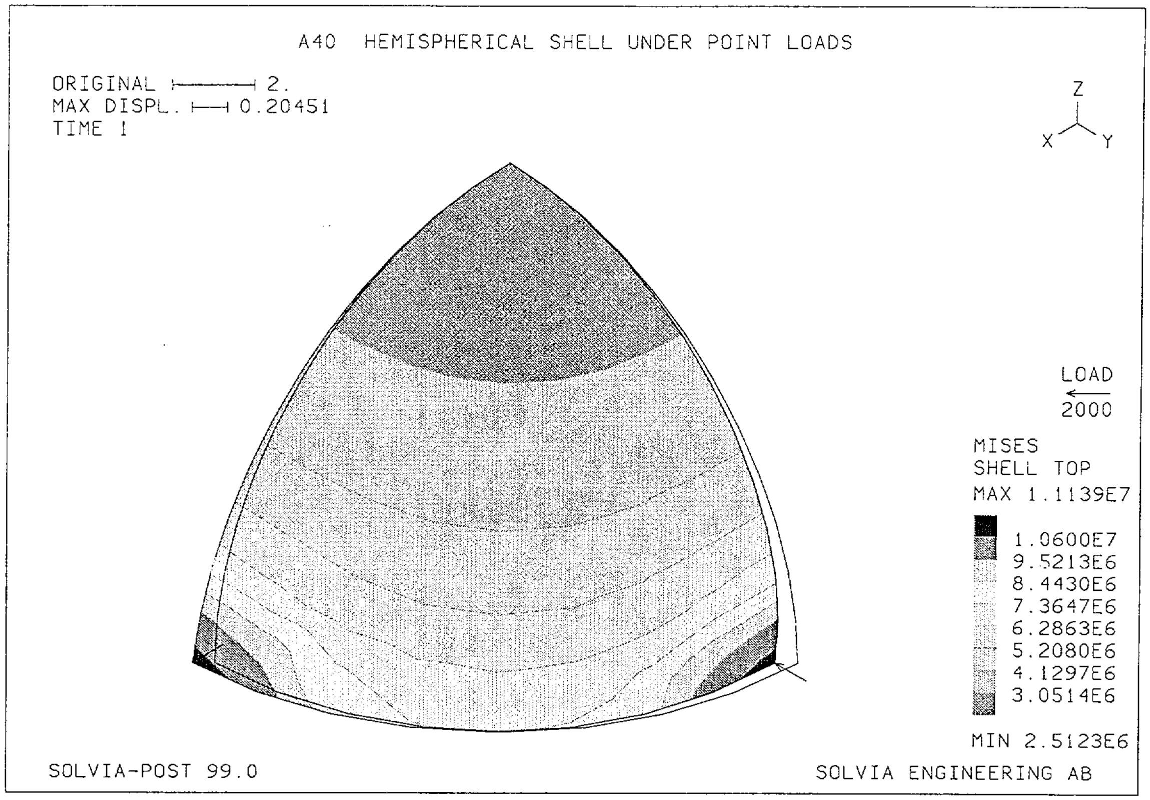
$$u_x = 0.1832 \text{ m}$$

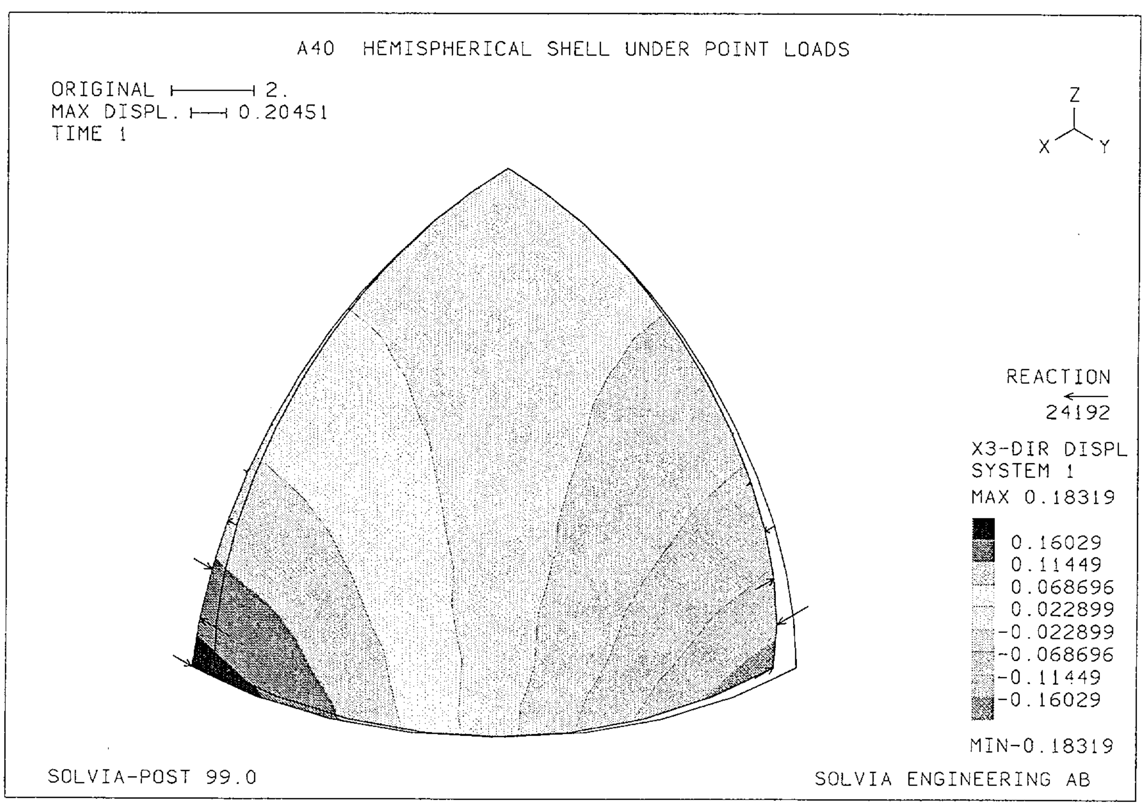
The figures on page A40.3 show a contour plot of the von Mises stress distribution and a contour plot of the radial displacements (in the x_3 direction of the Local Spherical System). Note that the bending in the structure is concentrated to the corners where the loads are acting.

Reference

[1] NAFEMS, Background to Benchmarks, 1993.




A40 HEMISPHERICAL SHELL UNDER POINT LOADS
ORIGINAL 2.
MAX DISPL. 0.20451
TIME 1
Z
X
Y
LOAD
2000
MISES
SHELL TOP
MAX 1.1139E7
1.0600E7
9.5213E6
8.4430E6
7.3647E6
6.2863E6
5.2080E6
4.1297E6
3.0514E6
MIN 2.5123E6
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A40 HEMISPHERICAL SHELL UNDER POINT LOADS
ORIGINAL 2.
MAX DISPL. 0.20451
TIME 1
Z
X
Y
REACTION
24192
X3-DIR DISPL
SYSTEM 1
MAX 0.18319
0.16029
0.11449
0.068696
0.022899
-0.022899
-0.068696
-0.11449
-0.16029
MIN-0.18319
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A40  HEMISPHERICAL SHELL UNDER POINT LOADS'
*
DATABASE  CREATE
*
SYSTEM  1  SPHERICAL
COORDINATES  /  ENTRIES  NODE  R  THETA  PHI
  1  10.  0.  TO  13  10.  0. 90.
 14  10. 90.  TO  26  10. 90. 90.
DELETE  14
SKEWSYSTEMS  EULERANGLES
  1   0.  TO  13    0. 90.
 14   0.  TO  26  -90.
NSKEWS  INPUT=NODES
  1 1  TO  26 26
*
MATERIAL  1  ELASTIC  E=68.25E9 NU=0.3
*
EGROUP  1  SHELL
THICKNESS  1  0.04
GSURFACE   1 13 26 1  EL1=12 EL2=8 NODES=4 SYSTEM=1,
           BLENDING=ANGLES
*
FIXBOUNDARIES  156  /  1  15 TO 26
FIXBOUNDARIES  246  /  1 TO 13
FIXBOUNDARIES  3    /  1
LOADS  CONCENTRATED
 13 3   2000
 26 3 -2000
*
MESH  NSYMBOLS=MYNODES NNUMBERS=MYNODES VECTOR=LOAD BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A40  HEMISPHERICAL SHELL UNDER POINT LOADS
*
DATABASE  CREATE
SYSTEM 1 SPHERICAL
*
WRITE  FILENAME='a40.lis'
*
SET  OUTLINE=YES ORIGINAL=YES
MESH  CONTOUR=MISES VECTOR=LOAD
MESH  CONTOUR=D3    VECTOR=REACTION SYSTEM=1
*
NLIST  ZONENAME=MYNODES DIRECTION=123 SYSTEM=1
SUMMATION  KIND=ENERGY
END
```

EXAMPLE A41

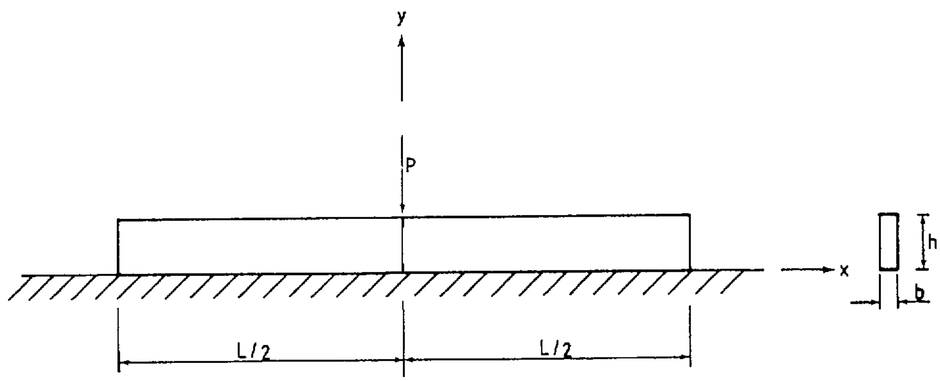
BEAM ON ELASTIC FOUNDATION

Objective

To verify the capability to model a beam supported by an elastic foundation.

Physical Problem

A beam, shown in the figure below, resting on an elastic foundation and subjected to a concentrated transverse load at mid-span is considered.



Beam properties:	$L = 100$ in.	$E = 2.1 \cdot 10^6$ psi
	$b = 1$ in.	$\nu = 0.3$
	$h = 10$ in.	
Foundation stiffness:	$k_o = 5.0 \cdot 10^5$ psi	
Concentrated loads data:	$P = 1.0 \cdot 10^4$ lbf	

Finite Element Model

The finite element model used for the analysis is shown in the figure on page A40.2. Thirteen BEAM elements are used to model one-half of the beam. Axial-translational SPRING elements are used to represent the lumped stiffness of the elastic foundation.

Solution Results

The input data on pages A40.3 and A40.4 is used in the finite element analysis. The analytical solution for the beam on elastic foundation is found in [1].

Deflection at mid-span, (in.):

Theory	SOLVIA
$-1.6348 \cdot 10^{-3}$	$-1.6348 \cdot 10^{-3}$

Bending moment at mid-span, (lbf-in.):

Theory	SOLVIA
$-1.53 \cdot 10^4$	$-1.49 \cdot 10^4$

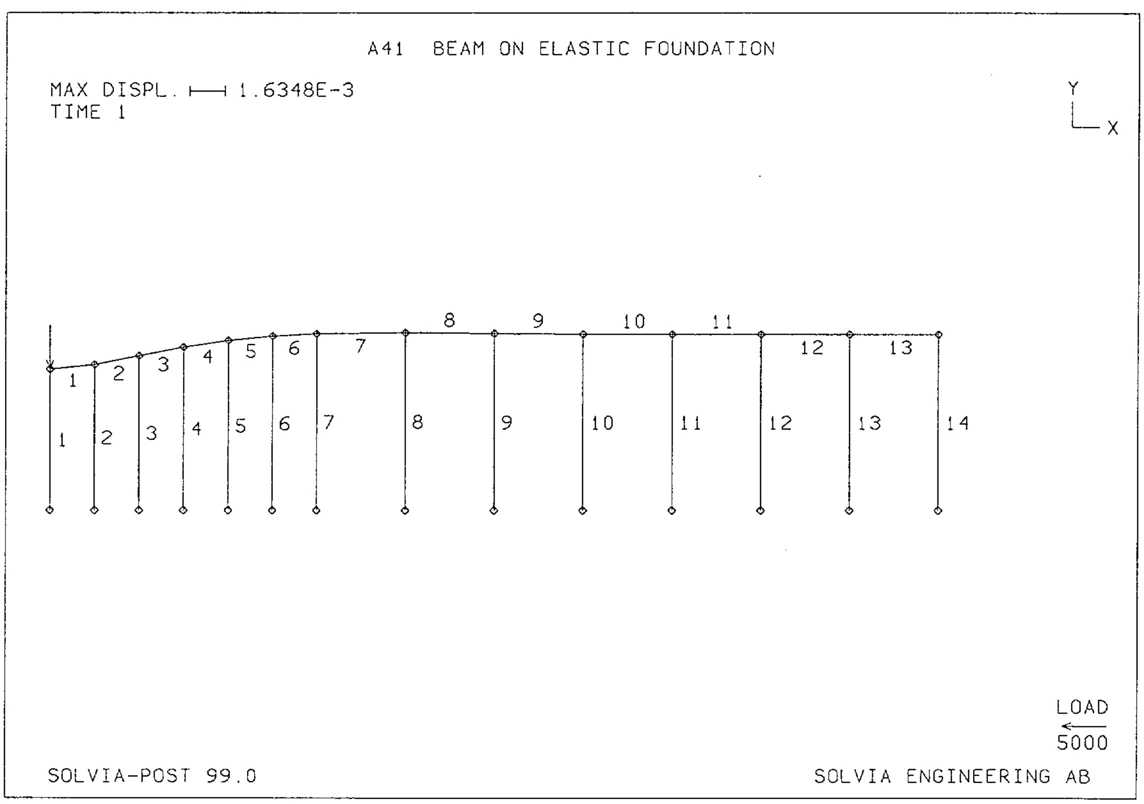
The distribution of the transverse displacement and the bending moment along the beam predicted by the finite element analysis are shown in the figures on page A40.3.

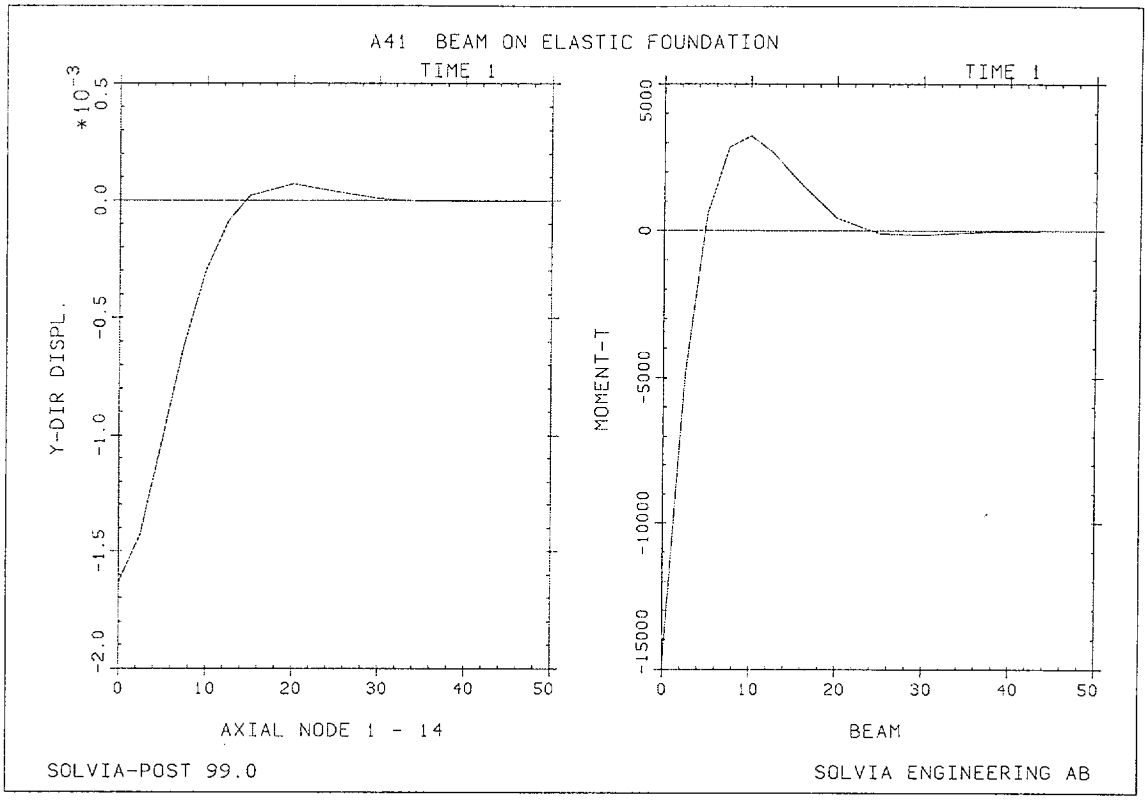
User Hints

- The differential equation for the beam on an elastic foundation is of the same form as the differential equation for an axisymmetric cylinder, see Example A.2. The finite element representation is, however, here accomplished by BEAM and SPRING elements.
- It is important to have a fine enough mesh close to the applied load in this example since the significant displacements occur close to the load and decrease rapidly with increasing distance from the applied load.
- The stiffness of the SPRING elements is calculated based on lumping of the foundation stiffness, so that half of the foundation stiffness under each BEAM element is represented by the SPRING elements at each of the two BEAM end nodes. If an ISOBEAM element with, say, 3 nodes were used, a consistent lumping of the stiffness would be recommended so that, for each ISOBEAM element, 1/6th of the foundation stiffness is attributed to the end nodes, and 4/6th to the midside node.
- Note that a small positive (upward) displacement occurs for a few nodes, see page A40.3. If no tension can develop in the foundation, a nonlinear analysis could be carried out using the nonlinear DISP-FORCE option for the SPRING elements. The foundation material would then be modeled to be elastic in compression but with zero stiffness in tension.

Reference

[1] Timoshenko, S.P., Strength of Materials, Part II, Third Edition, D. Van Nostrand Comp., 1958.





SOLVIA-PRE input

```
HEADING   'A41  BEAM ON ELASTIC FOUNDATION'
*
DATABASE  CREATE
MASTER  IDOF=001110
COORDINATES
  1  0.    0.  TO   7  15.    0.  TO  14  50.
 15  0. -10.  TO  21  15. -10.  TO  28  50. -10.
*
MATERIAL  1 ELASTIC E=2.1E6 NU=0.3
*
EGROUP  1  BEAM  RESULT=FORCES
SECTION  1 GENERAL RINERTIA=3.1233 SINERTIA=0.8333333,
                   TINERTIA=83.333333 AREA=10.
ENODES
 1  15 1 2  TO  13  15 13 14
*
EGROUP  2  SPRING  DIRECTION=AXIALTRANSLATION
PROPERTYSET  1  K=6.25E5
PROPERTYSET  2  K=1.25E6
PROPERTYSET  3  K=1.875E6
PROPERTYSET  4  K=2.50E6
ENODES
 1  1 15  TO  14  14 28
EDATA  /  ENTRIES  EL  PROPERTYSET
 1 1  /  2 2  TO  6 2  /  7 3  /  8 4  TO  13 4  /  14 2
*
```

SOLVIA-PRE input (cont.)

```
FIXBOUNDARIES      /  15   TO   28
FIXBOUNDARIES  16  /  1
LOADS  CONCENTRATED
 1 2 -5000.
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A41  BEAM ON ELASTIC FOUNDATION
*
DATABASE  CREATE
*
WRITE  FILENAME='a41.lis'
*
MESH  VIEW=Z ENUMBER=YES NSYMBOL=YES VECTOR=LOAD
*
NPLINE  NAME=AXIAL  /  1  TO  14
NLINE  LINENAME=AXIAL DIRECTION=2 SUBFRAME=21
*
EGROUP  1
EPLINE  NAME=BEAM
 1  1 2  TO  13  1 2
ELINE  LINENAME=BEAM KIND=MT OUTPUT=ALL
*
NLIST
ELIST
END
```

EXAMPLE A42

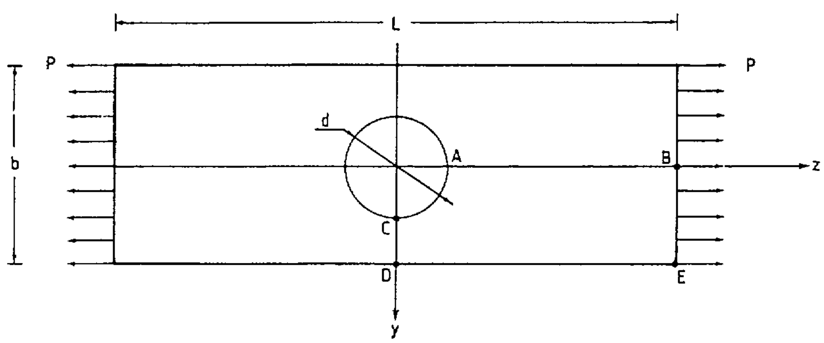
PERFORATED TENSION STRIP

Objective

To verify the performance of the PLANE STRESS element when employed to model stress concentrations around holes.

Physical Problem

The rectangular perforated strip, shown in the figure below, subjected to a uniform tension at two opposite sides, is considered.



L = 56 mm
b = 20 mm
d = 10 mm
h = 1 mm (thickness)

$E = 7.0 \cdot 10^4$ N/m^2
$\nu = 0.25$
$p = 25.0$ N/m^2

Finite Element Model

The finite element model considered for the analysis is shown on page A42.3. Using symmetry considerations, one-quarter of the strip is modeled with thirty 8-node PLANE STRESS elements.

Solution Results

The analytical solution for the stress σ_{zz} at location C and D in the figure above is given in [1].

The input data on pages A42.7 and A42.8 gives the following results:

Stress σ_{zz} at location C, $\left(\mathrm{N/mm^2}\right)$:

Theory	SOLVIA
107.5	108.6

Stress σ_{zz} at location D, $\left(\mathrm{N/mm^2}\right)$:

Theory	SOLVIA
18.75	16.49

The calculated stress values are output at nodes 5 and 6, which correspond to the locations C and D, respectively.

The contour lines of von Mises effective stress and the mean stress (negative hydrostatic pressure) are shown on page A42.4. The mean stress is defined as

$$\sigma_{\text{mean}} = \tfrac{1}{3}\left(\sigma_{xx} + \sigma_{yy} + \sigma_{zz}\right)$$

The contour lines of maximum deviation from nodal mean values of effective stress and pressure are also shown on page A42.4. The left top figure on page A42.5 shows the sum of the effective stress deviation and the pressure deviation. A contour plot of the displacement is also shown on page A42.5. The stress σ_{zz} along the symmetry line C-D and the radial displacement along a line from A to C are shown in bottom figure on page A42.5.

For comparison, the contour lines of effective stress and deviation of effective stress are shown on page A42.9 for a refined mesh. We may note that the maximum value of effective stress deviation have decreased about a factor of 4 compared to the corresponding value on page A42.4 for the coarser model.

The value of σ_{zz} for the finer mesh at locations C and D are 108.8 $\left(N / mm^2\right)$ and 17.55 $\left(N / mm^2\right)$, respectively.

The figures on page A42.6 show principal stresses and Tresca effective stress in the first finite element model as displayed by SOLVIA-POST when element stresses are output at integration points by SOLVIA.

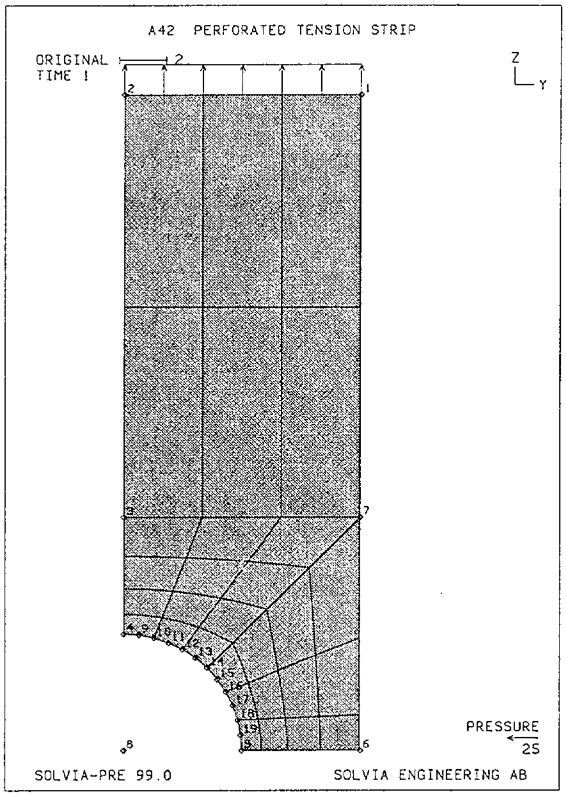
User Hints

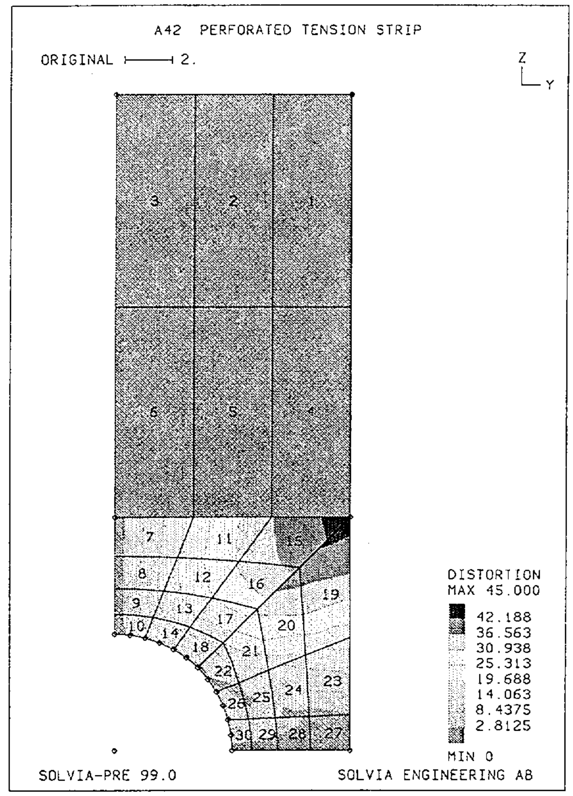
- The stress deviation plots may be used as a tool for determining whether a good enough mesh has been used. In this example one can see that elements, which are distorted from the rectangular shape, see figure on page A42.3, give a higher stress and pressure deviation. Note that no deviation can occur, for example, at the locations C and D since these nodes are coupled only to one element so that the mean value at these nodes is based only on one stress value. A deviation with value zero can, therefore, not be used as a proof that the corresponding stress value is associated with zero error. An exact solution would have zero stress deviations but requires also that other criteria are satisfied. One such criterion is that the boundary conditions for stresses must be satisfied [2].

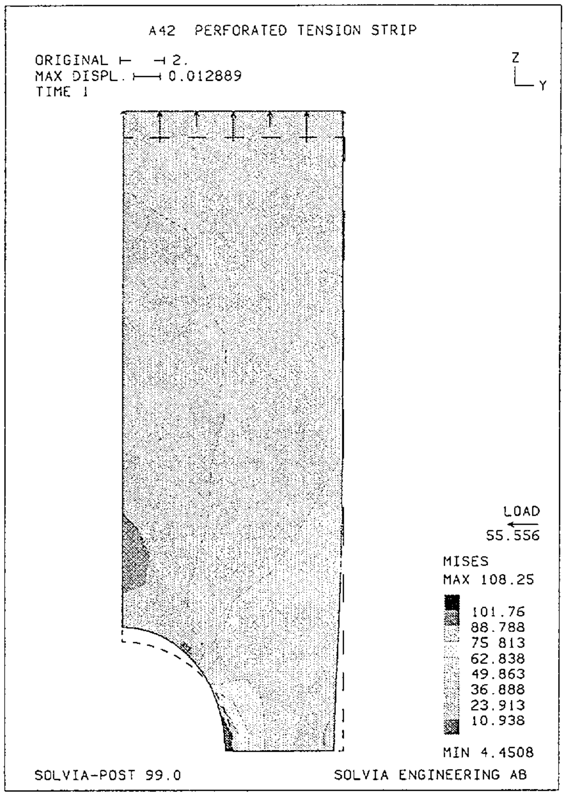
References

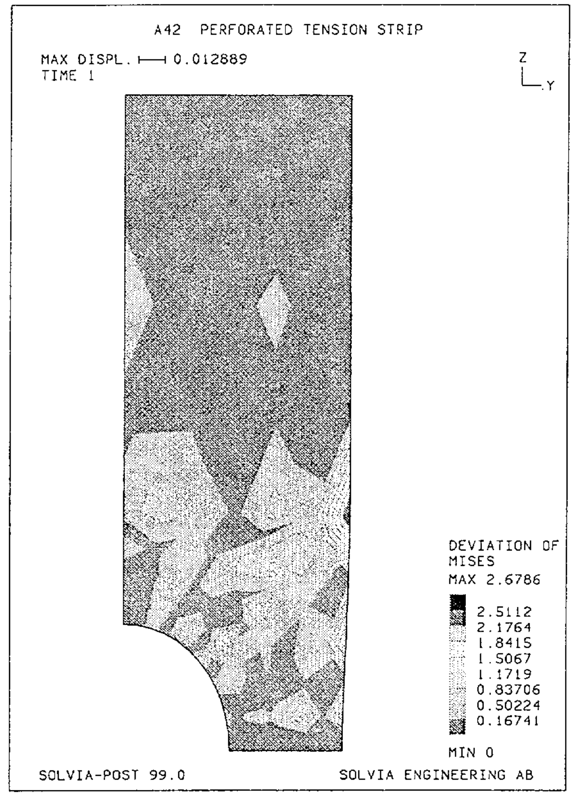
[1] Timoshenko, S.P., and Goodier, J.N., Theory of Elasticity, Third Edition, McGraw-Hill, 1970, pp. 94-95.

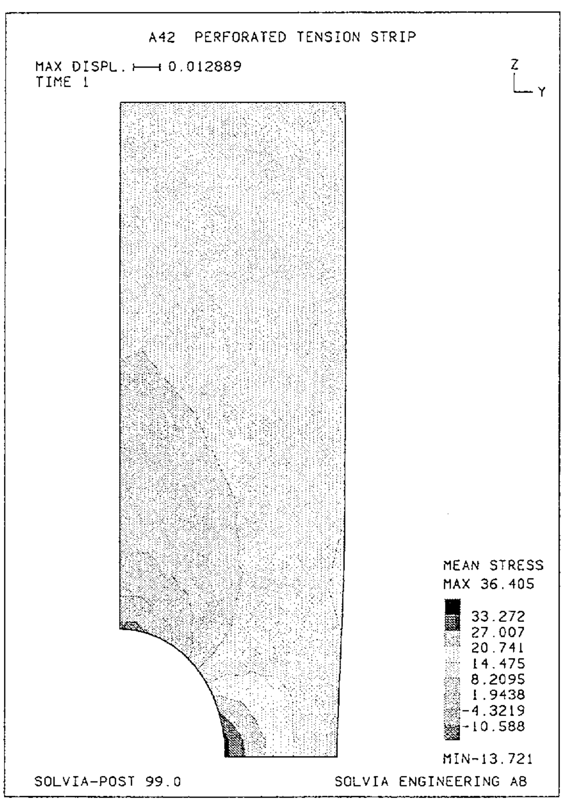
[2] Larsson, G., and Olsson, H., "An Engineering Error Measure for Finite Element Analysis", Finite Element News, April, 1988.

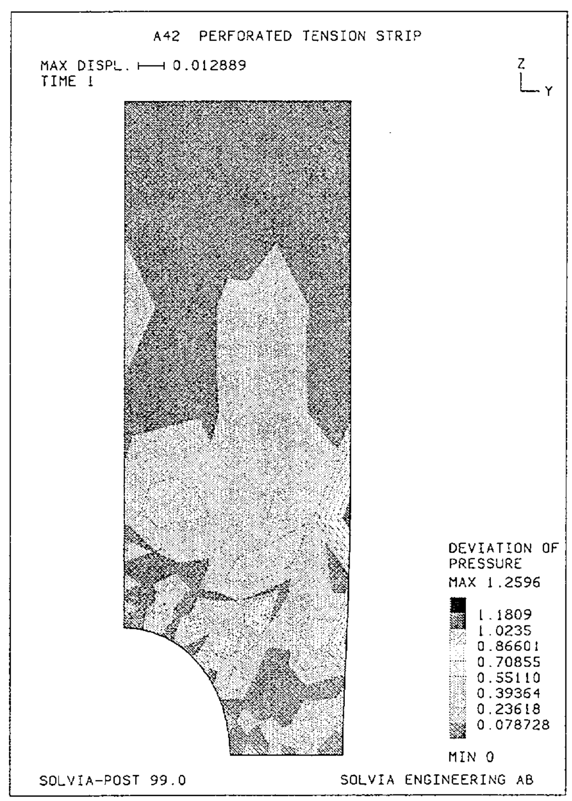

A42 PERFORATED TENSION STRIP
ORIGINAL 2.
TIME 1
Z
Y
PRESSURE
25
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

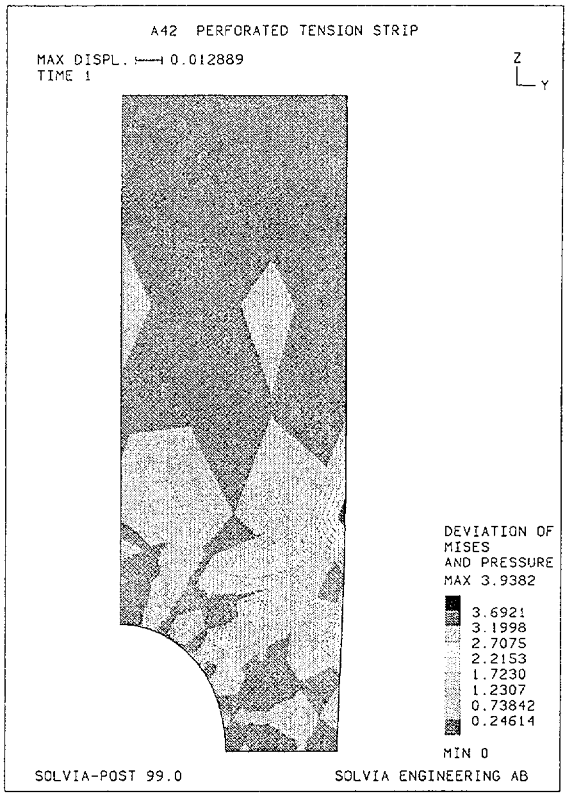

A42 PERFORATED TENSION STRIP
ORIGINAL 2.
Z
Y
DISTORTION
MAX 45.000
42.188
36.563
30.938
25.313
19.688
14.063
8.4375
2.8125
MIN 0
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

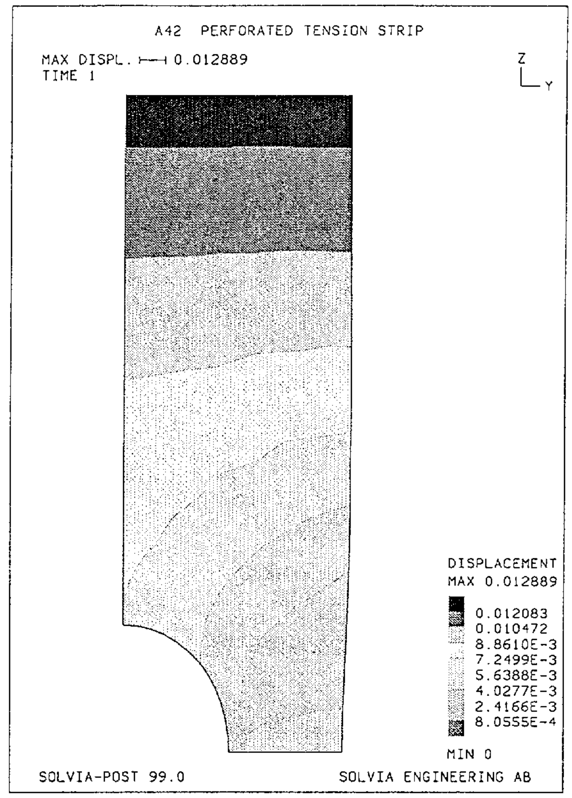

A42 PERFORATED TENSION STRIP
ORIGINAL 2.
MAX DISPL. 0.012889
TIME 1
Z
Y
LOAD
55.556
MISES
MAX 108.25
101.76
88.788
75.813
62.838
49.863
36.888
23.913
10.938
MIN 4.4508
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

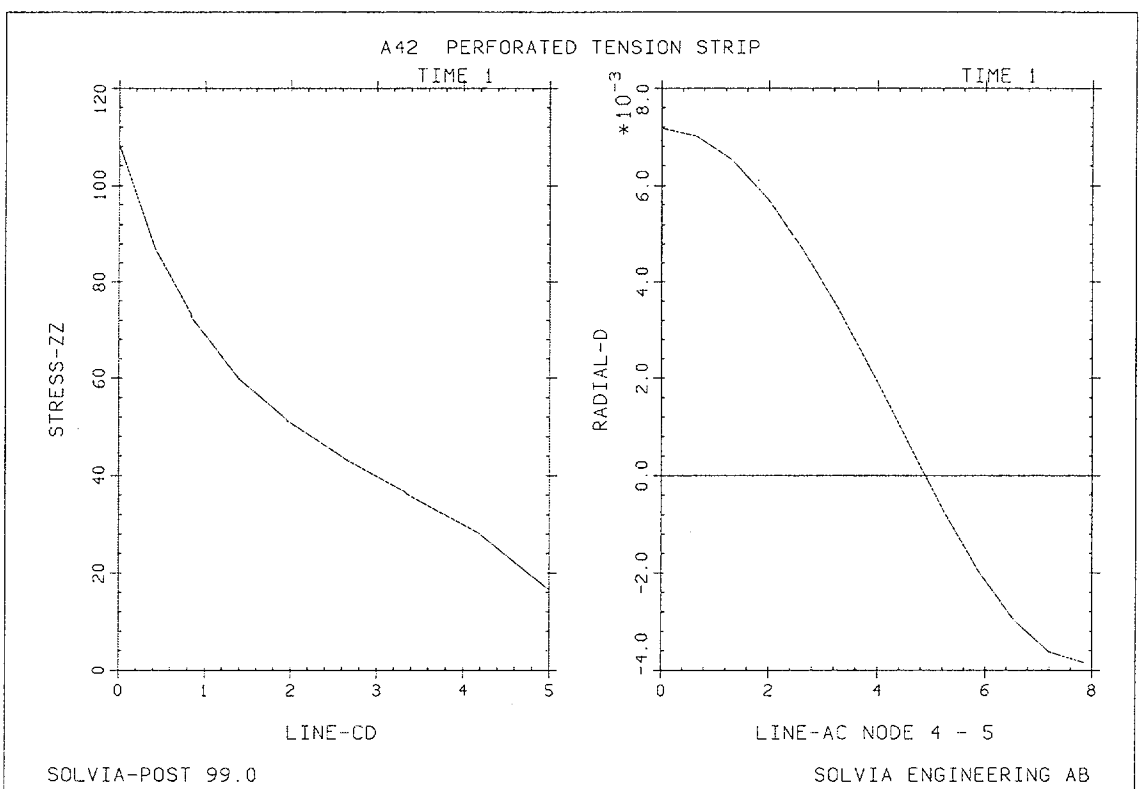

A42 PERFORATED TENSION STRIP
MAX DISPL. 0.012889
TIME 1
Z
Y
DEVIATION OF
MISES
MAX 2.6786
2.5112
2.1764
1.8415
1.5067
1.1719
0.83706
0.50224
0.16741
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

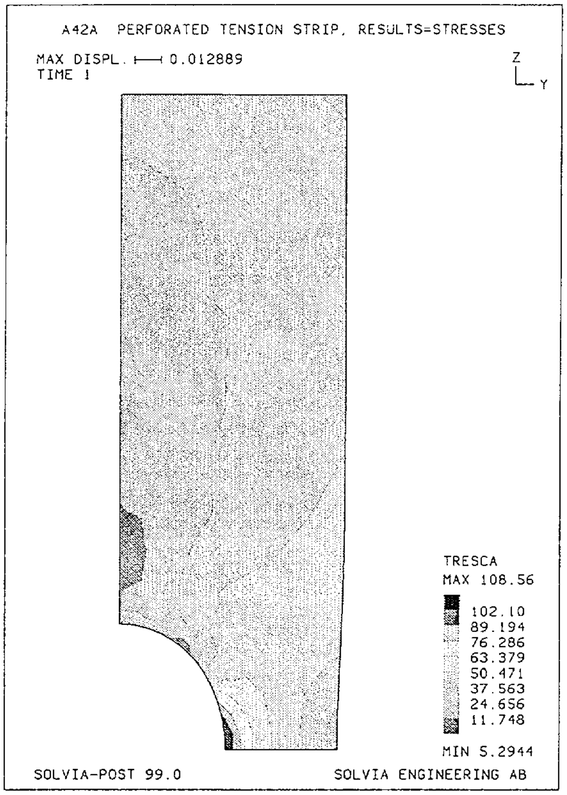

A42 PERFORATED TENSION STRIP
MAX DISPL. 0.012889
TIME 1
Z
Y
MEAN STRESS
MAX 36.405
33.272
27.007
20.741
14.475
8.2095
1.9438
-4.3219
-10.588
MIN-13.721
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

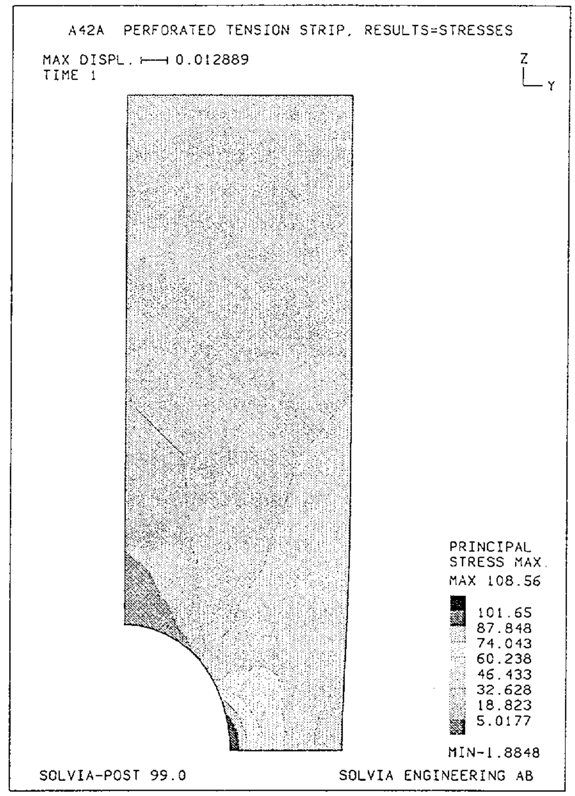

A42 PERFORATED TENSION STRIP
MAX DISPL. 0.012889
TIME 1
Z
Y
DEVIATION OF
PRESSURE
MAX 1.2596
1.1809
1.0235
0.86601
0.70855
0.55110
0.39364
0.23618
0.078728
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

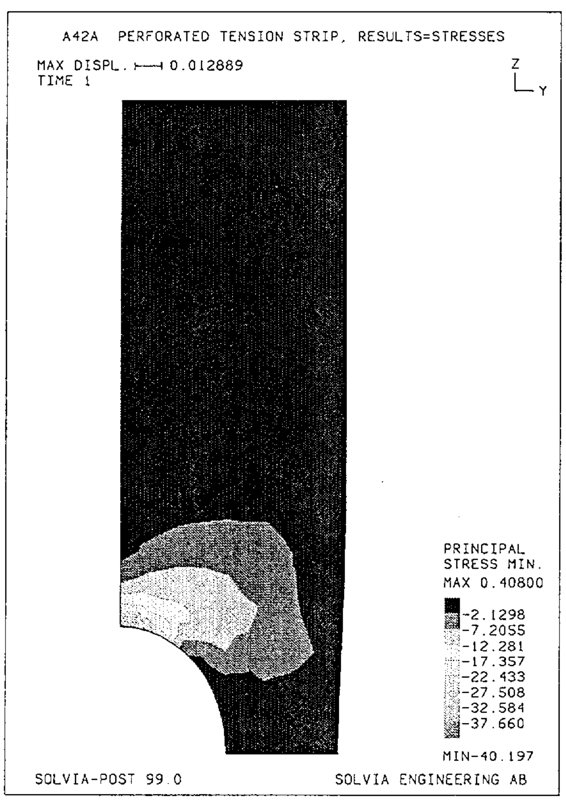

A42 PERFORATED TENSION STRIP
MAX DISPL. 0.012889
TIME 1
Z
Y
DEVIATION OF
MISES
AND PRESSURE
MAX 3.9382
3.6921
3.1998
2.7075
2.2153
1.7230
1.2307
0.73842
0.24614
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

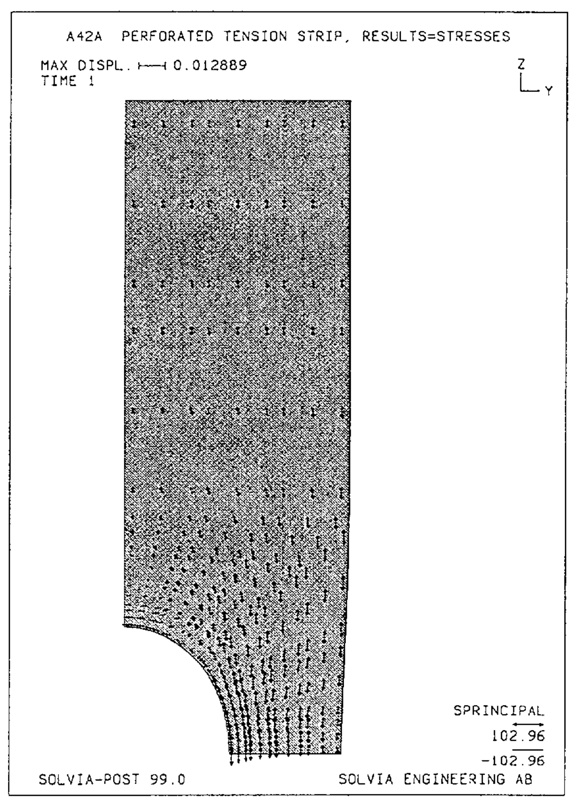

A42 PERFORATED TENSION STRIP
MAX DISPL. 0.012889
TIME 1
Z
Y
DISPLACEMENT
MAX 0.012889
0.012083
0.010472
8.8610E-3
7.2499E-3
5.6388E-3
4.0277E-3
2.4166E-3
8.0555E-4
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A42 PERFORATED TENSION STRIP
TIME 1
STRESS-ZZ
0
20
40
60
80
100
120
0
1
2
3
4
5
LINE-CD
TIME 1
*10^-3
RADIAL-D
-4.0
-2.0
0.0
2.0
4.0
6.0
8.0
0
2
4
6
8
LINE-AC NODE 4 - 5
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A42A PERFORATED TENSION STRIP, RESULTS=STRESSES
MAX DISPL. 0.012889
TIME 1
Z
Y
TRESCA
MAX 108.56
102.10
89.194
76.286
63.379
50.471
37.563
24.656
11.748
MIN 5.2944
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A42A PERFORATED TENSION STRIP, RESULTS=STRESSES
MAX DISPL. 0.012889
TIME 1
Z
Y
PRINCIPAL
STRESS MAX.
MAX 108.56
101.65
87.848
74.043
60.238
46.433
32.628
18.823
5.0177
MIN-1.8848
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A42A PERFORATED TENSION STRIP, RESULTS=STRESSES
MAX DISPL. 0.012889
TIME 1
Z
Y
PRINCIPAL
STRESS MIN.
MAX 0.40800
-2.1298
-7.2055
-12.281
-17.357
-22.433
-27.508
-32.584
-37.660
MIN-40.197
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

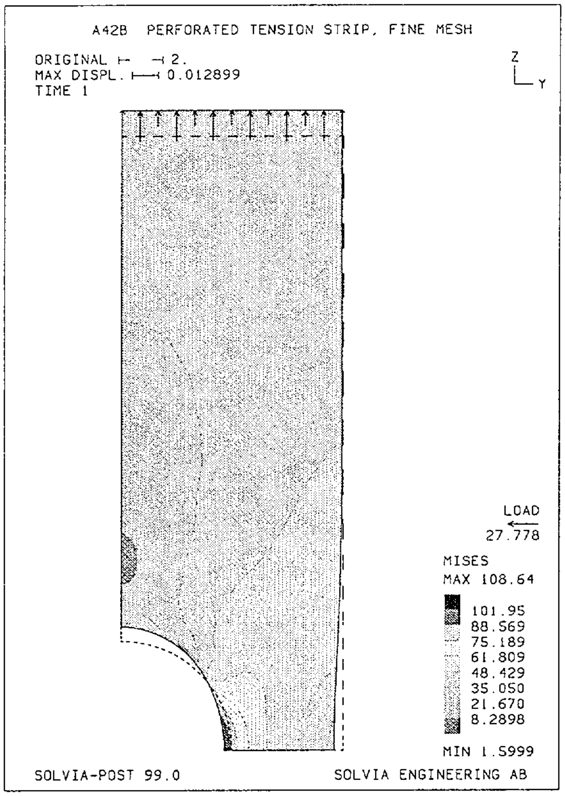

A42A PERFORATED TENSION STRIP, RESULTS=STRESSES
MAX DISPL. 0.012889
TIME 1
Z
Y
SPRINCIPAL
102.96
-102.96
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

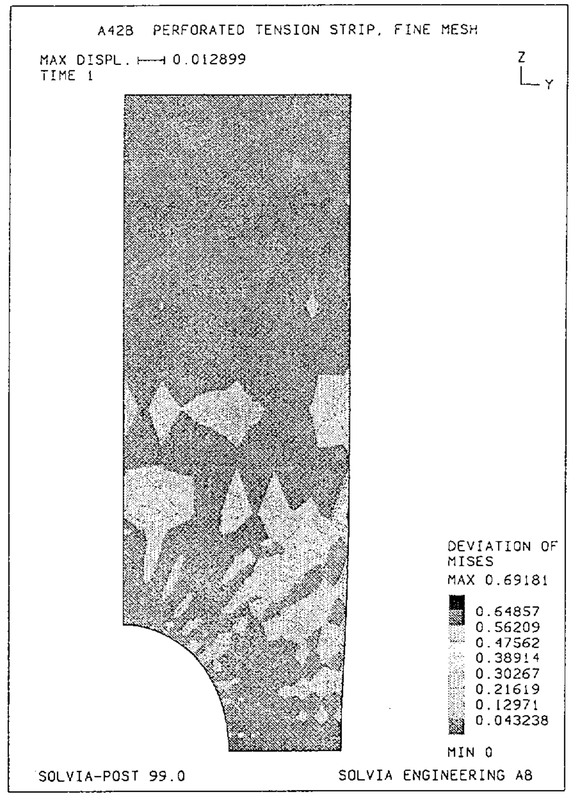
SOLVIA-PRE input

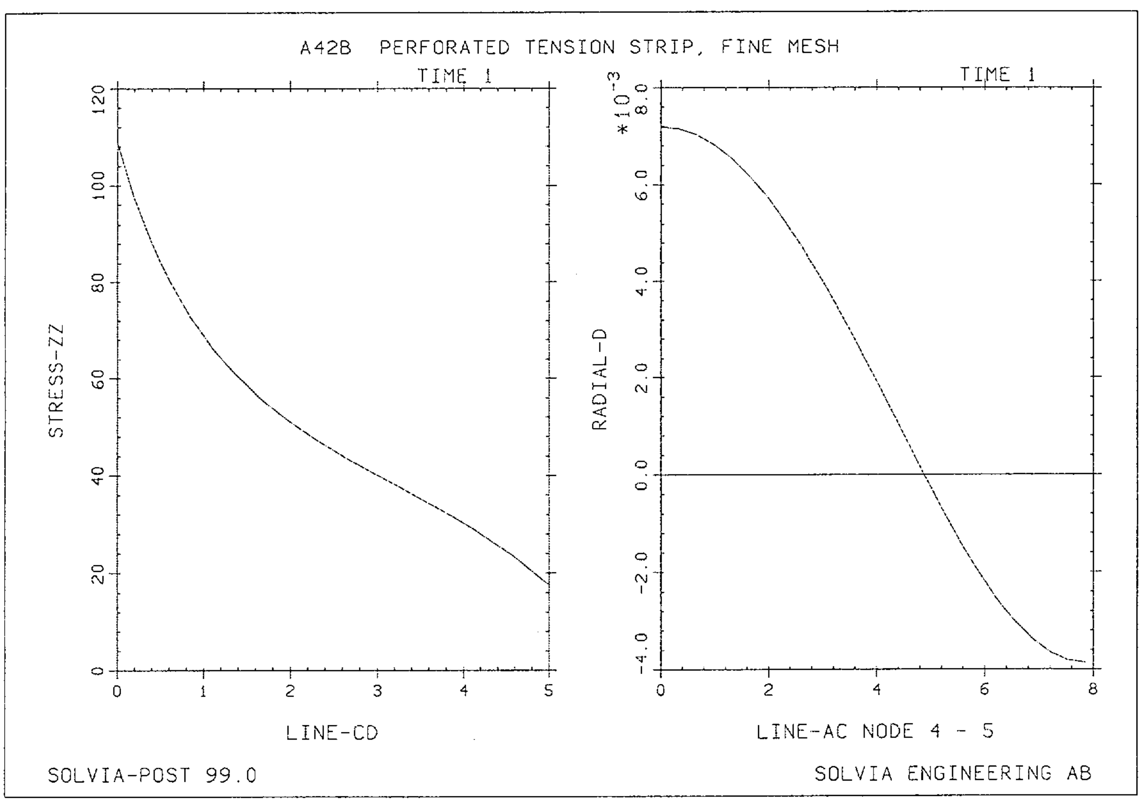
```
HEADING  'A42  PERFORATED TENSION STRIP'
*
DATABASE  CREATE
MASTER  IDOF=100111
*
COORDINATES
 ENTRIES  NODES     Y      Z
           1       10.    28.
           2        0.    28.
           3        0.    10.
           4        0.     5.
           5        5.     0.
           6       10.     0.
           7       10.    10.
           8        0.     0.
*
LINE  ARC        4 5  EL=6 MIDNODES=1 NCENTER=8 NFIRST=9
LINE  STRAIGHT   3 7  EL=3 MIDNODES=1
LINE  STRAIGHT   7 6  EL=3 MIDNODES=1 RATIO=0.1
LINE  COMBINED   3 6 7
*
MATERIAL  1  ELASTIC  E=7.0E4 NU=0.25
*
EGROUP  1  PLANE STRESS2  RESULTS=NSTRESSES
GSURFACE  1 2 3 7  EL1=3 EL2=2 NODES=8
GSURFACE  3 4 5 6  EL1=4 EL2=6 NODES=8
EDATA  /  1 1.
*
FIXBOUNDARIES  2 INPUT=LINES  /  2 3  /  3 4
FIXBOUNDARIES  3 INPUT=LINES  /  5 6
FIXBOUNDARIES  /  8
*
LOADS  ELEMENT  INPUT=LINE
 1 2 -25.
*
SET  PLOTORIENTATION=PORTRAIT NSYMBOLS=MYNODES
MESH  NNUMBERS=MYNODES VECTOR=LOAD
MESH  CONTOUR=DISTORTION ENUMBERS=YES
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A42  PERFORATED TENSION STRIP
*
DATABASE  CREATE
WRITE  FILENAME='a42.lis'
*
SET  PLOTORIENTATION=PORTRAIT
SET  OUTLINE=YES
MESH  ORIGINAL=DASHED CONTOUR=MISES VECTOR=LOAD
MESH  CONTOUR=SDEVIATION
*
MESH  CONTOUR=SMEAN
MESH  CONTOUR=PDEVIATION
*
MESH  CONTOUR=SPDEVIATION
MESH  CONTOUR=DISPLACEMENTS
*
EPLINE  NAME=LINE-CD
 30  3 7 4  TO  27  3 7 4
SET  PLOTORIENTATION=LANDSCAPE
ELINE   LINENAME=LINE-CD KIND=SZZ SUBFRAME=21
*
NPLINE  NAME=LINE-AC
 4  9  TO  19  5
NVARIABLE  NAME=Y  DIRECTION=2 KIND=COORDINATE
NVARIABLE  NAME=Z  DIRECTION=3 KIND=COORDINATE
NVARIABLE  NAME=DY DIRECTION=2 KIND=DISPLACEMENT
NVARIABLE  NAME=DZ DIRECTION=3 KIND=DISPLACEMENT
RESULTANT  NAME=RADIAL-D STRING='(DY*Y+DZ*Z)/(SQRT(Y*Y+Z*Z))'
*
RLINE  LINENAME=LINE-AC RESULTANTNAME=RADIAL-D
*
ZONE  NAME=EDGE INPUT=ELEMENTS  /  27  TO  30
ELIST  ZONENAME=EDGE
END
```


A42B PERFORATED TENSION STRIP, FINE MESH
ORIGINAL ⊢ ⊣ 2.
MAX DISPL. ⊢⊣ 0.012899
TIME 1
Z
Y
LOAD
27.778
MISES
MAX 108.64
101.95
88.569
75.189
61.809
48.429
35.050
21.670
8.2898
MIN 1.5999
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A42B PERFORATED TENSION STRIP, FINE MESH
MAX DISPL. ⊢⊣ 0.012899
TIME 1
Z
Y
DEVIATION OF
MISES
MAX 0.69181
0.64857
0.56209
0.47562
0.38914
0.30267
0.21619
0.12971
0.043238
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A42B PERFORATED TENSION STRIP, FINE MESH
TIME 1
STRESS-ZZ
LINE-CD
TIME 1
RADIAL-D
*10^-3
LINE-AC NODE 4 - 5
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

EXAMPLE A43

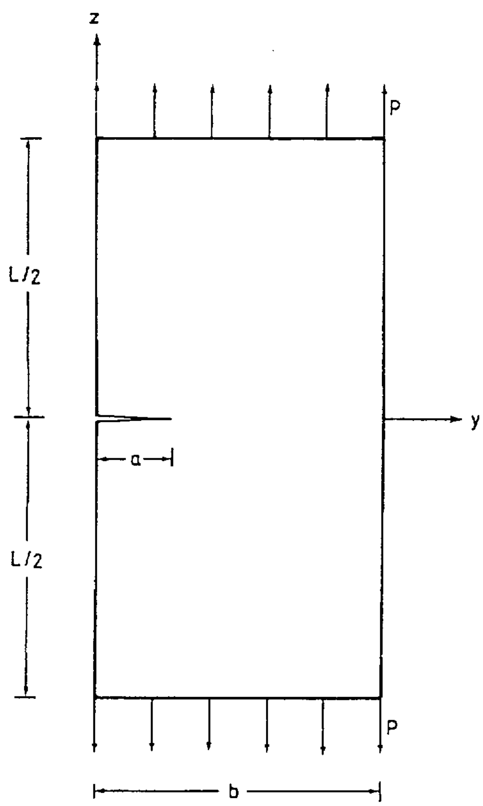
FRACTURE MECHANICS ANALYSIS OF A TENSILE SPECIMEN

Objective

To verify the performance of the PLANE STRAIN element when used for linear elastic fracture mechanics analysis.

Physical Problem

The tensile specimen with an edge crack, as shown in the figure below, is analyzed for its elastic response. The stress intensity factor K_I [1] is to be determined for the given geometry and load conditions.



$L = 0.32$ m
$b = 0.16$ m
$a = 0.040$ m
$h = 1.0$ m (thickness)
$E = 2.0 \cdot 10^{11}$ N/m^2
$\nu = 0.29$
$p = 1.0 \cdot 10^3$ N/m^2

Finite Element Model

The finite element model considered is shown on page A43.4. Twenty 8-node PLANE STRAIN elements are used to model the top half of the specimen. At the crack tip, node 11 in the model, six quarter-point triangular elements with $1/\sqrt{R}$ stress singularities are used.

Solution Results

The analytical solution for K_I is as follows:

$$K_I = \sqrt{\frac{E}{(1-\nu^2)} \cdot G}$$

and

$$G = -\frac{d\Pi}{dA}$$

where

G = energy release rate

E = Young's modulus

ν = Poisson's ratio

dA = change in crack area

Π = total potential energy

The strain energy release rate at the crack tip node is obtained as

$$G = -\frac{\partial \Pi}{\partial x_i^c} \cdot \frac{a_i^c}{t}$$

where

x_i^c = coordinates of the crack tip nodal point (node 11 in this example)

a_i^c = components of the unit vector in the direction of crack propagation

t = 1.0 = specimen thickness

The following SOLVIA numerical solution is obtained by using the input data on pages A43.6 and A43.7:

$$\frac{\partial \Pi}{\partial y} = -\ 0.627677 \cdot 10^{-6}$$

The above value represents only the contribution of the material in the upper half of the specimen. Thus, the total energy release rate, G, is obtained as

$$G = 2\left(0.627677 \cdot 10^{-6}\right)\frac{1.0}{1.0} = 0.125535 \cdot 10^{-5}$$

This yields that

$$K_I = 532.7$$

The value compares very well with the reference solution $K_I = 531.7$ given in ref. [1].

The deformed mesh is shown on page A43.5 as displayed by SOLVIA-POST.

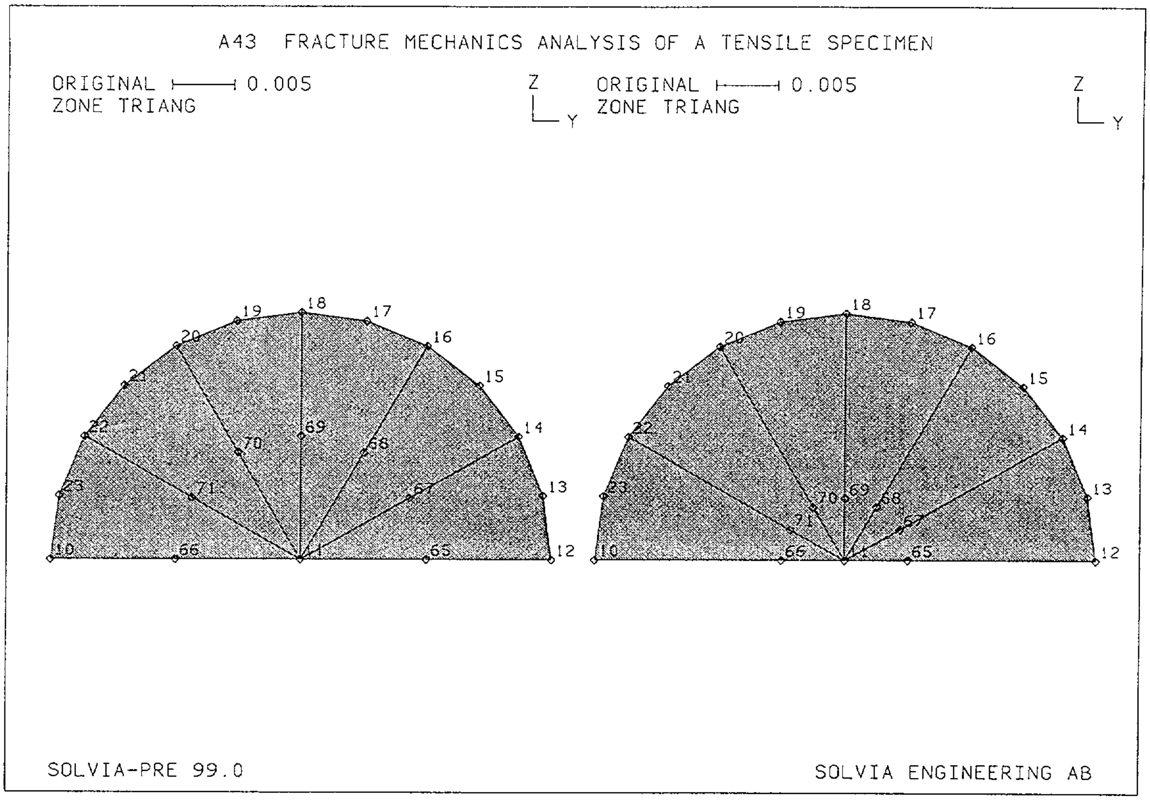
User Hints

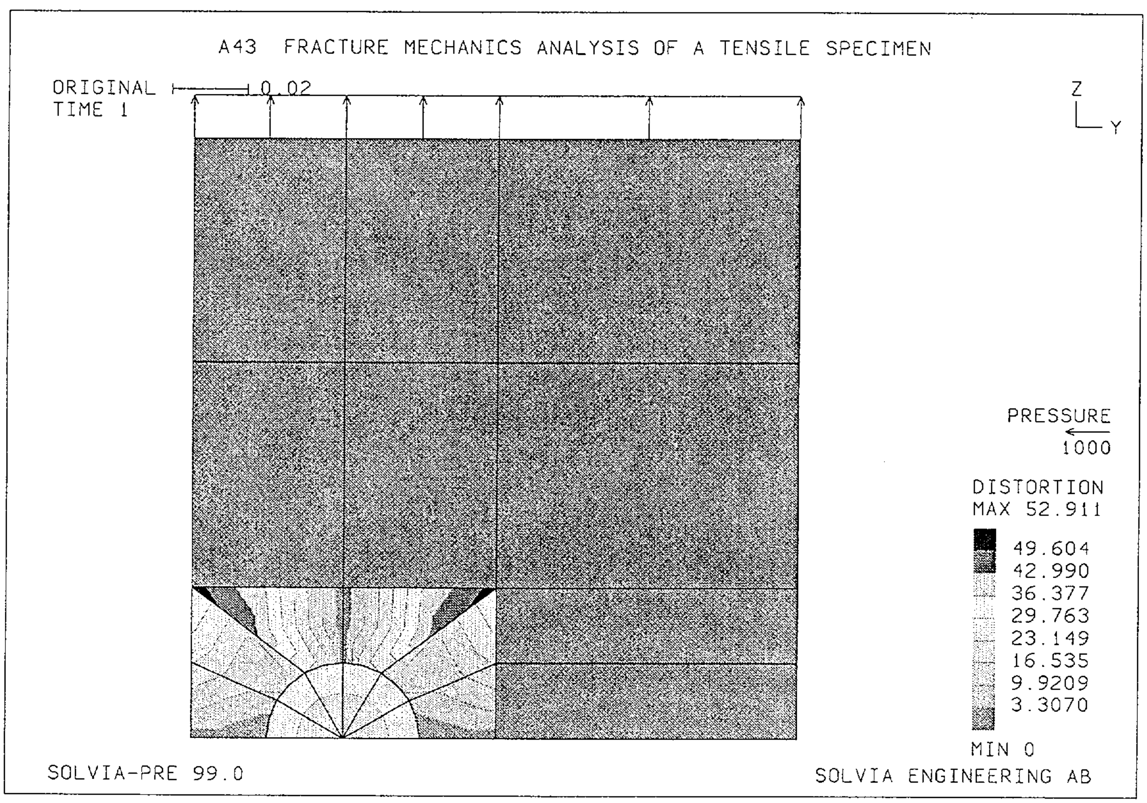
- Accurate results are obtained in this solution, although a coarse finite element discretization is used.

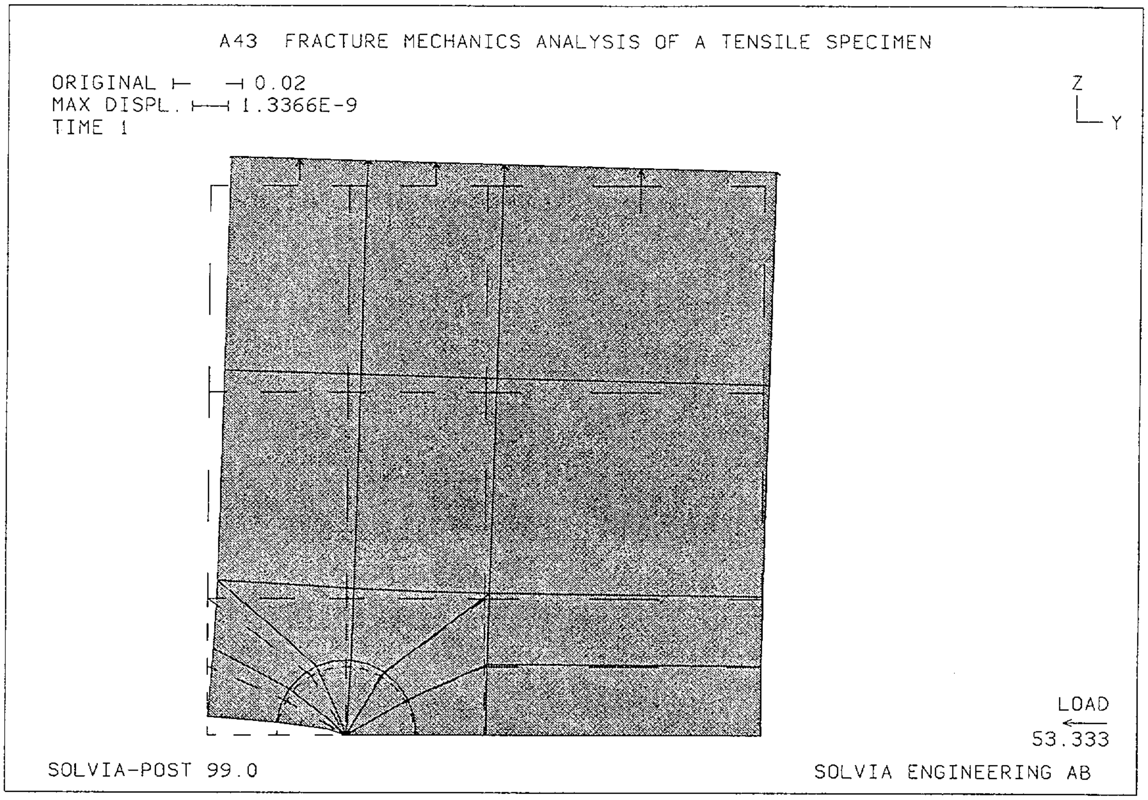
- Note that the zone definitions made in SOLVIA-PRE prior to the SOLVIA command can also be used in SOLVIA-POST. The zone definitions are transferred by the SOLVIA program to the porthole file. When loading the SOLVIA-POST database the zone definitions are stored in the database.

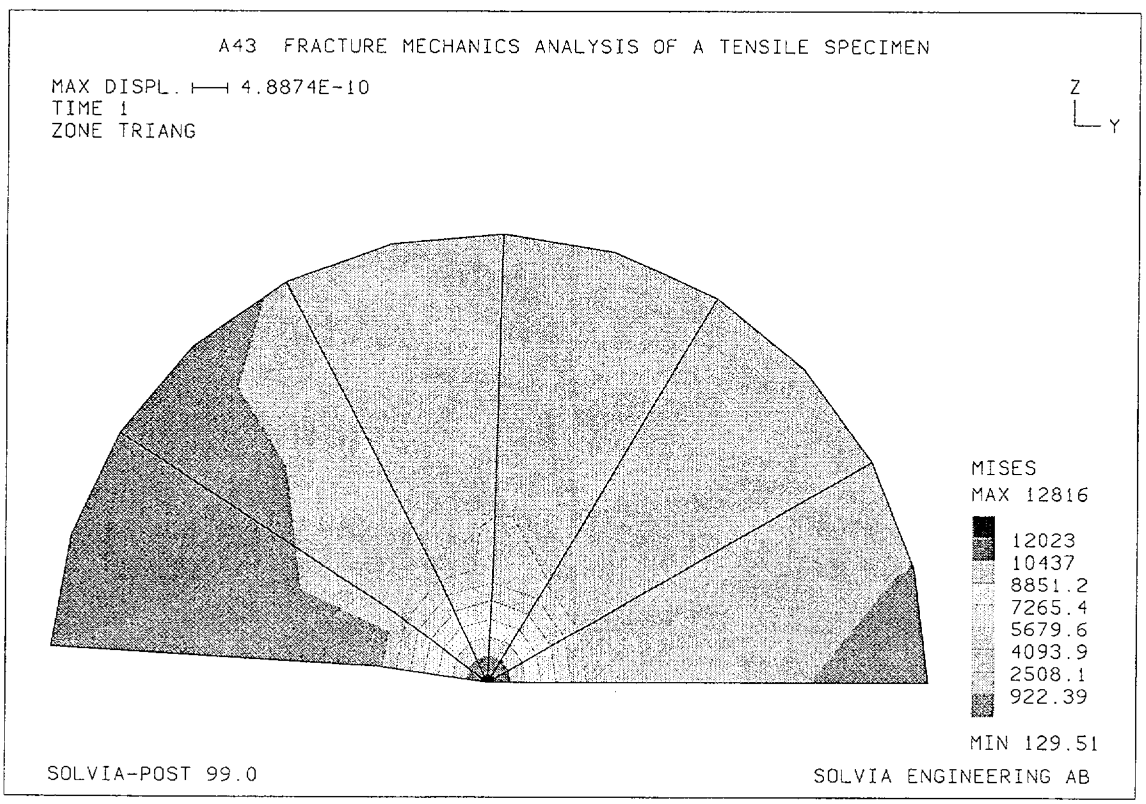
Reference

[1] Tada, H., Paris, P.C. and Irwin, G.R., The Stress Analysis of Cracks Handbook, Del Research Corp., Hellertown, PENN., 1973.


A43 FRACTURE MECHANICS ANALYSIS OF A TENSILE SPECIMEN
ORIGINAL 0.005
ZONE TRIANG
Z
Y
ORIGINAL 0.005
ZONE TRIANG
Z
Y
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A43 FRACTURE MECHANICS ANALYSIS OF A TENSILE SPECIMEN
ORIGINAL 0.02
TIME 1
Z
Y
PRESSURE
1000
DISTORTION
MAX 52.911
49.604
42.990
36.377
29.763
23.149
16.535
9.9209
3.3070
MIN 0
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A43 FRACTURE MECHANICS ANALYSIS OF A TENSILE SPECIMEN
ORIGINAL ⊢ ⊣ 0.02
MAX DISPL. ⊢⊣ 1.3366E-9
TIME 1
Z
Y
LOAD
53.333
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A43 FRACTURE MECHANICS ANALYSIS OF A TENSILE SPECIMEN
MAX DISPL. ⊢⊣ 4.8874E-10
TIME 1
ZONE TRIANG
Z
Y
MISES
MAX 12816
12023
10437
8851.2
7265.4
5679.6
4093.9
2508.1
922.39
MIN 129.51
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A43  FRACTURE MECHANICS ANALYSIS OF A TENSILE SPECIMEN'
*
DATABASE  CREATE
MASTER  IDOF=100111
COORDINATES  /  ENTRIES  NODE   Y    Z
                          1    .08  .04
                          2    .08
                          3    .16
                          4    .16  .04
                          5    .16  .16  TO
                          7    .0   .16
                          8    .0   .04
                          9    .0
SYSTEM  1  CYLINDRICAL  Y=0.04
COORDINATES  /  ENTRIES  NODE   R    THETA
                          10   .02  180.
                          11   .0     0.
                          12   .02    0.
SET  MYNODES=0 NODES=8
LINE  CYLINDRICAL  12 10 EL=6 MIDNODES=1
*
MATERIAL  1 ELASTIC E=2.07E11 NU=0.29
*
EGROUP  1  PLANE STRAIN
GSURFACE  1 2 3 4  EL1=2 EL2=1
GSURFACE  6 1 4 5  EL1=2 EL2=1
GSURFACE  7 8 1 6  EL1=2 EL2=2
LINE  STRAIGHT  8 9  EL=2 MIDNODES=1
LINE  COMBINED  9 2 8 1
GSURFACE  9  10 12 2   EL1=1 EL2=6
GSURFACE  11 12 10 11  EL1=1 EL2=6
*
ZONE  NAME=TRIANG INPUT=CYLINDRICAL-LIMITS SYSTEM=1 RMAX=0.02
MESH  ZONENAME=TRIANG NSYMBOLS=YES NNUMBERS=YES SUBFRAME=21
COORDINATES  / ENTRIES  NODE   R    THETA
                         65   .005    0
                         67   .005   30  TO
                         71   .005  150
                         66   .005  180
MESH  ZONENAME=TRIANG NNUMBERS=YES NSYMBOLS=YES
*
STRAINENERGY  11
*
LOADS  ELEMENT
 3 R -1.E3  STEP 2 TO  7 R -1.E3
*
FIXBOUNDARIES  3 INPUT=LINES  /  11 12  /  12 2  /  2 3
FIXBOUNDARIES  2 INPUT=NODES  /  11
*
MESH  VECTOR=LOAD CONTOUR=DISTORTION SUBFRAME=11
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A43  FRACTURE MECHANICS ANALYSIS OF A TENSILE SPECIMEN
*
DATABASE  CREATE
*
WRITE  FILENAME='a43.lis'
*
ENERGYRELEASERATE
*
MESH  VECTOR=LOAD ORIGINAL=DASHED
MESH  ZONENAME=TRIANG CONTOUR=MISES
END
```

EXAMPLE A44

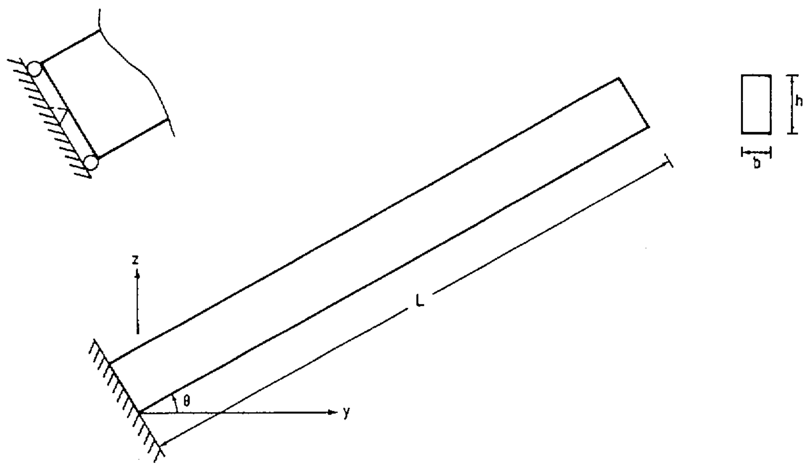
FUNDAMENTAL FREQUENCY OF CANTILEVER, PLANE STRESS

Objective

To verify the dynamic behaviour of the PLANE STRESS element in frequency analysis when employing SKEW degree-of-freedom Systems.

Physical Problem

A cantilever beam of rectangular cross-section as shown in the figure below is considered. The cantilever is the same as previously analyzed for a static load in Example A14.



$L = 1.0$ m

$b = 0.05$ m

$h = 0.10$ m

$\theta = 30°$

$E = 2.0 \cdot 10^{11}$ N/m^2

$\nu = 0.30$

$\rho = 7800$ kg/m^3

Finite Element Model

The cantilever beam is modeled using ten 8-node PLANE STRESS elements with a consistent mass matrix, see the top figure on page A44.3. Since the model is inclined in the Global System, a SKEW degree-of-freedom System is used. The subspace iteration method is used for the frequency calculations.

Solution Results

The lowest natural frequency of a cantilever beam is given in [1], p. 108 as:

$$f = \frac{\lambda^2}{2\pi L^2}\sqrt{\frac{EI}{m}}$$

where

$\lambda = 1.87510407$

m = mass/unit length

In this formula rotary inertia and shear deformations are not considered.

Insertion of the numerical values gives for the fundamental mode in the Y-Z plane:

$$f = 81.80 \text{ Hz}$$

The SOLVIA solution using the input data on pages A44.4 and A44.5 gives the following result:

$$f = 81.19 \text{ Hz}$$

The bottom figure on page A44.3 and the top figure on page A44.4 show the mode shape of the two lowest frequencies as displayed by SOLVIA-POST.

User Hints

- Note that the consistent element mass matrix for the PLANE element is always calculated using 3×3 Gauss integration and that the user has no control over it. This ensures that the entire mass of the element is accounted for in the mass matrix.

- For a pinned-pinned beam [1] p. 181 gives the following expression for the ratio between the frequency including rotary inertia and shear deformations and the frequency including flexural effects only:

$$\frac{(f)_{\text{rot.+shear}}}{(f)_{\text{flex.}}} = 1 - \frac{\pi^2}{2L^2}\left(\frac{I}{A}\right)\cdot\left(1+\frac{E}{KG}\right)$$

  where

$$K = \frac{10(1+\nu)}{12+11\nu} \qquad \text{for a rectangular section}$$

  Assuming the effects of rotary inertia and shear deformation to be of the same order for a cantilever, we obtain the following estimate

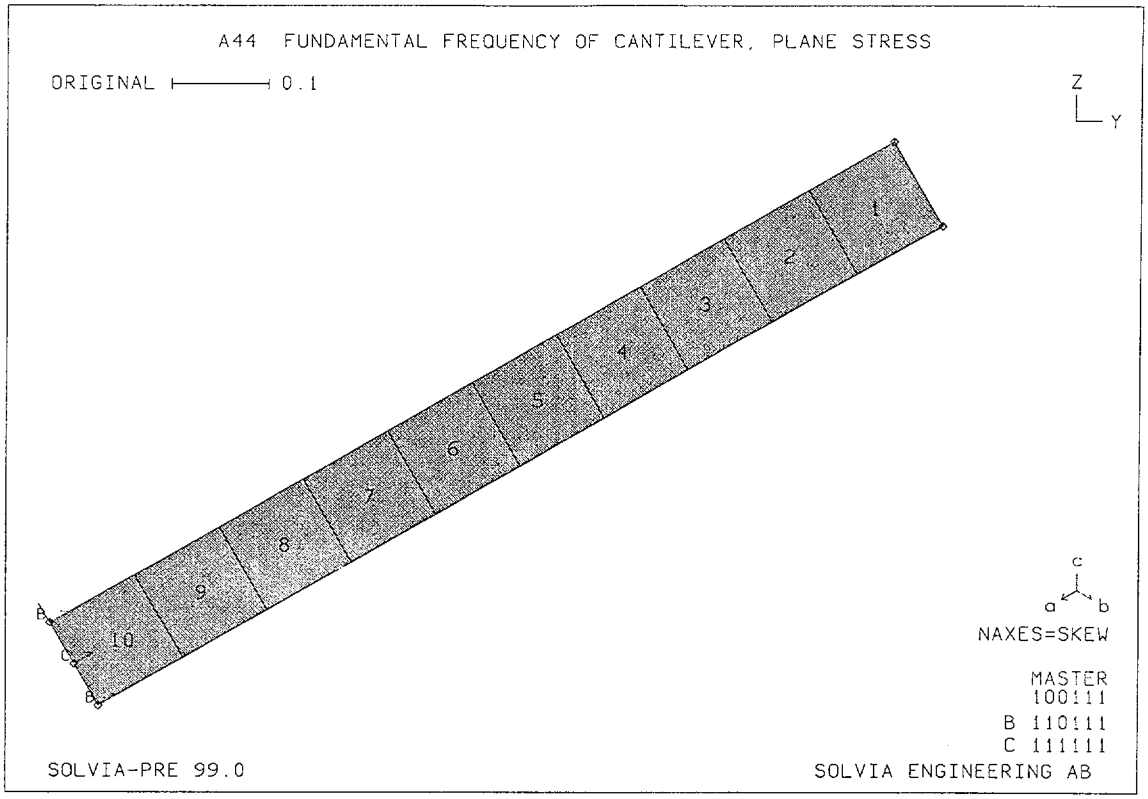
$$(f)_{\text{rot.+shear}} = 80.43 \text{ Hz}$$

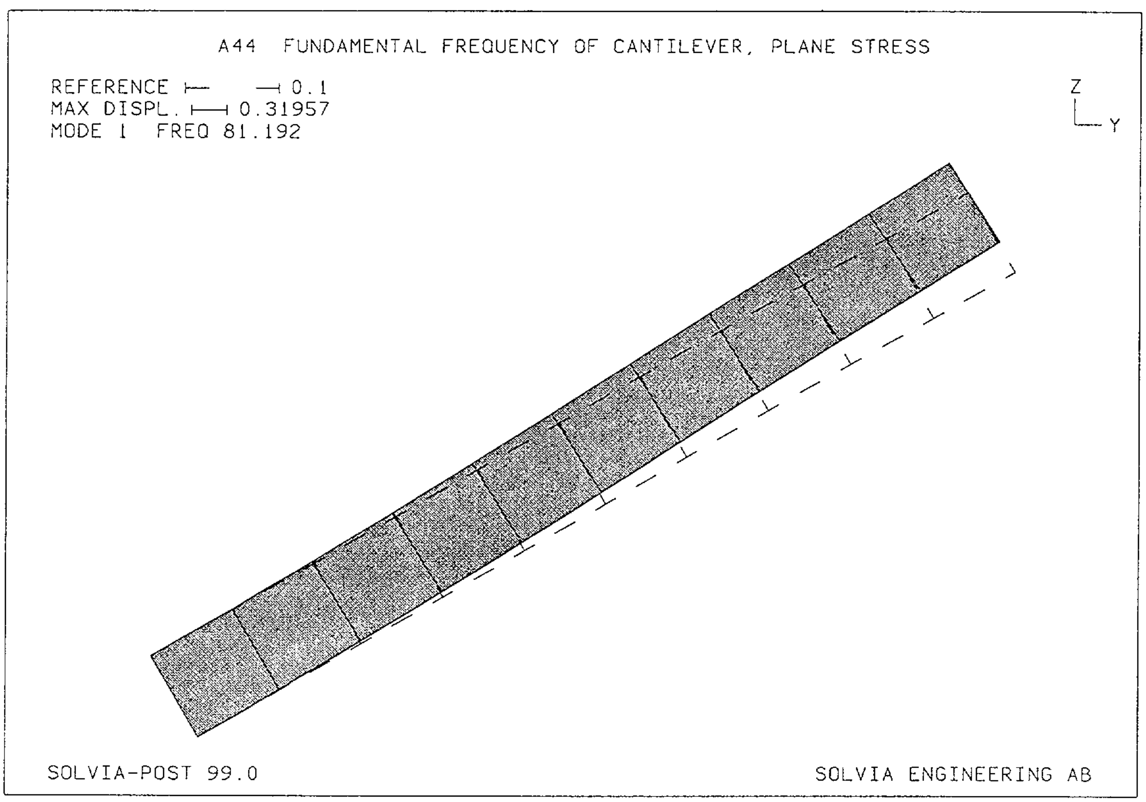
  The SOLVIA solution for a consistent mass matrix gives a higher value for the lowest natural frequency than this analytical estimate.

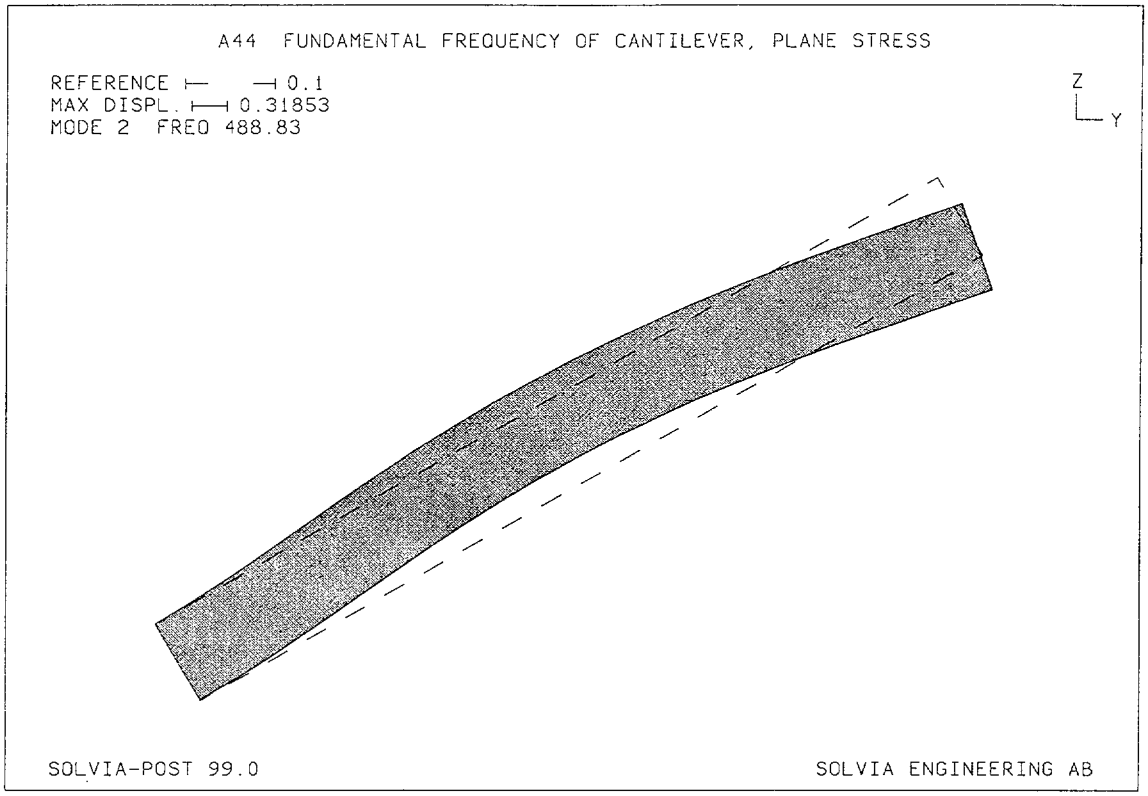
- Of course, the determinant search method can also be used and it gives the same frequency results.

Reference

[1] Blevins, R.D., Formulas for Natural Frequency and Mode Shape, Van Nostrand Reinhold Company, 1979.


A44 FUNDAMENTAL FREQUENCY OF CANTILEVER, PLANE STRESS
ORIGINAL 0.1
Z
Y
1
2
3
4
5
6
7
8
9
10
c
a
b
NAXES=SKEW
MASTER
100111
B 110111
C 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A44 FUNDAMENTAL FREQUENCY OF CANTILEVER, PLANE STRESS
REFERENCE 0.1
MAX DISPL. 0.31957
MODE 1 FREQ 81.192
Z
Y
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB



SOLVIA-PRE input

```
HEADING  'A44  FUNDAMENTAL FREQUENCY OF CANTILEVER, PLANE STRESS'
*
DATABASE  CREATE
MASTER  IDOF=100111
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=CONSISTENT
FREQUENCIES  SUBSPACE-ITERATION NEIG=2 SSTOL=1.E-8
*
SKEWSYSTEMS  EULERANGLES
 1  30.
SYSTEM  1 CARTESIAN  PHI=30
COORDINATES
 ENTRIES NODE   YL    ZL
           1     0     0
           2     0     0.05
           3     0     0.1
           4     1     0
           5     1     0.1
*
MATERIAL  1  ELASTIC E=2.E11 NU=0.3 DENSITY=7800
*
EGROUP  1  PLANE STRESS2
GSURFACE  5 3 1 4  EL1=10 EL2=1 NODES=8
EDATA  /  ENTRIES EL THICK
  1  0.05
*
```

SOLVIA-PRE input (cont.)

```
NSKEWS  INPUT=LINE
 1 3  1
FIXBOUNDARIES  2  /  1 3
FIXBOUNDARIES     /  2
*
SET  NSYMBOLS=MYNODES
MESH  BCODE=ALL NAXES=SKEW ENUMBERS=YES
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A44  FUNDAMENTAL FREQUENCY OF CANTILEVER, PLANE STRESS
*
DATABASE  CREATE
*
WRITE  FILENAME='a44.lis'
*
SET  RESPONSETYPE=VIBRATIONMODE
*
MESH  TIME=1 ORIGINAL=DASHED
MESH  TIME=2 ORIGINAL=DASHED OUTLINE=YES
*
FREQUENCIES
MASS-PROPERTIES
END
```

EXAMPLE A45

FUNDAMENTAL FREQUENCY OF CANTILEVER, PLANE STRAIN

Objective

To verify the dynamic behaviour of the PLANE STRAIN element in frequency analysis and when employing SKEW degree-of-freedom Systems.

Physical Problem

Same as in figure on page A44.1.

Finite Element Model

Same as in figure on page A44.3 except that PLANE STRAIN elements and a lumped mass distribution are used. The determinant search method is used for the frequency solution.

Solution Results

To obtain the same theoretical solution as for the plane stress case of Example A44 the following material data is used:

$$E^* = \frac{1+2\nu}{(1+\nu)^2} \cdot E = 1.89349 \cdot 10^{11}\, N/m^2$$

$$\nu^* = \frac{\nu}{(1+\nu)} = 0.230769$$

where E and ν are the Young's modulus and Poisson's ratio, respectively, as used in Example A44.

The theoretical solution given in Example A44 is

$$f = 81.80 \text{ Hz}$$

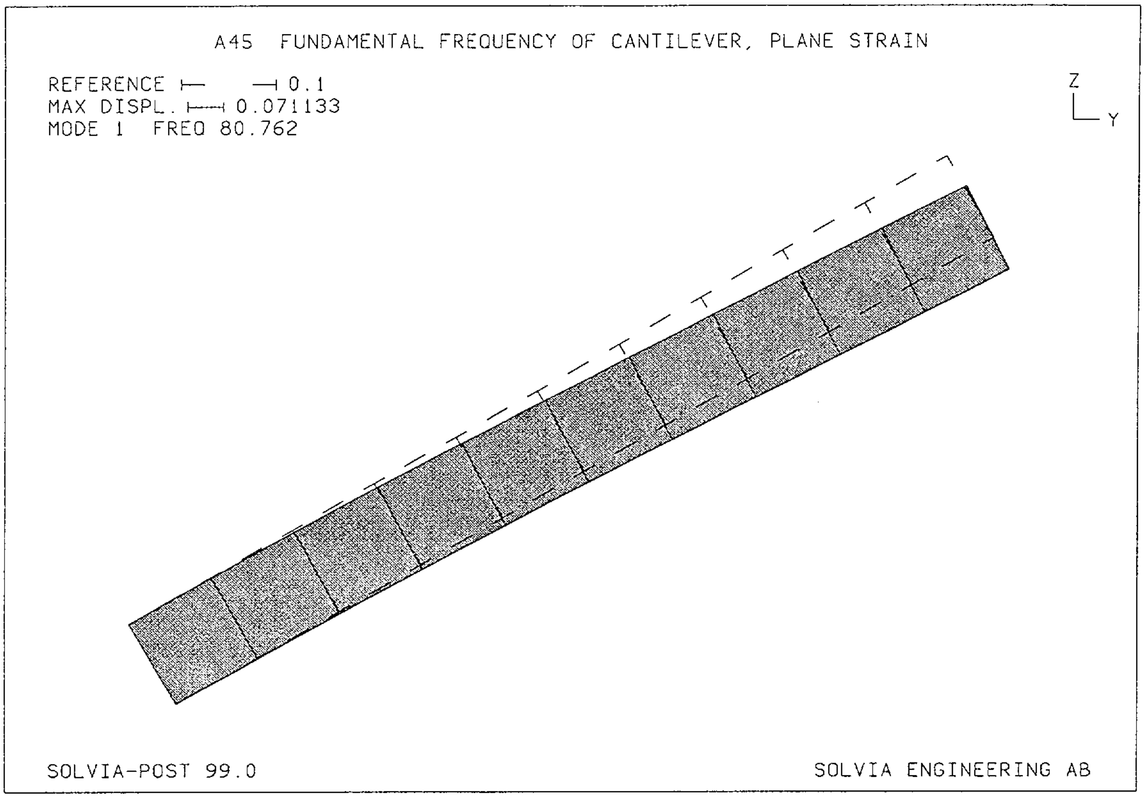
The input data on page A45.3 gives the following result:

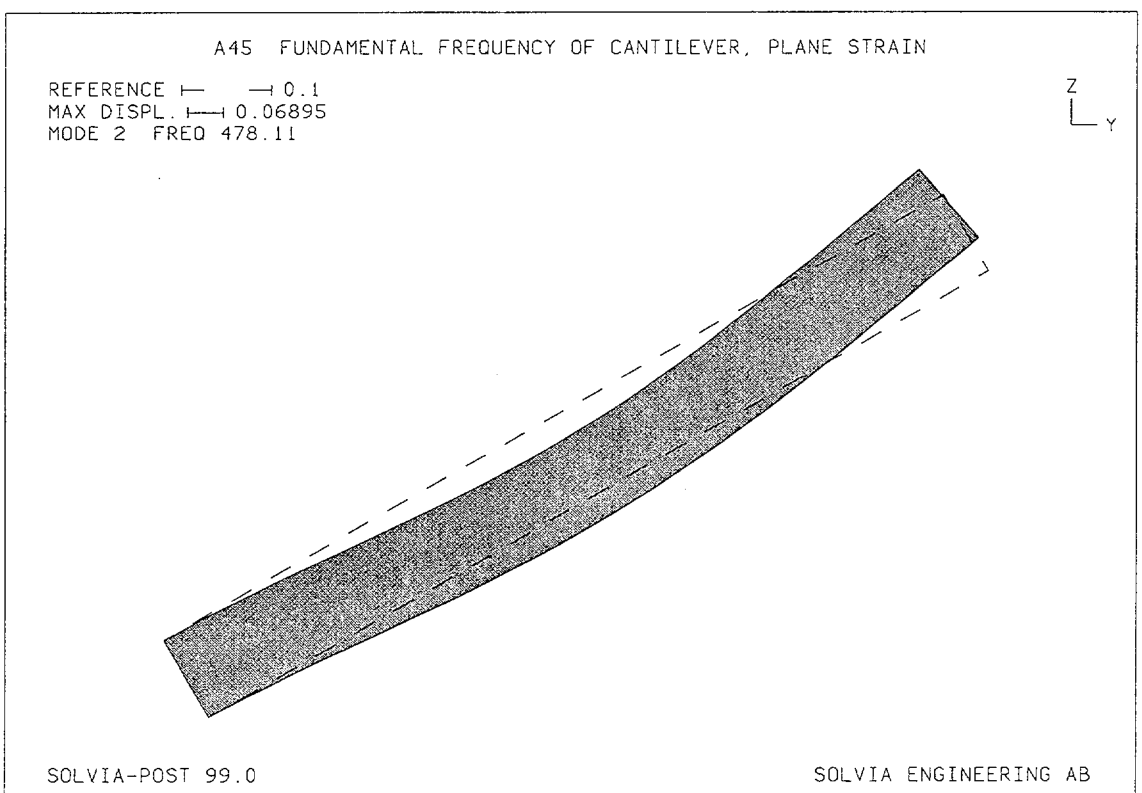
$$f = 80.76 \text{ Hz}$$

The figures on page A45.2 show the mode shape of the two lowest frequencies as displayed by SOLVIA-POST.

User Hints

- An analysis using the consistent mass discretization yields the result f = 81.19 Hz which is exactly equal to the result obtained in Example A44 (which should be the case).


A45 FUNDAMENTAL FREQUENCY OF CANTILEVER, PLANE STRAIN
REFERENCE 0.1
MAX DISPL. 0.071133
MODE 1 FREQ 80.762
Z
Y
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A45 FUNDAMENTAL FREQUENCY OF CANTILEVER, PLANE STRAIN
REFERENCE 0.1
MAX DISPL. 0.06895
MODE 2 FREQ 478.11
Z
Y
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A45  FUNDAMENTAL FREQUENCY OF CANTILEVER, PLANE STRAIN'
*
DATABASE  CREATE
MASTER  IDOF=100111
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=LUMPED
FREQUENCIES  DETERMINANT-SEARCH NEIG=2
*
SKEWSYSTEMS  EULERANGLES
 1  30.
SYSTEM  1 CARTESIAN  PHI=30
COORDINATES
 ENTRIES NODE   YL    ZL
           1     0     0
           2     0     0.05
           3     0     0.1
           4     1     0
           5     1     0.1
*
MATERIAL  1  ELASTIC E=1.89349E11 NU=0.230769 DENSITY=7800
*
EGROUP  1  PLANE STRAIN
GSURFACE  5 3 1 4  EL1=10 EL2=1 NODES=8
*
NSKEWS  INPUT=LINE
 1 3  1
FIXBOUNDARIES  2  /  1 3
FIXBOUNDARIES     /  2
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A45  FUNDAMENTAL FREQUENCY OF CANTILEVER, PLANE STRAIN
*
DATABASE  CREATE
*
WRITE  FILENAME='a45.lis'
*
SET  RESPONSETYPE=VIBRATIONMODE
*
MESH  TIME=1 ORIGINAL=DASHED
MESH  TIME=2 ORIGINAL=DASHED OUTLINE=YES
*
FREQUENCIES
MASS-PROPERTIES
END
```

EXAMPLE A46

FUNDAMENTAL FREQUENCY OF CANTILEVER, SOLID

Objective

To verify the dynamical behaviour of the SOLID element in frequency analysis when employing SKEW degree-of-freedom Systems.

Physical Problem

Same as in the figure on page A44.1.

Finite Element Model

The finite element model considered is shown on page A46.2. Ten 20-node SOLID elements are used in the model. A consistent mass discretization and the determinant search method of frequency analysis are used.

Solution Results

The theoretical solution for this problem is given in Example A44.

The input data on pages A46.4 and A46.5 is used in the finite element analysis and gives the following results:

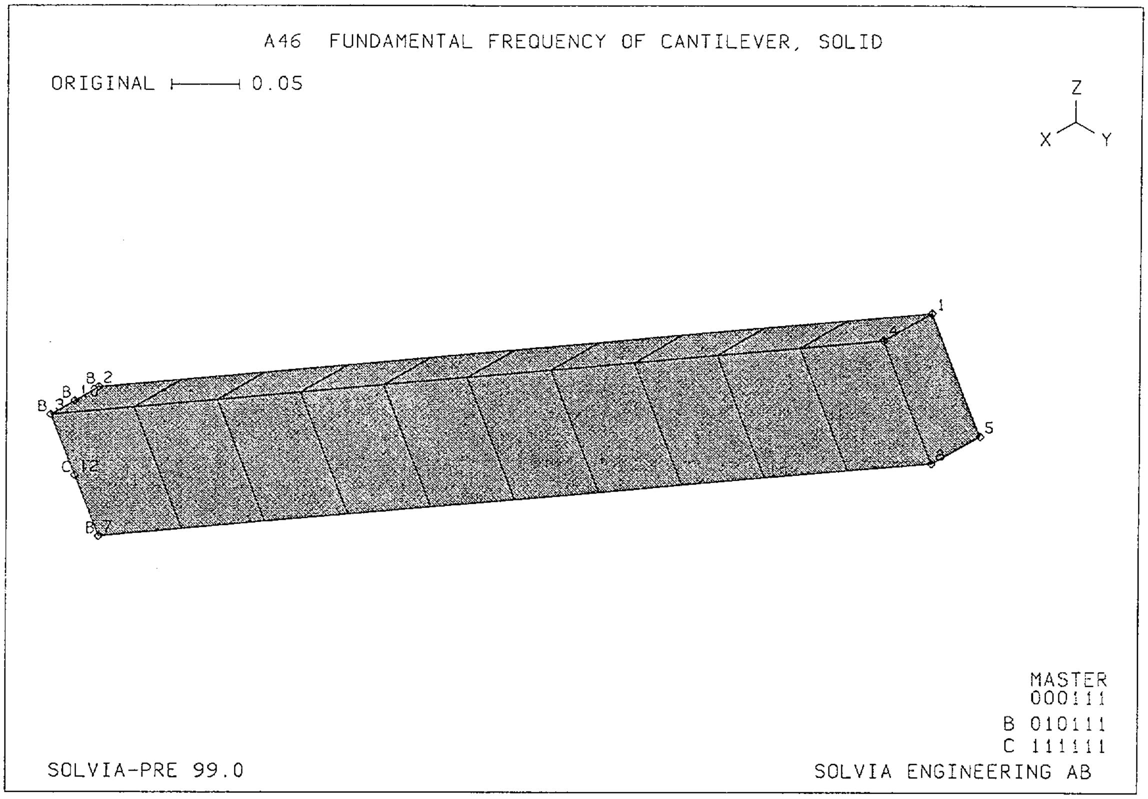
Fundamental frequency for motion in the Y-Z plane (mode 2):

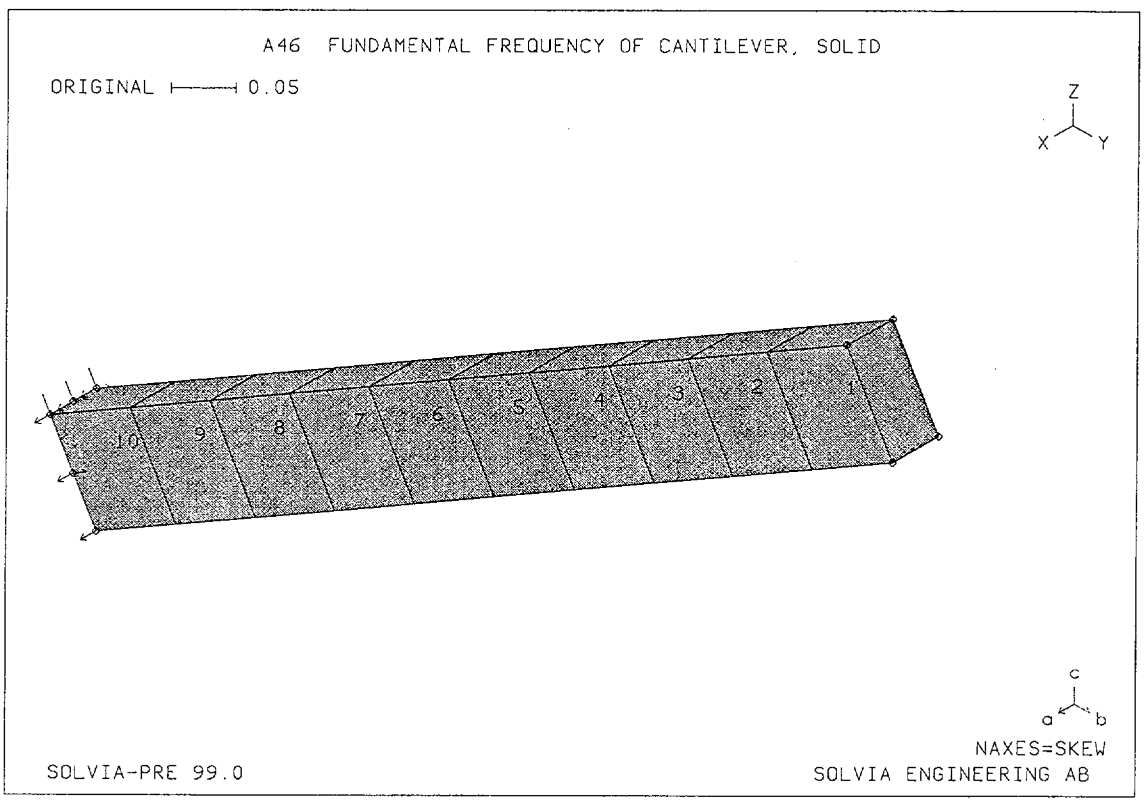
Beam theory	SOLVIA
81.80	81.16

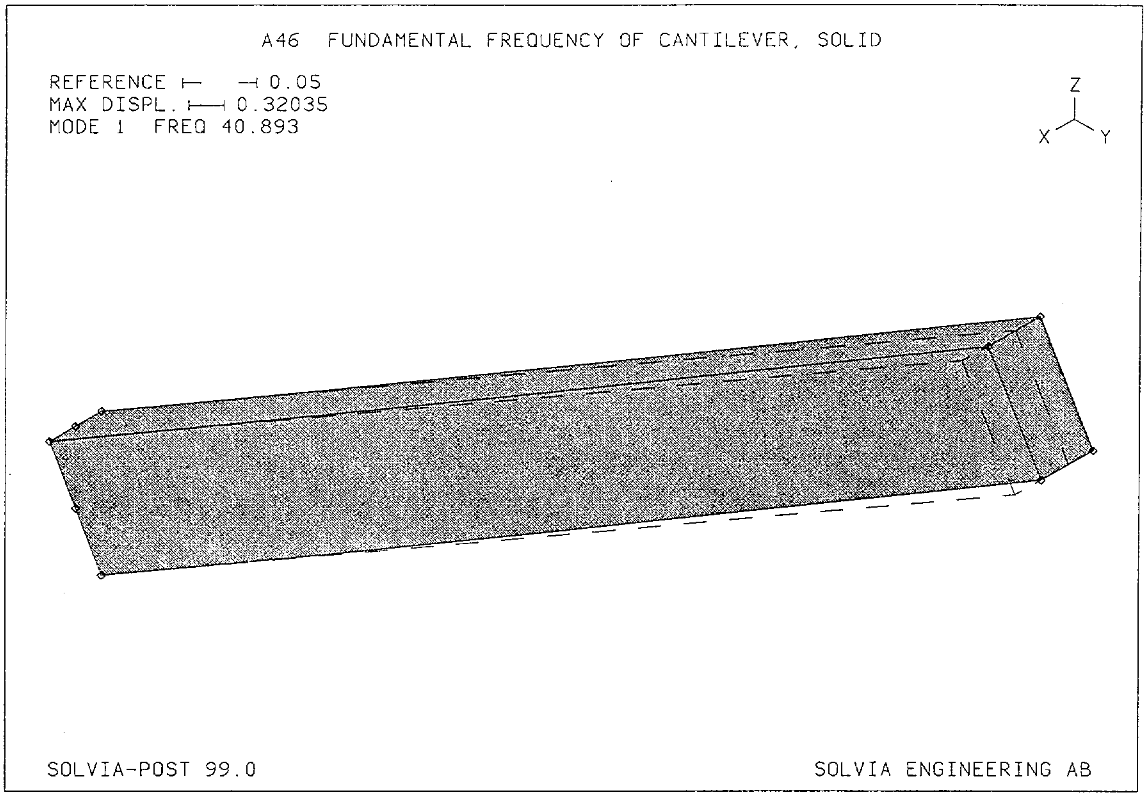
The mode shapes of the two lowest frequencies as calculated in the finite element analysis are shown on page A46.3.

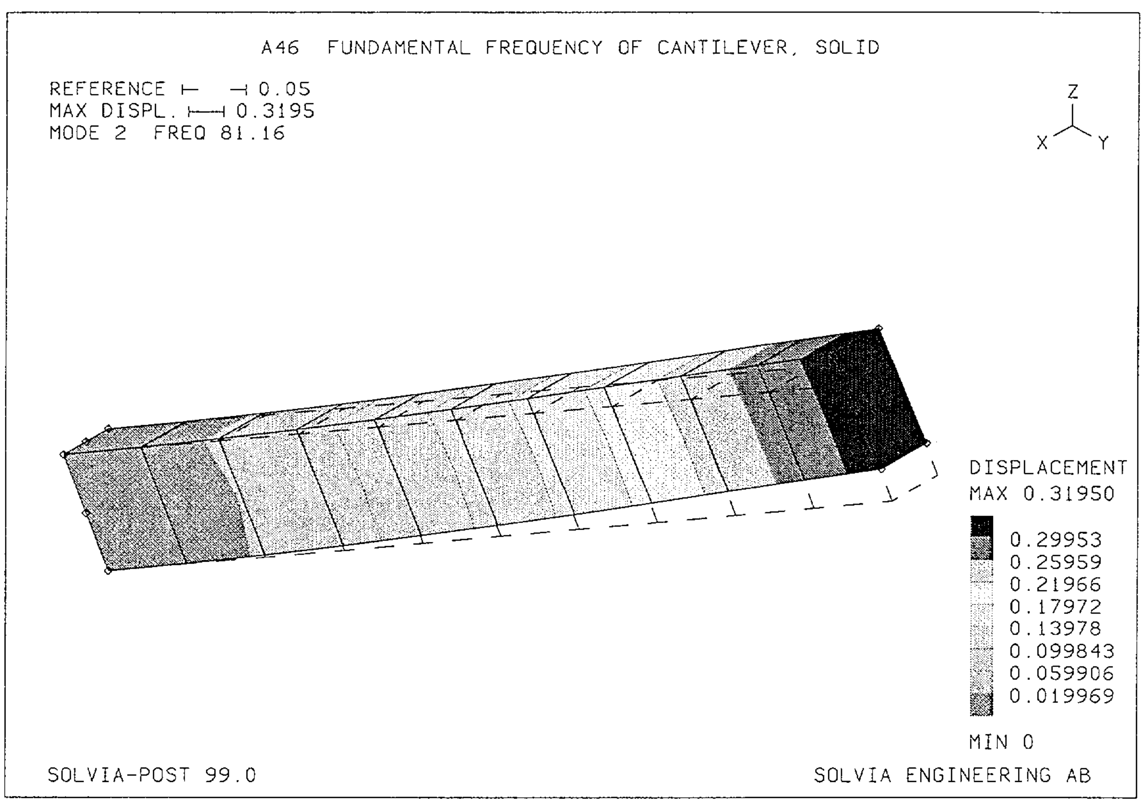
User Hints

- Note that the consistent element mass matrix for the SOLID element is always calculated using 3×3×3 Gauss points and that the user has no control over it. This ensures that the entire mass of the element is accounted for in the mass matrix.


A46 FUNDAMENTAL FREQUENCY OF CANTILEVER, SOLID
ORIGINAL 0.05
Z
X
Y
MASTER
000111
B 010111
C 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A46 FUNDAMENTAL FREQUENCY OF CANTILEVER, SOLID
ORIGINAL 0.05
Z
X
Y
10
9
8
7
6
5
4
3
2
1
c
a
b
NAXES=SKEW
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A46 FUNDAMENTAL FREQUENCY OF CANTILEVER, SOLID
REFERENCE 0.05
MAX DISPL. 0.32035
MODE 1 FREQ 40.893
Z
X
Y
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A46 FUNDAMENTAL FREQUENCY OF CANTILEVER, SOLID
REFERENCE 0.05
MAX DISPL. 0.3195
MODE 2 FREQ 81.16
Z
X
Y
DISPLACEMENT
MAX 0.31950
0.29953
0.25959
0.21966
0.17972
0.13978
0.099843
0.059906
0.019969
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A46  FUNDAMENTAL FREQUENCY OF CANTILEVER, SOLID'
*
DATABASE  CREATE
*
MASTER  IDOF=000111
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=CONSISTENT
FREQUENCIES  DETERMINANT-SEARCH  NEIG=2
*
SKEWSYSTEMS  EULERANGLES  /  1 30.
SYSTEM  1  CARTESIAN  PHI=30
COORDINATES
 ENTRIES NODE    XL       YL      ZL
           1   -0.025   1.0     0.1
           2   -0.025   0.0     0.1
           3    0.025   0.0     0.1
           4    0.025   1.0     0.1
           5   -0.025   1.0     0.0
           6   -0.025   0.0     0.0
           7    0.025   0.0     0.0
           8    0.025   1.0     0.0
           9    0.0     0.0     0.0
          10    0.0     0.0     0.1
          11   -0.025   0.0     0.05
          12    0.025   0.0     0.05
*
MATERIAL  1  ELASTIC  E=2.E11 NU=0.3 DENSITY=7800
*
EGROUP  1  SOLID
GVOLUME  1 2 3 4 5 6 7 8  EL1=10 EL2=1 EL3=1 NODES=20
*
NSKEWS  INPUT=SURFACE
 2 3 6 7  1
*
FIXBOUNDARIES  123  /  11 12
FIXBOUNDARIES   2   /  2 3 6 7 9 10
*
SET  NSYMBOLS=MYNODES
MESH  NNUMBERS=MYNODES BCODE=ALL
MESH  NAXES=SKEW ENUMBERS=YES
*
SOLVIA
END
```

SOLVIA-POST input

```
*   A46   FUNDAMENTAL FREQUENCY OF CANTILEVER, SOLID
*
DATABASE  CREATE
*
WRITE  FILENAME='a46.lis'
*
FREQUENCIES
MASS-PROPERTIES
*
SET  RESPONSETYPE=VIBRATIONMODE ORIGINAL=DASHED NSYMBOLS=MYNODES
*
MESH  TIME=1 OUTLINE=YES
MESH  TIME=2 CONTOUR=DISPLACEMENT
END
```

EXAMPLE A47

FUNDAMENTAL FREQUENCY OF CANTILEVER, BEAM

Objective

To verify the dynamical behaviour of the BEAM element in frequency analysis when employing SKEW degree-of-freedom Systems.

Physical Problem

Same as in figure on page A44.1.

Finite Element Model

The inclined cantilever is modeled using ten BEAM elements as shown on page A47.2. The element stiffness matrices are calculated in closed form and the mass is represented using a lumped mass matrix. The subspace iteration method of analysis is used.

Solution Results

The theoretical solution for this problem is presented in Example A44.

The input data on page A47.4 gives the following result:

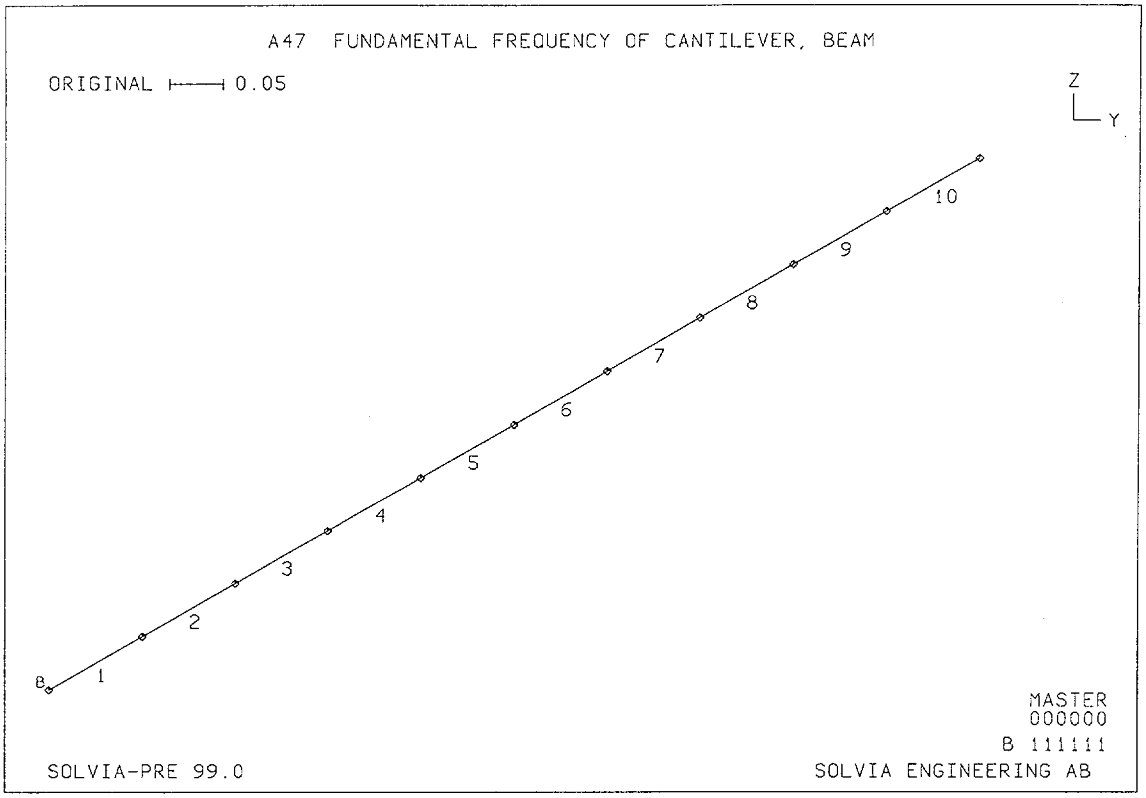
Fundamental frequency (Hz) for motion in the Y-Z plane (mode 2):

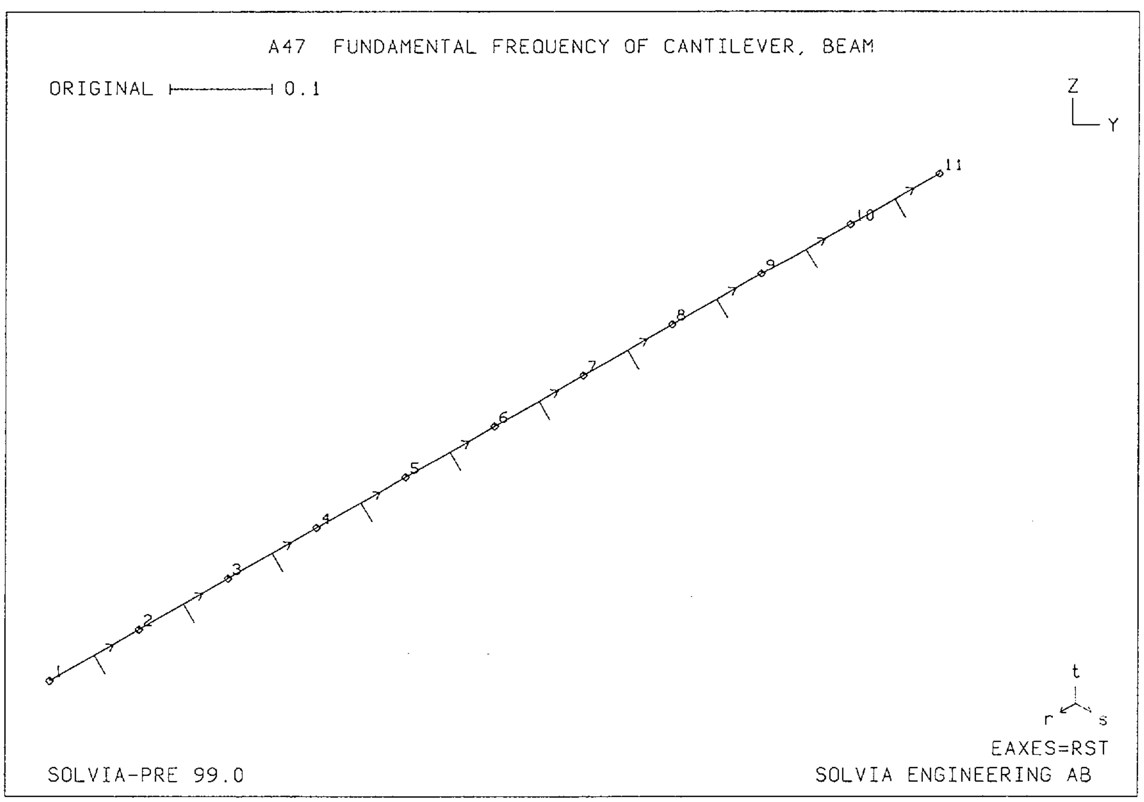
Beam theory	SOLVIA
81.80	80.94

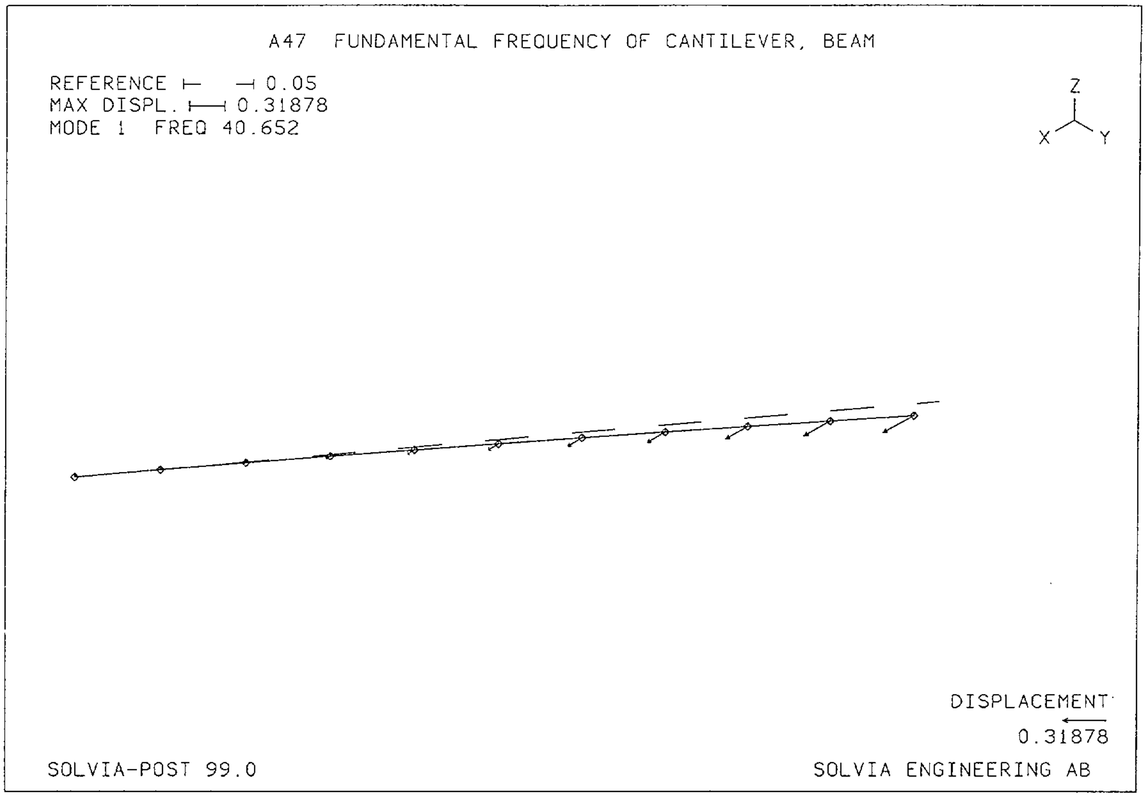
The mode shapes of the fundamental frequencies calculated in the finite element analysis for the X-Y and Y-Z planes are shown on page A47.3 as displayed by SOLVIA-POST.

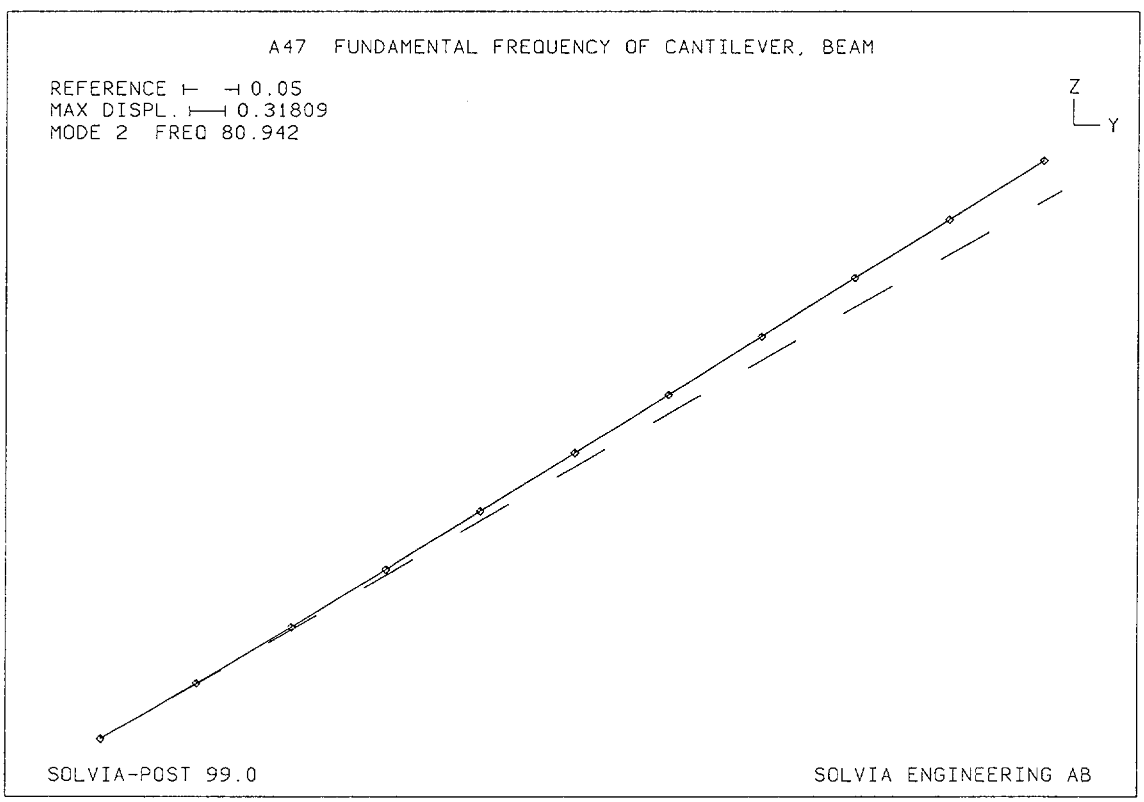
User Hints

- Employing consistent mass matrix in this finite element analysis gives a fundamental frequency, f = 81.15 Hz. Note that the consistent mass matrix for the BEAM element does not include contributions due to shear deformation.


A47 FUNDAMENTAL FREQUENCY OF CANTILEVER, BEAM
ORIGINAL 0.05
Z
Y
1
2
3
4
5
6
7
8
9
10
B
MASTER
000000
B 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A47 FUNDAMENTAL FREQUENCY OF CANTILEVER, BEAM
ORIGINAL 0.1
Z
Y
11
t
r
s
EAXES=RST
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A47 FUNDAMENTAL FREQUENCY OF CANTILEVER, BEAM
REFERENCE ⊢ ⊣ 0.05
MAX DISPL. ⊢⊣ 0.31878
MODE 1 FREQ 40.652
Z
X
Y
DISPLACEMENT
0.31878
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A47 FUNDAMENTAL FREQUENCY OF CANTILEVER, BEAM
REFERENCE ⊢ ⊣ 0.05
MAX DISPL. ⊢⊣ 0.31809
MODE 2 FREQ 80.942
Z
Y
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A47  FUNDAMENTAL FREQUENCY OF CANTILEVER, BEAM'
*
DATABASE  CREATE
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=LUMPED
FREQUENCIES  SUBSPACE-ITERATION NEIG=2
*
SKEWSYSTEMS EULERANGLES
 1 30
SYSTEM 1 CARTESIAN  PHI=30
COORDINATES  /  ENTRIES  NODE YL ZL
  1  0 0  TO  11  1 0
NSKEWS
  1 1  TO  11 1
*
MATERIAL  1  ELASTIC  E=2.E11 NU=0.3 DENSITY=7800
*
EGROUP  1  BEAM
SECTION  1  RECTANGULAR  WTOP=0.05 D=0.1
BEAMVECTOR  /  1  1.
ENODES
 1  -1 1 2  TO  10  -1 10 11
*
FIXBOUNDARIES  /  1
*
SET  VIEW=X NSYMBOLS=YES
MESH  ENUMBER=YES BCODE=ALL
MESH  NNUMBER=YES EAXES=RST
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A47  FUNDAMENTAL FREQUENCY OF CANTILEVER, BEAM
*
DATABASE  CREATE
*
WRITE  FILENAME='a47.lis'
*
FREQUENCIES
MASS-PROPERTIES
*
SET  RESPONSETYPE=VIBRATIONMODE ORIGINAL=DASHED NSYMBOLS=YES
*
MESH  VIEW=I TIME=1 VECTOR=DISPLACEMENT
MESH  VIEW=X TIME=2
*
NMAX  DIRECTION=1346 NUMBER=5
END
```

EXAMPLE A48

FUNDAMENTAL FREQUENCY OF CANTILEVER, ISOBEAM

Objective

To verify the dynamical behaviour of the ISOBEAM element in frequency analysis when employing SKEW degree-of-freedom Systems.

Physical Problem

Same as in figure on page A44.1.

Finite Element Model

The inclined cantilever beam is modeled using ten parabolic ISOBEAM elements as shown on page A48.2. The element stiffness matrices are evaluated using 2×4×4 Gauss points and the mass is represented in a diagonal mass matrix. The determinant search method of frequency analysis is used.

Solution Results

The theoretical solution is the same as in Example A44.

The input data on page A48.4 gives the following result:

Fundamental frequency (Hz):

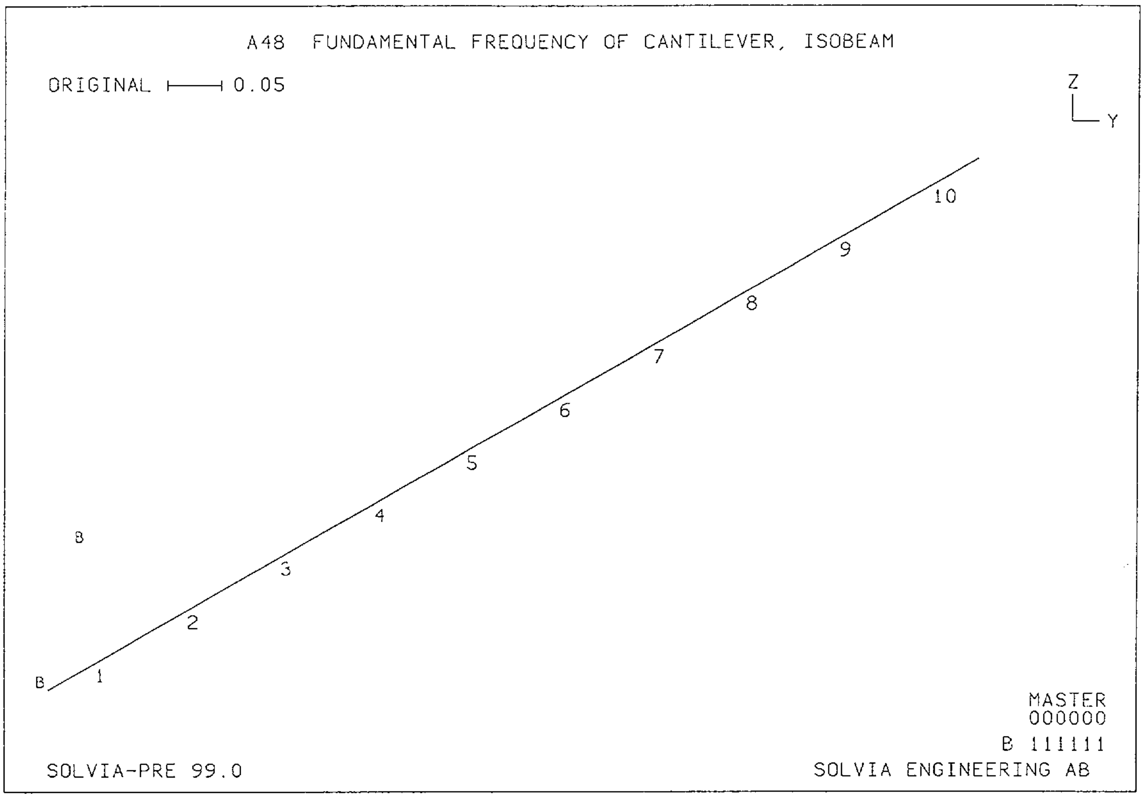
Beam theory	SOLVIA
81.80	81.30

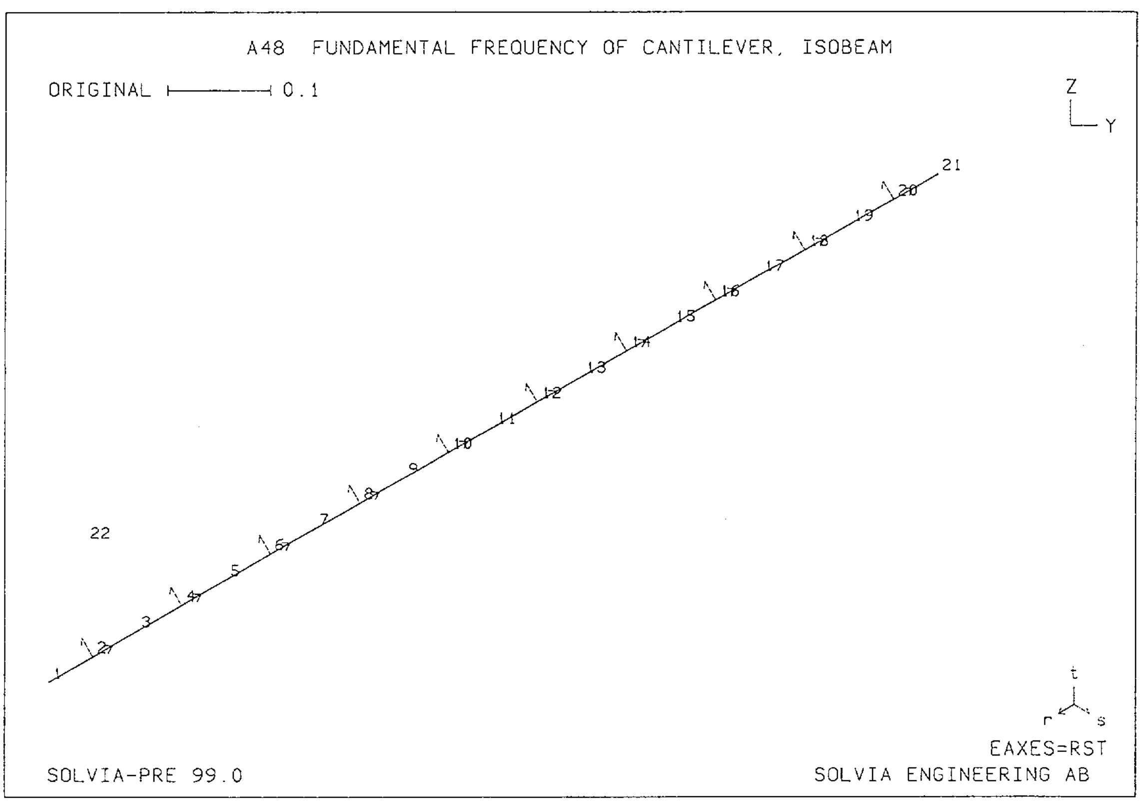
The mode shapes of the fundamental frequencies in the finite element analysis for the X-Y and Y-Z planes are shown on page A48.3.

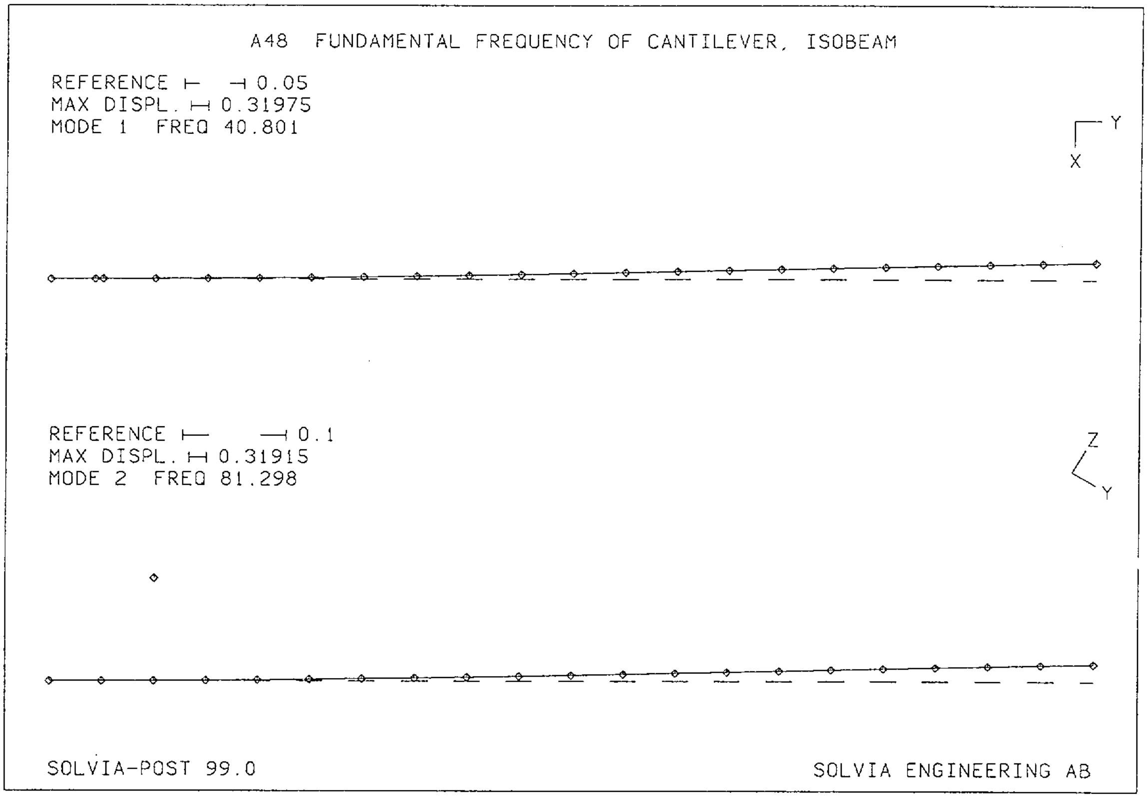
User Hints

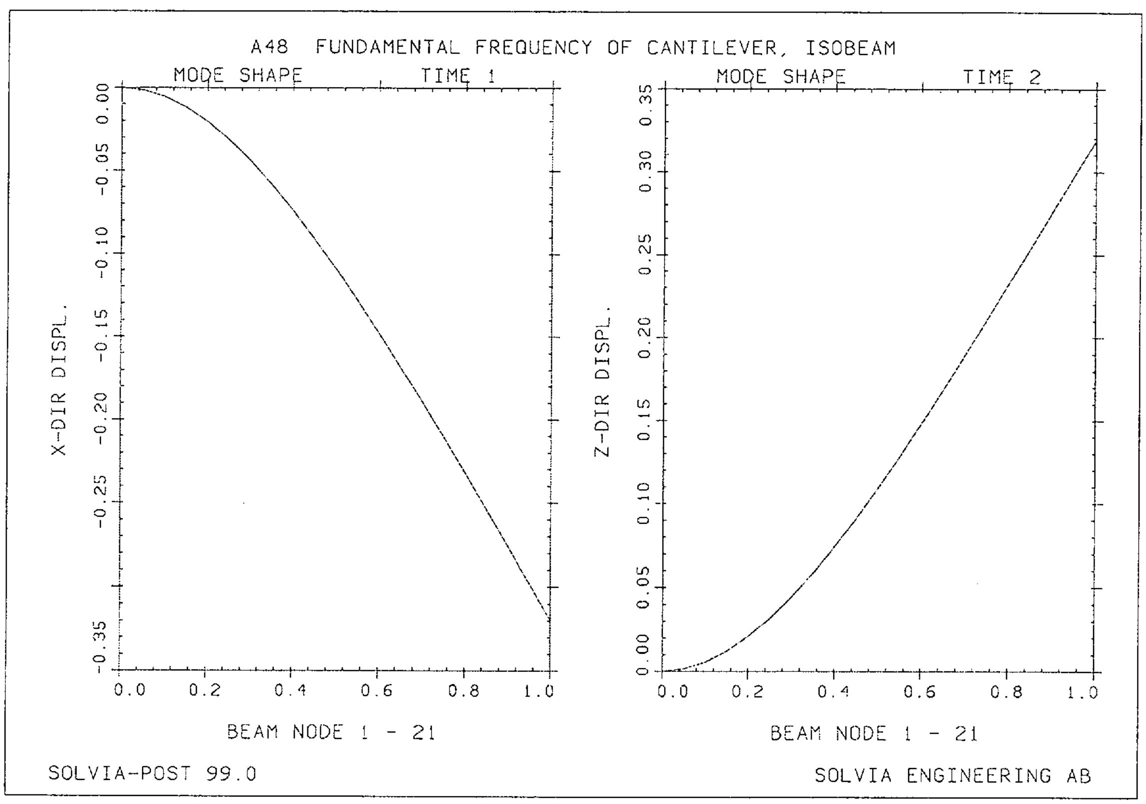
- Employing a consistent mass matrix in the finite element analysis gives a fundamental frequency f = 81.24 Hz

- Note that the shear factor for the ISOBEAM element is always equal to 1.0 and the user has not control over it.


A48 FUNDAMENTAL FREQUENCY OF CANTILEVER, ISOBEAM
ORIGINAL 0.05
Z
Y
10
9
8
7
6
5
4
3
2
1
B
B
MASTER
000000
B 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A48 FUNDAMENTAL FREQUENCY OF CANTILEVER, ISOBEAM
ORIGINAL 0.1
Z
Y
21
22
t
r
s
EAXES=RST
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A48 FUNDAMENTAL FREQUENCY OF CANTILEVER, ISOBEAM
REFERENCE 0.05
MAX DISPL. 0.31975
MODE 1 FREQ 40.801
Y
X
REFERENCE 0.1
MAX DISPL. 0.31915
MODE 2 FREQ 81.298
Z
Y
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A48 FUNDAMENTAL FREQUENCY OF CANTILEVER, ISOBEAM
MODE SHAPE TIME 1
X-DIR DISPL.
BEAM NODE 1 - 21
MODE SHAPE TIME 2
Z-DIR DISPL.
BEAM NODE 1 - 21
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A48  FUNDAMENTAL FREQUENCY OF CANTILEVER, ISOBEAM'
*
DATABASE  CREATE
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=LUMPED
FREQUENCIES  DETERMINANT-SEARCH NEIG=2
*
SKEWSYSTEMS  EULERANGLES
 1 30
SYSTEM  1  CARTESIAN  PHI=30
COORDINATES
 1  TO  21  0. 1.  /  22  0. 0.1 0.1
NSKEWS
 1 1  TO  22 1
*
MATERIAL  1  ELASTIC  E=2.E11 NU=0.3 DENSITY=7800
*
EGROUP  1  ISOBEAM
SECTION  1  SDIM=0.100 TDIM=0.050
ENODES
 ENTRIES  EL AUX N1 N2 N3
           1  22  1  3  2  TO  10  22 19 21 20
*
FIXBOUNDARIES  /  1 22
*
MESH  VIEW=X ENUMBERS=YES BCODE=ALL
MESH  VIEW=X NNUMBERS=YES EAXES=RST
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A48  FUNDAMENTAL FREQUENCY OF CANTILEVER, ISOBEAM
*
DATABASE CREATE
*
WRITE  FILENAME='a48.lis'
*
FREQUENCIES
MASS-PROPERTIES
*
VIEW  ID=1 ZVIEW=1. ROTATION=-90.
VIEW  ID=2 XVIEW=1. ROTATION=-30.
SET  RESPONSETYPE=VIBRATIONMODE ORIGINAL=DASHED NSYMBOLS=YES
*
MESH  VIEW=1 TIME=1 SUBFRAME=12
MESH  VIEW=2 TIME=2
*
NPLINE  NAME=BEAM  /  1  TO  21
NLINE  LINENAME=BEAM DIRECTION=1 TIME=1 SUBFRAME=21
NLINE  LINENAME=BEAM DIRECTION=3 TIME=2 OUTPUT=ALL
END
```

EXAMPLE A49

FUNDAMENTAL FREQUENCY OF CANTILEVER, PLATE

Objective

To verify the dynamical behaviour of the PLATE element in frequency analysis when employing SKEW degree-of-freedom Systems.

Physical Problem

Same as in figure on page A44.1.

Finite Element Model

The finite element model consists of twenty PLATE elements as shown in the top figure on page A49.2. A lumped mass matrix is employed in the analysis, i.e. 1/3rd of the element mass is attributed to each element translational degree of freedom. The displacements are constrained to be zero in the X-direction. The subspace iteration method of frequency analysis is used.

Solution Results

The theoretical solution for this problem is presented in Example A44.

Using the input data on page A49.3 the following result is obtained in the finite element analysis:

Fundamental frequency (Hz):

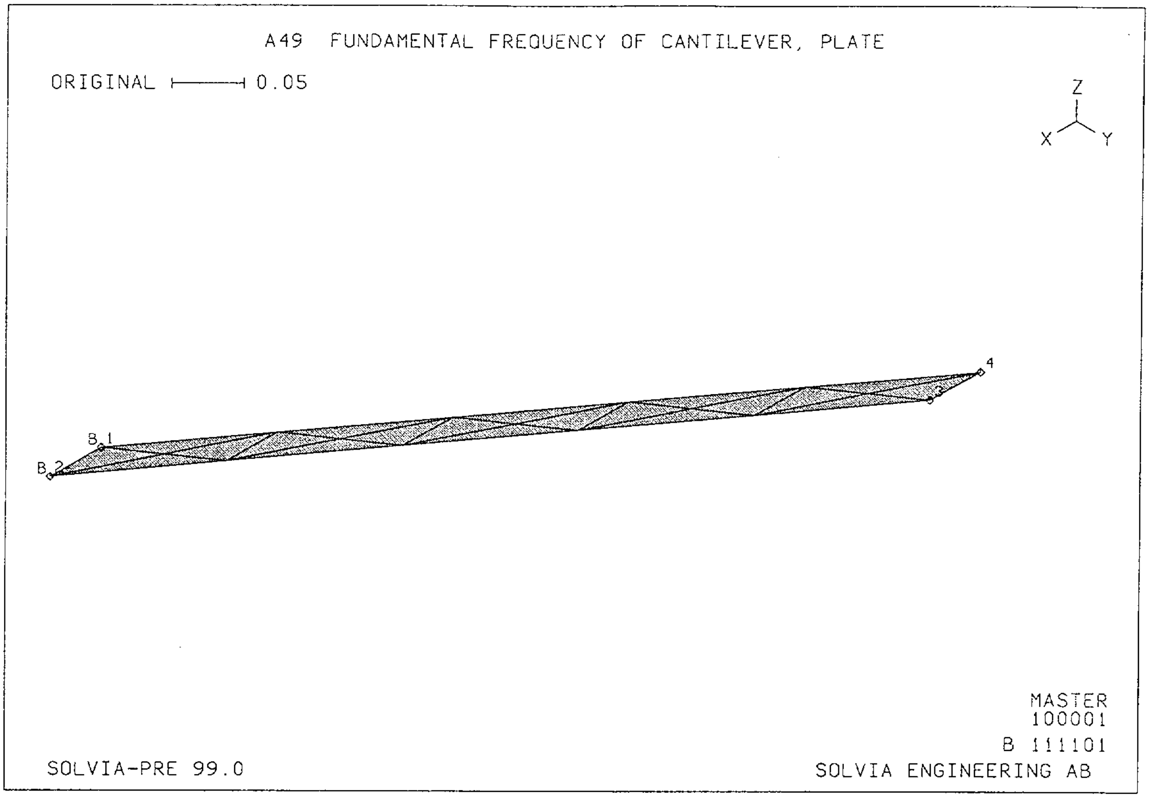
Beam theory	SOLVIA
81.80	81.16

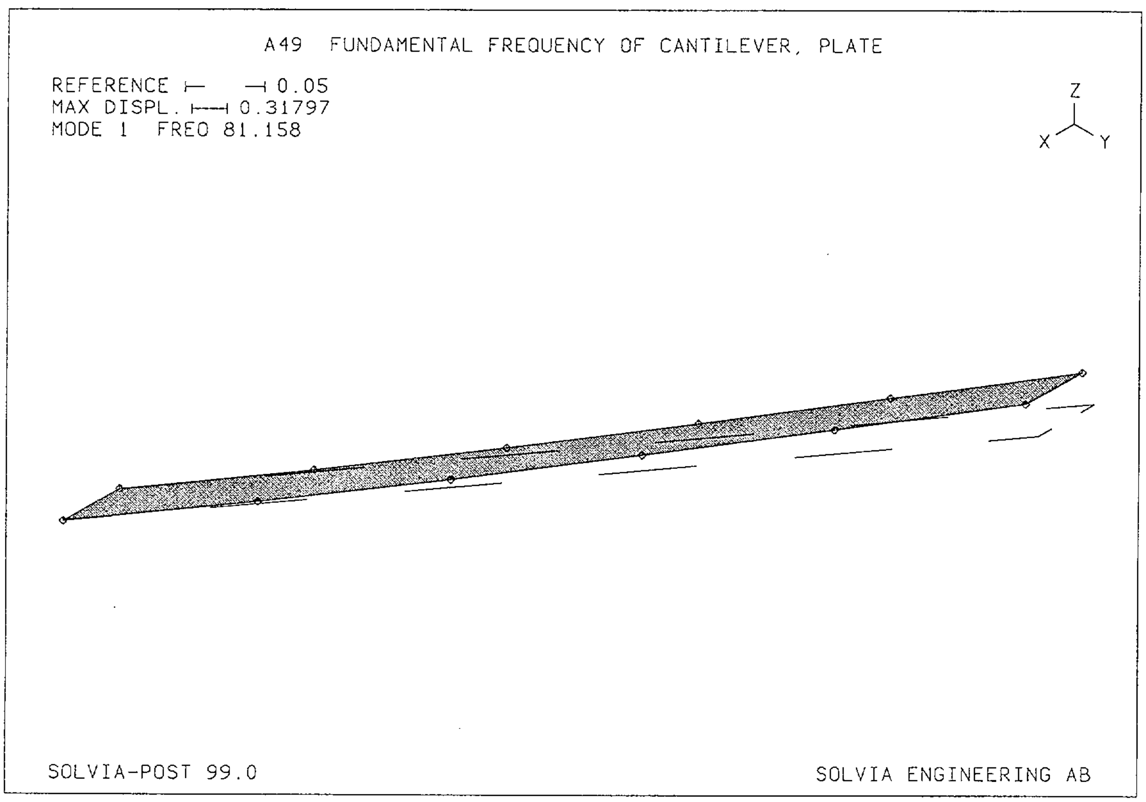
The bottom figure on page A49.2 shows the mode shape corresponding to the fundamental frequency calculated in the finite element analysis.

User Hints

- A more refined finite element model (40 PLATE elements) and a lumped mass matrix yields the result f = 81.61 Hz while using a consistent mass matrix for this refined model gives f = 81.76 Hz.

- Note that the consistent element mass matrix is not a consistent matrix in the usual sense, because linear variations in the displacements over the element are assumed and no mass is attributed to the rotational degrees of freedom.


A49 FUNDAMENTAL FREQUENCY OF CANTILEVER, PLATE
ORIGINAL 0.05
Z
X Y
B 1
B 2
4
MASTER
100001
B 111101
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A49 FUNDAMENTAL FREQUENCY OF CANTILEVER, PLATE
REFERENCE 0.05
MAX DISPL. 0.31797
MODE 1 FREQ 81.158
Z
X Y
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A49  FUNDAMENTAL FREQUENCY OF CANTILEVER, PLATE'
*
DATABASE  CREATE
MASTER  IDOF=100001
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=LUMPED
FREQUENCIES  SUBSPACE-ITERATION NEIG=1
*
SKEWSYSTEM  EULERANGLES
 1 30
SYSTEM  1  CARTESIAN PHI=30
COORDINATES
 1  /  2  0.05  /  3  0.05 1.  /  4  0. 1.
*
MATERIAL  1  ELASTIC  E=2.E11 NU=0.3 DENSITY=7800
*
EGROUP  1  PLATE
GSURFACE  1 2 3 4  EL1=1 EL2=5
EDATA  /  1 0.1
*
NSKEWS  INPUT=SURFACE
 1 2 3 4  1
FIXBOUNDARIES  234  /  1 2
*
MESH  NSYMBOLS=MYNODES NNUMBERS=MYNODES BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A49  FUNDAMENTAL FREQUENCY OF CANTILEVER, PLATE
*
DATABASE  CREATE
*
WRITE  FILENAME='a49.lis'
*
FREQUENCIES
MASS-PROPERTIES
*
SET  RESPONSETYPE=VIBRATIONMODE ORIGINAL=DASHED
*
MESH  OUTLINE=YES NSYMBOL=YES
NMAX  DIRECTION=345 NUMBER=3
END
```

EXAMPLE A50

FUNDAMENTAL FREQUENCY OF CANTILEVER, SHELL

Objective

To verify the dynamical behaviour of the SHELL element in frequency analysis when employing SKEW degree-of-freedom Systems.

Physical Problem

Same as in figure on page A44.1.

Finite Element Model

Five 9-node SHELL elements are used to model the cantilever beam as shown in figures on page A50.2. The element stiffness matrix is calculated using a shear factor of 5/6. A consistent mass discretization is used. The determinant search method of analysis is employed in the frequency analysis.

Solution Results

The theoretical solution for this problem is presented in Example A44.

The input data on page A50.4 gives the following result:

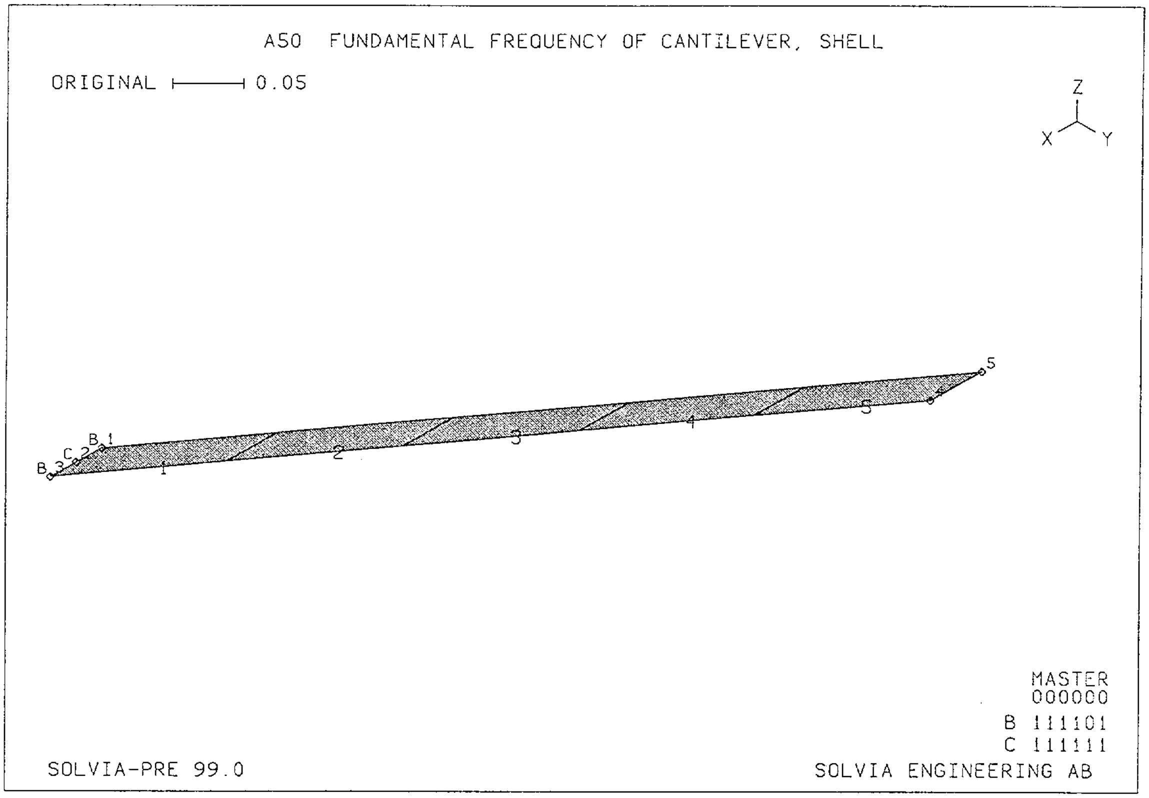
Fundamental frequency (Hz) for motion in the Y-Z plane (mode 2):

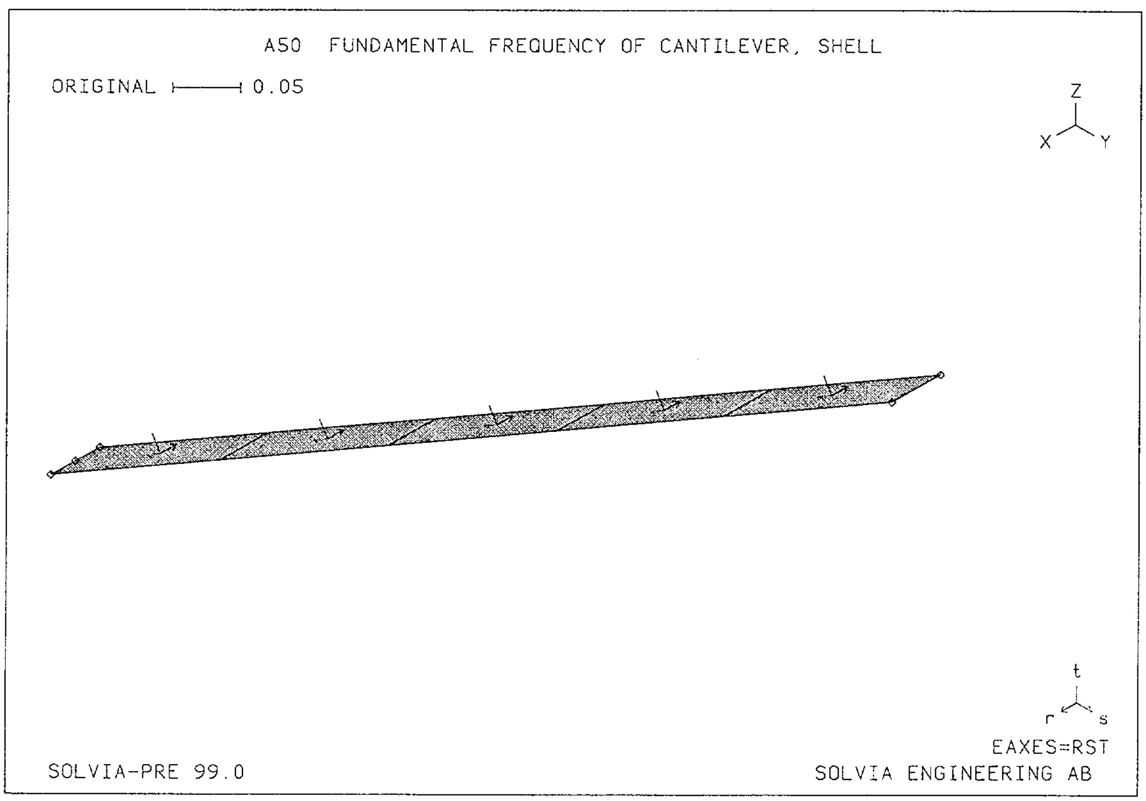
Beam theory	SOLVIA
81.80	81.00

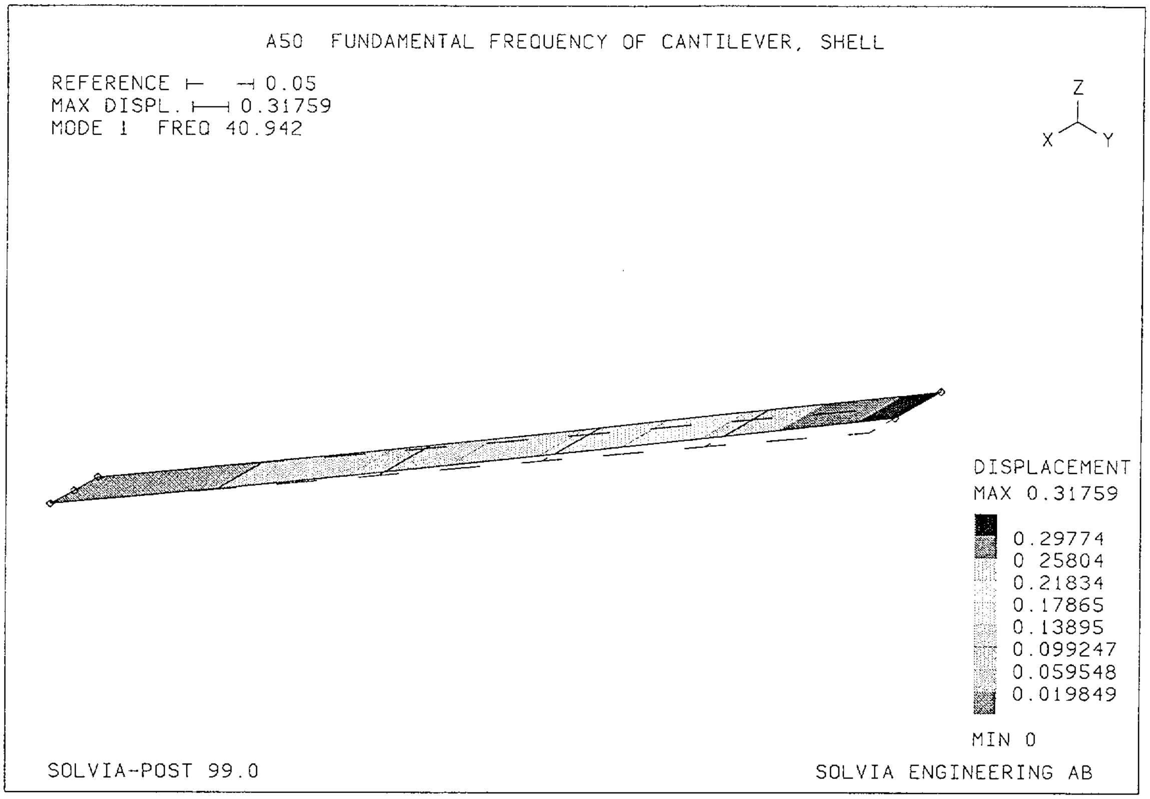
The figures on page A50.3 shows the mode shapes corresponding to the fundamental frequencies calculated in the finite element analysis.

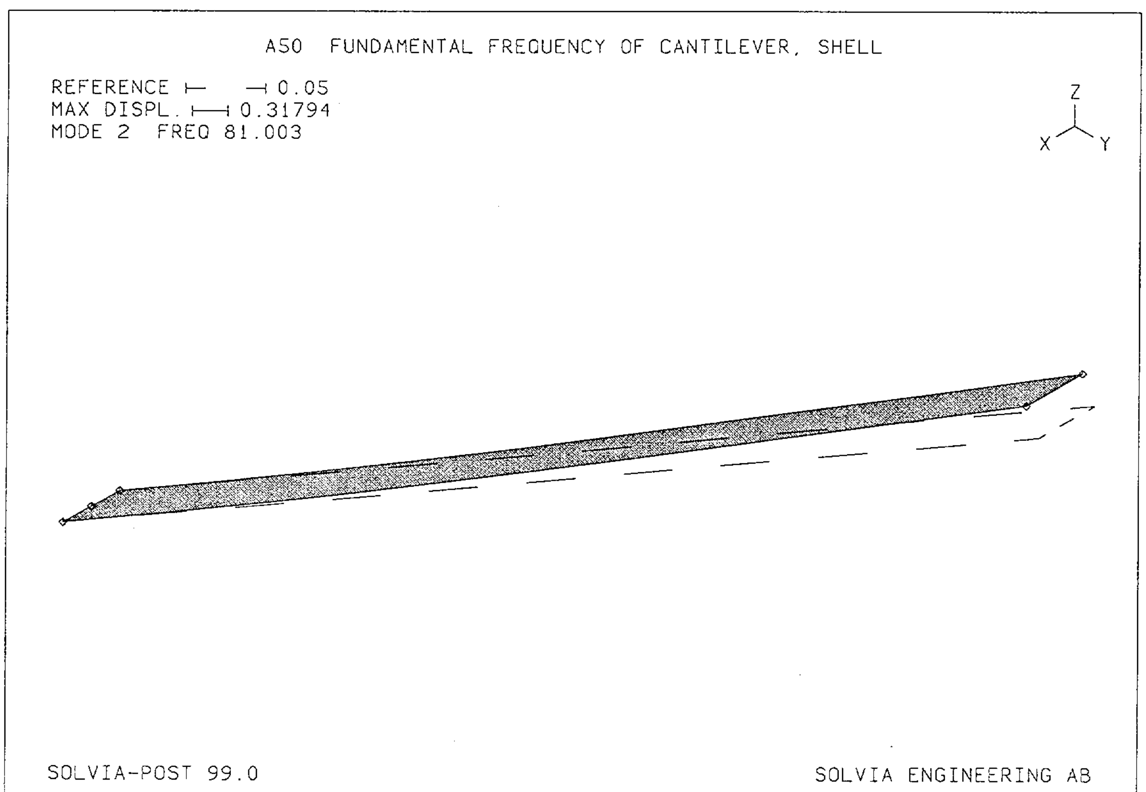
User Hints

- Employing a lumped mass matrix in the finite element analysis yields the result f = 80.73 Hz.


A50 FUNDAMENTAL FREQUENCY OF CANTILEVER, SHELL
ORIGINAL 0.05
Z
X Y
B 3
C 2
B 1
1
2
3
4
5
5
MASTER
000000
B 111101
C 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A50 FUNDAMENTAL FREQUENCY OF CANTILEVER, SHELL
ORIGINAL 0.05
Z
X Y
t
r s
EAXES=RST
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A50 FUNDAMENTAL FREQUENCY OF CANTILEVER, SHELL
REFERENCE 0.05
MAX DISPL. 0.31759
MODE 1 FREQ 40.942
Z
X
Y
DISPLACEMENT
MAX 0.31759
0.29774
0.25804
0.21834
0.17865
0.13895
0.099247
0.059548
0.019849
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A50 FUNDAMENTAL FREQUENCY OF CANTILEVER, SHELL
REFERENCE 0.05
MAX DISPL. 0.31794
MODE 2 FREQ 81.003
Z
X
Y
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A50  FUNDAMENTAL FREQUENCY OF CANTILEVER, SHELL'
*
DATABASE  CREATE
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=CONSISTENT
FREQUENCIES  DETERMINANT-SEARCH NEIG=2
*
SKEWSYSTEM  EULERANGLES
  1  30.
SYSTEM  1  CARTESIAN PHI=30
COORDINATES
 1  TO  3  0.05  /  4  0.05 1.  /  5  0. 1.
*
MATERIAL  1  ELASTIC  E=2.E11 NU=0.3 DENSITY=7800
*
EGROUP  1  SHELL
THICKNESS  1  T1=0.1
GSURFACE  1 3 4 5  EL1=1 EL2=5 NODES=9
*
NSKEWS  INPUT=SURFACE
 1 3 4 5  1
FIXBOUNDARIES  12346  /  1 3
FIXBOUNDARIES  / 2
*
SET  NSYMBOLS=MYNODES
MESH  NNUMBERS=MYNODES ENUMBERS=YES BCODE=ALL
MESH  EAXES=RST
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A.50  FUNDAMENTAL FREQUENCY OF CANTILEVER, SHELL
*
DATABASE  CREATE
*
WRITE  FILENAME='a50.lis'
*
FREQUENCIES
MASS-PROPERTIES
*
SET  RESPONSETYPE=VIBRATIONMODE ORIGINAL=DASHED NSYMBOL=MYNODES
*
MESH  TIME=1 CONTOUR=DISPLACEMENTS
MESH  TIME=2 OUTLINE=YES
*
NMAX  NUMBER=3
END
```

EXAMPLE A51

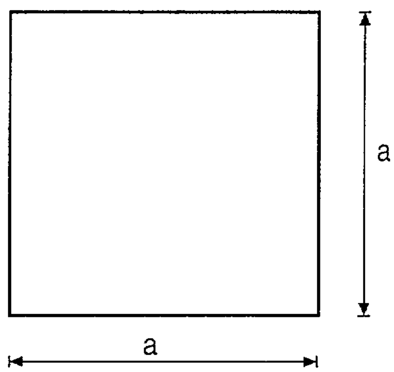
FUNDAMENTAL FREQUENCY OF A SIMPLY SUPPORTED PLATE

Objective

To verify the dynamical behaviour of the SHELL element in frequency analysis.

Physical Problem

The figure below shows the simply supported plate to be analyzed.



$E = 2.0 \cdot 10^{11}$ N/m^2
$\nu = 0.3$
$a = 2.00$ m
$h = 0.01$ m
$\rho = 7850$ kg/m^3

Finite Element Model

The figure on page A51.2 shows the finite element model. Because of symmetry, only one quarter of the plate need to be modeled. Four 9-node SHELL elements are used. A lumped mass matrix is employed and the subspace iteration method is used for the frequency calculation.

Solution Results

The first natural frequency of a simply supported plate is given for example in [1] p. 258 as follows:

$$f_1 = \frac{\pi}{a^2} \cdot \sqrt{\frac{E \cdot h^3}{12 \cdot \rho \cdot h \cdot (1 - \nu^2)}}$$

Insertion of the numerical values gives

$$f_1 = 12.00 \text{ Hz}$$

The SOLVIA solution using the input data on page A51.4 gives

$$f_1 = 12.17 \text{ Hz}$$

The corresponding mode shape and distributions of von Mises effective stress and strain energy density are given in figures on page A51.3.

The calculated sum of the strain energy for the first mode is 2923.41 Nm which is equal to

$$\tfrac{1}{2}\varphi_1^T \mathbf{K}\,\varphi_1 = \tfrac{1}{2}(2\pi f_1)^2 = 2923.41$$

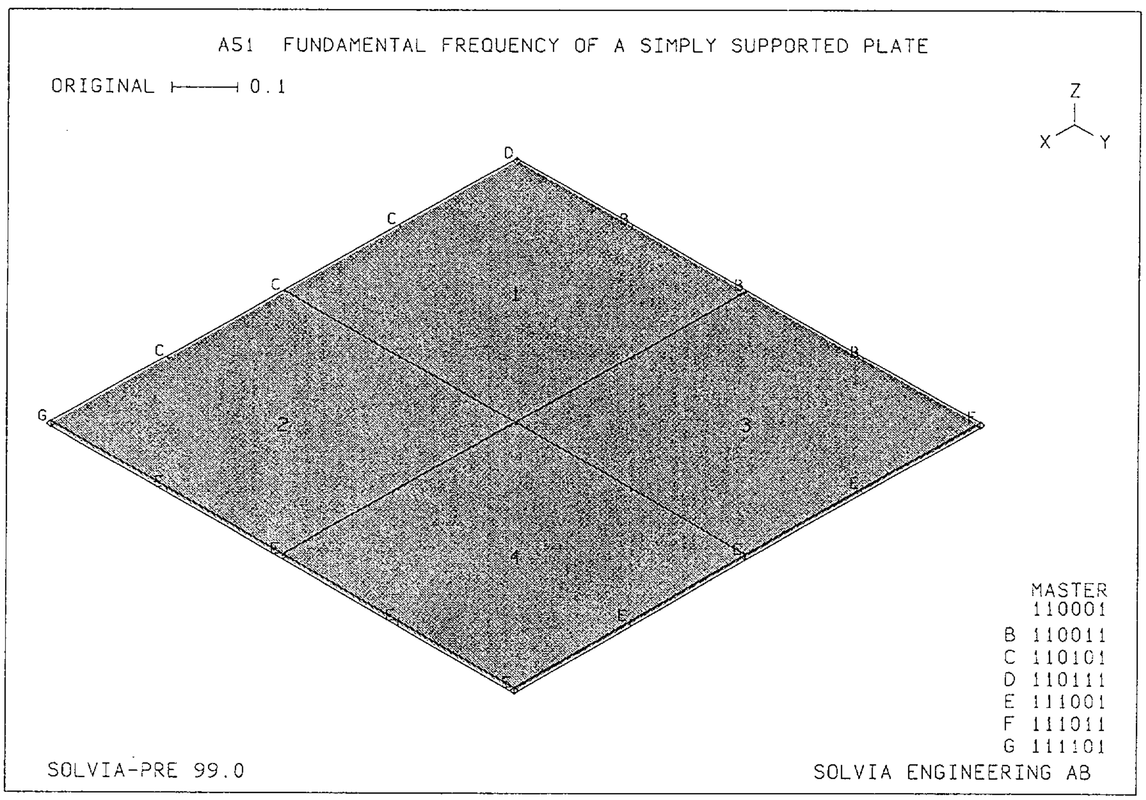
as it should.

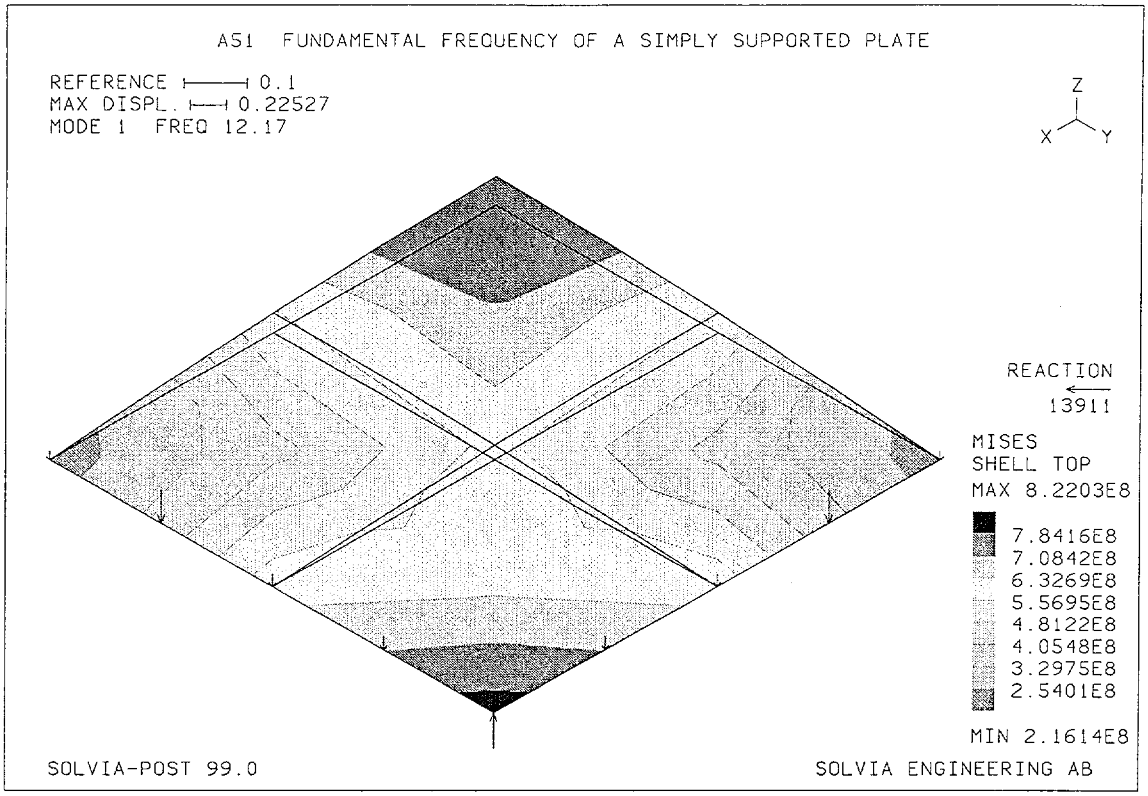
User Hints

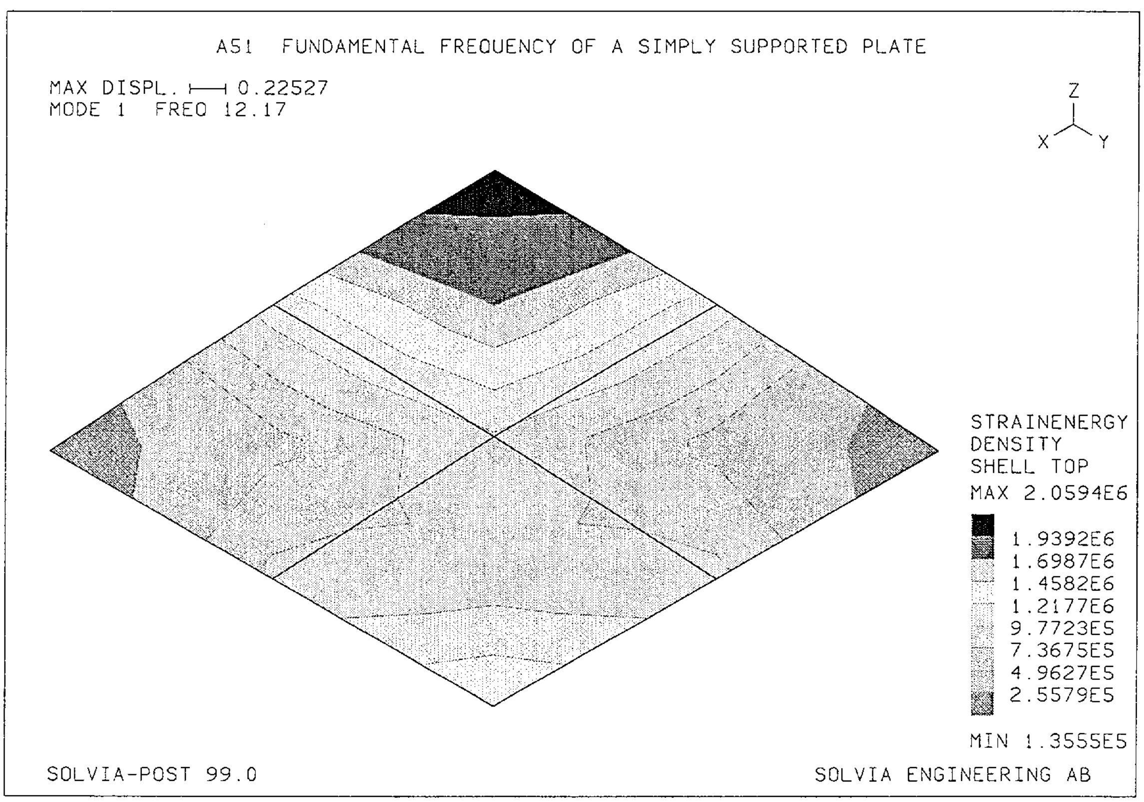
- Note that full integration (default) is necessary when a consistent mass matrix is used since reduced integration (2x2x2) gives a spurious mode of very small frequency for this example.
- A consistent mass matrix gives 12.15 Hz for the fundamental mode.
- Simply supported boundary conditions for a plate are discussed in Example A27.
- A finer mesh should be used if detailed stresses are to be calculated.

Reference

[1] Blevins, R.D., Formulas for Natural Frequency and Mode Shape, Van Nostrand Reinhold Company, 1979.




A51 FUNDAMENTAL FREQUENCY OF A SIMPLY SUPPORTED PLATE
REFERENCE 0.1
MAX DISPL. 0.22527
MODE 1 FREQ 12.17
Z
X
Y
REACTION
13911
MISES
SHELL TOP
MAX 8.2203E8
7.8416E8
7.0842E8
6.3269E8
5.5695E8
4.8122E8
4.0548E8
3.2975E8
2.5401E8
MIN 2.1614E8
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A51 FUNDAMENTAL FREQUENCY OF A SIMPLY SUPPORTED PLATE
MAX DISPL. 0.22527
MODE 1 FREQ 12.17
Z
X
Y
STRAINENERGY
DENSITY
SHELL TOP
MAX 2.0594E6
1.9392E6
1.6987E6
1.4582E6
1.2177E6
9.7723E5
7.3675E5
4.9627E5
2.5579E5
MIN 1.3555E5
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A51  FUNDAMENTAL FREQUENCY OF A SIMPLY SUPPORTED PLATE'
*
DATABASE  CREATE
MASTER  IDOF=110001
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=LUMPED
FREQUENCIES  SUBSPACE-ITERATION NEIG=1 MODALSTRESSES=YES
*
COORDINATES
 1  /  2  1.  /  3  1. 1.  /  4  0. 1.
*
MATERIAL  1  ELASTIC  E=2.E11 NU=0.3 DENSITY=7850
*
EGROUP  1  SHELL
GSURFACE  1 2 3 4  EL1=2 EL2=2 NODES=9
THICKNESS  1  0.01
*
FIXBOUNDARIES  3 INPUT=LINES  /  2 3  /  3 4
FIXBOUNDARIES  4 INPUT=LINES  /  1 2
FIXBOUNDARIES  5 INPUT=LINES  /  4 1
*
SET  NSYMBOLS=MYNODES MIDSURFACE=NO
MESH  ENUMBERS=YES BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A51  FUNDAMENTAL FREQUENCY OF A SIMPLY SUPPORTED PLATE
*
DATABASE  CREATE
*
WRITE  FILENAME='a51.lis'
*
FREQUENCIES
MASS-PROPERTIES
*
SET  RESPONSETYPE=VIBRATIONMODE
MESH  CONTOUR=MISES VECTOR=REACTION ORIGINAL=YES
MESH  CONTOUR=ENERGY
*
NMAX  NUMBER=3
SUMMATION  KIND=ENERGY DETAILS=YES
END
```

EXAMPLE A52

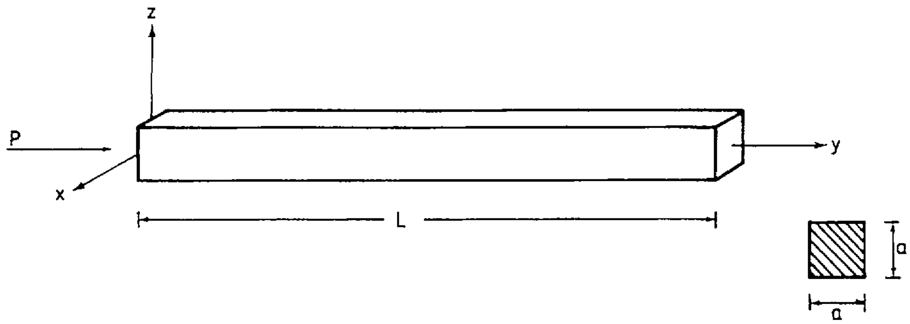
WAVE PROPAGATION IN A ROD

Objective

To verify the dynamical behaviour of the TRUSS element in wave propagation analysis.

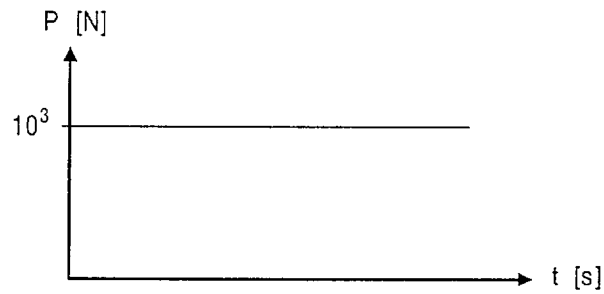
Physical Problem

A uniform bar free at both ends is considered, see figure below. An axial step force is applied at one end of the bar at time 0.



$a = 0.1$ m $\quad E = 2.0 \cdot 10^9$ N/m^2

$L = 1.0$ m $\quad \rho = 1000$ kg / m^3



Finite Element Model

The finite element model is shown in the top figure on page A52.4. The bar is modeled using ten 2-node TRUSS elements. The explicit central difference method is chosen for the time integration and a lumped mass matrix is employed.

The time step Δt is selected according to

$$\Delta t = \frac{L_e}{c} = L_e \cdot \sqrt{\frac{\rho}{E}}$$

where

L_e = length of one truss element

c = wave velocity in the material

ρ = mass density

E = Young's modulus

Since the step load is applied at time 0 it is necessary to specify the corresponding initial acceleration according to

$$a_1 = \frac{P}{m_1}$$

where

m_1 = mass at node 1

P = applied force

a_1 = acceleration at node 1

Solution Results

The theoretical solution for this problem is presented, for example, in [1].

The input data on page A52.5 gives the following results considering 4 time steps:

Longitudinal stress in bar, σ (N/m^2):

	Theory	SOLVIA
Before wave front	0	0
After wave front	$-1.0\cdot10^5$	$-1.0\cdot10^5$

The wave front can be seen in the SOLVIA solution to advance one element length per specified time step, which is in agreement with the theoretical solution.

The velocity and acceleration time history for the typical node 2 is shown in the bottom figure on page A52.4.

User Hints

- In this example with equal lengths of the TRUSS elements the exact theoretical solution is obtained by selecting the time step to be equal to the critical time step.

- Note that the load vector input by time functions at the start of the solution (time=0) is not used in SOLVIA. Instead the load vector ${}^{\circ}\mathbf{R}$ at time 0 is assumed to be

$${}^{\circ}\mathbf{R} = \mathbf{M}\,{}^{\circ}\ddot{\mathbf{U}} + \mathbf{C}\,{}^{\circ}\dot{\mathbf{U}} + {}^{\circ}\mathbf{F}$$

where ${}^{\circ}\dot{\mathbf{U}}$ and ${}^{\circ}\ddot{\mathbf{U}}$ are the initial velocity and acceleration vectors and **M** and **C** are the mass and damping matrices, respectively. ${}^{\circ}\mathbf{F}$ is a vector with the nodal equivalent forces corresponding to the initial displacements. Therefore, the initial acceleration is specified as described above to simulate the applied force at time 0. To start the central difference solution procedure ${}^{\Delta t}\mathbf{U}$ is calculated from the initial conditions as

$$^{\Delta t}\mathbf{U} = {}^{\circ}\mathbf{U} + \Delta t \cdot {}^{\circ}\dot{\mathbf{U}} + \frac{\Delta t^2}{2} \cdot {}^{\circ}\ddot{\mathbf{U}}$$

- Note that the finite element model is completely free to move in the X-direction. If this model is used for a static problem, no solution would be possible since the model contains rigid body modes (and the solution is non-unique). In the dynamic case, however, the inertia of the bar results in a unique solution, and no solution difficulties are encountered.

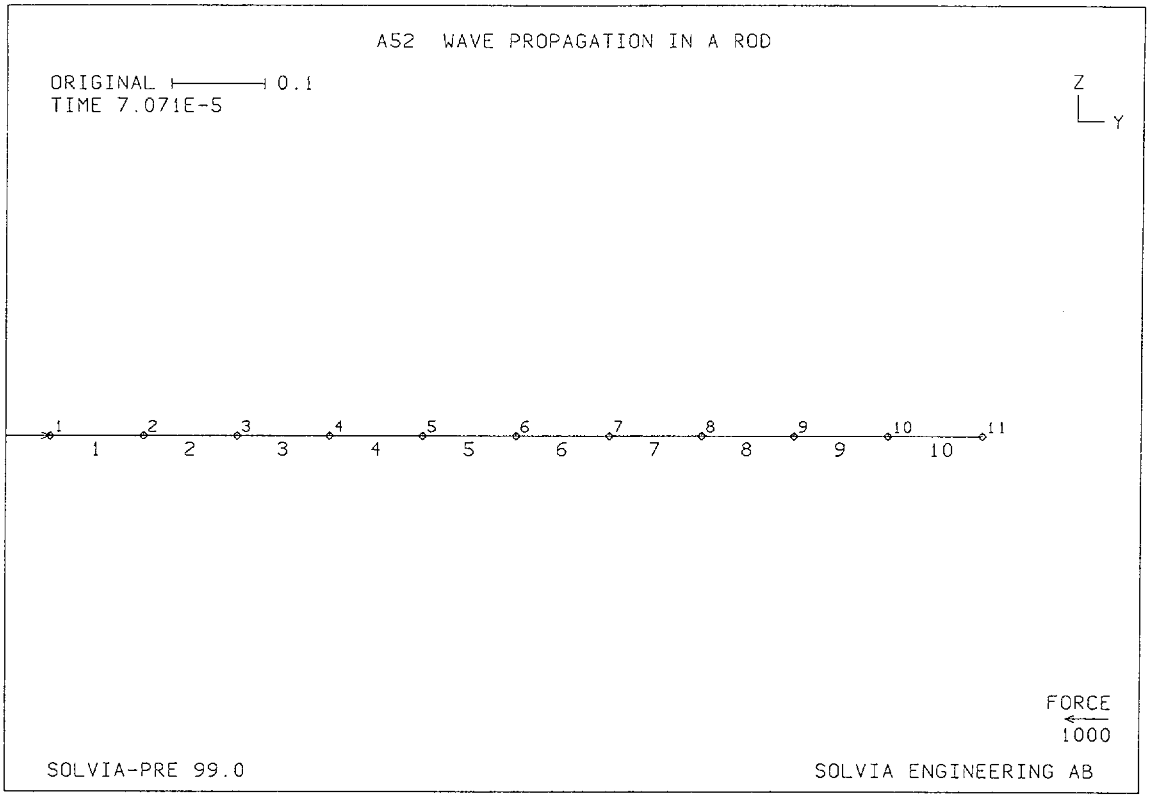
- In explicit time integration the total system stiffness matrix is not calculated and no matrix factorization is carried out because with explicit time integration the mass matrix in SOLVIA is required to be diagonal. The solution for one time step is therefore obtained very effectively but a large number of time steps may need to be used, since the time step Δt must satisfy

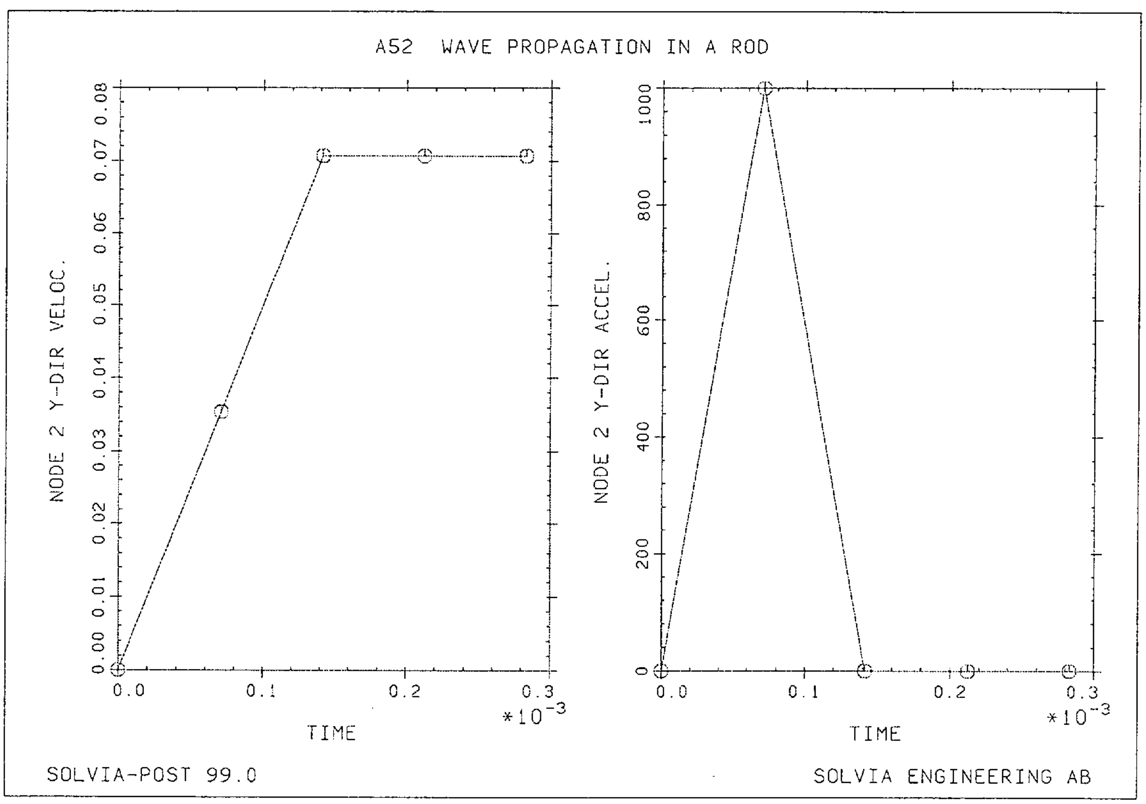
  $$\Delta t \leq \Delta t_{cr} = \frac{T_n}{\pi}$$

  where T_n is the smallest period of the finite element assemblage with n degrees of freedom.

Reference

[1] Zukas, J.A., Nicholas, T., Swift, H.F., Greszczuk, L.B. and Curran, D.R., Impact Dynamics, John Wiley & Sons, 1982.


A52 WAVE PROPAGATION IN A ROD
ORIGINAL 0.1
TIME 7.071E-5
Z
Y
1 2 3 4 5 6 7 8 9 10 11
1 2 3 4 5 6 7 8 9 10
FORCE
1000
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A52 WAVE PROPAGATION IN A ROD
NODE 2 Y-DIR VELOC.
0.00 0.01 0.02 0.03 0.04 0.05 0.06 0.07 0.08
0.0 0.1 0.2 0.3
*10^-3
TIME
NODE 2 Y-DIR ACCEL.
0 200 400 600 800 1000
0.0 0.1 0.2 0.3
*10^-3
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A52  WAVE PROPAGATION IN A ROD'
*
DATABASE  CREATE
MASTER  IDOF=101111 NSTEP=4 DT=7.071E-5
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=LUMPED METHOD=CENTRAL
*
COORDINATES
 1  TO  11  0. 1.
*
INITIAL  ACCELERATION
 1  0. 2.E3 0.
*
MATERIAL  1  ELASTIC  E=2.E9 DENSITY=1000
*
EGROUP  1 TRUSS
ENODES
 1  1 2  TO  10  10 11
EDATA  /  1  0.01
*
LOADS  CONCENTRATED
 1 2 1000.
*
SET  VIEW=X NSYMBOLS=YES
MESH  NNUMBERS=YES ENUMBERS=YES VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A52  WAVE PROPAGATION IN A ROD
*
DATABASE CREATE
*
WRITE  FILENAME='a52.lis'
*
MASS-PROPERTIES
*
SUBFRAME  21
NHISTORY  NODE=2 DIRECTION=2 KIND=VELOCITY SYMBOL=1
NHISTORY  NODE=2 DIRECTION=2 KIND=ACCELERATION SYMBOL=1
*
ELIST  TSTART=7.071E-5 TEND=2.8284E-4
END
```

EXAMPLE A53

WAVE PROPAGATION IN A WATER COLUMN

Objective

To verify the dynamical behaviour of the FLUID3 element in wave propagation analysis.

Physical Problem

Same as in the figure on page A52.1 except that the rod is replaced by a water column of the same wave velocity c.

Finite Element Model

Ten 8-node FLUID3 elements are modeling the water column as shown in the top figure on page A53.2. A lumped mass matrix is used in the analysis and the explicit central difference method is chosen for the time integration. The element stiffness matrices are calculated using one-point-integration. Time step and initial conditions are selected as for Example A52.

Solution Results

The theoretical solution is the one presented in Example A52.

The input data on page A53.3 gives the following results:

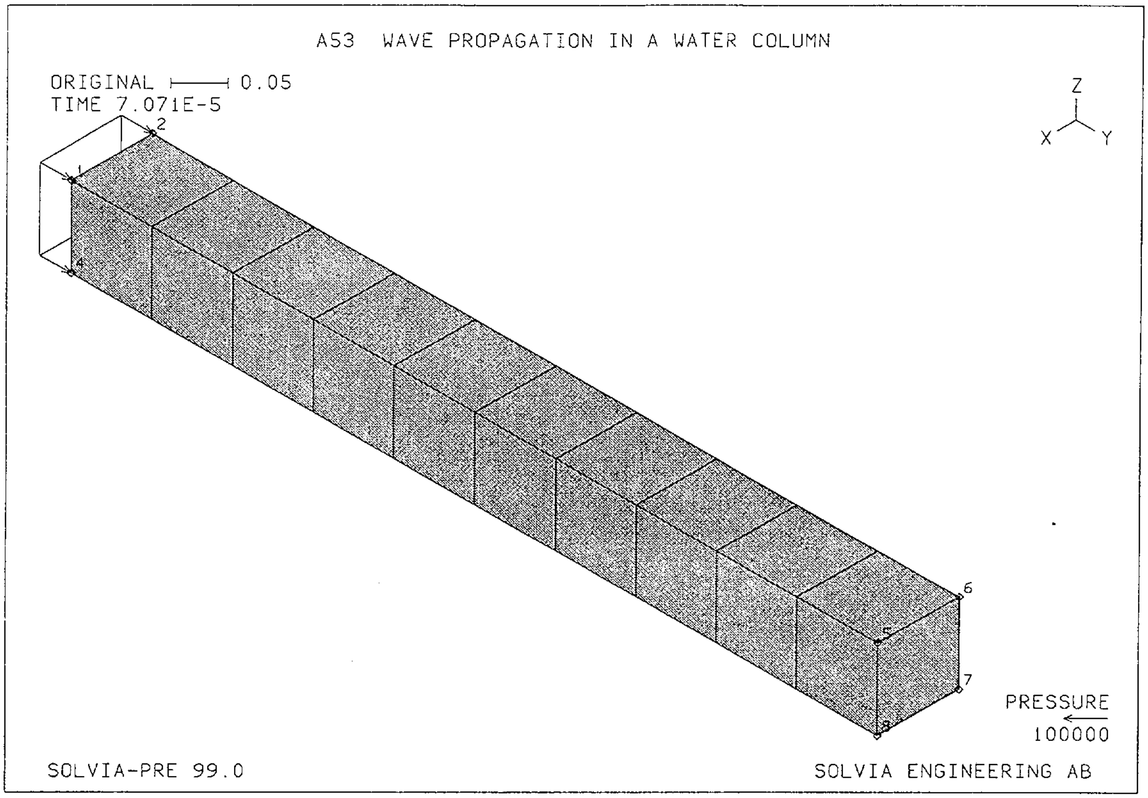
Pressure in water column, p $\left(N/m^2\right)$:

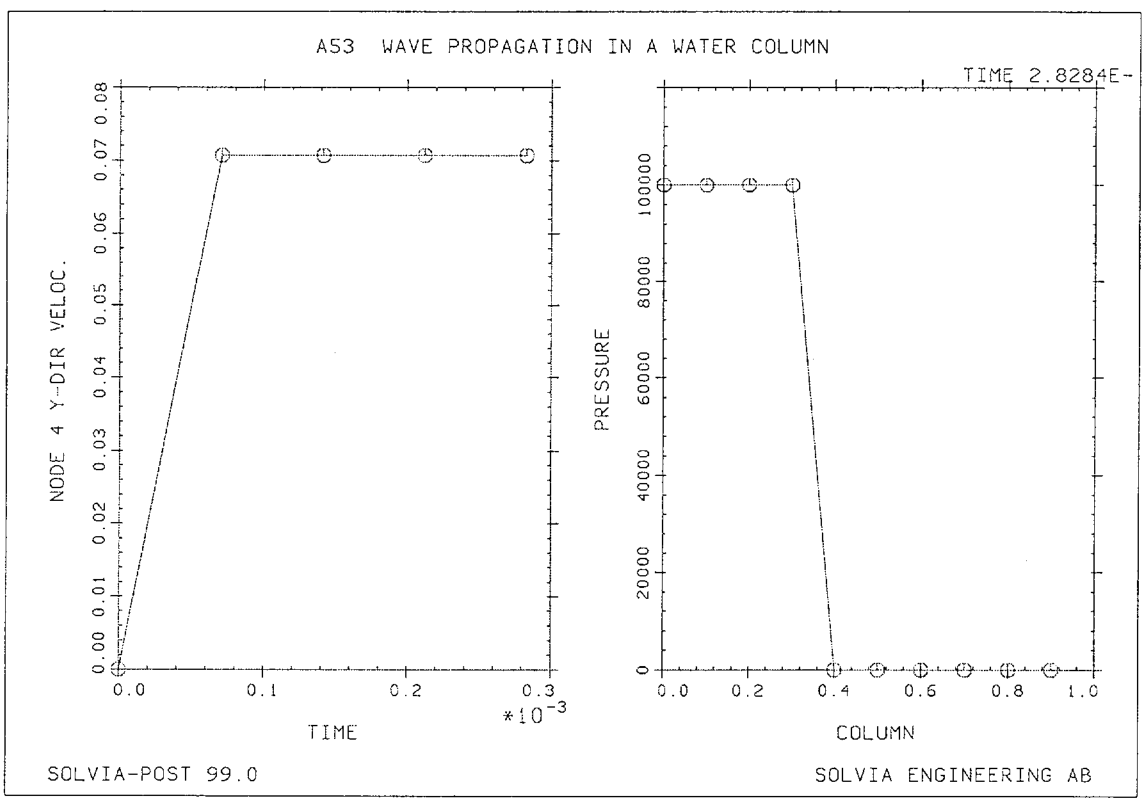
	Theory	SOLVIA
Before wave front	0	0
After wave front	$-1.0\cdot10^5$	$-1.0\cdot10^5$

The bottom figure on page A53.2 shows the pressure distribution in the water column at the fourth time step predicted by the finite element analysis which is in excellent agreement with the analytical solution.

User Hints

See Example A52.


A53 WAVE PROPAGATION IN A WATER COLUMN
ORIGINAL 0.05
TIME 7.071E-5
Z
X Y
PRESSURE
100000
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A53 WAVE PROPAGATION IN A WATER COLUMN
TIME 2.8284E-
NODE 4 Y-DIR VELOC.
TIME
*10^-3
PRESSURE
COLUMN
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A53  WAVE PROPAGATION IN A WATER COLUMN'
*
DATABASE  CREATE
MASTER  IDOF=101111 NSTEP=4 DT=7.0710E-5
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=LUMPED METHOD=CENTRAL
*
COORDINATES
 1  .1 0. .1  /  2  0. 0. .1  /  3  /  4  .1
*
INITIAL ACCELERATION
 1 0. 2.E3  TO  4 0. 2.E3
*
NGENERATION  NSTEP=4 YSTEP=1.  /  1  TO  4
*
MATERIAL  1  FLUID K=2.E9 DENSITY=1000.
*
EGROUP  1  FLUID3  RSINT=1 TINT=1
GVOLUME  1 2 3 4 5 6 7 8  EL1=1 EL2=1 EL3=10 NODES=8
*
LOADS  ELEMENT  INPUT=SURFACE
 1 2 3 4  1.E5
*
MESH  NSYMBOLS=MYNODES NNUMBERS=MYNODES VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A53  WAVE PROPAGATION IN A WATER COLUMN
*
DATABASE  CREATE
*
WRITE  FILENAME='a53.lis'
*
SUBFRAME  21
NHISTORY  NODE=4 DIRECTION=2 KIND=VELOCITY SYMBOL=1
*
EPLINE  NAME=COLUMN  /  1 1  TO  10 1
ELINE  LINENAME=COLUMN KIND=PRESSURE SYMBOL=1 OUTPUT=ALL
END
```

EXAMPLE A54

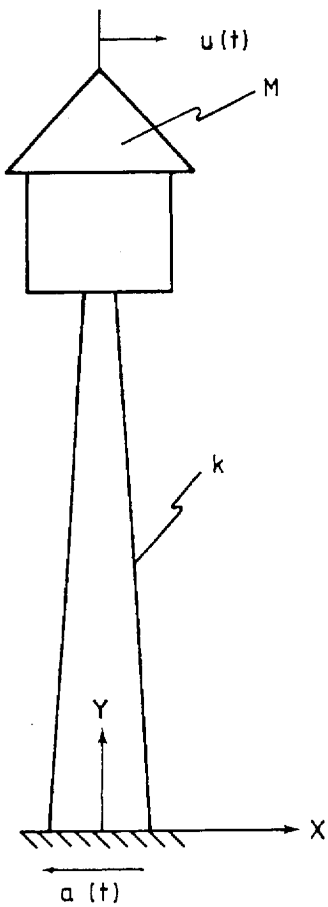
CANTILEVER SUBJECTED TO GROUND MOTION

Objective

To verify the dynamical behaviour of the BEAM element and the solution procedure for mass proportional loading due to a ground acceleration.

Physical Problem

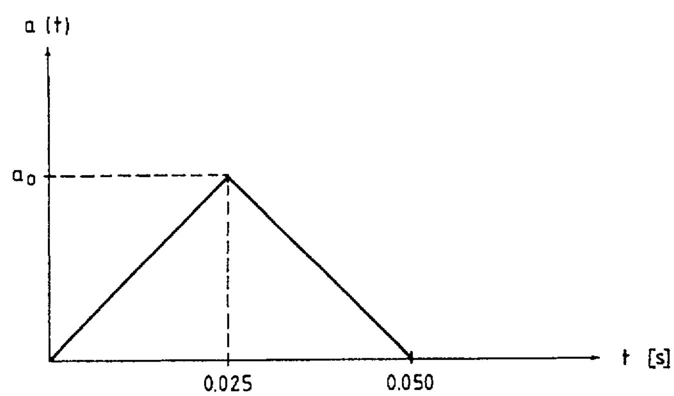
A water tower subjected to ground acceleration, as shown in the figure below, is considered. The time history of the ground motion is also shown in the figure.



$k = 2.7 \cdot 10^6$ lbf / ft

$M = 3000$ $\text{lbf} \cdot \text{s}^2$ / ft

$a_0 = 32.2$ ft / s^2



Finite Element Model

One BEAM element is used to model the tower and the water tank is modeled as a concentrated mass. The transverse stiffness is set equal to the spring constant k for the tower. Thus

$$\frac{3EI}{L^3} = k = 2.7 \cdot 10^6 \ \text{lbf/ft}$$

and we select (for ease of computation)

$I = 1 \ ft^4$

$L = 10 ft$

$E = 0.9 \cdot 10^9 \ lbf / ft^2$

The Newmark method of time integration is chosen with $\alpha = 0.25$ and $\delta = 0.50$. The time step size is 0.005 sec.

Solution Results

The input data on page A54.4 is used in the finite element analysis. The table below shows the calculated results for some time steps in the analysis together with the analytical solution and results reported in [1].

	Transverse displacement, u(t) [ft]		
Time	Analytical	Ref. [1]	SOLVIA
0.010	0.000214	0.0002	0.000239
0.020	0.00169	0.0017	0.00173
0.030	0.00551	0.0055	0.00554
0.040	0.0113	0.0114	0.0113
0.050	0.0174	0.0176	0.0174
0.075	0.0255	-	0.0254

The figures on page A54.3 shows the time history of the transverse displacement, velocity and acceleration at the top of the water tower and the bending moment at the built-in end of the BEAM element, as displayed by SOLVIA-POST.

User Hints

- Note that the forces to which the model is subjected are calculated as

  $$\mathbf{M}\ddot{\mathbf{U}} + \mathbf{K}\mathbf{U} = -\mathbf{M}\mathbf{d}_x \ddot{u}_g$$

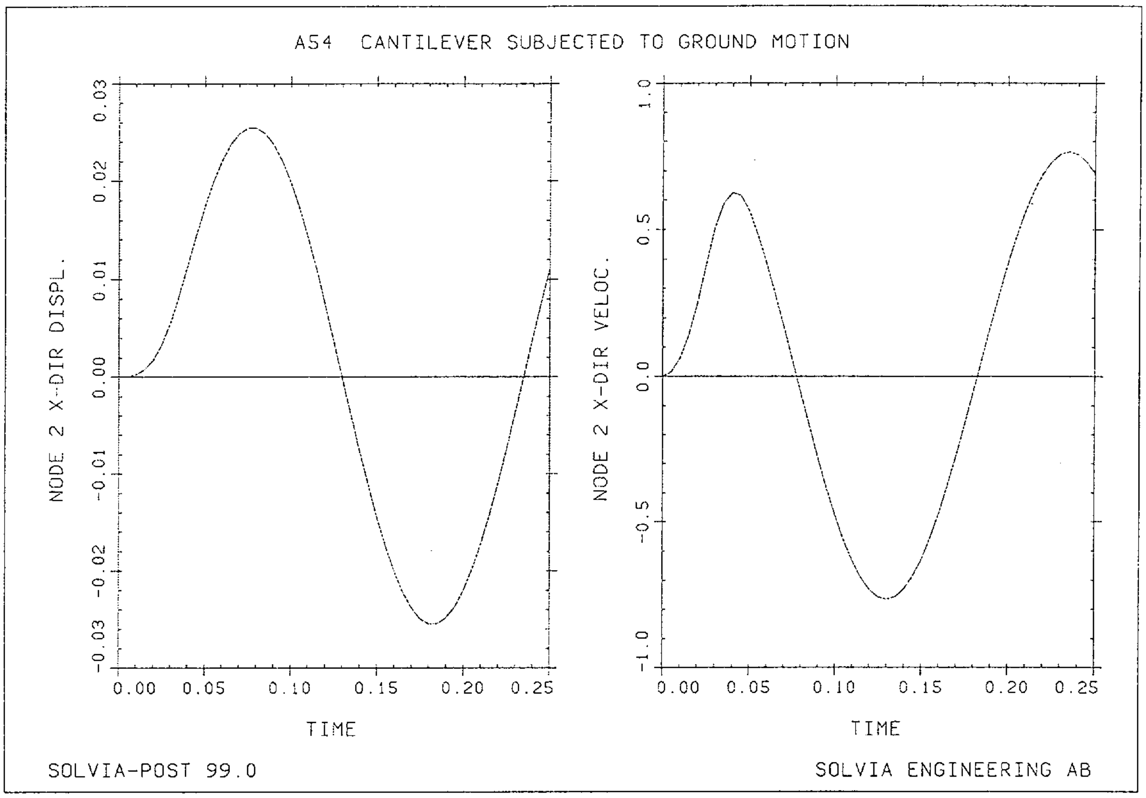
  where $\mathbf{d}_x$ is a vector with zeroes in all components except +1 (plus one) in each of the components corresponding to the X-displacements. Hence, the same acceleration forces are applied to all X-displacements degrees of freedom, which means that arrival time differences in the accelerations at different supports are not modeled.

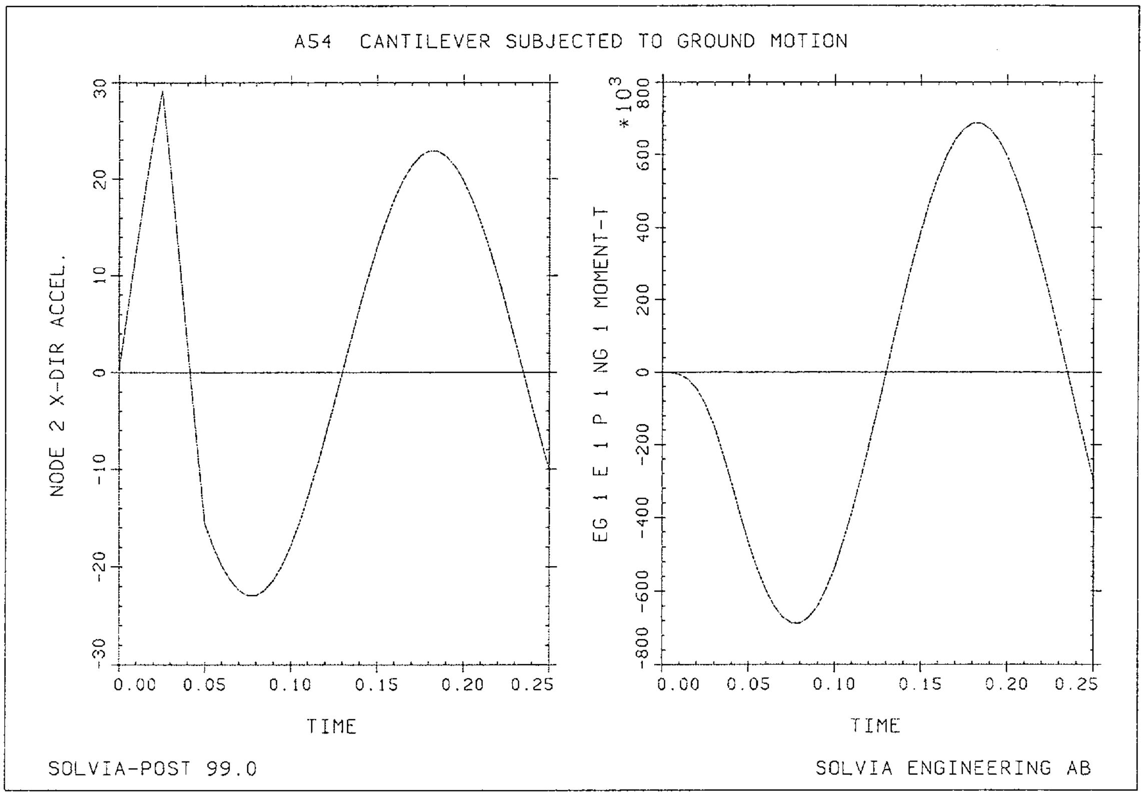
- Note that a ground acceleration in the negative X-direction corresponds to an applied mass proportional loading in the positive X-direction.

- Note also that the X-Y-Z system is moving with the ground and we obtain a response measured in this moving system. Therefore, to obtain, for example, the absolute acceleration response (which is measured in a fixed system) it is necessary to add or subtract the ground acceleration time history depending on whether the ground acceleration acts in the positive or negative coordinate direction.

Reference

[1] Clough, R.W. and Penzien, J., Dynamics of Structures, McGraw-Hill, 1975, pp. 102-105.


A54 CANTILEVER SUBJECTED TO GROUND MOTION
NODE 2 X-DIR DISPL.
NODE 2 X-DIR VELOC.
TIME
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A54 CANTILEVER SUBJECTED TO GROUND MOTION
NODE 2 X-DIR ACCEL.
EG 1 E 1 P 1 NG 1 MOMENT-T
*10^3
TIME
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A54  CANTILEVER SUBJECTED TO GROUND MOTION'
*
DATABASE  CREATE
MASTER  IDOF=001110 NSTEP=50 DT=0.005
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=LUMPED METHOD=NEWMARK,
          DELTA=0.5 ALPHA=0.25
*
TIMEFUNCTION  1
 0. 0. /  .025 1. /  .050 0. /  1. 0.
*
COORDINATES
 1  /  2  0. 10.  /  3  -3.
*
MASSES  /  2  3000. 3000. 3000.
*
MATERIAL  1  ELASTIC  E=0.9E9
*
EGROUP  1  BEAM  RESULT=FORCES
SECTION  1  GENERAL  RINERTIA=1. SINERTIA=1. TINERTIA=1. AREA=1.
ENODES  /  1  3 1 2
*
FIXBOUNDARIES  /  1 3
LOADS  MASSPROPORTIONAL  XFACTOR=1. ACCGRA=32.2
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A54  CANTILEVER SUBJECTED TO GROUND MOTION
*
DATABASE  CREATE
*
WRITE  FILENAME='a54.lis'
*
SUBFRAME  21
NHISTORY  NODE=2 DIRECTION=1 KIND=DISPLACEMENT OUTPUT=ALL
NHISTORY  NODE=2 DIRECTION=1 KIND=VELOCITY
*
SUBFRAME  21
NHISTORY  NODE=2 DIRECTION=1 KIND=ACCELERATION
EHISTORY  EL=1 POINT=1 KIND=MT
END
```

EXAMPLE A55

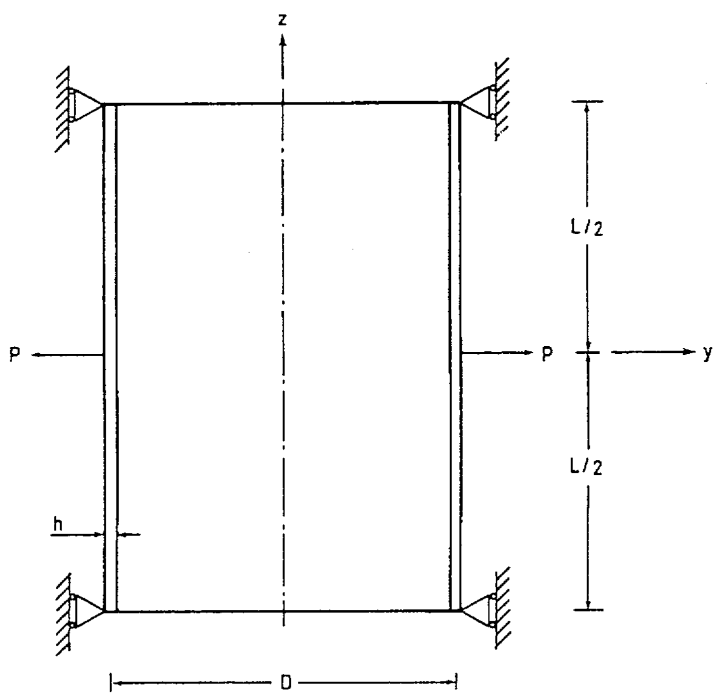
CYLINDRICAL TUBE UNDER STEP LOADING

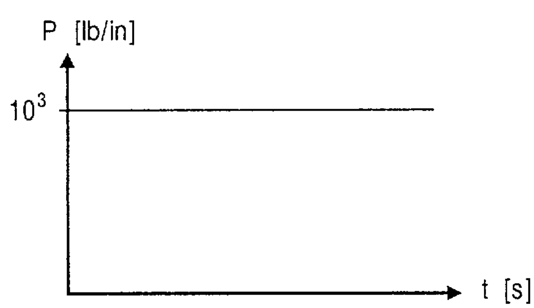
Objective

To verify the dynamical behaviour of the PLANE AXISYMMETRIC element and the use of direct time integration.

Physical Problem

A cylindrical tube initially at rest is subjected to a uniform circumferential line load at its midspan at time $t=0^+$, as shown in the figure below.





L = 18.0 in

D = 6.0 in

h = 0.3 in

E = $3.0 \cdot 10^7$ psi

ν = 0.3

$\rho = 3.663 \cdot 10^{-2}$ lbf.sec^2/ in.^{4}

Finite Element Model

The finite element model used in the analysis is shown in the left figure on page A55.2. Using symmetry considerations only one-half of the tube is modeled with sixteen 8-node PLANE AXISYMMETRIC elements. The trapezoidal rule (the Newmark method with α=0.25 and δ=0.50) is employed in the analysis to obtain the step-by-step dynamic response. A step size of 0.00001 sec. is used and the solution response is evaluated for 60 steps in the analysis. A consistent mass matrix is used.

Solution Results

The theoretical solution for this problem is presented in [1] pp. 55-56. The input data on page A55.3 is used in the finite element analysis.

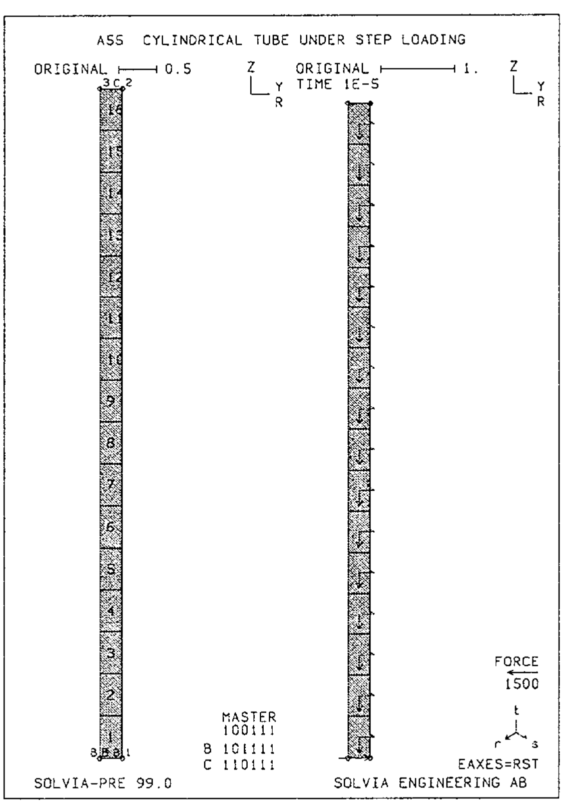
The right figure on page A55.2 shows the deformed finite element mesh at time t=0.00060 sec. and a time-history curve of the radial displacements at the symmetry boundary. The calculated results are in good agreement with the theoretical solution.

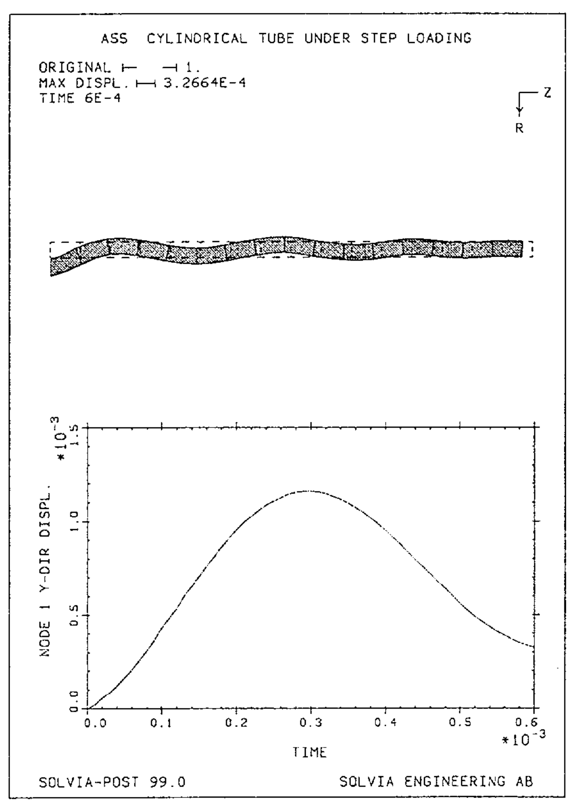
User Hints

- The trapezoidal rule is unconditionally stable, and the time step size is therefore selected based on accuracy considerations only. Note that a time step for use of the central difference method would be considerably smaller.

Reference

[1] "SAP IV - A Structural Analysis Program for Static and Dynamic Response of Linear Systems", Report EERC 73-11, Univ. of Calif., Berkeley, Calif., 1974 (revised).





SOLVIA-PRE input

```
HEADING  'A55  CYLINDRICAL TUBE UNDER STEP LOADING'
*
DATABASE  CREATE
MASTER  IDOF=100111 NSTEP=60 DT=1.E-5
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=CONSISTENT METHOD=NEWMARK
*
TIMEFUNCTION  1
 0. 3000.  /  1 3000.
*
COORDINATES  /  ENTRIES  NODE   Y     Z
 1  3.15  /  2  3.15  9.  /  3  2.85  9.  /  4  2.85
*
MATERIAL  1  ELASTIC  E=3.0E7 NU=0.3 DENSITY=3.663E-2
*
EGROUP  1  PLANE AXISYMMETRIC
GSURFACE  1 2 3 4  EL1=16 EL2=1 NODES=8
*
FIXBOUNDARIES  3  INPUT=LINES  /  1 4
FIXBOUNDARIES  2  INPUT=NODES  /  2
*
LOADS  CONCENTRATED
 1 2 0.5
*
SET  PLOTORIENTATION=PORTRAIT NSYMBOLS=MYNODES
MESH  NNUMBERS=MYNODES ENUMBERS=YES BCODE=ALL SUBFRAME=21
MESH  VECTOR=LOAD EAXES=RST
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A55  CYLINDRICAL TUBE UNDER STEP LOADING
*
DATABASE  CREATE
WRITE  FILENAME='a55.lis'
*
SET  PLOTORIENTATION=PORTRAIT
VIEW  ID=1 XVIEW=1 ROT=-90
MESH  VIEW=1 ORIGINAL=DASHED SUBFRAME=12
*
NHISTORY  NODE=1 DIRECTION=2 OUTPUT=ALL
END
```

EXAMPLE A56

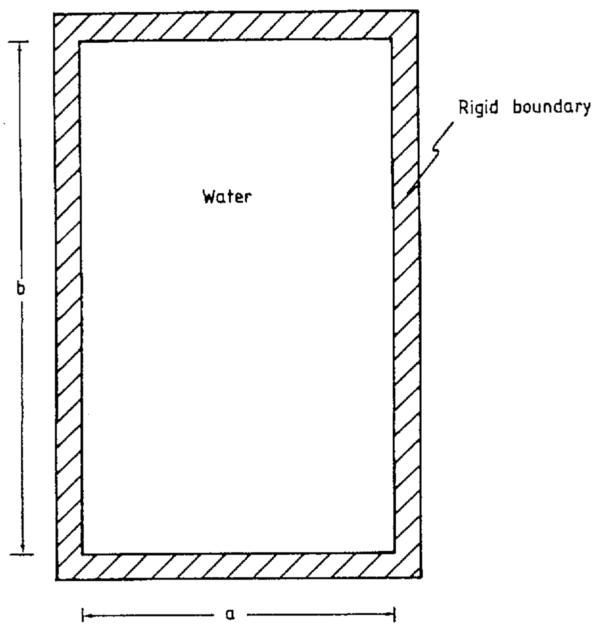
FREQUENCIES OF A WATER-FILLED ACOUSTIC CAVITY

Objective

To verify the PLANE conduction element in SOLVIA-TEMP when used for frequency analysis.

Physical Problem

The acoustic cavity considered for the analysis is shown in the figure below. The cavity is bounded by rigid walls and filled with water which is assumed to be inviscid [1]. The natural frequencies of the cavity are to be determined.



$a = 12$ in

$b = 20$ in

$\beta = 3.16 \cdot 10^5$ psi (bulk modulus)

$\rho = 9.35 \cdot 10^{-5}$ $\mathrm{lb\,sec^2/in^4}$ (density)

Material specification to SOLVIA-TEMP:

$3.16 \cdot 10^5$ (conductivity)

$9.35 \cdot 10^{-5}$ (specific heat per unit volume)

Finite Element Model

The wave equation governing the motion of the fluid inside the cavity is

$$\frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = \frac{1}{c^2} \cdot \frac{\partial^2 \phi}{\partial t^2}; \qquad c = \sqrt{\frac{\beta}{\rho}}$$

$$\frac{\partial \phi}{\partial n} = 0 \qquad \text{at the boundary}$$

where ϕ is the velocity potential, c is the velocity of sound in water, β is the bulk modulus and ρ is the density of the water. The natural frequencies of the cavity can be obtained by performing a frequency analysis of a planar heat flow problem in which the conductivity (k) and specific heat (c) correspond to β and ρ, respectively. Using this approach, the finite element model shown in the top figure on page A56.3 is employed in the analysis. Four 8-node PLANE conduction elements are used to model the cavity. The degrees-of-freedom of all the nodal points are left free corresponding to the boundary conditions of the cavity. In the finite element frequency analysis, a consistent heat capacity assumption is used.

Solution Results

The input data shown on page A56.4 is used in the finite element frequency analysis. The lowest four frequencies predicted by the finite element model are shown in the table below. Analytical solutions evaluated using formulas given in [1] are also shown for comparison.

	SOLVIA-TEMP	Analytical *[1]
$\omega_1(n=0, m=0)$	0	0
$\omega_2(n=0, m=1)$	9166	9132
$\omega_3(n=1, m=0)$	15277	15220
$\omega_4(n=1, m=1)$	17763	17749

*) $\omega = \pi \cdot c \cdot \sqrt{\left(\frac{n}{a}\right)^2 + \left(\frac{m}{b}\right)^2}$;m, n = 0,1,2,3,..

Contour plots of temperature eigenvectors can be seen in the bottom figure on page A56.3.

User Hints

- SOLVIA-TEMP calculates the eigenvalues to the equation

  $$\mathbf{K}\,\boldsymbol{\phi} = \mathbf{C}\,\boldsymbol{\phi}\,\boldsymbol{\Lambda}$$

  where

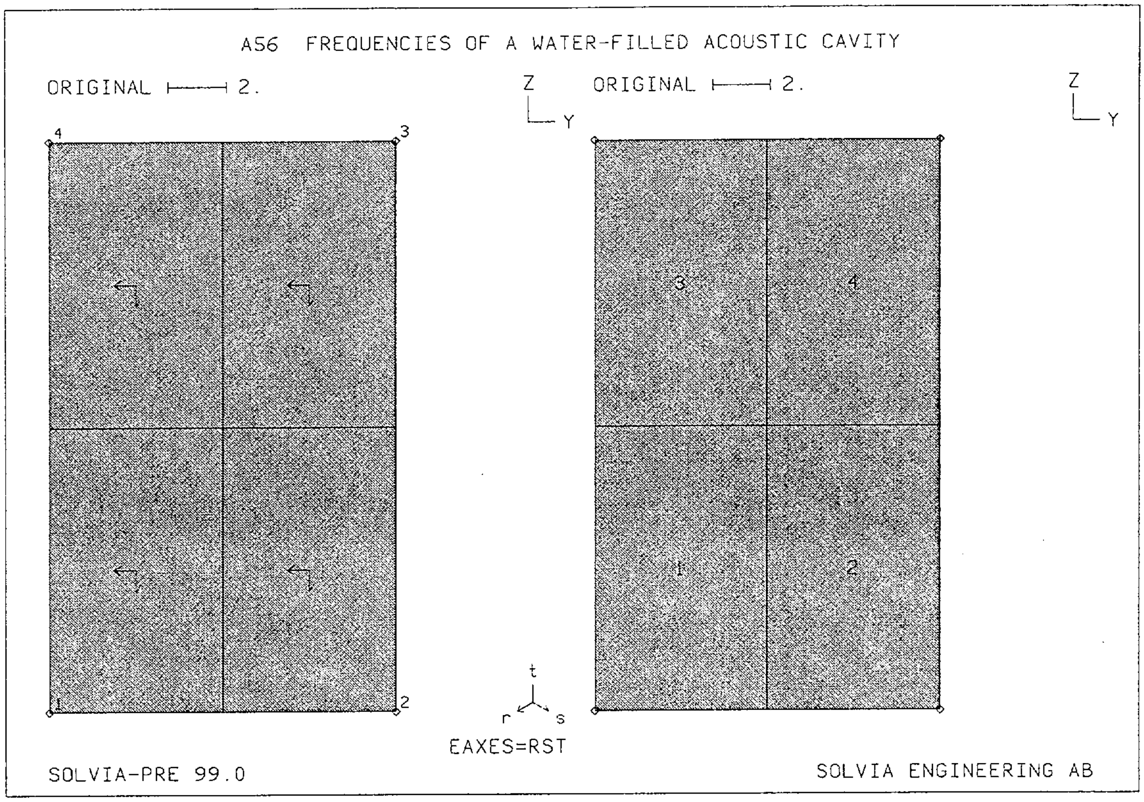
  $\mathbf{K}$ = heat conductivity matrix
  $\mathbf{C}$ = heat capacity matrix
  $\boldsymbol{\phi}$ = matrix containing eigenvectors
  $\boldsymbol{\Lambda}$ = diagonal matrix with eigenvalues

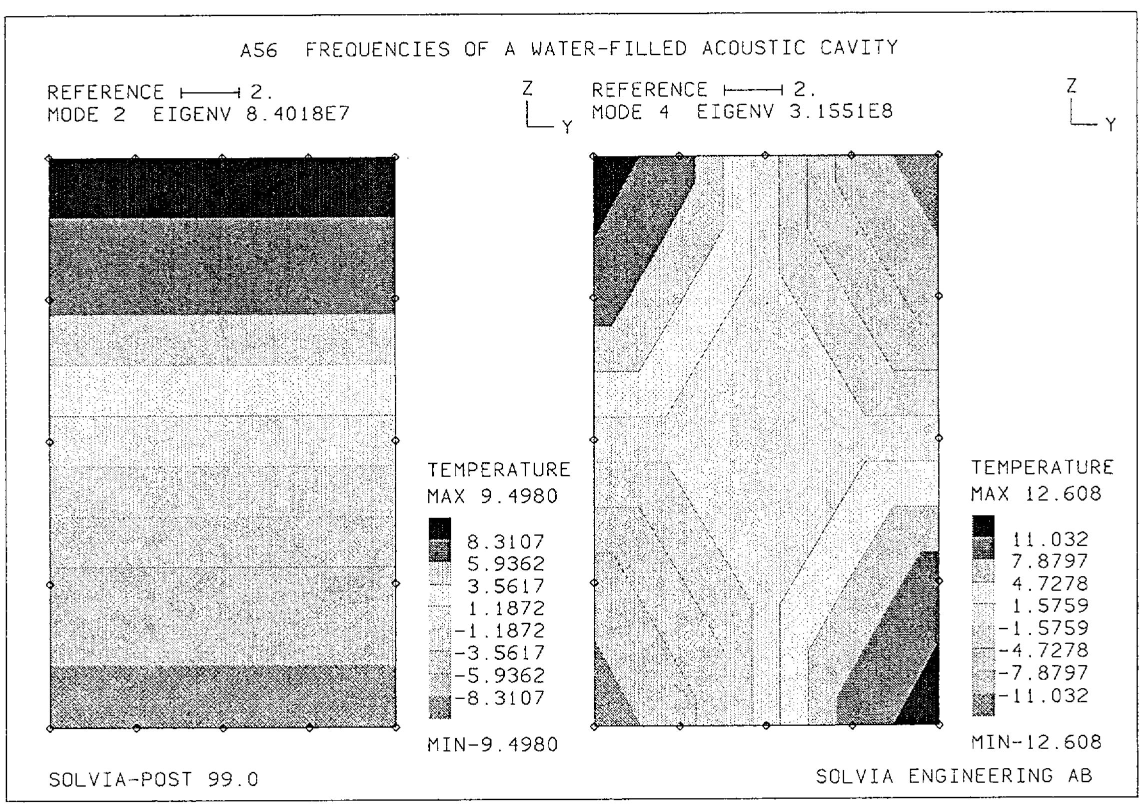
  To obtain the circular eigenfrequencies ω_i of the wave equation, we must, therefore, take the square root of the SOLVIA-TEMP eigenvalues.

- Since only the gradient of the potential is zero at the boundaries while the potential is free, there is one zero frequency (rigid body) mode in this problem.

Reference

[1] Blevins, R.D., Formulas for Natural Frequency and Mode Shape, Van Nostrand Reinhold, 1979, pp. 337-341.


A56 FREQUENCIES OF A WATER-FILLED ACOUSTIC CAVITY
ORIGINAL 2.
Z
Y
ORIGINAL 2.
Z
Y
4
3
1
2
3
4
1
2
t
r
s
EAXES=RST
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A56 FREQUENCIES OF A WATER-FILLED ACOUSTIC CAVITY
REFERENCE 2.
MODE 2 EIGENV 8.4018E7
Z
Y
REFERENCE 2.
MODE 4 EIGENV 3.1551E8
Z
Y
TEMPERATURE
MAX 9.4980
8.3107
5.9362
3.5617
1.1872
-1.1872
-3.5617
-5.9362
-8.3107
MIN-9.4980
TEMPERATURE
MAX 12.608
11.032
7.8797
4.7278
1.5759
-1.5759
-4.7278
-7.8797
-11.032
MIN-12.608
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A56  FREQUENCIES OF A WATER-FILLED ACOUSTIC CAVITY'
*
DATABASE  CREATE
*
T-ANALYSIS  HEATMATRIX=CONSISTENT
T-FREQUENCIES  NEIG=4 IRBM=1
COORDINATES
 ENTRIES NODE  Y   Z
           1    0.  0.
           2   12.  0.
           3   12. 20.
           4    0. 20.
*
T-MATERIAL  1  CONDUCTION  K=3.16E5 SPECIFICHEAT=9.35E-5
*
EGROUP  1  PLANE STRAIN
GSURFACE  1 2 3 4  EL1=2 EL2=2 NODES=8
*
SET  NSYMBOLS=MYNODES
MESH  NNUMBERS=MYNODES EAXES=RST SUBFRAME=21
MESH  ENUMBERS=YES GSCALE=OLD
*
SOLVIA-TEMP
END
```

SOLVIA-POST input

```
*  A56  FREQUENCIES OF A WATER-FILLED ACOUSTIC CAVITY
*
T-DATABASE  CREATE
*
WRITE  FILENAME='a56.lis'
*
FREQUENCIES
*
SET  RESPONSETYPE=VIBRATIONMODE OUTLINE=YES NSYMBOLS=YES
*
MESH  CONTOUR=TEMPERATURE TIME=2 SUBFRAME=21
MESH  CONTOUR=TEMPERATURE TIME=4
END
```