ATTACHMENT 5 TO AEP:NRC:2520

SOLVIA ENGINEERING REPORT SE 99-4,
"SOLVIA VERIFICATION MANUAL LINEAR EXAMPLES"

SOLVIA® Finite Element System
Version 99.0

SOLVIA Verification Manual
Linear Examples

Report SE 99-4

SOLVIA Engineering AB

SOLVIA Verification Manual, Linear Examples.

SOLVIA Engineering AB
Trefasgatan 3
SE-721 30 Västerås
Sweden

Tel +46-21-144050
Fax +46-21-188890
engineering@solvia.se
www.solvia.com

INTRODUCTORY REMARKS

The objective with this report is to present example solutions obtained with the SOLVIA-PRE, SOLVIA and SOLVIA-POST computer programs (the SOLVIA System) that verify and demonstrate their usage. Solutions to linear analyses are presented in this report. Nonlinear example solutions are presented in the companion report SE 99-5.

Since the aim is to compare the analysis results with analytical solutions, relatively small problems are solved, that also allow insight into the results. The analyses reported upon can be directly rerun with version 99.0 of the SOLVIA System. Complete input data for SOLVIA-PRE and SOLVIA-POST is given for each example. All plot pictures created by the input data have been output to Microsoft Word in the PostScript graphical language.

We intend to update this report with further example solutions as we continue our work on the SOLVIA System. If you have any suggestions regarding the example solutions presented in this manual or suggestions on additional problems, we would be glad to hear from you.

CONTENTS

EXAMPLE A1

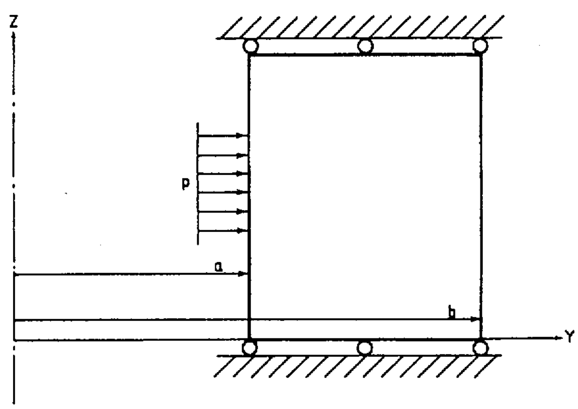
THICK CYLINDER UNDER INTERNAL PRESSURE

Objective

To verify the stress and displacement variation of the PLANE AXISYMMETRIC element and the application of axisymmetric pressure loading.

Physical Problem

The figure below shows the cylinder to be analyzed. The cylinder is assumed to be guided so that no axial displacement can occur and it is acted upon by internal pressure.



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

$\nu = 0.3$

$p = 10.0 \cdot 10^{6}\ N/m^2$

$a = 0.5\ m$

$b = 1.0\ m$

Finite Element Model

The top figure on page A1.3 shows the finite element model. It consists of 8 PLANE AXISYMMETRIC elements having parabolic displacement variation. The element pressure is applied along the generated line between nodes 1 and 4.

Solution Results

The theoretical stress solution is given in [1] p. 60:

$$\sigma_y = \frac{a^2 p}{b^2 - a^2}\left(1 - \frac{b^2}{y^2}\right)$$

$$\sigma_x = \frac{a^2 p}{b^2 - a^2}\left(1 + \frac{b^2}{y^2}\right)$$

$$\sigma_z = \nu\left(\sigma_x + \sigma_y\right)$$

The displacement in the radial direction can be calculated from the circumferential strain:

$$u_y = y \cdot \varepsilon_x = \frac{y}{E}\left(\sigma_x - \nu\left(\sigma_y + \sigma_z\right)\right)$$

or

$$u_y = \frac{a^2 p\left(y^2 + b^2 - 2y^2 \nu\right)(1+\nu)}{Ey\left(b^2 - a^2\right)}$$

The SOLVIA numerical solution obtained using the input data on pages A1.4 and A1.5 is as follows:

Stresses in 10^6 N/m^2 at node 1 and node 2:

y-coord.	σ_x		σ_y		σ_z	
(m)	SOLVIA	Theory	SOLVIA	Theory	SOLVIA	Theory
0.5	16.73	16.67	-9.85	-10.00	2.066	2.000
1.0	6.672	6.667	0.012	0.	2.005	2.000

Radial displacements:

y-coord. (m)	SOLVIA (m)	Theory (m)
0.5	$4.7667 \cdot 10^{-5}$	$4.7667 \cdot 10^{-5}$
1.0	$3.0333 \cdot 10^{-5}$	$3.0333 \cdot 10^{-5}$

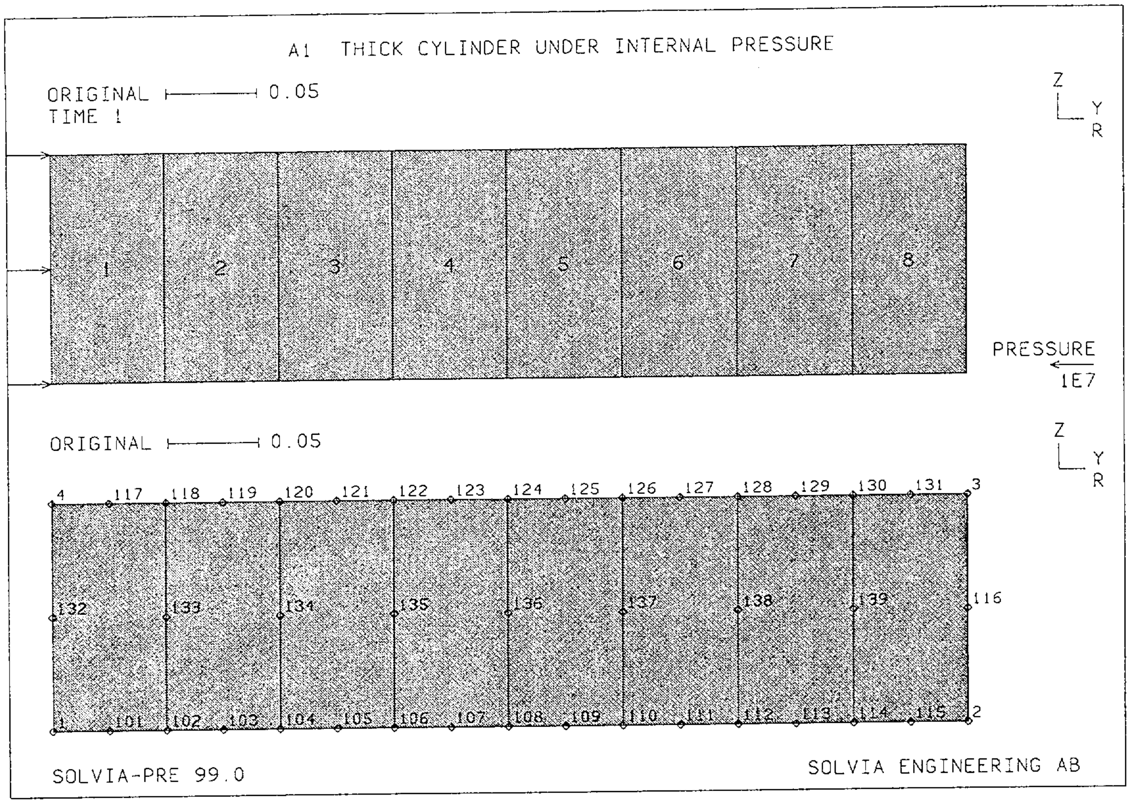
The plots from SOLVIA-POST on pages A1.3 and A1.4 show the variation of radial displacement, circumferential stress, radial stress and axial stress along a line in the radial direction.

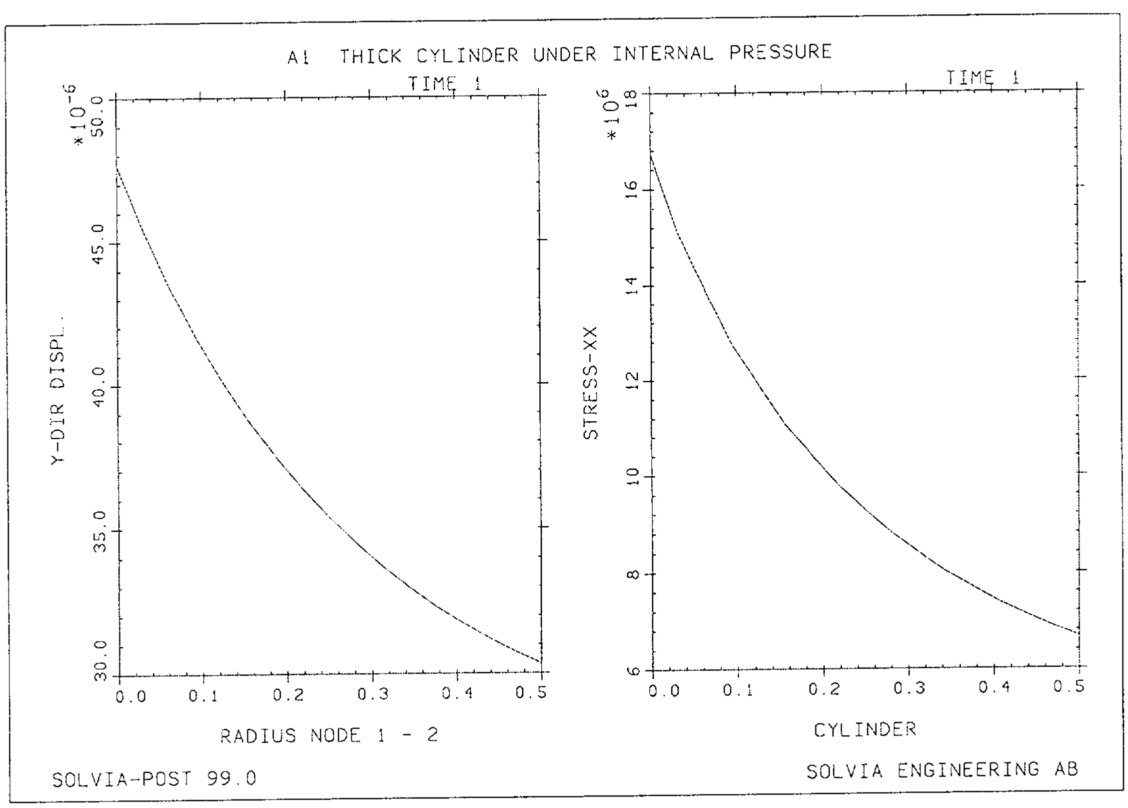
User Hints

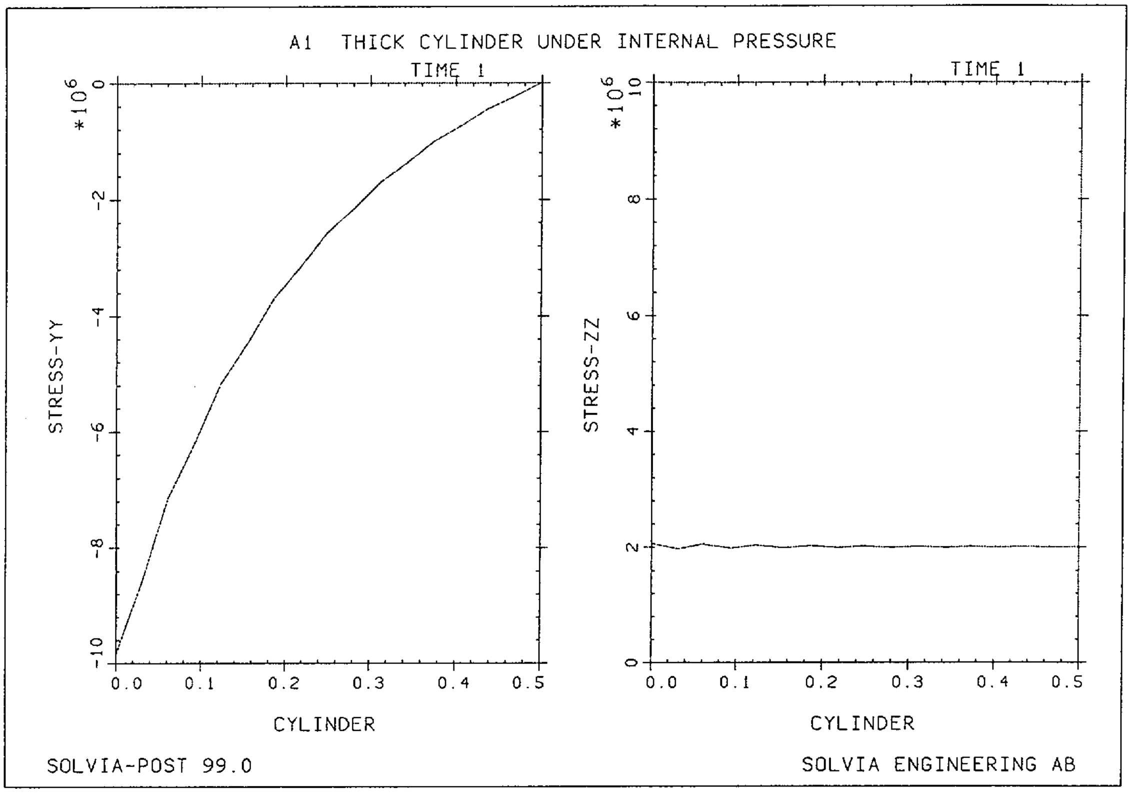
- Symmetry gives that no variation of stresses can occur in the axial (Z) direction. A linear displacement assumption in the axial direction would therefore be sufficient and the higher order displacement assumption in the axial direction can not improve the results.
- Note that MYNODES is by default set to 100. The generated node numbers start with 101 as seen in the figure on page A1.3.

Reference

[1] Timoshenko, S., Goodier, J.N., Theory of Elasticity, Second Edition, McGraw-Hill, 1951.


A1 THICK CYLINDER UNDER INTERNAL PRESSURE
ORIGINAL 0.05
TIME 1
Z
Y
R
1
2
3
4
5
6
7
8
PRESSURE
1E7
ORIGINAL 0.05
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A1 THICK CYLINDER UNDER INTERNAL PRESSURE
TIME 1
*10^-6
Y-DIR DISPL.
RADIUS NODE 1 - 2
*10^6
STRESS-XX
CYLINDER
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB



SOLVIA-PRE input

```
HEAD  'A1  THICK CYLINDER UNDER INTERNAL PRESSURE'
*
DATABASE  CREATE
MASTER  IDOF=101111
*
COORDINATES
 ENTRIES NODE   Y    Z
           1   0.5
           2   1.
           3   1.   0.125
           4   0.5  0.125
*
MATERIAL  1  ELASTIC E=2.E11 NU=0.3
*
EGROUP  1  PLANE AXISYMMETRIC RESULTS=NSTRESSES
GSURFACE  1 2 3 4   EL1=8 EL2=1 NODES=8
LOADS ELEMENT  INPUT=LINE
 1 4  10.E6
*
SUBFRAME  12
MESH  ENUMBER=YES VECTOR=LOAD
MESH  NSYMBOL=YES NNUMBER=YES GSCALE=OLD
*
SOLVIA
END
```

SOLVIA-POST input

```
*   A1   THICK CYLINDER UNDER INTERNAL PRESSURE
*
DATABASE  CREATE
*
WRITE  FILENAME='a1.lis'
*
EPLINE  NAME=CYLINDER
 1  1 5 2  TO  8  1 5 2
NPLINE  NAME=RADIUS
 1  101 TO 115  2
AXIS  ID=1 VMIN=0 VMAX=100.E5 LABEL='STRESS-ZZ'
*
NLINE  LINENAME=RADIUS DIRECTION=2 OUTPUT=ALL SUBFRAME=21
ELINE  LINENAME=CYLINDER KIND=SXX OUTPUT=ALL
ELINE  LINENAME=CYLINDER KIND=SYY OUTPUT=ALL SUBFRAME=21
ELINE  LINENAME=CYLINDER KIND=SZZ YAXIS=1 OUTPUT=ALL
END
```

EXAMPLE A2

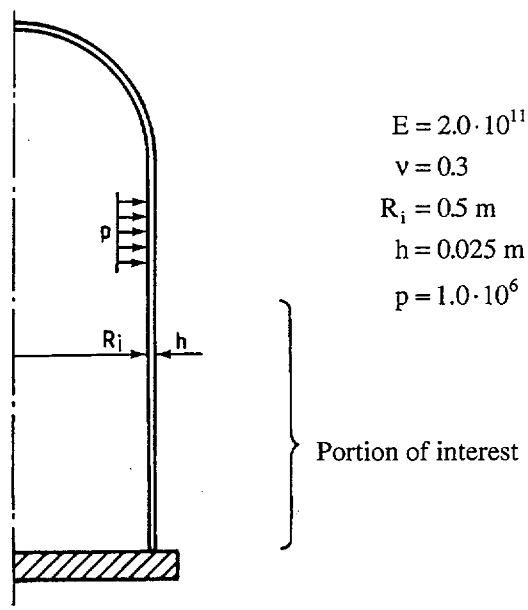
AXISYMMETRIC SHELL UNDER INTERNAL PRESSURE

Objective

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

Physical Problem

The figure below shows the cylinder to be analyzed. The radius to thickness ratio is 20 so thin shell theory is applicable. The cylinder is fixed at one end and it is loaded by internal pressure. The cylinder is long so that the effects of bending at one end do not affect the other end.



$$E = 2.0 \cdot 10^{11}\ \text{N/m}^2$$
$$\nu = 0.3$$
$$R_i = 0.5\ \text{m}$$
$$h = 0.025\ \text{m}$$
$$p = 1.0 \cdot 10^{6}\ \text{N/m}^2$$

Finite Element Model

The figure on page A2.4 shows the finite element model. The 8-node PLANE AXISYMMETRIC element with 2×2 integration is used to describe the bending behaviour. Since most of the bending is concentrated towards the end a finer mesh is used for that part.

Solution Results

The behaviour of the cylindrical shell under edge loading and internal pressure is described for example in [1] p. 140.

Using

$$n^4 = \frac{3(1-\nu)^2}{h^2 a^2}$$

$$D = \frac{E h^3}{12(1-\nu^2)}$$

$$C = \frac{a}{a - h/2}$$

$$M_o = \frac{p_a}{2n^2}\left(C - \frac{\nu}{2}\right)$$

$$T_o = -\frac{p_a}{n}\left(C - \frac{\nu}{2}\right)$$

$$p_a = \frac{p_i}{C^2}$$

where

- E = Young's modulus
- ν = Poisson's ratio
- a = mean radius
- h = thickness
- p_i = internal pressure applied at radius a - h/2. Note that the factor C is introduced to calculate equivalent loads acting at the midsurface.

we obtain the radial displacement as

$$w = \frac{p_a}{4n^4 D}\left(C - \frac{\nu}{2}\right) - e^{-nx}(\cos nx + \sin nx)$$

and the axial bending moment

$$M = \left(\left(\frac{T_o}{n} + M_o\right)\sin nx + M_o \cos nx\right)e^{-nx}$$

The stresses in the axial (z) and circumferential (x) directions are

$$\sigma_{zz} = \frac{p_a a}{2h} + \frac{6Mr}{h^3}$$

$$\sigma_{xx} = \frac{wE}{a} + \nu\sigma_{xx}$$

where r is the distance from the mean radius of the cylinder.

Using the input data on pages A2.6 and A2.7 we obtain the following results from SOLVIA:

	Radial displacement $(10^{-6}\,m)$	
z-coord. (m)	SOLVIA	Theory
0.025	3.76	2.91
0.050	10.74	9.51
0.075	18.73	17.41
0.100	26.32	25.08
0.200	43.72	43.21
0.300	45.81	45.53
0.400	44.61	44.29
0.500	44.07	43.71
0.900	44.11	43.75

The SOLVIA results are taken from the nodes on the mean surface of the cylinder. The displacements are also shown by the deformed mesh on page A2.5.

	Stresses at inner Gauss points (MN/m^2)			
z-coord.	σ_{xx}		σ_{zz}	
	SOLVIA	Theory	SOLVIA	Theory
0.0026	7.16	7.99	26.07	26.59
0.0099	7.26	7.36	23.39	23.87
0.0151	7.22	7.06	21.59	22.02
0.0223	6.92	6.84	19.31	19.71
0.0526	8.08	7.73	12.15	12.35
0.1151	13.81	13.32	6.38	6.34
0.2151	19.86	19.52	7.63	7.57
0.9000	20.29	20.00	9.76	9.76

The stress results for the inner Gauss points are also plotted on page A2.5.

User Hints

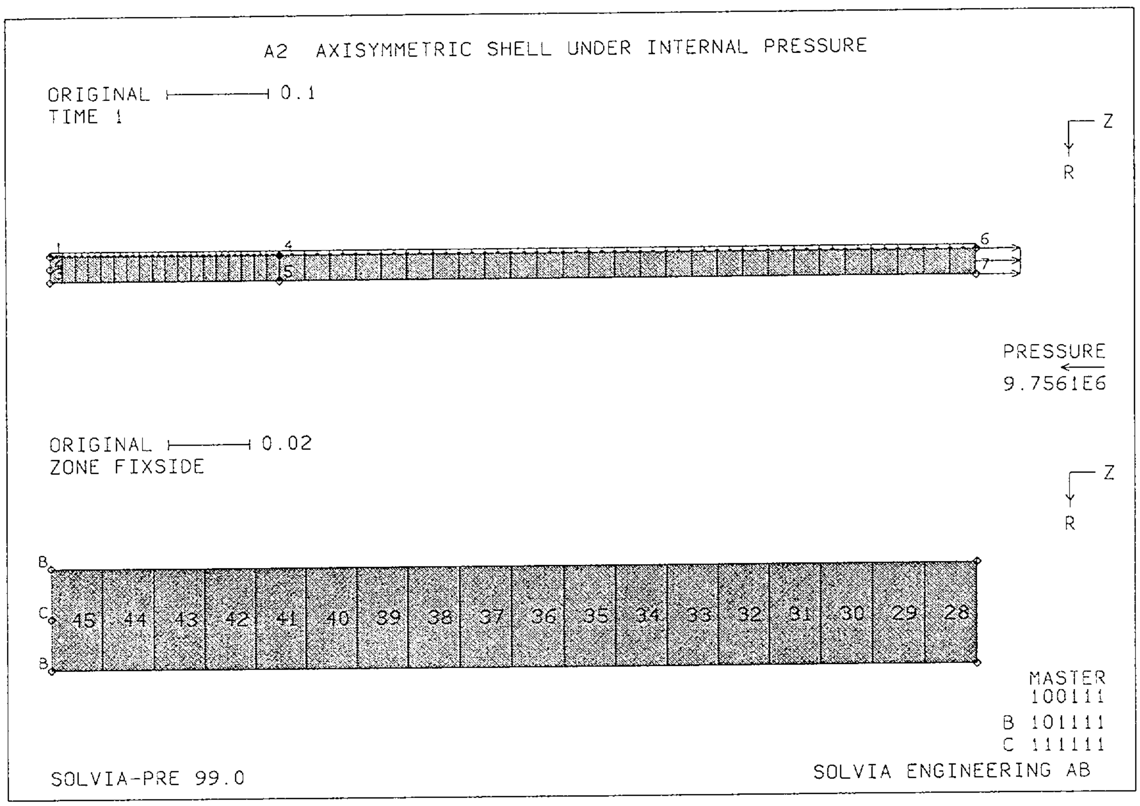
- The 8-node PLANE element describes the bending behaviour quite good. The 4-node element, which has a linear assumption on the displacement behaviour in the radial and the axial direction, would not be satisfactory.

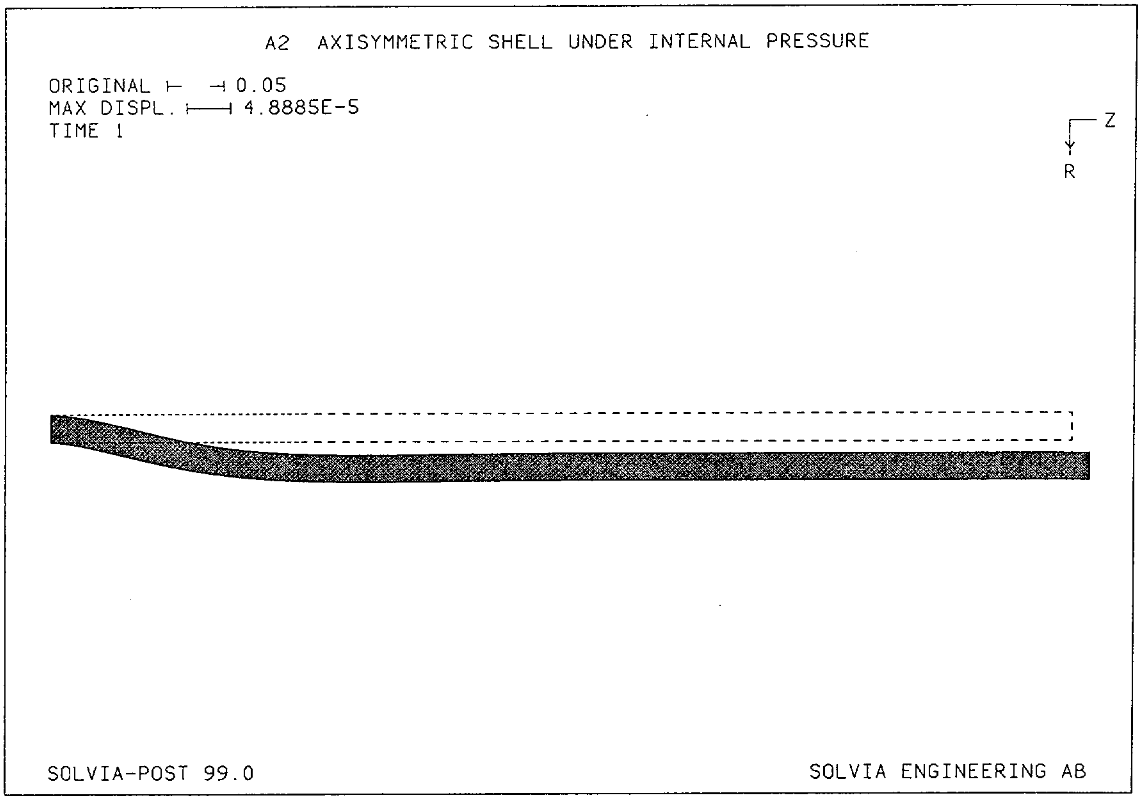
- The portion of most interest in edge bending of an axisymmetric cylindrical shell extends approximately the length $2 \cdot \sqrt{ah}$ in the axial direction, where a is the mean radius and h is the thickness. This portion should, therefore, be modeled with a finer mesh.

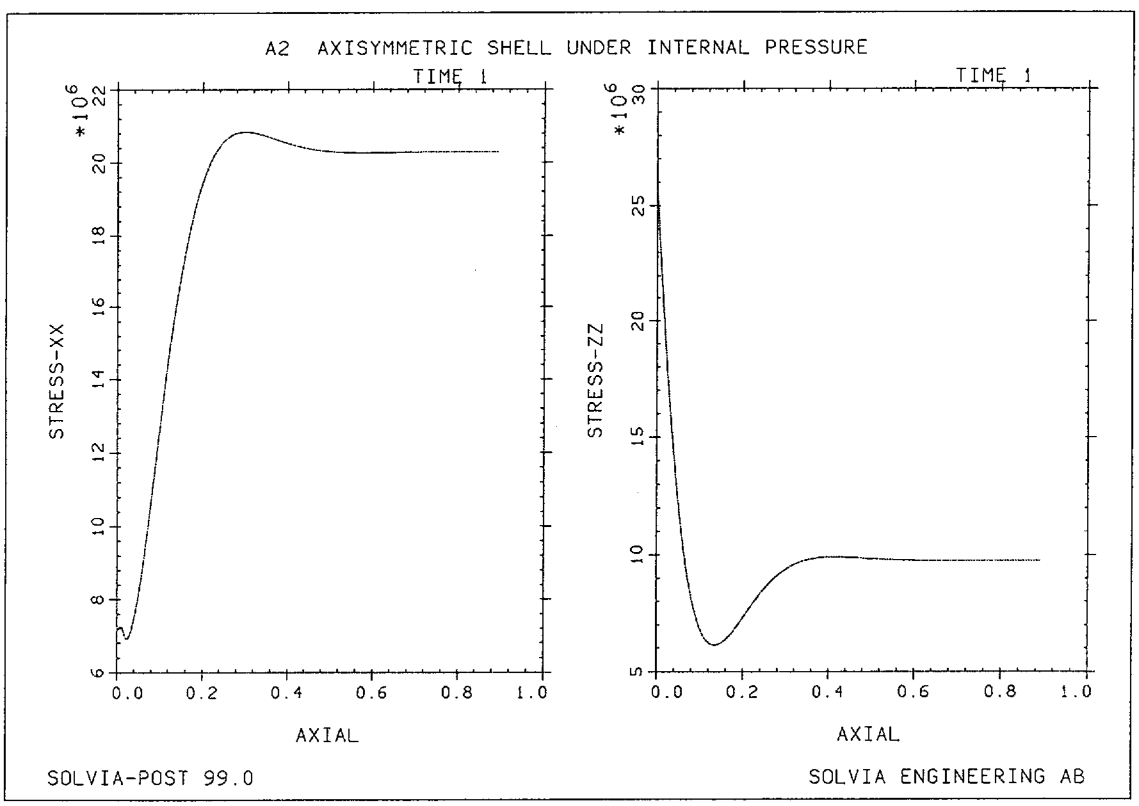
- In thin shell theory the pressure loading is applied at the midsurface of the shell. For PLANE elements, however, the pressure loading is applied at the element boundaries. A correction factor C is, therefore, introduced in the formulas obtained using thin shell theory in order to calculate equivalent loads acting at the midsurface.

Reference

[1] Kraus, H., Thin Elastic Shells, John Wiley & Sons, 1967




A2 AXISYMMETRIC SHELL UNDER INTERNAL PRESSURE
ORIGINAL 0.05
MAX DISPL. 4.8885E-5
TIME 1
Z
R
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A2 AXISYMMETRIC SHELL UNDER INTERNAL PRESSURE
TIME 1
*10^6
STRESS-XX
AXIAL
TIME 1
*10^6
STRESS-ZZ
AXIAL
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A2  AXISYMMETRIC SHELL UNDER INTERNAL PRESSURE'
*
DATABASE  CREATE
*
MASTER  IDOF=100111
COORDINATES
 ENTRIES  NODE   Y      Z
            1   0.5          TO
            3   0.525
            4   0.5   0.225
            5   0.525 0.225
            6   0.5   0.9
            7   0.525 0.9
*
MATERIAL  1  ELASTIC E=2.E11 NU=0.3
*
EGROUP  1  PLANE AXISYMMETRIC INT=2
GSURFACE  6 4 5 7  EL1=27 EL2=1 NODES=8
GSURAFCE  4 1 3 5  EL1=18 EL2=1 NODES=8
*
LOADS ELEMENT  INPUT=LINE
 1 4  1.E6
 4 6  1.E6
 6 7 -9.7561E6
*
FIXBOUNDARIES   3  /  1 3
FIXBOUNDARIES  23  /  2
*
VIEW  ID=1 XVIEW=1 ROTATION=-90
SET  NSYMBOLS=MYNODES VIEW=1
MESH  NNUMBERS=MYNODES VECTOR=LOAD SUBFRAME=12
ZONE  NAME=FIXSIDE INPUT=GLOBAL-LIMITS ZMAX=0.230
MESH  ZONENAME=FIXSIDE ENUMBER=YES BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*   A2   AXISYMMETRIC SHELL UNDER INTERNAL PRESSURE
*
DATABASE  CREATE
*
WRITE  FILENAME='a2.lis'
*
VIEW  ID=1 XVIEW=1 ROTATION=-90
SET  ORIGINAL=DASHED DEFORMED=YES VIEW=1
MESH  OUTLINE=YES DMAX=1
*
EPLINE  NAME=AXIAL
 45  2 4  TO  1  2 4
*
ELINE  LINENAME=AXIAL KIND=SXX OUTPUT=ALL SUBFRAME=21
ELINE  LINENAME=AXIAL KIND=SZZ OUTPUT=ALL
*
NLIST  KIND=DISPLACEMENT
NLIST  KIND=REACTION
END
```

EXAMPLE A3

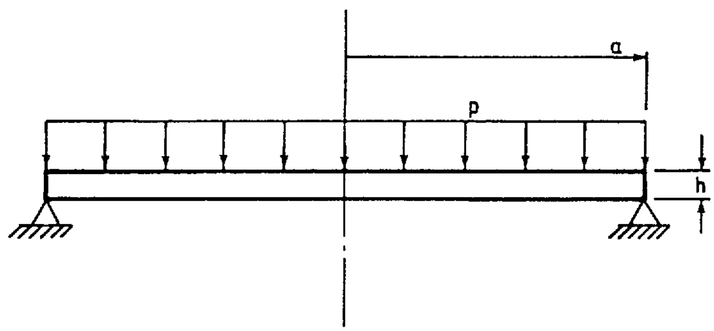
SIMPLY SUPPORTED CIRCULAR PLATE UNDER PRESSURE LOAD

Objective

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

Physical Problem

The figure below shows the circular plate to be analyzed. It is a thin plate since the diameter to thickness ratio is 20. The boundary of the plate is simply supported.



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

$\nu = 0.3$

$a = 0.5$ m

$h = 0.05$ m

$p = 1.0 \cdot 10^6$ N/m^2

Finite Element Model

The top figure on page A3.3 shows the finite element model. Ten 8-node PLANE AXISYMMETRIC elements are used.

Solution Results

The theoretical solution is given for example in [1] p. 56 as follows:

Center deflection:

$$w_{max} = \frac{(5+\nu)pa^4}{64(1+\nu)D} = 0.1739 \cdot 10^{-2} \text{ m}$$

$$D = \frac{Eh^3}{12(1-\nu^2)}$$

Moment:

$$M_y = \frac{p}{16}(3+\nu)(a^2 - y^2)$$

$$M_x = \frac{p}{16}\left(a^2(3+\nu) - y^2(1+3\nu)\right)$$

Stresses at x=0 and y=0:

$$\sigma_{yy} = \frac{6M_y}{h^2} = 123.75 \cdot 10^6 \text{ Nm}^2$$
$$\sigma_{xx} = \frac{6M_x}{h^2} = 123.75 \cdot 10^6 \text{ Nm}^2$$

Using the input data on page A3.5 the following results are obtained:

Center deflection	
SOLVIA	Theory
$-0.1752 \cdot 10^{-2}$	$-0.1739 \cdot 10^{-2}$

Stresses of element 10 at node 1 (at y = z = 0):

σ_{xx} (N/m^2)		σ_{yy} (N/m^2)	
SOLVIA	Theory	SOLVIA	Theory
124.02	123.75	124.02	123.75

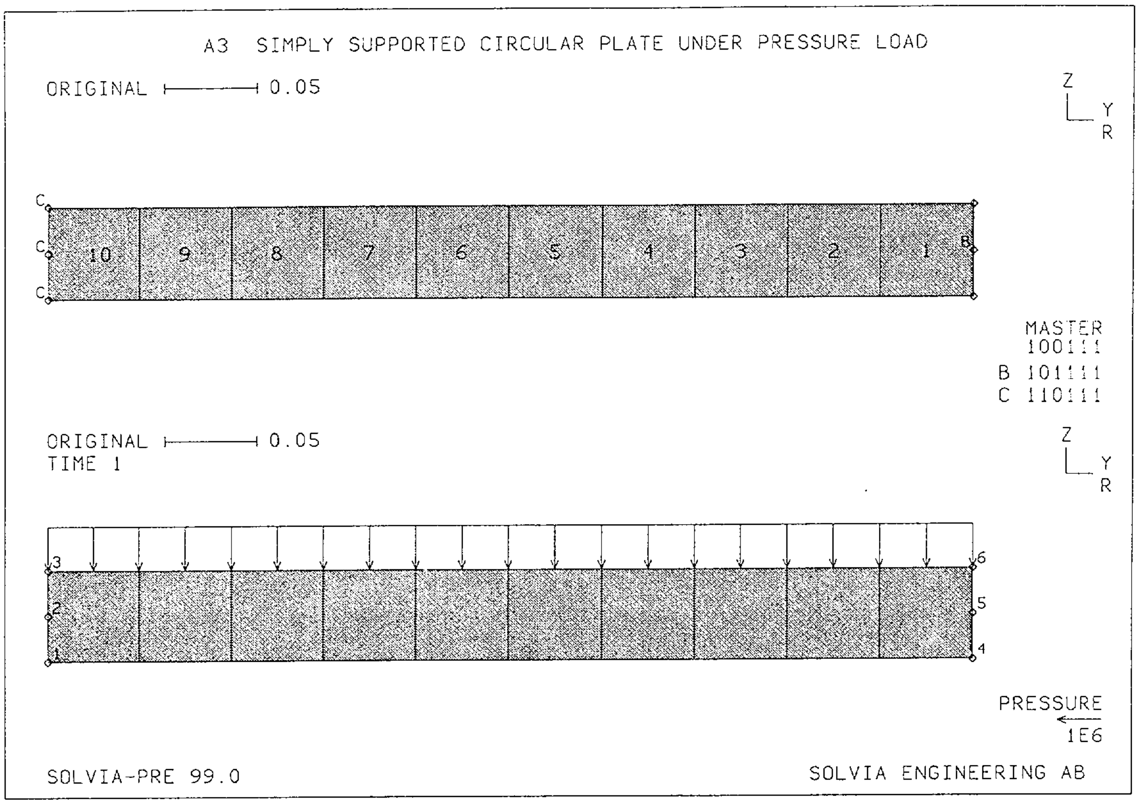
The distribution of the von Mises effective stress and the stresses σ_{xx} and σ_{yy} along the lower surface of the plate as obtained from SOLVIA are shown in the bottom figure on page A3.3 and the top figure on page A3.4.

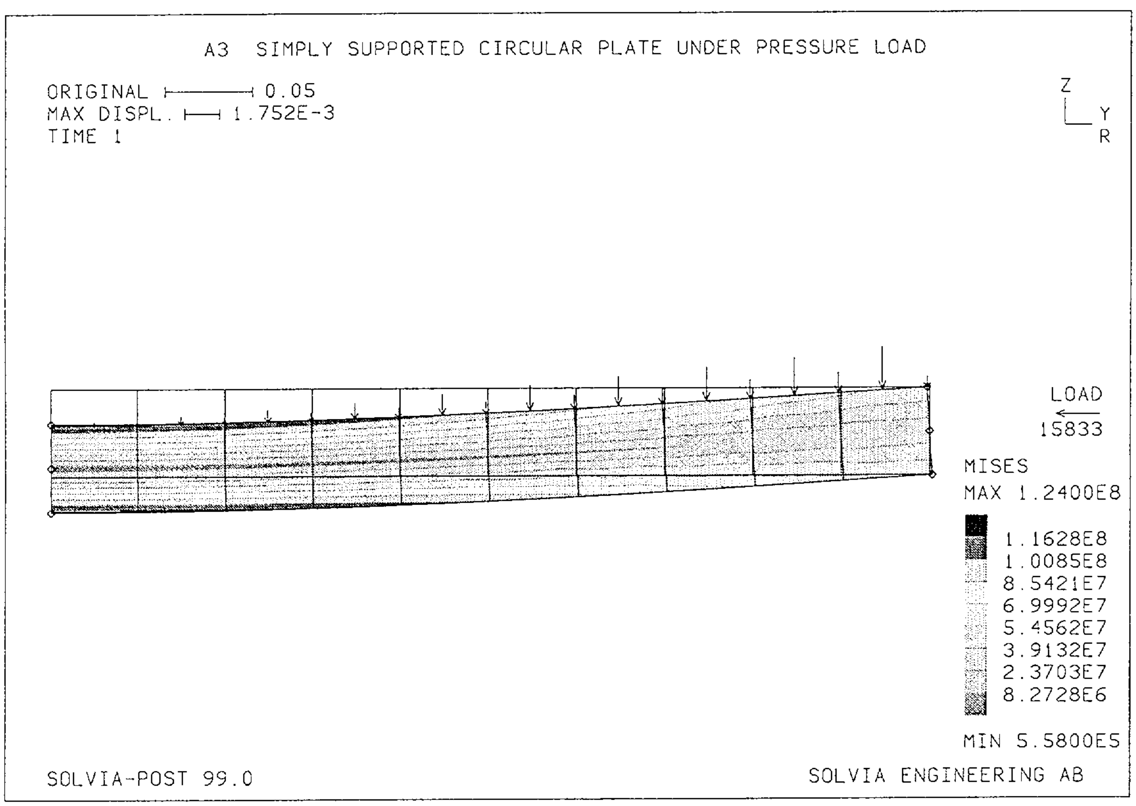
User Hints

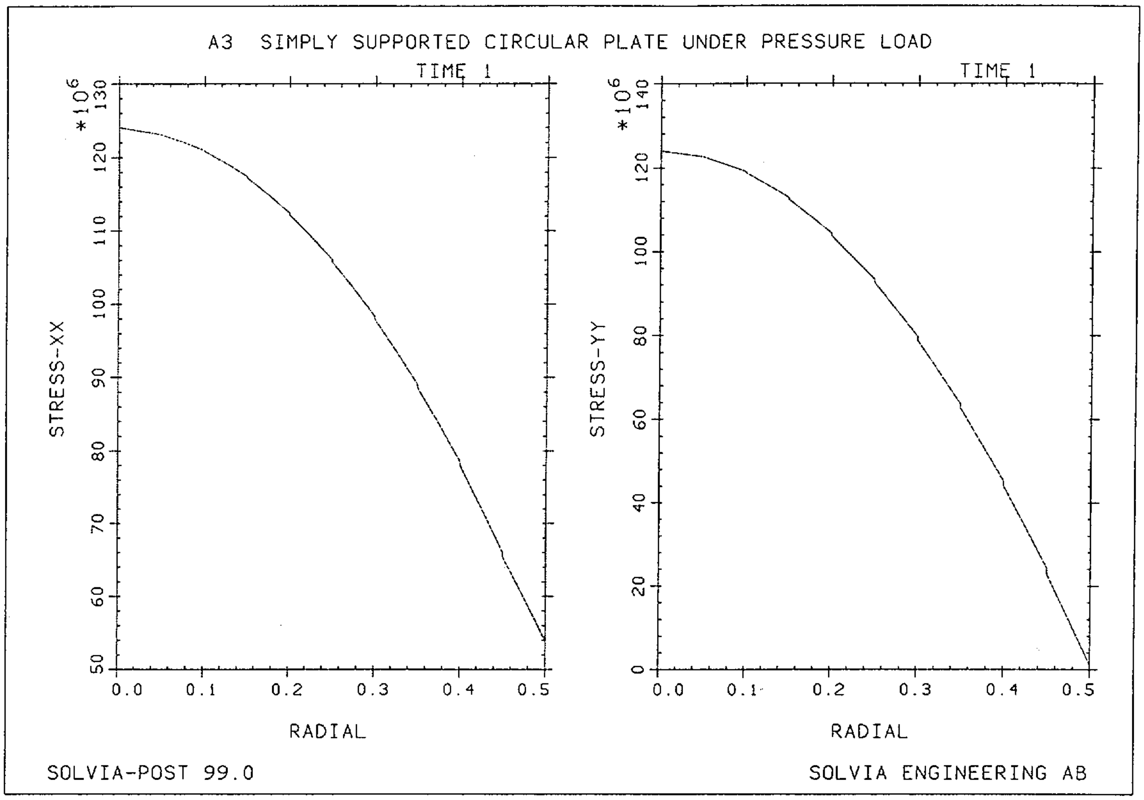
- The 8-node PLANE element should be used in order to model the bending behaviour.
- Note that the PLANE AXISYMMETRIC element extends 1 radian in the circumferential direction.

Reference

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


A3 SIMPLY SUPPORTED CIRCULAR PLATE UNDER PRESSURE LOAD
ORIGINAL 0.05
Z
Y
R
C
C
C
10
9
8
7
6
5
4
3
2
1
B
MASTER
100111
B 101111
C 110111
ORIGINAL 0.05
TIME 1
Z
Y
R
3
2
1
6
5
4
PRESSURE
1E6
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A3 SIMPLY SUPPORTED CIRCULAR PLATE UNDER PRESSURE LOAD
ORIGINAL 0.05
MAX DISPL. 1.752E-3
TIME 1
Z
Y
R
LOAD
15833
MISES
MAX 1.2400E8
1.1628E8
1.0085E8
8.5421E7
6.9992E7
5.4562E7
3.9132E7
2.3703E7
8.2728E6
MIN 5.5800E5
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A3 SIMPLY SUPPORTED CIRCULAR PLATE UNDER PRESSURE LOAD
TIME 1
STRESS-XX
*10^6
130
120
110
100
90
80
70
60
50
0.0
0.1
0.2
0.3
0.4
0.5
RADIAL
TIME 1
STRESS-YY
*10^6
140
120
100
80
60
40
20
0
RADIAL
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A3  SIMPLY SUPPORTED CIRCULAR PLATE UNDER PRESSURE LOAD'
*
DATABASE  CREATE
MASTER  IDOF=100111
*
COORDINATES
 ENTRIES  NODE    Y    Z
            1               TO
            3    0.0  0.05
            4    0.5         TO
            6    0.5  0.05
*
MATERIAL  1  ELASTIC E=2.E11 NU=0.3
*
EGROUP  1  PLANE AXISYMMETRIC RESULTS=NSTRESSES
GSURFACE  6 3 1 4  EL1=10 EL2=1 NODES=8
LOADS ELEMENT  INPUT=LINE
 3 6  1.E6
*
FIXBOUNDARIES  2  /  1 2 3
FIXBOUNDARIES  3  /  5
*
SET  NSYMBOLS=MYNODES
SUBFRAME  12
MESH  ENUMBER=YES BCODE=ALL
MESH  NNUMBER=MYNODES VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A3  SIMPLY SUPPORTED CIRCULAR PLATE UNDER PRESSURE LOAD
*
DATABASE  CREATE
WRITE  FILENAME='a3.lis'
*
EPLINE  NAME=RADIAL
 10 3 7 4  TO  1 3 7 4
*
SET  NSYMBOLS=MYNODES
MESH  ORIGINAL=YES CONTOUR=MISES VECTOR=LOAD
*
SUBFRAME  21
ELINE  LINENAME=RADIAL KIND=SXX OUTPUT=ALL
ELINE  LINENAME=RADIAL KIND=SYY OUTPUT=ALL
*
NMAX  KIND=DISPLACEMENT NUMBER=3
SUMMATION  KIND=REACTION DETAILS=YES
END
```

EXAMPLE A4

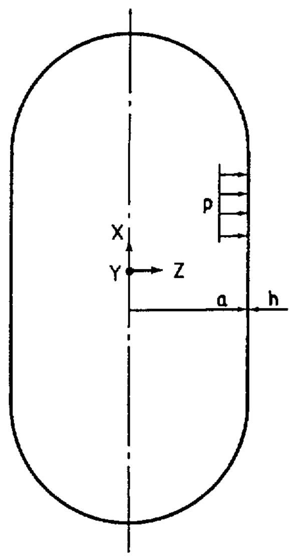
CIRCULAR CYLINDRICAL SHELL UNDER PRESSURE LOAD

Objective

To verify the membrane behaviour of a curved thin SHELL element under distributed loading.

Physical Problem

The pressure vessel shown in the figure below is loaded by internal pressure. A section of the shell subjected only to membrane action is to be analyzed.



$E = 6.625 \cdot 10^{10}\ N/m^2$

$\nu = 0.33$

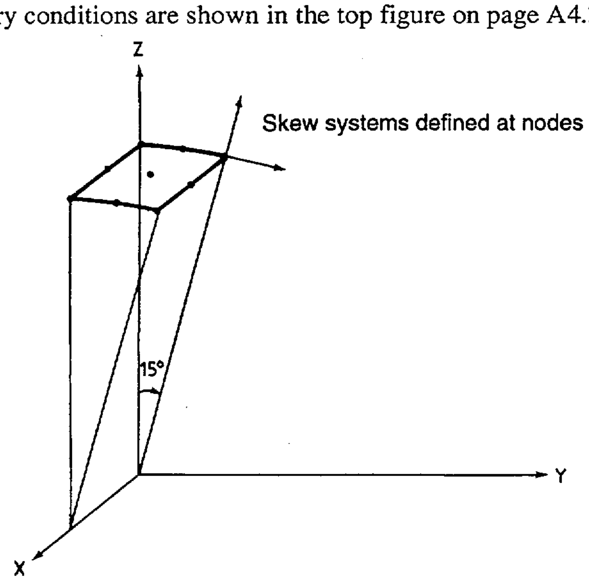
$a = 2.0000\ m$

$h = 0.0010\ m$

$p = 0.1 \cdot 10^5\ N/m^2$

Finite Element Model

The figure below shows the finite element model. We use one 9-node SHELL element extending 15 degrees and skew systems oriented in the radial and circumferential directions for ease of applying the boundary conditions. The axial loading is simulated by SHELL edge loading. The numbering of nodes and the boundary conditions are shown in the top figure on page A4.3.



Solution Results

The radial deformation is

$$w = \frac{pa^2}{Eh}\left(1 - \frac{\nu}{2}\right)$$

and the stresses are:

$$\sigma_{rad} = \frac{pa}{2h}$$

$$\sigma_{hoop} = \frac{pa}{h}$$

The SOLVIA numerical results obtained using the input data shown on page A4.4 are as follows:

Displacement		Stresses			
$w(10^{-3}m)$		$\sigma_{rad}(10^6 N/m^2)$		$\sigma_{hoop}(10^6 N/m^2)$	
SOLVIA	Theory	SOLVIA	Theory	SOLVIA	Theory
0.5041	0.5041	10.0	10.0	20.0	20.0

The deformed mesh is shown in the bottom figure on page A4.3.

User Hints

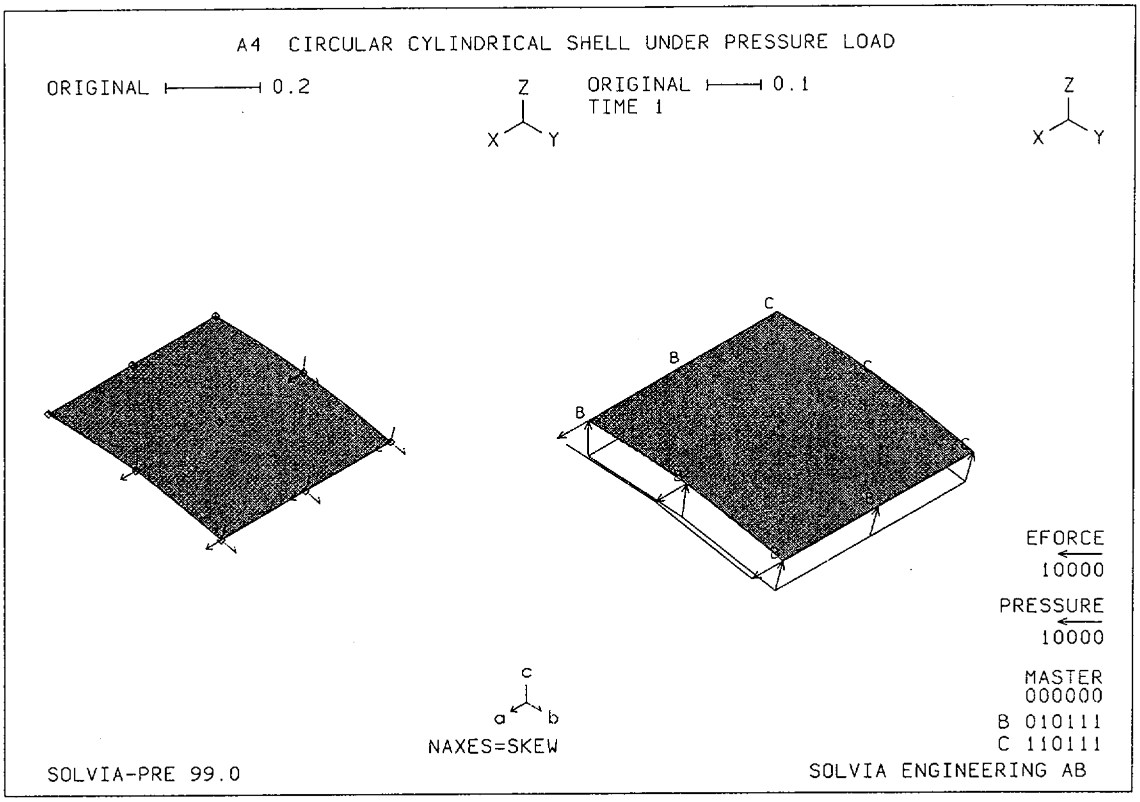
- The quadratic SHELL element is chosen in order to approximate the circular shell. This approximation leads to some very small variation in the displacement and stress results since the curvature of the element varies slightly over the element.

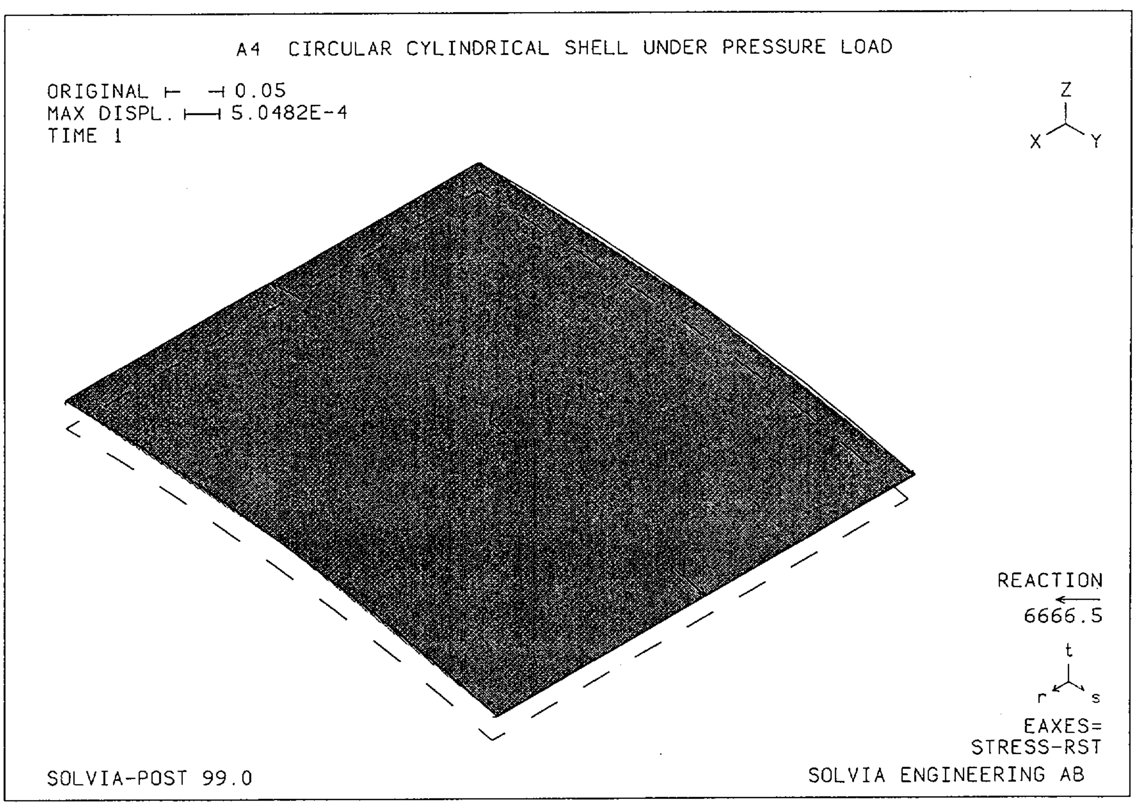
- Note that SOLVIA-PRE automatically assigns GLOBAL rotations for a SHELL midsurface node when the node has a boundary condition for rotation specified [1].

- If the 9-node SHELL element shall model bending then three or more elements are recommended for the 15 degree portion, thus at least 5 degrees per element.

Reference

[1] SOLVIA-PRE 99.0 Users Manual, Stress Analysis, Report SE 99-1, p. 7.23 - 7.34.


A4 CIRCULAR CYLINDRICAL SHELL UNDER PRESSURE LOAD
ORIGINAL 0.2
Z
X Y
ORIGINAL 0.1
TIME 1
Z
X Y
C
B
B
C
EFORCE
10000
PRESSURE
10000
MASTER
000000
B 010111
C 110111
c
a b
NAXES=SKEW
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A4 CIRCULAR CYLINDRICAL SHELL UNDER PRESSURE LOAD
ORIGINAL 0.05
MAX DISPL. 5.0482E-4
TIME 1
Z
X Y
REACTION
6666.5
t
r s
EAXES=
STRESS-RST
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A4  CIRCULAR CYLINDRICAL SHELL UNDER PRESSURE LOAD'
*
DATABASE  CREATE
*
SYSTEM  1 CYLINDRICAL
COORDINATES
 ENTRIES NODE    R  THETA  XL
 1   2. 90. 0.    TO  3   2. 75. 0.
 4   2. 90. 0.5   TO  6   2. 75. 0.5
SKEWSYSTEM  EULERANGLES
 1  -7.5
 2 -15.
*
MATERIAL  1  ELASTIC E=6.625E10 NU=0.33
*
EGROUP  1  SHELL STRESSREFERENCE=ELEMENT
THICKNESS  1 1.E-3
GSURFACE  3 1 4 6  EL1=1 EL2=1 NODES=9 SYSTEM=1
LOADS ELEMENT  TYPE=PRESSURE INPUT=ELEMENTS
 1 -T 1.E4
LOADS ELEMENT  TYPE=FORCE INPUT=LINE
 4 6 OUT 1.E4
*
NSKEWS  INPUT=NODES  /  2 1  /  5 1
NSKEWS  INPUT=LINES  /  3 6  2
*
FIXBOUNDARIES  12456 INPUT=LINE  / 1 3
FIXBOUNDARIES   2456 INPUT=LINE  / 1 4 / 4 6 / 3 6
*
SET  SMOOTHNESS=YES
MESH  NSYMBOLS=YES NAXES=SKEW SUBFRAME=21
MESH  VECTOR=LOAD BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A4  CIRCULAR CYLINDRICAL SHELL UNDER PRESSURE LOAD
*
*
DATABASE  CREATE
WRITE  FILENAME='a4.lis'
*
SET  SMOOTHNESS=YES
MESH  VECTOR=REACTION ORIGINAL=DASHED EAXES=STRESS-RST
*
NLIST  DIRECTION=123
EMAX
END
```

EXAMPLE A5

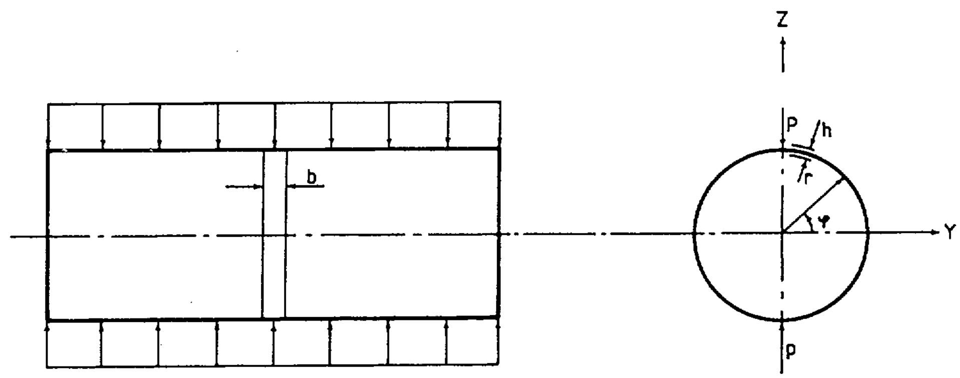
CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, SHELL

Objective

To verify the bending behaviour of the curved thin SHELL element.

Physical Problem

The figure below shows the circular cylindrical shell to be analyzed. It is acted upon by equal and opposite line loads. The solution is sought for the part of the shell which is not disturbed by end effects.



$r = 2.00$ m

$h = 0.02$ m

$b = 0.50$ m

$P = 400\ \mathrm{N/m^2}$

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

$\nu = 0.3$

Finite Element Model

The figure on page A5.3 shows the finite element model used. Symmetry gives that only a 90° portion of the cylinder need be modelled. All 4 boundary faces have symmetry boundary conditions. In addition, the displacements in the X-direction are zero. The 16-node SHELL element with Gauss integration orders 4×4×2 as well as 3×3×2 have been used for comparison.

Solution Results

The theoretical solution is given in [1] p. 381 for a thin ring under equal and opposite forces. A correction to account for the plane strain condition in the axial direction gives then

$$\delta_z = -\tfrac{1}{2}\left(1-\nu^2\right)\left(\frac{\pi}{4}-\frac{2}{\pi}\right)\frac{bPr^3}{EI}$$

$$\delta_y = \tfrac{1}{2}\left(1-\nu^2\right)\left(\frac{2}{\pi}-\frac{1}{2}\right)\frac{bPr^3}{EI}$$

where δ_z is the radial displacement at the application of the load and δ_y is the radial displacement 90° from the load (along the Y-axis).

The following displacement results are obtained using the input data shown on page A5.7:

	Theory	SOLVIA 6 elements 4×4×2 int.	SOLVIA 12 elements 4×4×2 int.	SOLVIA 6 elements 3×3×2 int.
δ_z (mm)	-1.625	-1.607	-1.625	-1.625
δ_y (mm)	1.492	1.480	1.492	1.492

Compare also the bending stress at the outer face for ϕ=0° and ϕ=90°.

The theoretical bending moment is ([1] p. 380):

$$M = \frac{bPr}{2}\left(\cos\phi - \frac{2}{\pi}\right)$$

$$M(\phi = 0) = 72.676[\mathrm{Nm}] \quad (145.352\,\mathrm{Nm/m})$$

$$M(\phi = 90°) = -127.324[\mathrm{Nm}] \quad (-254.648\,\mathrm{Nm/m})$$

The following results are obtained

Bending stress [MPa]				
	Theory	SOLVIA 6 elements 4×4×2 int.	SOLVIA 12 elements 4×4×2 int.	SOLVIA 6 elements 3×3×2 int.
$\phi = 0°$	2.180	2.249	2.182	2.264
$\phi = 90°$	-3.820	-3.915	-3.871	-4.383

The variations of bending stress (STRESS-RR) and axial stress (STRESS-SS) along the shell are shown in figures on pages A5.4 and A5.5 for 4×4×2 integration and in figures on page A5.6 for 3×3×2 integration.

User Hints

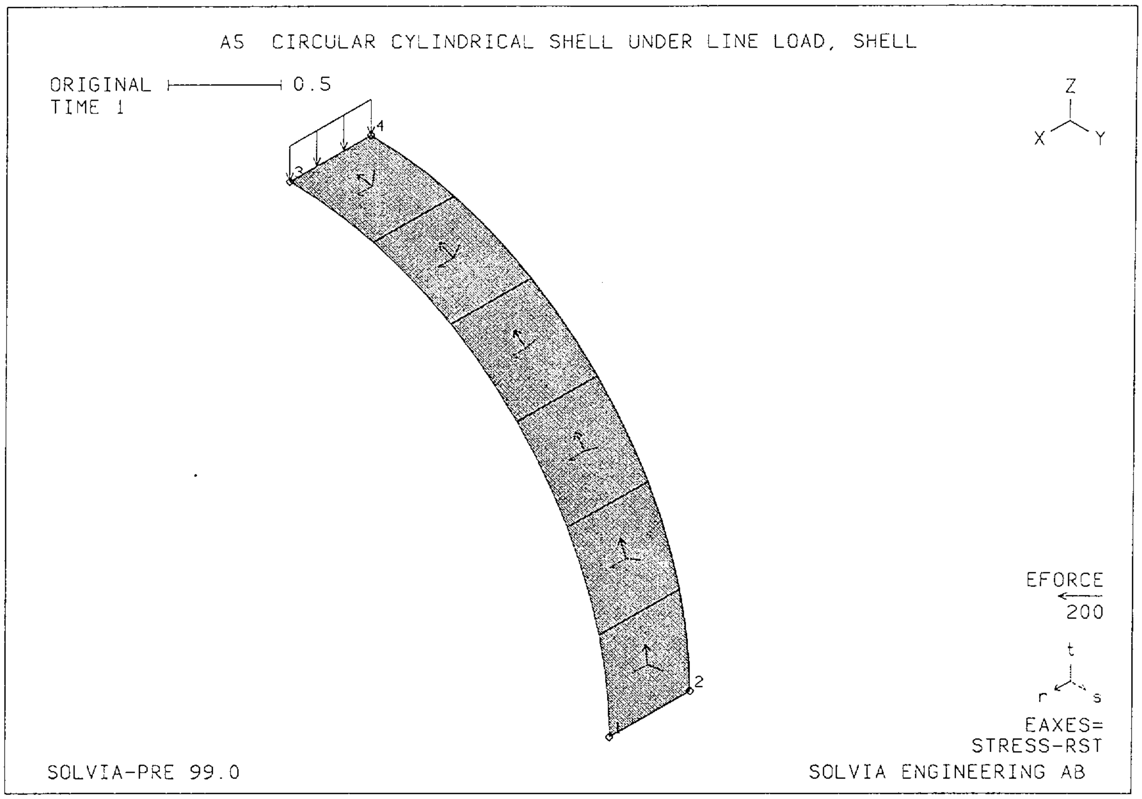
- Note that the stress distribution resulting from the analysis using 4×4×2 integration is more accurate than the 3×3×2 stresses.

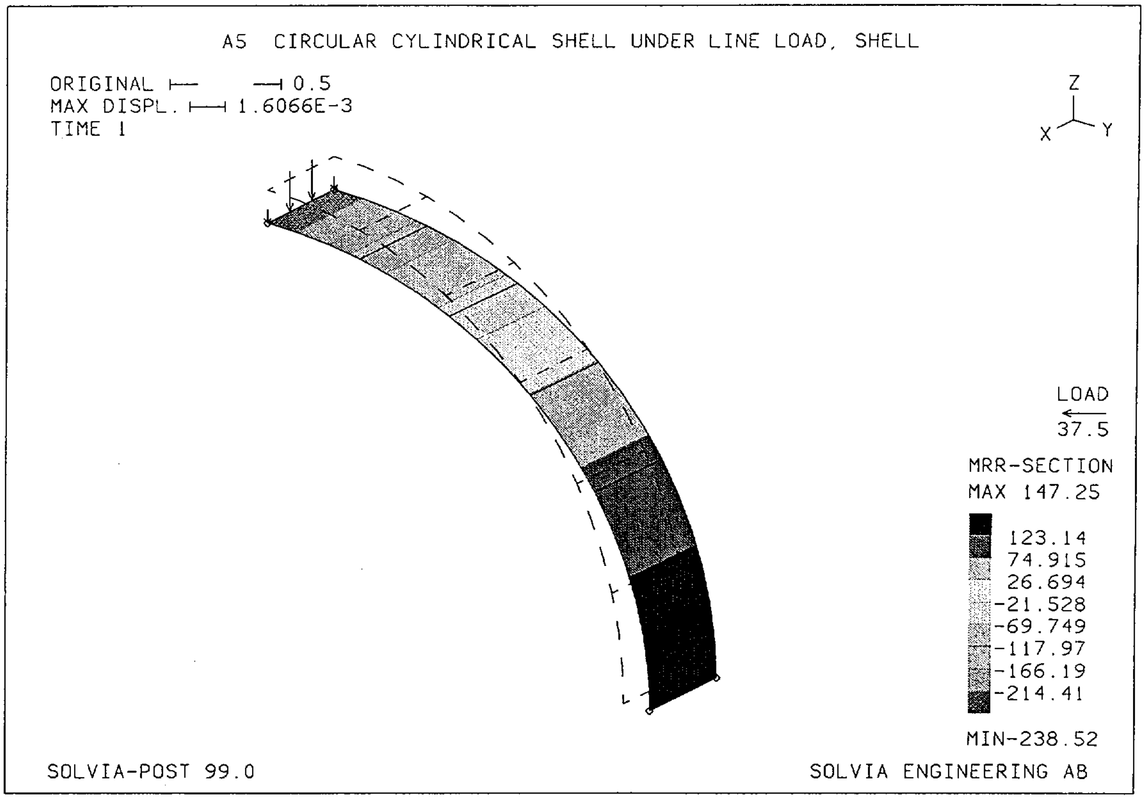
- Note that SOLVIA-PRE automatically assigns GLOBAL rotations for a SHELL midsurface node when the node has a boundary condition for rotation specified [2].

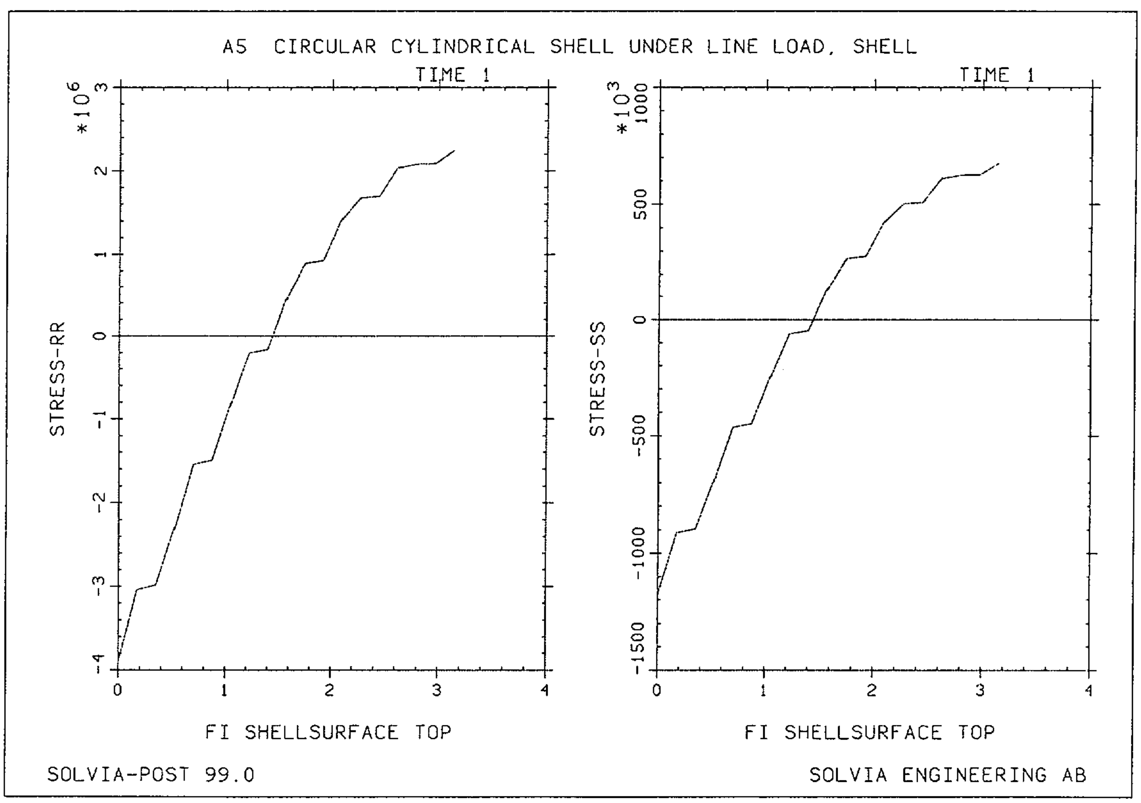
References

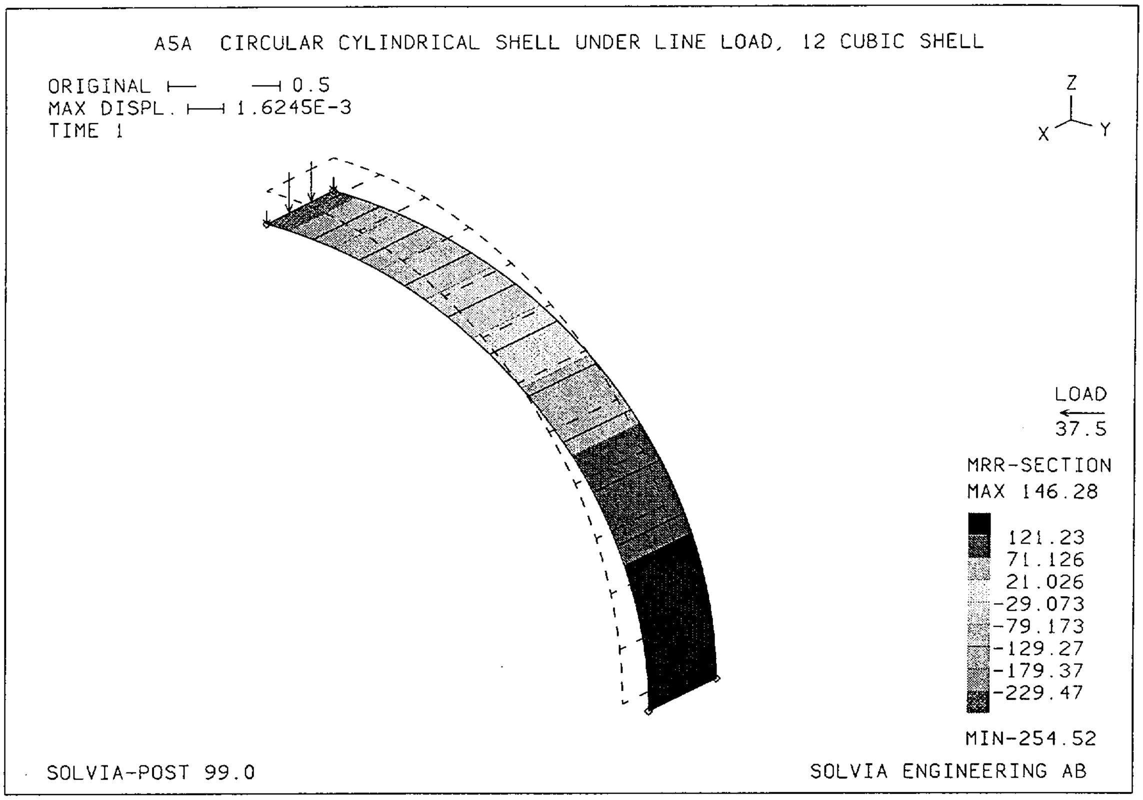
[1] Timoshenko, S., Strength of Materials, Part I, Elementary Theory and Problems, Third Edition, D.Van Nostrand, 1955.

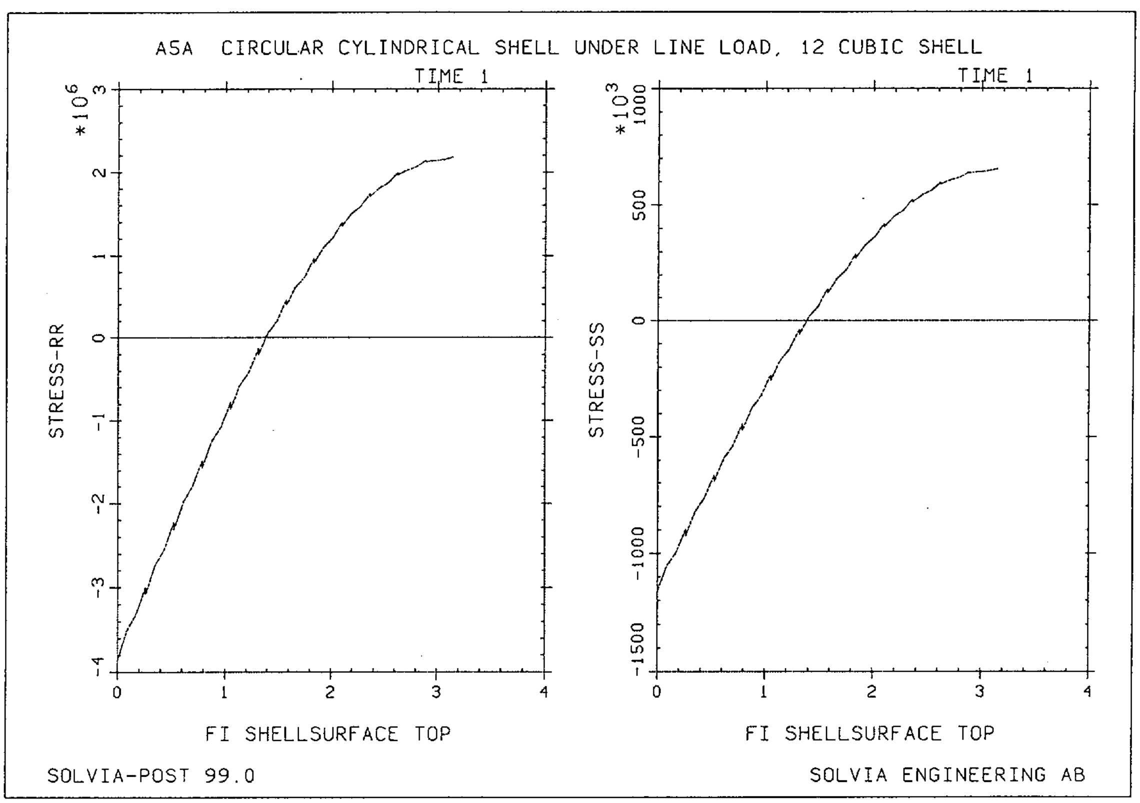
[2] SOLVIA-PRE 99.0 Users Manual, Stress Analysis, Report SE 99-1, p. 7.23 - 7.34.

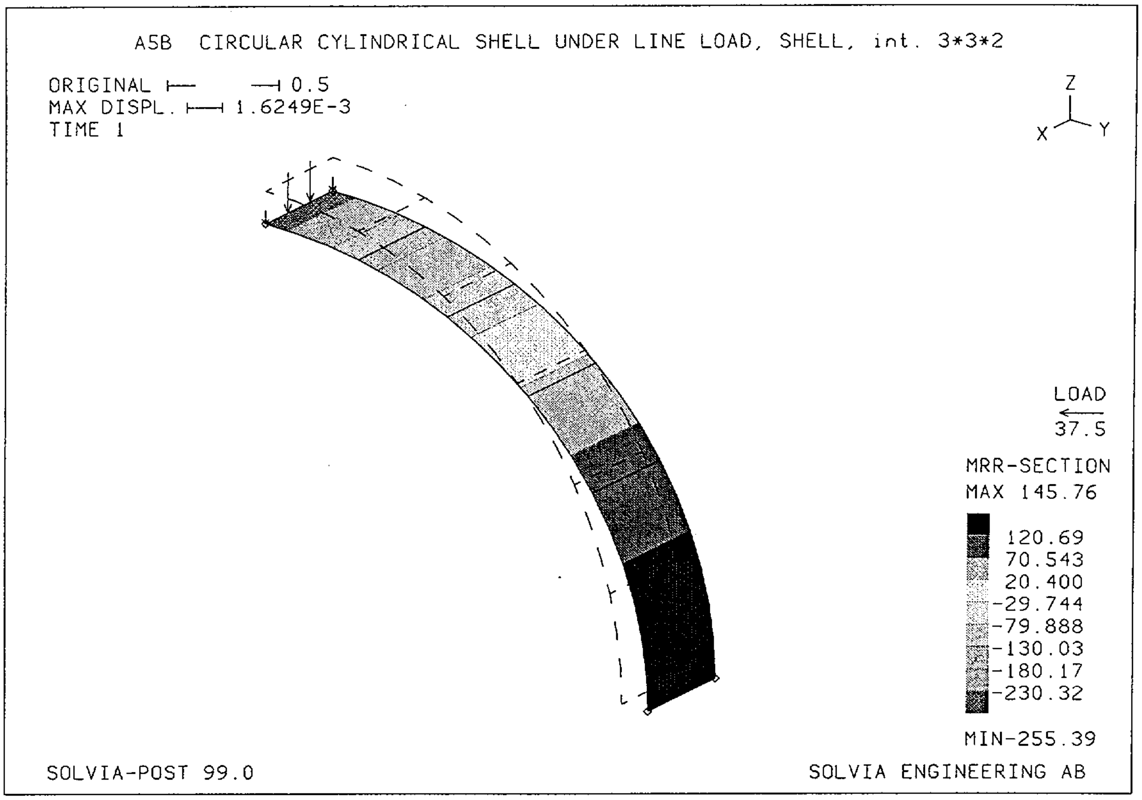

A5 CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, SHELL
ORIGINAL 0.5
TIME 1
Z
X Y
4
3
2
1
EFORCE
200
t
r s
EAXES=
STRESS-RST
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A5 CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, SHELL
ORIGINAL 0.5
MAX DISPL. 1.6066E-3
TIME 1
Z
X Y
LOAD
37.5
MRR-SECTION
MAX 147.25
123.14
74.915
26.694
-21.528
-69.749
-117.97
-166.19
-214.41
MIN-238.52
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

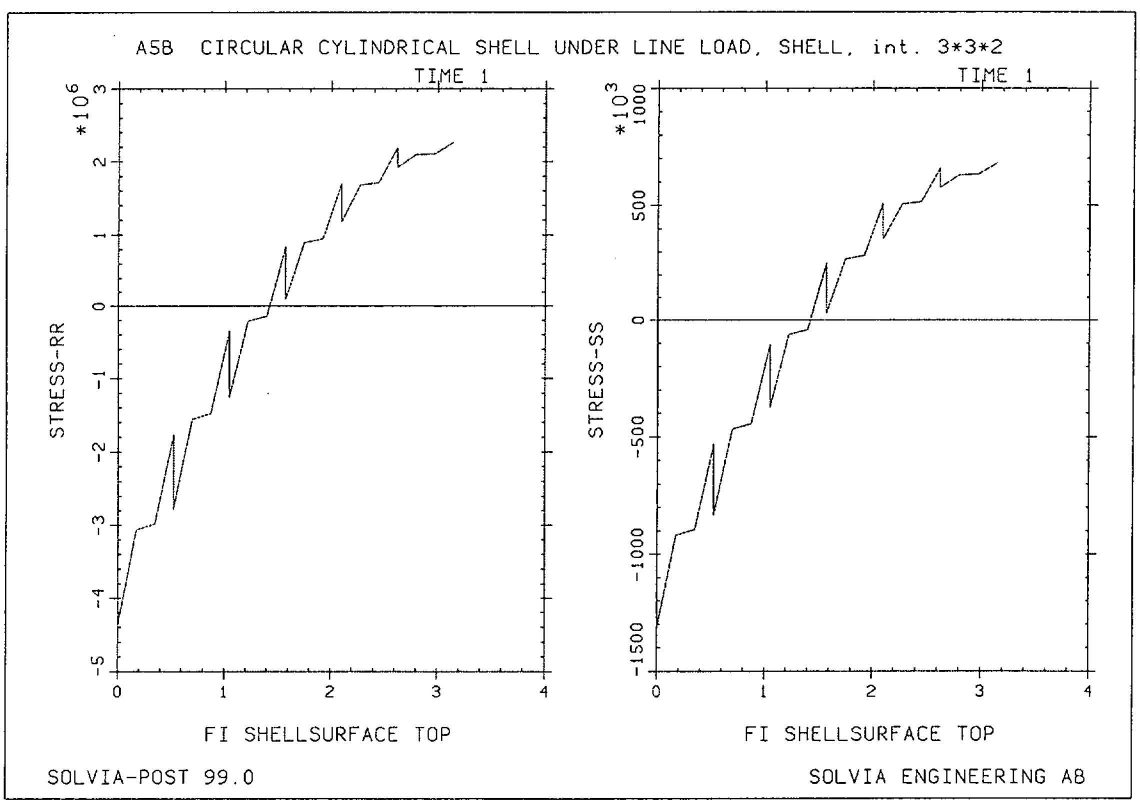

A5 CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, SHELL
TIME 1
*10^6
STRESS-RR
FI SHELLSURFACE TOP
TIME 1
*10^3
STRESS-SS
FI SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A5A CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, 12 CUBIC SHELL
ORIGINAL 0.5
MAX DISPL. 1.6245E-3
TIME 1
Z
X
Y
LOAD
37.5
MRR-SECTION
MAX 146.28
121.23
71.126
21.026
-29.073
-79.173
-129.27
-179.37
-229.47
MIN-254.52
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A5A CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, 12 CUBIC SHELL
TIME 1
TIME 1
*10^6
STRESS-RR
*10^3
STRESS-SS
FI SHELLSURFACE TOP
FI SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB



A5B CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, SHELL, int. 3*3*2
ORIGINAL 0.5
MAX DISPL. 1.6249E-3
TIME 1
Z
X Y
LOAD
37.5
MRR-SECTION
MAX 145.76
120.69
70.543
20.400
-29.744
-79.888
-130.03
-180.17
-230.32
MIN-255.39
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB



A5B CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, SHELL, int. 3*3*2
TIME 1
TIME 1
*10^6
STRESS-RR
*10^3
STRESS-SS
FI SHELLSURFACE TOP
FI SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input (6 elements, 4×4×2 int.)

```
HEAD  'A5  CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, SHELL'
*
DATABASE  CREATE
*
SYSTEM  1  CYLINDRICAL
COORDINATES
ENTRIES  NODE  R  THETA   XL
          1    2.   0.    0.5
          2    2.   0.    0.
          3    2.  90.    0.5
          4    2.  90.    0.
*
MATERIAL  1  ELASTIC  E=2.E11 NU=.3
*
EGROUP  1  SHELL  RINT=4 RESULTS=NSTRESSES STRESSREFERENCE=ELEMENT
THICKNESS  1 .02
GSURFACE  3 1 2 4  EL1=6 EL2=1 NODES=16 SYSTEM=1
LOADS ELEMENT  TYPE=FORCE INPUT=LINE
 3 4  THICKNESS -200
*
FIXBOUNDARIES  345 INPUT=LINES  /  1 2
FIXBOUNDARIES  246 INPUT=LINES  /  3 4
FIXBOUNDARIES  156 INPUT=LINES  /  1 3  /  2 4
*
SET  NSYMBOLS=MYNODES NNUMBERS=MYNODES
MESH  VECTOR=LOAD EAXES=STRESS-RST
*
SOLVIA
END
```

SOLVIA-POST input (6 elements, 4×4×2 int.)

```
*  A5  CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, SHELL
*
DATABASE  CREATE
*
WRITE  FILENAME='a5.lis'
*
VIEW  ID=1  XVIEW=1. YVIEW=0.75 ZVIEW=0.5
SET  VIEW=1 ORIGINAL=DASHED
MESH  CONTOUR=MRR-SECTION NSYMBOLS=MYNODES VECTOR=LOAD
*
EPLINE  NAME=FI
 1  1 5 9 2  TO  6  1 5 9 2
ELINE  LINENAME=FI KIND=SRR OUTPUT=ALL SUBFRAME=21
ELINE  LINENAME=FI KIND=SSS OUTPUT=ALL
*
ZONE  NAME=EDGES INPUT=NODES  /  1 TO 4
NLIST  ZONENAME=EDGES DIRECTION=123
NLIST  KIND=REACTION DIRECTION=34
END
```

EXAMPLE A6

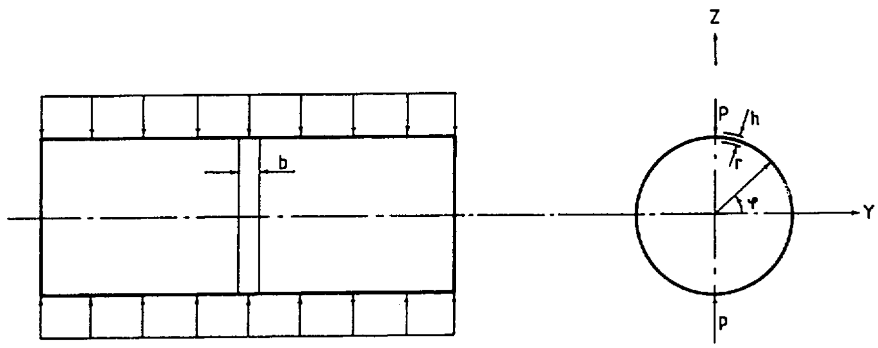
CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, PLATE

Objective

To verify the bending behaviour of the PLATE element when applied to a curved shell structure.

Physical Problem

The cylindrical shell to be analyzed is the same as in example A.5. It is shown again in the figure below.



$r = 2.00$ m

$h = 0.02$ m

$b = 0.50$ m

$P = 400$ N/m

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

$\nu = 0.3$

Finite Element Model

As in the previous example only a 90° portion of the cylinder is modeled due to symmetry reasons, see figures on page A6.3. Note that the boundary conditions are such that the displacements in the X-direction are zero. In addition, all 4 boundary faces have symmetry boundary conditions.

Solution Results

The theoretical solution is the same as for example A.5. Using the input data on pages A6.5 and A6.6 the following results are obtained:

Radial displacements:

δ_z (mm) $\phi = 90$ degrees		δ_y (mm) $\phi = 0$ degrees	
Theory	SOLVIA	Theory	SOLVIA
-1.625	-1.627	1.492	1.492

The distribution of radial displacements along a line from node 3 to node 5 is shown in figure on page A6.4.

The theoretical bending moment per unit length is ([1] p. 380):

$$M = \frac{P\,r}{2}\left(\cos\phi - \frac{2}{\pi}\right)$$

The bending moment at the point of application of the line load (ϕ=90 degrees) can then be compared with the results obtained using the input shown on pages A6.5 and A6.6.

Bending moment (Nm/m)

Theory	SOLVIA	
	el.45 pt 2	el.46 pt 4
-255	-276	-251

The distribution of the bending moment per unit length along two lines is shown on page A6.4. The line "MID" is located in the symmetry plane of the model (X=0.25) while the line "SIDE" is located along one side of the model (X=0).

The distribution of the membrane force per unit length in the circumferential direction is also shown on page A6.4.

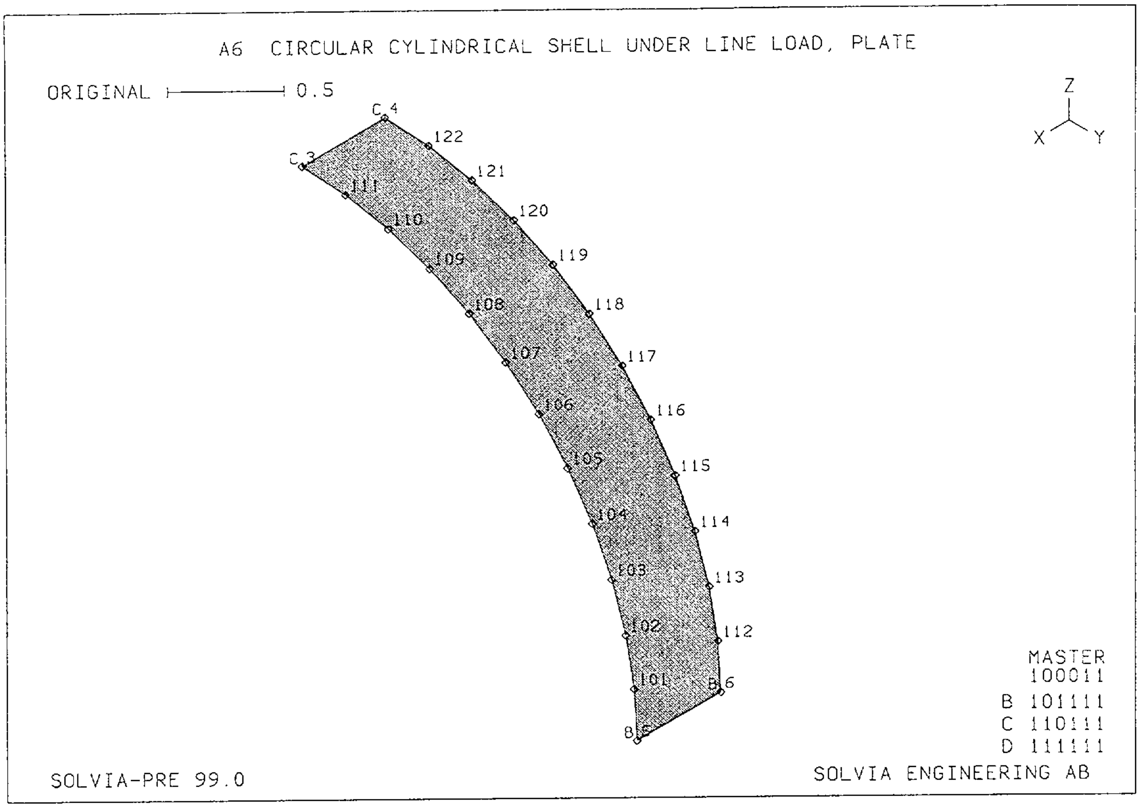
A Local Cylindrical System is used in displaying the bending moment (M11) and the membrane force (F11) per unit length. The x_1-direction of the Local System is in the circumferential direction.

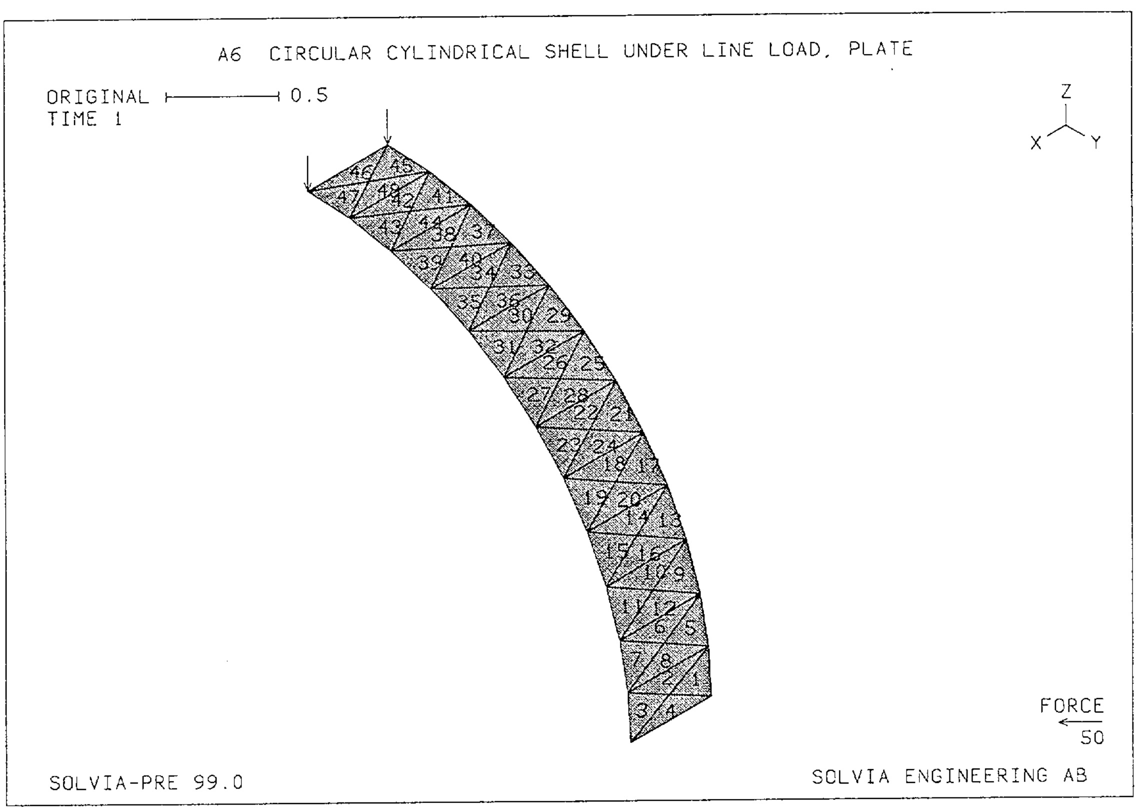
User Hints

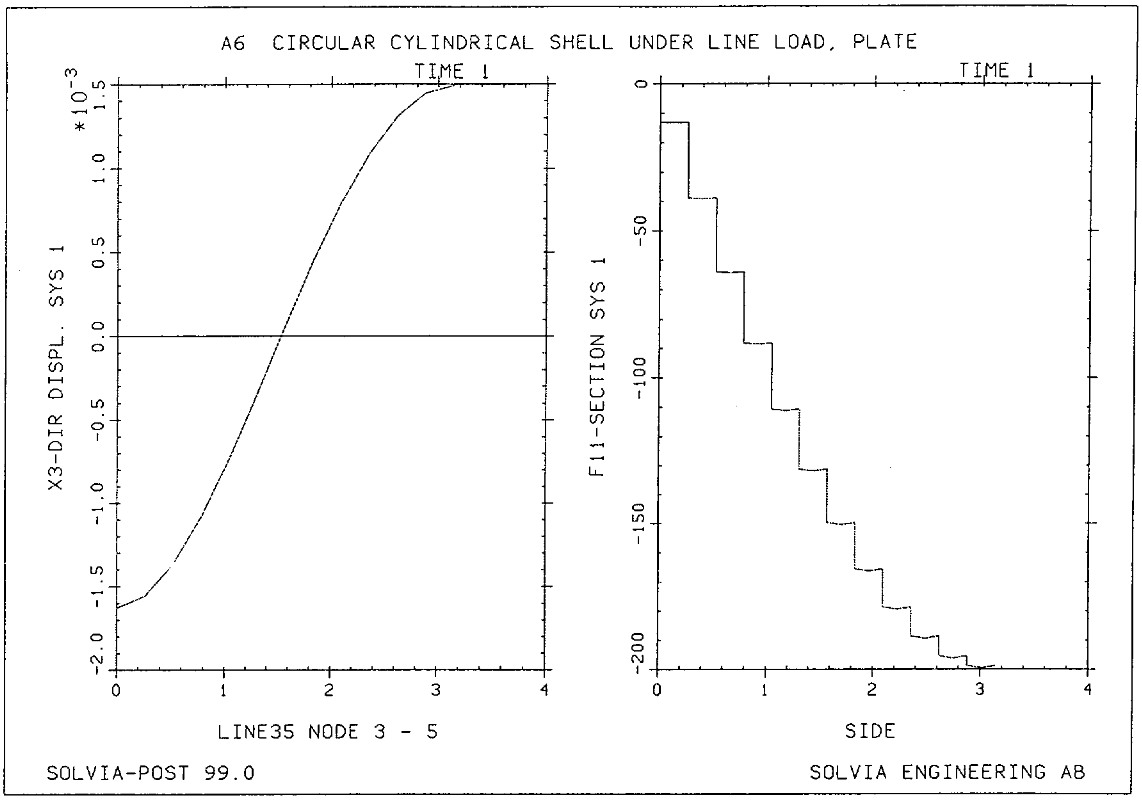
- The PLATE element does not include shear deformation effects; hence
  - even when the shell modelled is thick, shear deformation effects are not included in the model
  - the element can reliably be employed for thin and even very thin plates/shells.
- The membrane action in the PLATE element corresponds to a constant strain assumption. Hence, the membrane forces are constant over each element, see for example the variation of the F11-force per unit length on page A6.4.
- Note that significantly larger moment jumps between neighboring elements occur for the line "SIDE" than for the line "MID".

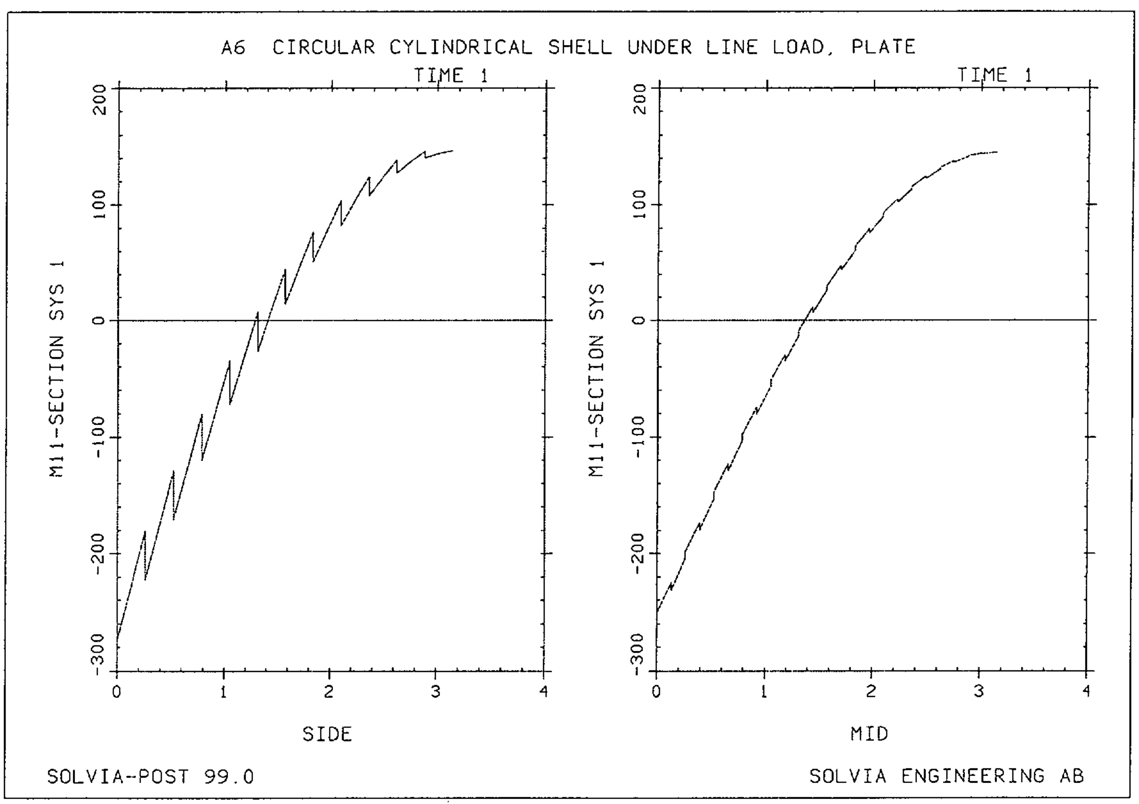
Reference

[1] Timoshenko, S., Strength of Materials, Part I, Elementary Theory and Problems, Third Edition, D Van Nostrand, 1955.


A6 CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, PLATE
ORIGINAL 0.5
Z
X Y
C 4
C 3
122
121
120
119
118
117
116
115
114
113
112
111
110
109
108
107
106
105
104
103
102
101
B 6
B 5
MASTER
100011
B 101111
C 110111
D 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A6 CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, PLATE
ORIGINAL 0.5
TIME 1
Z
X Y
FORCE
50
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A6 CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, PLATE
TIME 1
X3-DIR DISPL. SYS 1
*10^-3
LINE35 NODE 3 - 5
F11-SECTION SYS 1
SIDE
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A6 CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, PLATE
TIME 1
M11-SECTION SYS 1
SIDE
MID
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A6  CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, PLATE'
*
DATABASE  CREATE
*
MASTER  IDOF=100011
*
COORDINATES
 1 .5  /  2  /  3 .5 0 2  /  4 0 0 2  /  5 .5 2  /  6 0 2
*
LINE  ARC  N1=5 N2=3 NCENTER=1 EL=12
LINE  ARC  N1=6 N2=4 NCENTER=2 EL=12
*
MATERIAL  1  ELASTIC  E=2.E11 NU=.3
*
EGROUP  1  PLATE  RESULTS=TABLES
STRESSTABLE  1  1 2 3 4 5 6
GSURFACE  6 4 3 5  EL1=12 EL2=1
EDATA  /  1  0.02
*
LOADS  CONCENTRATED
 3 3 -50
 4 3 -50
*
FIXBOUNDARIES   34  /  5 6
FIXBOUNDARIES   24  /  3 4
FIXBOUNDARIES  234  /  1 2
*
MESH  NSYMBOL=YES NNUMBER=YES OUTLINE=YES BCODE=ALL
MESH  ENUMBER=YES VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A6  CIRCULAR CYLINDRICAL SHELL UNDER LINE LOAD, PLATE
*
DATABASE  CREATE
SYSTEM 1 CYLINDRICAL
*
WRITE  FILENAME='a6.lis'
*
NPLINE   NAME=LINE35
 3 111   STEP -1 TO 101 5
EPLINE   NAME=SIDE
 45  2 4 1  STEP -4 TO  1  2 4 1
*
SUBFRAME  21
NLINE  LINENAME=LINE35 DIRECTION=3 OUTPUT=ALL SYSTEM=1
ELINE  LINENAME=SIDE KIND=F11 OUTPUT=ALL SYSTEM=1
SUBFRAME  21
ELINE  LINENAME=SIDE KIND=M11 OUTPUT=ALL SYSTEM=1
*
EPLINE  NAME=MID
 46 4 3 / 48 3 4 / 42 4 3 / 44 3 4
 38 4 3 / 40 3 4 / 34 4 3 / 36 3 4
 30 4 3 / 32 3 4 / 26 4 3 / 28 3 4
 22 4 3 / 24 3 4 / 18 4 3 / 20 3 4
 14 4 3 / 16 3 4 / 10 4 3 / 12 3 4
  6 4 3 /  8 3 4 /  2 4 3 /  4 3 4
*
ELINE  LINENAME=MID KIND=M11 OUTPUT=ALL SYSTEM=1
*
NLIST  KIND=REACTION DIRECTION=34
END
```

EXAMPLE A7

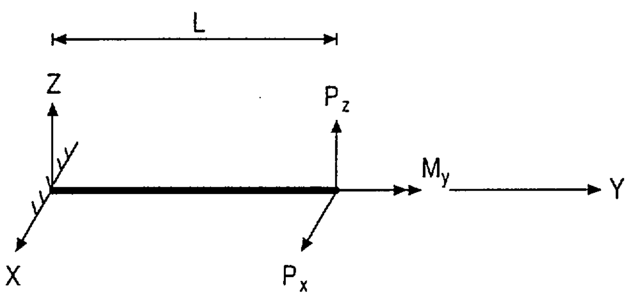
CANTILEVER BEAM UNDER TIP LOADS, BEAM ELEMENT

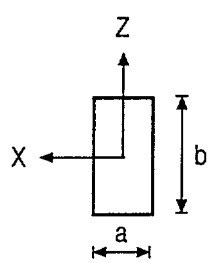
Objective

To verify the three-dimensional action of the straight BEAM element under end loads.

Physical Problem

A cantilever beam as shown in figure below under transverse end load and torsion.





$$L = 1 \text{ m}$$
$$a = 0.01 \text{ m}$$
$$b = 0.02 \text{ m}$$
$$E = 2.0 \cdot 10^{11} \text{ N/m}^2$$
$$\nu = 0.3$$
$$P_x = 5 \text{ N}$$
$$P_z = 10 \text{ N}$$
$$M_y = 1 \text{ Nm}$$

Finite Element Model

The behaviour of the cantilever is described using one BEAM element only.

Solution Results

The theoretical solution is for example given in [1] for the end displacement δ and for the end rotations ϕ_b due to bending and ϕ_t due to torsion.

$$\delta = \frac{PL^3}{3EI} + \frac{PL}{A_s G}$$

$$\phi_b = \frac{PL^2}{2EI} \quad \text{(bending)} \qquad \phi_t = \frac{M_t L}{GJ} \quad \text{(torsion)}$$

The following solution is obtained using the input data on page A7.4:

Displacements (mm)

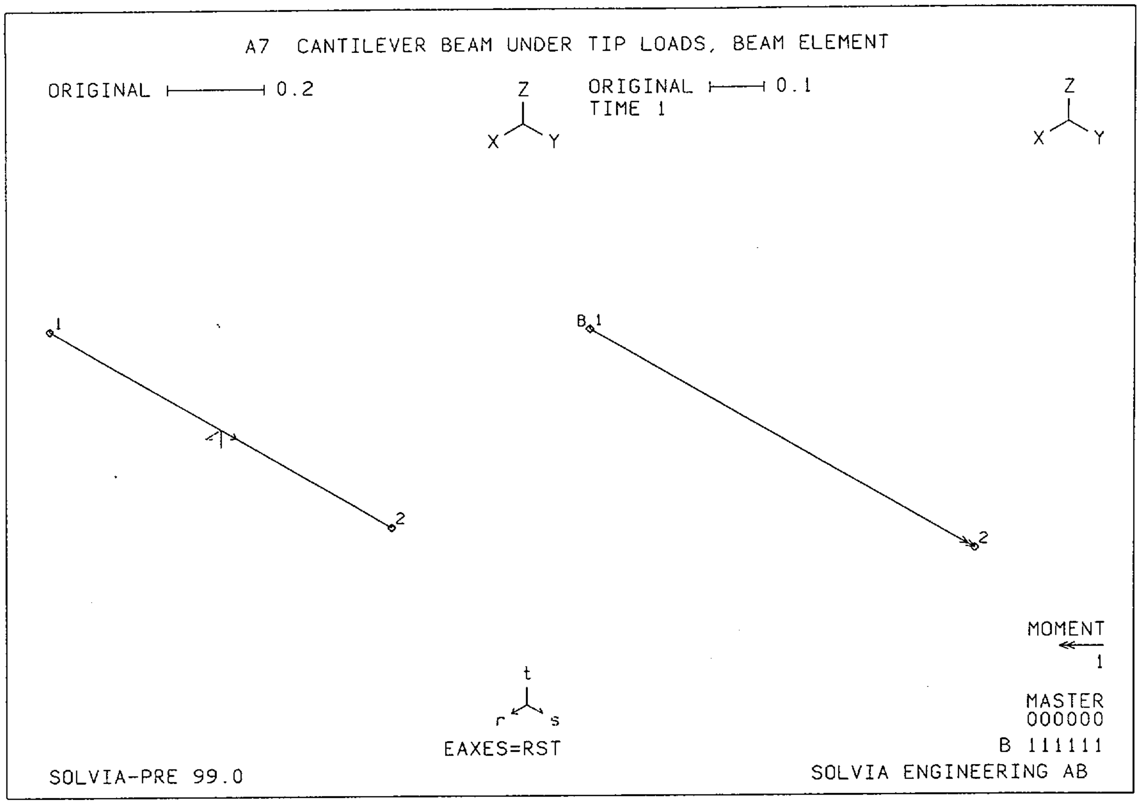
Theory		SOLVIA	
δ_x	δ_z	δ_x	δ_z
5.0004	2.5008	5.0004	2.5008

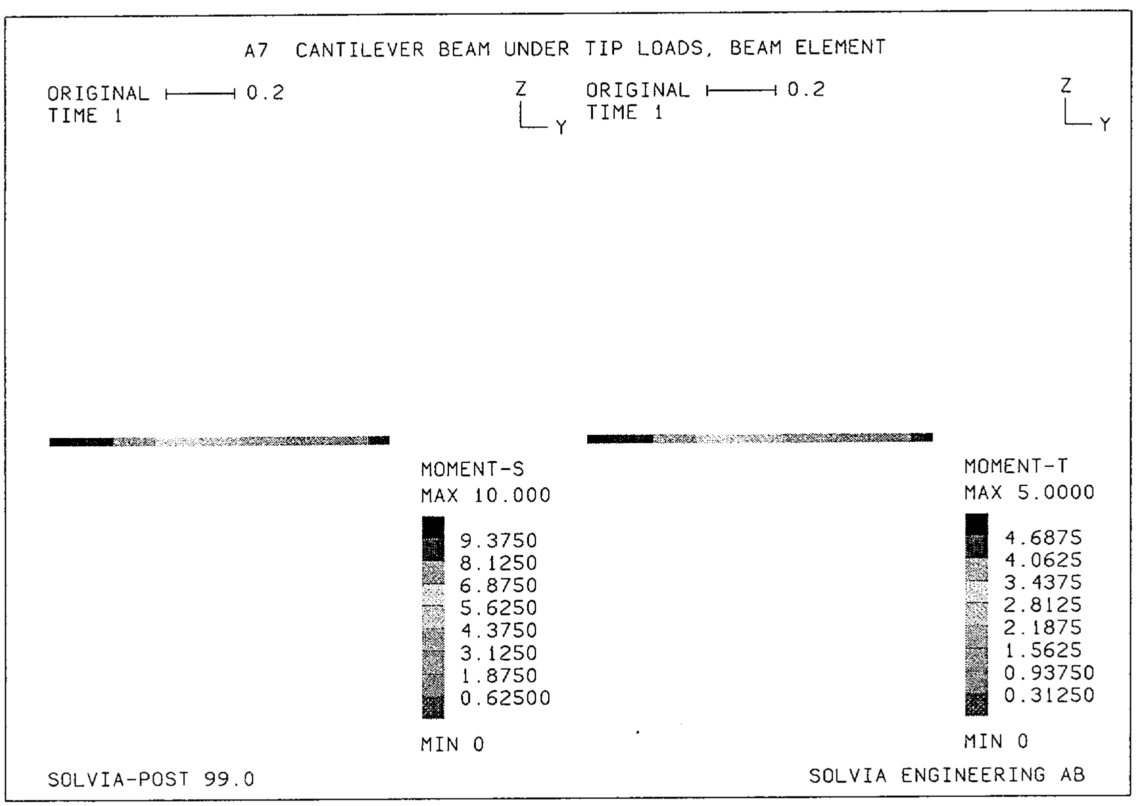
Rotations $(\text{radians} \cdot 10^{-3})$

Theory			SOLVIA		
ϕ_x	ϕ_y	ϕ_z	ϕ_x	ϕ_y	ϕ_z
3.7500	2.8424	-7.5000	3.7500	2.8424	-7.5000

Reference

[1] Roark, R.J., Formulas for Stress and Strain, Fourth Edition, McGraw Hill, 1965.


A7 CANTILEVER BEAM UNDER TIP LOADS, BEAM ELEMENT
ORIGINAL 0.2
Z
X Y
ORIGINAL 0.1
TIME 1
Z
X Y
1
2
B 1
2
MOMENT
1
t
r s
EAXES=RST
MASTER
000000
B 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A7 CANTILEVER BEAM UNDER TIP LOADS, BEAM ELEMENT
ORIGINAL 0.2
TIME 1
Z
Y
ORIGINAL 0.2
TIME 1
Z
Y
MOMENT-S
MAX 10.000
9.3750
8.1250
6.8750
5.6250
4.3750
3.1250
1.8750
0.62500
MIN 0
MOMENT-T
MAX 5.0000
4.6875
4.0625
3.4375
2.8125
2.1875
1.5625
0.93750
0.31250
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A7  CANTILEVER BEAM UNDER TIP LOADS, BEAM ELEMENT'
*
DATABASE  CREATE
*
COORDINATES
 1
 2  0. 1.
*
MATERIAL  1  ELASTIC E=2.E11 NU=.3
*
EGROUP  1  BEAM  RESULT=FORCES
SECTION  1  RECTANGULAR WTOP=0.01 D=0.02
BEAMVECTOR
 1  1.
*
ENODES
 1  -1 1 2
*
LOADS  CONCENTRATED
 2 1  5.
 2 3 10.
 2 5  1.
*
FIXBOUNDARIES  /  1
*
SET  NSYMBOLS=YES NNUMBERS=YES
MESH  EAXES=RST SUBFRAME=21
MESH  BCODE=ALL VECTOR=MOMENT
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A7  CANTILEVER BEAM UNDER TIP LOADS, BEAM ELEMENT
*
DATABASE  CREATE
*
WRITE  FILENAME='a7.lis'
*
SET  VIEW=X ORIGINAL=YES DEFORMED=NO
MESH  CONTOUR=MS SUBFRAME=21
MESH  CONTOUR=MT
*
NLIST  KIND=DISPLACEMENT
NLIST  KIND=REACTION
ELIST
END
```

EXAMPLE A8

CANTILEVER BEAM UNDER TIP LOADS, ISOBEAM ELEMENT

Objective

To verify the three-dimensional action of the straight ISOBEAM element under end loads.

Physical Problem

Same as in Example A7.

Finite Element Model

One 4-node ISOBEAM element as shown in the top figure on page A8.3 is chosen to model the cantilever.

Solution Results

The theoretical solution is the same as given in Example A7. The following SOLVIA results are obtained using the input data on page A8.4.

Displacements (mm):

Theory δ_x	Theory δ_z	SOLVIA δ_x	SOLVIA δ_z
5.0004	2.5008	5.0003	2.5007

Rotations $\left(\text{radians} \cdot 10^{-3}\right)$:

Theory ϕ_x	Theory ϕ_y	Theory ϕ_z	SOLVIA ϕ_x	SOLVIA ϕ_y	SOLVIA ϕ_z
3.7500	2.8424	-7.5000	3.7500	2.8188	-7.5000

User Hints

- The 4-node ISOBEAM element has cubic assumptions for all displacements along the beam and can in this case describe the bending behaviour exactly. Special assumptions are used for the warping displacements due to torsion.

- The ISOBEAM element is more expensive to use than the corresponding linear BEAM element. The reason is that the ISOBEAM stiffness matrix and stresses are evaluated by numerical integration while the linear BEAM is formulated in closed form.

- Comparing the performance of the 4-node ISOBEAM element versus the BEAM element, we may note

  - for linear analysis of straight beam sections, always use the BEAM element;

  - for linear analysis of curved beams the ISOBEAM element can be effective (because transverse and longitudinal displacements are interpolated to the same order), but an alternative is to model a curved beam as an assemblage of straight BEAM elements (see Examples A9 and A10);

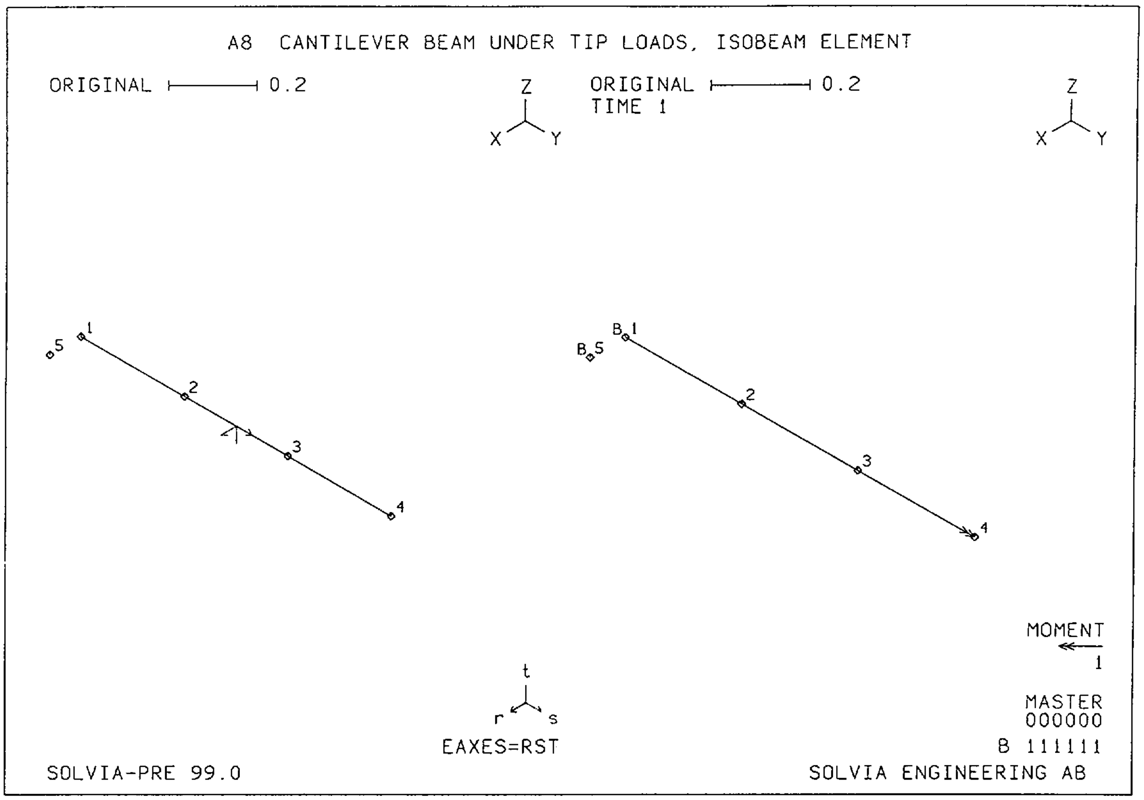
  - for nonlinear analysis the ISOBEAM element can be more effective, because in large displacement analysis geometry changes are modelled more accurately, (and transverse and longitudinal displacements are interpolated to the same order);

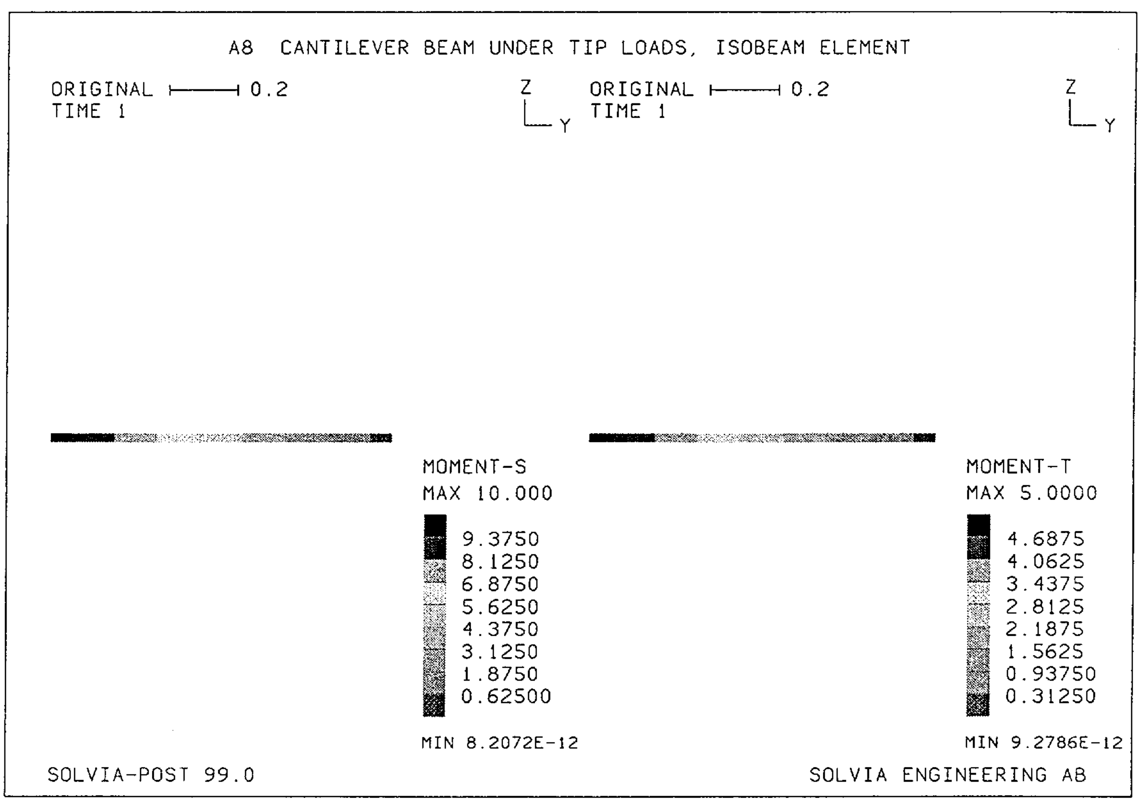
  - the ISOBEAM element is compatible with the isoparametric SHELL element and can be employed to model stiffeners.

- Note that the nodal point forces and moments output by SOLVIA-POST are virtual work equivalent nodal point forces/moments,

$$\mathbf{F} = \int_V \mathbf{B}^T \tau \, dV$$

where B is the strain-displacement matrix and t is the vector of element stresses. This vector **F** is equal to the externally applied forces/moments. Hence the forces/moments output by SOLVIA-POST corresponding to the element internal nodes are zero, because no loads are applied (in this problem) at these nodes.


A8 CANTILEVER BEAM UNDER TIP LOADS, ISOBEAM ELEMENT
ORIGINAL 0.2
ORIGINAL 0.2
TIME 1
MOMENT
1
MASTER
000000
B 111111
EAXES=RST
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A8 CANTILEVER BEAM UNDER TIP LOADS, ISOBEAM ELEMENT
ORIGINAL 0.2
TIME 1
ORIGINAL 0.2
TIME 1
MOMENT-S
MAX 10.000
9.3750
8.1250
6.8750
5.6250
4.3750
3.1250
1.8750
0.62500
MIN 8.2072E-12
MOMENT-T
MAX 5.0000
4.6875
4.0625
3.4375
2.8125
2.1875
1.5625
0.93750
0.31250
MIN 9.2786E-12
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A8  CANTILEVER BEAM UNDER TIP LOADS, ISOBEAM ELEMENT'
*
DATABASE  CREATE
*
COORDINATES
 1  TO  4  0. 1.  /  5  0.1
*
MATERIAL  1  ELASTIC  E=2.E11 NU=.3
*
EGROUP  1  ISOBEAM RESULTS=FORCES
SECTION  1  SDIM=0.01 TDIM=0.02
ENODES
 1  5 1 4 2 3
*
LOADS  CONCENTRATED
 4 1  5.
 4 3 10.
 4 5  1.
*
FIXBOUNDARIES  /  1 5
*
SET  NNUMBERS=YES NSYMBOLS=YES
MESH  EAXES=RST SUBFRAME=21
MESH  BCODE=ALL VECTOR=MOMENT
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A8  CANTILEVER BEAM UNDER TIP LOADS, ISOBEAM ELEMENT
*
DATABASE  CREATE
*
WRITE  FILENAME='a8.lis'
*
SET  VIEW=X ORIGINAL=YES DEFORMED=NO
MESH  CONTOUR=MS SUBFRAME=21
MESH  CONTOUR=MT
*
NLIST  DIRECTION=13456
NLIST  DIRECTION=13456 KIND=REACTION
ELIST
END
```

EXAMPLE A9

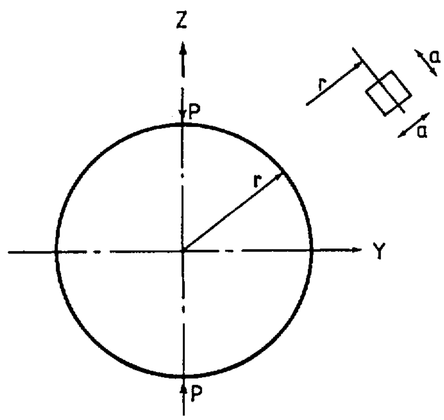
PINCHED CIRCULAR RING, ISOBEAM ELEMENTS

Objective

To verify the curved ISOBEAM element under bending.

Physical Problem

A circular ring of square cross section is subjected to equal and opposite concentrated forces, see figure below. The thickness of the ring and the mean radius are the same as for the shell in Examples A5 and A6.



$r = 2.000 \quad m$

$a = 0.020 \quad m$

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

$\nu = 0.3$

$P = 8.000 \quad N$

Finite Element Model

Symmetry gives that only a quarter of the ring needs to be modeled, see top figures on page A9.3 Four cubic ISOBEAM elements are used.

Solution Results

The theoretical solution is given in [1] p. 381 as follows:

$$\delta_z = -\frac{1}{2}\left(\frac{\pi}{4} - \frac{2}{\pi}\right)\frac{Pr^3}{EI}$$

$$\delta_y = \frac{1}{2}\left(\frac{2}{\pi} - \frac{1}{2}\right)\frac{Pr^3}{EI}$$

where δ_z is the radial displacement at the application of the load and δ_y is the radial displacement 90° from the load (along the y-axis).

The following displacement results are obtained:

δ_z (mm)		δ_y (mm)	
Theory	SOLVIA	Theory	SOLVIA
-1.785	-1.786	1.639	1.640

The SOLVIA results are obtained using the input data on pages A9.4 and A9.5.

The variation of radial displacements along a line from node 2 to node 1 is shown in the top figure on page A9.4.

The theoretical bending moment at the application of the load is ([1] p. 380):

$$M = -\frac{Pr}{\pi}$$

Comparison with the SOLVIA results yields:

Bending moment (Nm)	
Theory	SOLVIA
-5.09	-5.09

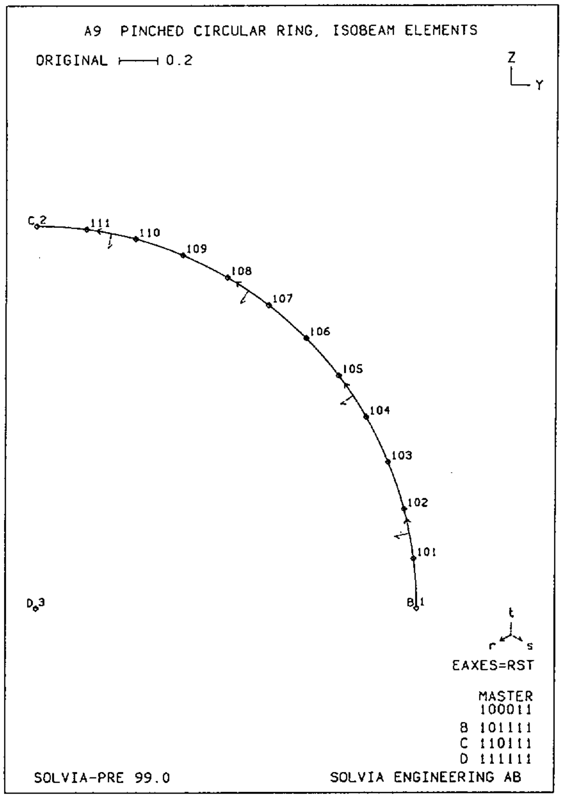
The variation of the bending moment is shown in the top figure on page A9.4.

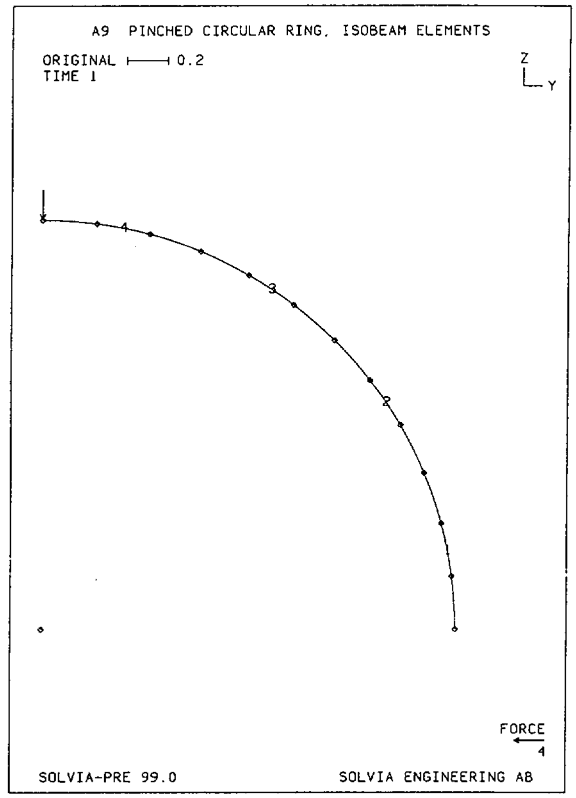
User Hints

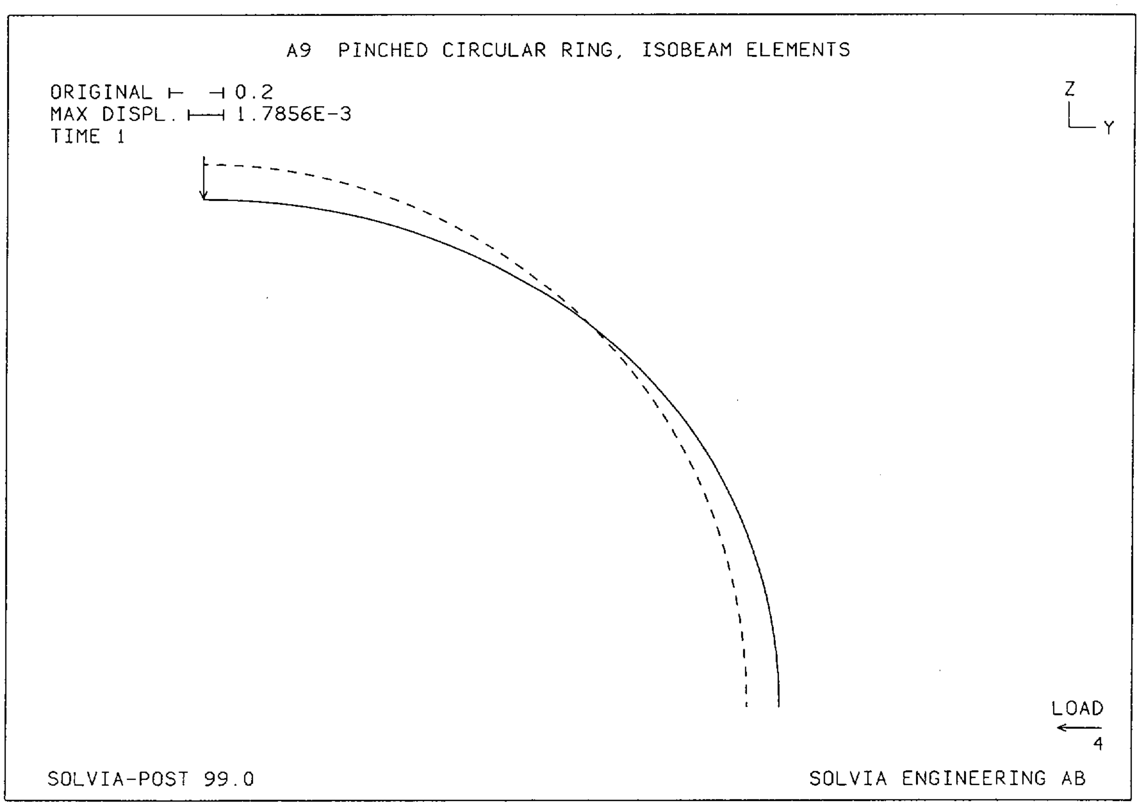
- The SOLVIA results are obtained using three point integration along the centroidal axis of each element. The four point integration results give about 4% less displacements.
- The present example structure is in a state of plane stress while the previous Examples A5 and A6 were plane strain solutions. As for these previous examples the corresponding 8-node PLANE element could also effectively be employed.
- Radial displacements are calculated in SOLVIA-POST using the x_3 displacements of a Local Cylindrical System.

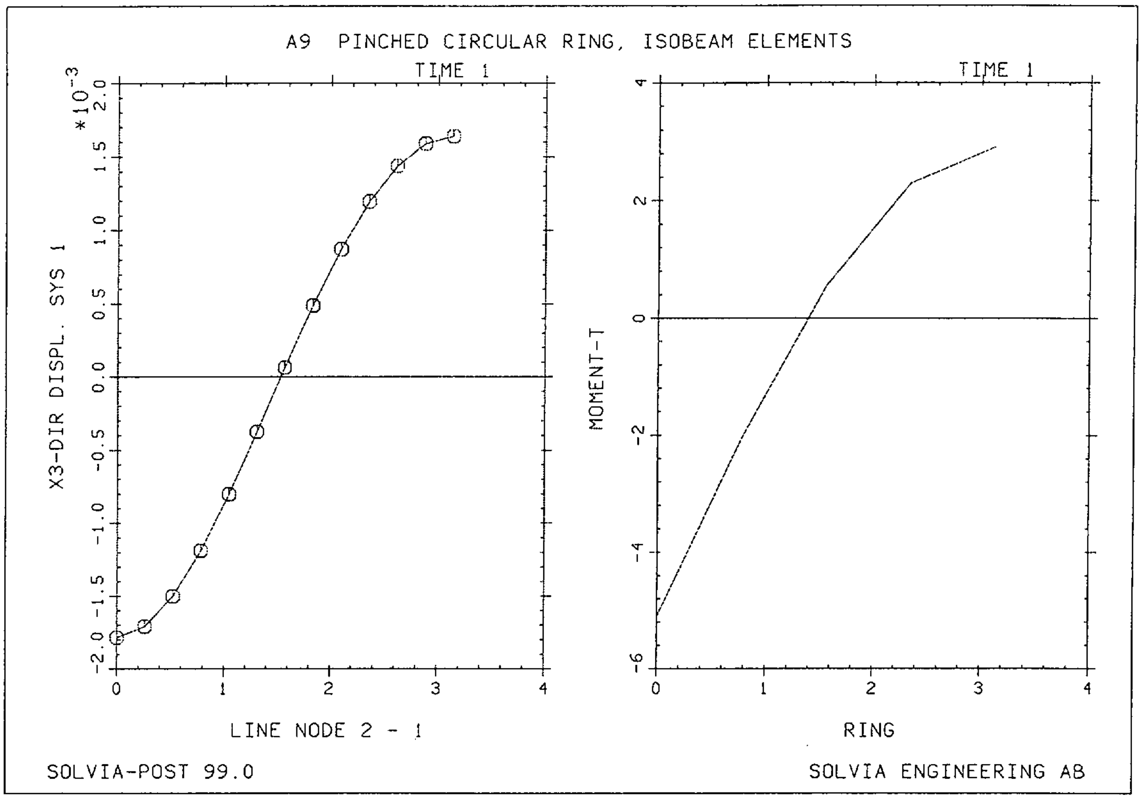
Reference

[1] Timoshenko, S., Strength of Materials, Part I, Elementary Theory and Problems, Third Edition, D. Van Nostrand, 1955.


A9 PINCHED CIRCULAR RING, ISOBEAM ELEMENTS
ORIGINAL 0.2
Z
Y
C 2
111
110
109
108
107
106
105
104
103
102
101
D 3
B 1
t
r
s
EAXES=RST
MASTER
100011
B 101111
C 110111
D 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A9 PINCHED CIRCULAR RING, ISOBEAM ELEMENTS
ORIGINAL 0.2
TIME 1
Z
Y
4
3
2
FORCE
4
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A9 PINCHED CIRCULAR RING, ISOBEAM ELEMENTS
ORIGINAL 0.2
MAX DISPL. 1.7856E-3
TIME 1
Z
Y
LOAD
4
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB



SOLVIA-PRE input

```
HEAD  'A9  PINCHED CIRCULAR RING, ISOBEAM ELEMENTS'
*
DATABASE  CREATE
*
MASTER  IDOF=100011
COORDINATES
 ENTRIES NODE  Y   Z
            1  2
            2  0   2
            3
*
LINE  ARC  N1=1 N2=2 NCENTER=3 EL=4 MIDNODES=2
*
MATERIAL  1  ELASTIC E=2.E11 NU=0.3
*
EGROUP  1  ISOBEAM RESULTS=FORCES
SECTION  1  SDIM=0.02 TDIM=0.02
GLINE  1 2 3  EL=4 NODES=4
*
LOADS  CONCENTRATED
 2 3 -4
*
FIXBOUNDARIES  34  /  1
FIXBOUNDARIES  24  /  2
FIXBOUNDARIES      /  3
*
```

SOLVIA-PRE input (cont.)

```
SET   VIEW=X SMOOTHNESS=YES NSYMBOLS=YES PLOTORIENTATION=PORTRAIT
MESH   NNUMBERS=YES EAXES=RST BCODE=ALL
MESH   ENUMBER=YES VECTOR=LOAD
*
SOLVIA
END
*
```

SOLVIA-POST input

```
*  A9  PINCHED CIRCULAR RING, ISOBEAM ELEMENTS
*
DATABASE  CREATE
SYSTEM 1 CYLINDRICAL
*
WRITE  FILENAME='a9.lis'
*
MESH  VIEW=X SMOOTHNESS=YES ORIGINAL=DASHED VECTOR=LOAD
*
NPLINE  NAME=LINE  /  2  111 STEP -1 TO 101  1
SUBFRAME  21
NLINE  LINENAME=LINE DIRECTION=3 SYMBOL=1 OUTPUT=ALL SYSTEM=1
*
EPLINE  NAME=RING
 4 2 / 4 1 / 3 2 / 3 1
 2 2 / 2 1 / 1 2 / 1 1
ELINE  LINENAME=RING KIND=MT OUTPUT=ALL
END
```

EXAMPLE A10

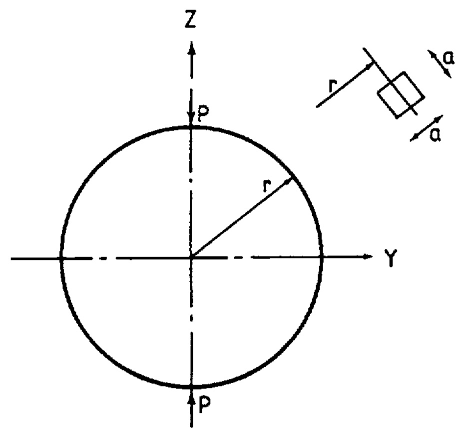
PINCHED CIRCULAR RING, BEAM ELEMENTS

Objective

To verify the BEAM element when applied to a curved structure.

Physical Problem

A circular ring of square cross-section subject to equal and opposite concentrated forces, see figure below, is considered. The problem is the same as in Example A9.



$r = 2.000$ m

$a = 0.020$ m

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

$\nu = 0.3$

$P = 8.000$ N

Finite Element Model

Symmetry gives that only a quarter of the ring need be modelled, see the bottom figures on page A10.2. Twelve BEAM elements are used.

Solution Results

The theoretical solution is the same as for Example A9. The following SOLVIA results are obtained using the input data shown on page A10.4.

Displacements:

δ_z = radial displacement at the point of load application

δ_y = radial displacement 90° from the point of load application

δ_z (mm)		δ_y (mm)	
Theory	SOLVIA	Theory	SOLVIA
-1.785	-1.779	1.639	1.634

The radial displacements of the ring are shown in the left bottom figure on page A10.3. The x_3 direction in the Local Cylindrical System is in the radial direction.

Forces and moments:

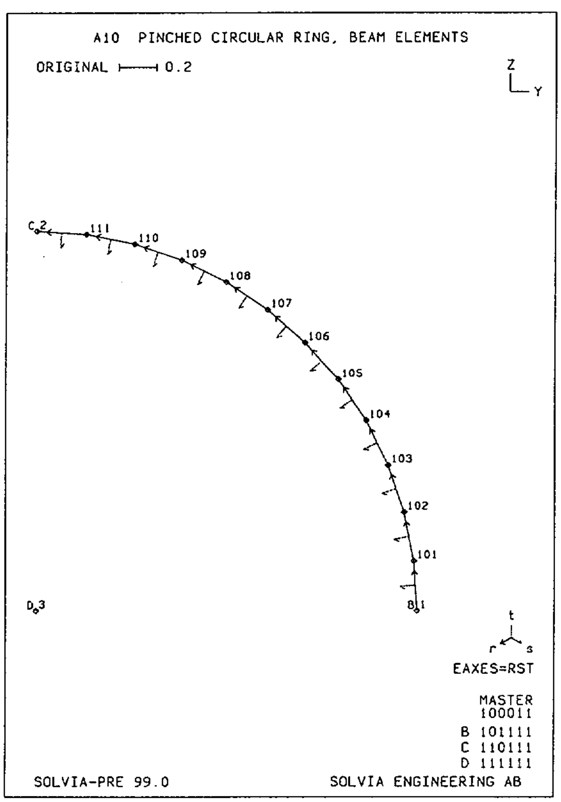
Bending moment (Nm) (at the point of load application) Theory	SOLVIA
-5.09	-5.09

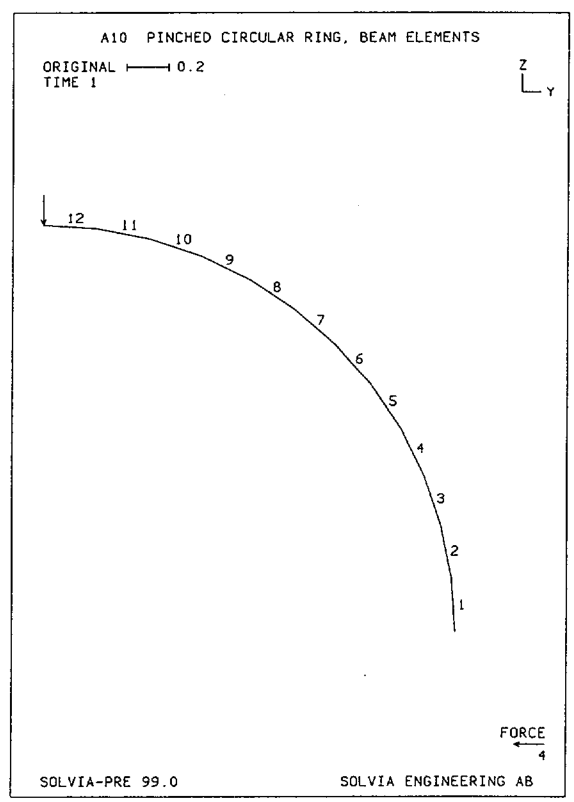
A contour plot of the bending moment is shown in the top figure on page A10.3. The variation of the axial force is shown in the bottom figure on page A10.3.

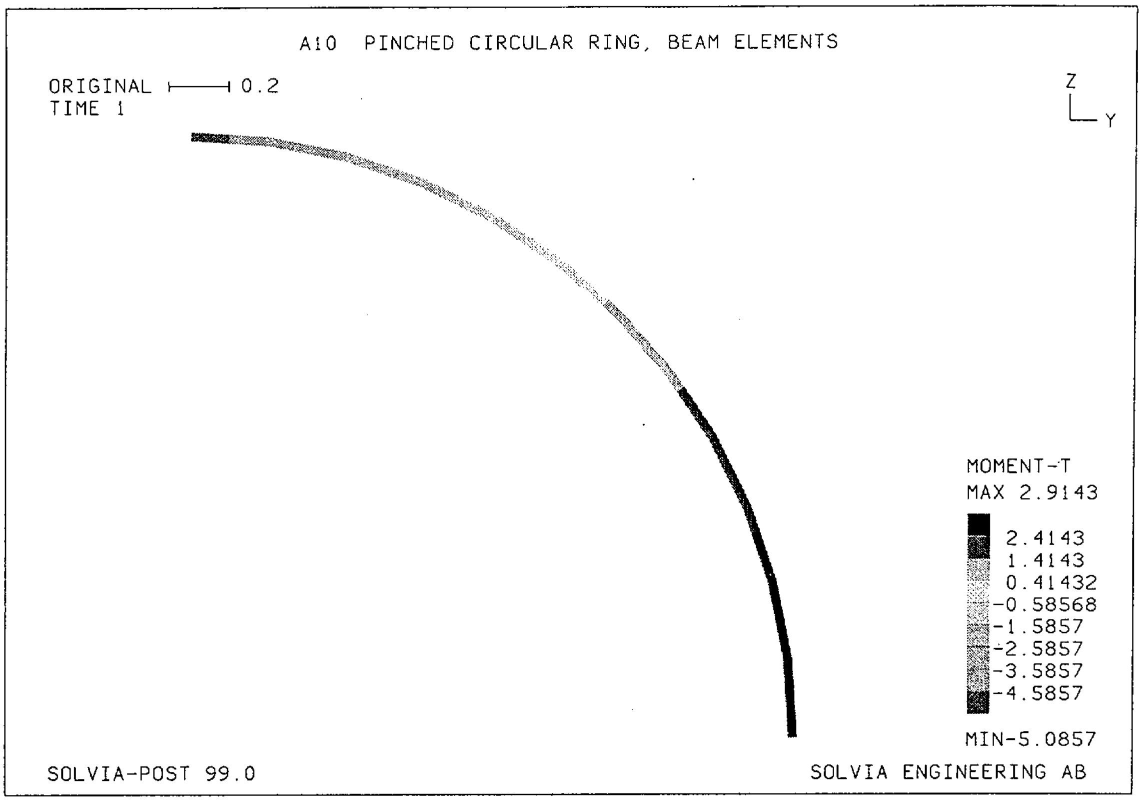
User Hints

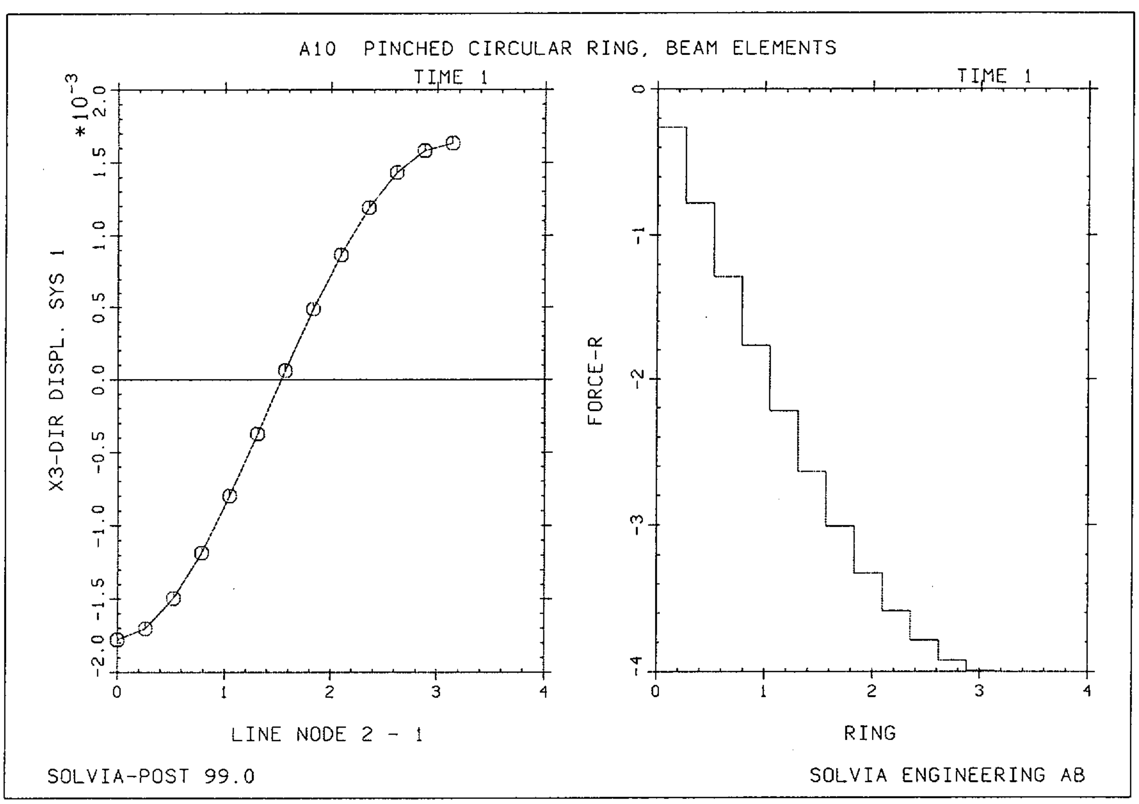
- The BEAM element is straight and each element has a cubic displacement assumption for bending and a linear displacement assumption for axial displacements. Hence, several elements are required to model a curved structure. The linear stiffness matrix is evaluated in closed form which is effective, and even when many elements are used it results in a relatively low cost of analysis.

- The axial force shown in the top figure on page A10.3 refers to the element coordinate system. Discontinuities, therefore, occur between elements due to the different orientation of the elements.






A10 PINCHED CIRCULAR RING, BEAM ELEMENTS
ORIGINAL 0.2
TIME 1
Z
Y
MOMENT-T
MAX 2.9143
2.4143
1.4143
0.41432
-0.58568
-1.5857
-2.5857
-3.5857
-4.5857
MIN-5.0857
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A10 PINCHED CIRCULAR RING, BEAM ELEMENTS
TIME 1
X3-DIR DISPL. SYS 1
*10^-3
LINE NODE 2 - 1
TIME 1
FORCE-R
RING
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input:

```
HEAD  'A10  PINCHED CIRCULAR RING, BEAM ELEMENTS'
*
DATABASE  CREATE
*
MASTER  IDOF=100011
COORDINATES
 1  0. 2.  /  2  0. 0. 2.  /  3  0.
*
LINE  ARC  N1=1 N2=2 NCENTER=3 EL=12
*
MATERIAL  1  ELASTIC E=2.E11 NU=0.3
*
EGROUP  1  BEAM RESULT=FORCES
SECTION  1  RECTANGULAR WTOP=0.02 D=0.02
GLINE  1 2 3  EL=12
*
FIXBOUNDARIES  34  /  1
FIXBOUNDARIES  24  /  2
FIXBOUNDARIES      /  3
*
LOADS  CONCENTRATED
 2 3 -4
*
SET  VIEW=X PLOTORIENTATION=PORTRAIT
MESH  NNUMBER=YES NSYMBOL=YES EAXES=RST BCODE=ALL
MESH  ENUMBER=YES VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input:

```
*  A10  PINCHED CIRCULAR RING, BEAM ELEMENTS
*
DATABASE  CREATE
SYSTEM 1 CYLINDRICAL
*
WRITE  FILENAME='a10.lis'
*
MESH  VIEW=X CONTOUR=MT ORIGINAL=YES DEFORMED=NO
*
NPLINE  NAME=LINE  /  2  111 STEP -1 TO 101  1
EPLINE  NAME=RING  /  12  2 1  TO  1  2 1
*
SUBFRAME  21
NLINE  LINENAME=LINE DIRECTION=3 SYMBOL=1 OUTPUT=ALL SYSTEM=1
ELINE  LINENAME=RING KIND=MT OUTPUT=LIST
ELINE  LINENAME=RING KIND=FR OUTPUT=ALL
END
```

EXAMPLE A11

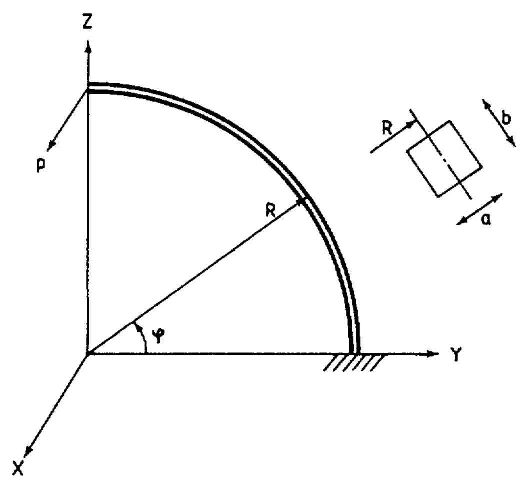
CURVED BEAM UNDER OUT-OF-PLANE LOAD, ISOBEAM ELEMENTS

Objective

To verify the three-dimensional action of curved ISOBEAM elements including torsion.

Physical Problem

The figure below shows the curved beam to be analyzed. The beam is relatively slender and fixed at one end. The other end is loaded by a concentrated force in the out-of-plane direction.



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

$\nu = 0.3$

$R = 1.0$ m

$a = 0.02$ m

$b = 0.02$ m

$P = 100$ N

Finite Element Model

The figure on page A11.3 shows the finite element model. Six 3-node ISOBEAM elements are used to approximate the 90-degree circular bend.

Solution Results

The theoretical solution is given in [1] p. 412.

Tip displacement in the direction of force:

$$\delta_x = \frac{PR^3}{EI_s}\left(\frac{\pi}{4} + \frac{EI_s}{C}\left(\frac{3\pi}{4} - 2\right)\right)$$

where $C = 0.141 \cdot a^4 \cdot G$ ([1] p. 290)

Torsional moment M_r and bending moment M_s:

$$|M_r| = PR(1 - \sin\phi)$$

$$|M_s| = PR \cdot \cos\phi$$

A comparison with results from SOLVIA using the input data on page A11.5 yields:

	Theory	SOLVIA
δ_x (m)	0.0500	0.0500

ϕ (degree)	M_r (Nm) Theory	M_r (Nm) SOLVIA	M_s (Nm) Theory	M_s (Nm) SOLVIA
0	100.0	99.9	100.0	100.1
15	74.1	74.1	96.6	96.6
30	50.0	50.0	86.6	86.6
45	29.3	29.3	70.7	70.7
60	13.4	13.4	50.0	50.0
75	3.4	3.4	25.9	25.9
90	0	0	0	0

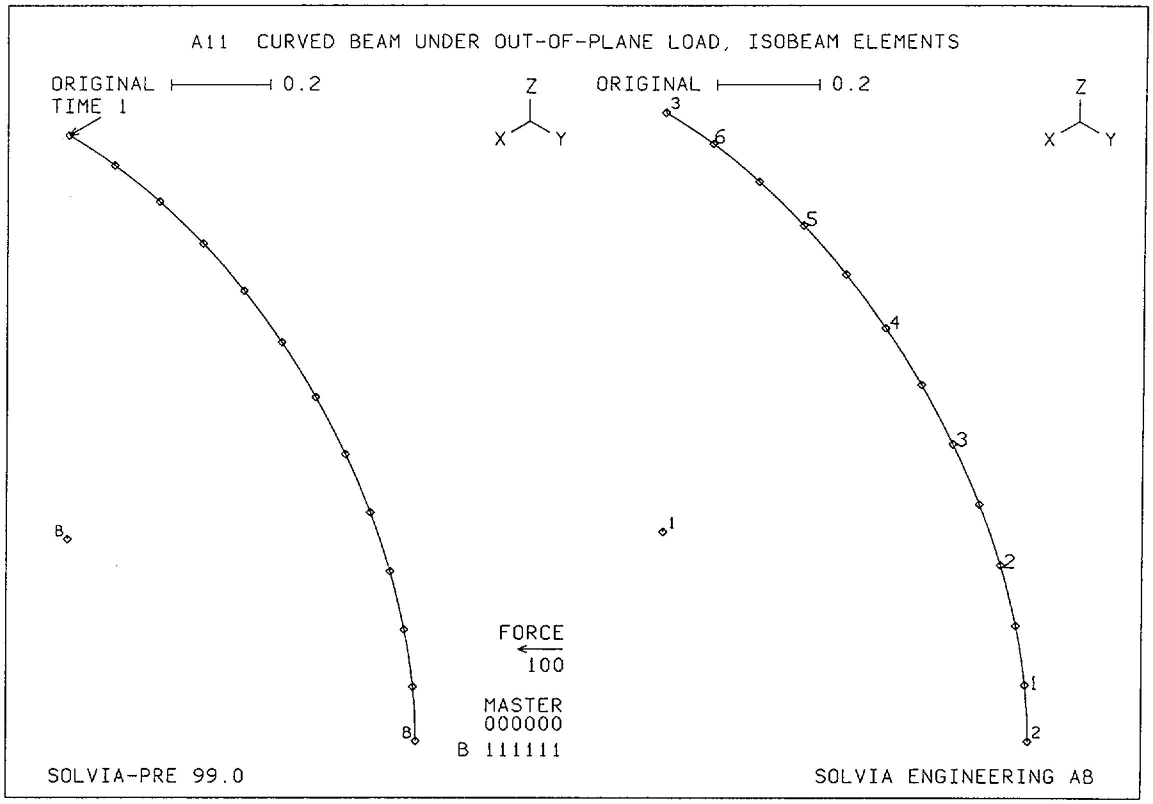
The deformed shape, contour plot of torsional moment and the variation of the moments M_r and M_s along the curved beam are shown in figures on page A11.4.

User Hints

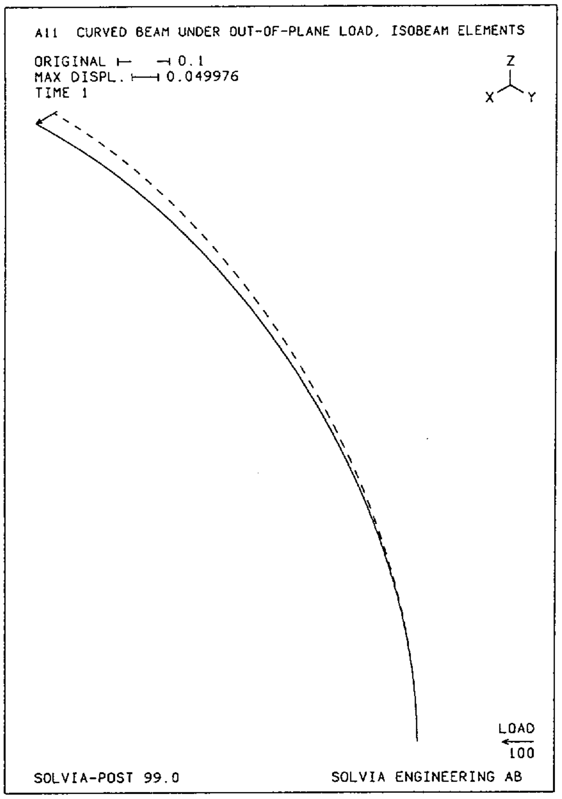
- Torsional effects are very important in this example. The ISOBEAM element contains special displacement assumptions for the torsional behaviour.
- The nodal forces and moments at an internal node of the ISOBEAM element are zero unless the node is acted upon by an externally applied force or moment. The forces and moments of the element end nodes are, however, those which balance the forces and moments of the adjoining elements and the applied loads, see Example A8.
- The constant curvature of the circular bend is only approximated by the 3-node (parabolic) ISOBEAM elements. Since six elements are used for the bend the solution still agrees very well with the exact solution.

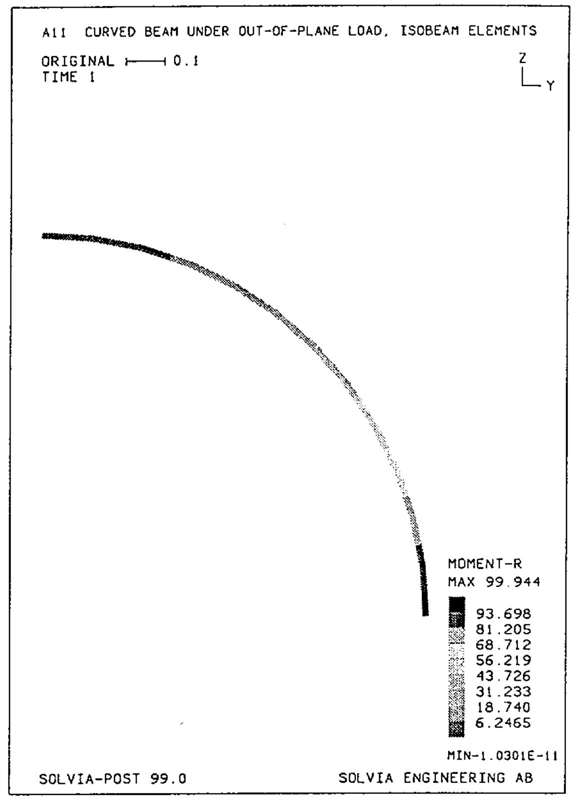
Reference

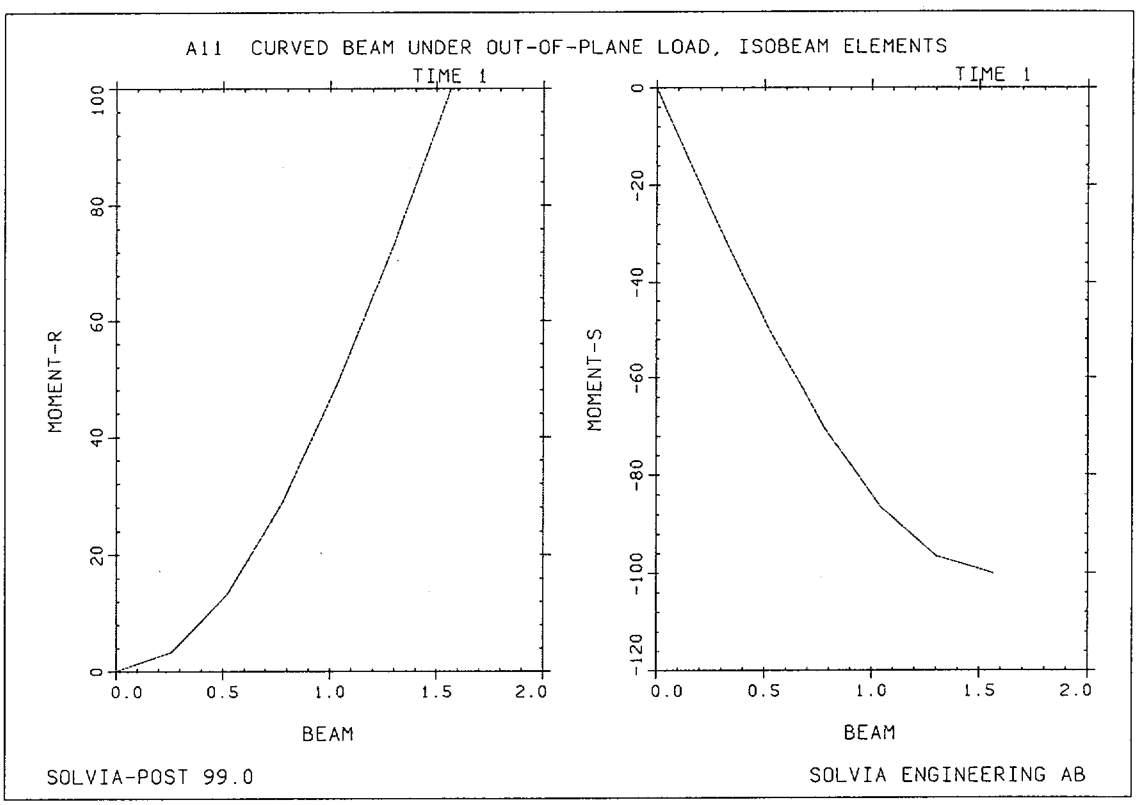
[1] Timoshenko, S., Strength of Materials, Part I, Elementary Theory and Problems, Third Edition, D. Van Nostrand, 1955.



A11 CURVED BEAM UNDER OUT-OF-PLANE LOAD, ISOBEAM ELEMENTS
ORIGINAL 0.2
TIME 1
Z
X Y
B
B
FORCE
100
MASTER
000000
B 111111
ORIGINAL 0.2
Z
X Y
3
6
5
4
3
1
2
1
2
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A11 CURVED BEAM UNDER OUT-OF-PLANE LOAD, ISOBEAM ELEMENTS
ORIGINAL 0.1
MAX DISPL. 0.049976
TIME 1
Z
X Y
LOAD
100
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A11 CURVED BEAM UNDER OUT-OF-PLANE LOAD, ISOBEAM ELEMENTS
ORIGINAL 0.1
TIME 1
Z
Y
MOMENT-R
MAX 99.944
93.698
81.205
68.712
56.219
43.726
31.233
18.740
6.2465
MIN-1.0301E-11
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A11 CURVED BEAM UNDER OUT-OF-PLANE LOAD, ISOBEAM ELEMENTS
TIME 1
MOMENT-R
0
20
40
60
80
100
0.0
0.5
1.0
1.5
2.0
BEAM
TIME 1
MOMENT-S
0
-20
-40
-60
-80
-100
-120
0.0
0.5
1.0
1.5
2.0
BEAM
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A11  CURVED BEAM UNDER OUT-OF-PLANE LOAD, ISOBEAM
ELEMENTS'
*
DATABASE  CREATE
*
SYSTEM  1  CYLINDRICAL
COORDINATES  /  ENTRIES  NODE  R  THETA
 1  /  2  1.  /  3  1. 90.
*
MATERIAL  1  ELASTIC  E=2.E11 NU=0.3
*
EGROUP  1  ISOBEAM  RESULTS=FORCES
SECTION  1  SDIM=0.02 TDIM=0.02
GLINE  N1=2 N2=3 AUX=1  EL=6 NODES=3 SYSTEM=1
*
FIXBOUNDARIES  /  1 2
*
LOADS  CONCENTRATED
 3 1 100.
*
SET  SMOOTHNESS=YES NSYMBOLS=YES
SUBFRAME  21
MESH  VECTOR=LOAD BCODE=ALL
MESH  ENUMBER=YES NNUMBERS=MYNODES
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A11  CURVED BEAM UNDER OUT-OF-PLANE LOAD, ISOBEAM ELEMENTS
*
DATABASE  CREATE
*
WRITE  FILENAME='a11.lis'
*
SET  PLOTORIENTATION=PORTRAIT
MESH  ORIGINAL=DASHED VECTOR=LOAD SMOOTHNESS=YES
MESH  CONTOUR=MR ORIGINAL=YES DEFORMED=NO VIEW=X
*
EPLINE  NAME=BEAM
 6  2 1  TO  1  2 1
*
SET  PLOTORIENTATION=LANDSCAPE
ELINE  LINENAME=BEAM KIND=MR OUTPUT=ALL SUBFRAME=21
ELINE  LINENAME=BEAM KIND=MS OUTPUT=ALL
*
NLIST  ZONENAME=N3 DIRECTION=156
NLIST  DIRECTION=156 KIND=REACTION
END
```

EXAMPLE A12

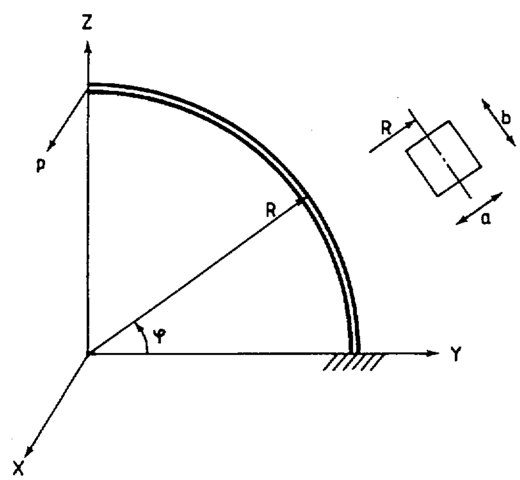
CURVED BEAM UNDER OUT-OF-PLANE LOAD, BEAM ELEMENTS

Objective

To verify the three-dimensional action of the BEAM element, including torsion, when applied to a curved structure.

Physical Problem

The curved beam extends 90° and is loaded by a concentrated force at one end and is fixed at the other end. The problem is the same as in Example A11, see figure below.



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

$\nu = 0.3$

$R = 1.0$ m

$a = 0.02$ m

$b = 0.02$ m

$P = 100$ N

Finite Element Model

The figure on page A12.2 shows the finite element model. Twelve BEAM elements are used.

Solution Results

The theoretical solution is given in Example A11. The following SOLVIA results are obtained using the input data on page A12.4.

Tip displacement (m) in the direction of the force:

Theory	SOLVIA
0.0500	0.0499

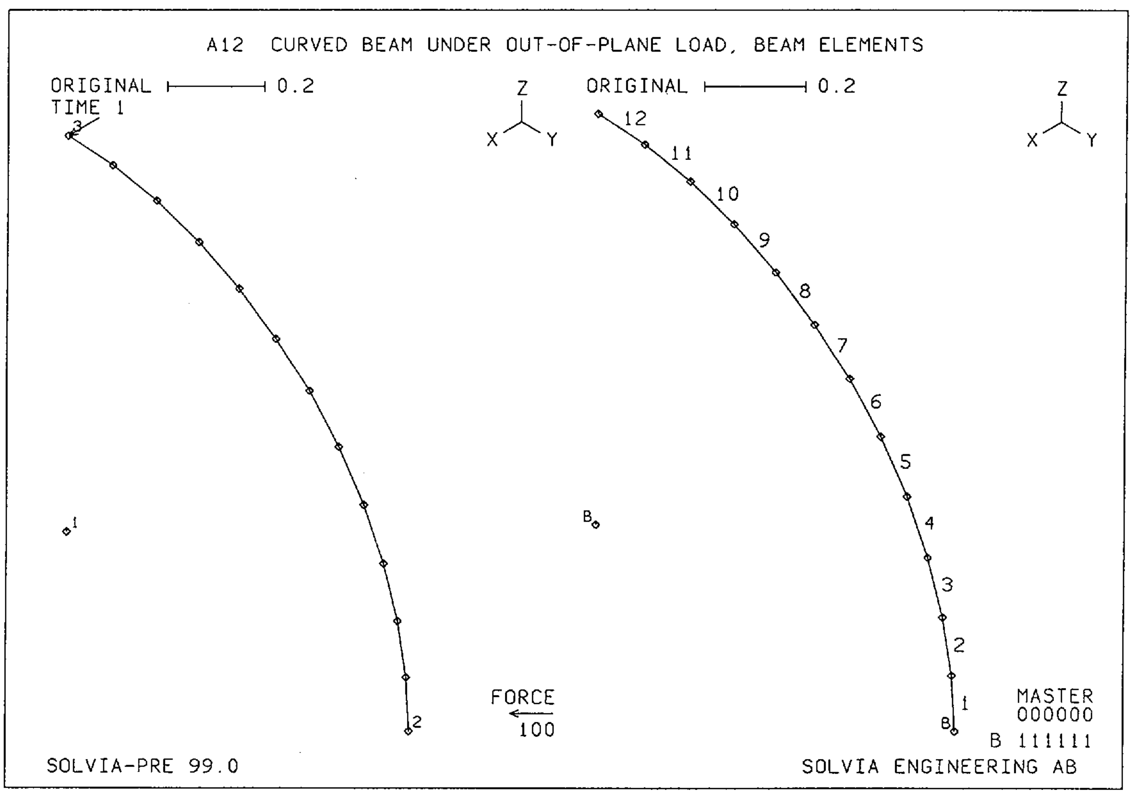
Moments (Nm) at the built-in end transformed to the Z-axis (torsion) and the Y-axis (bending):

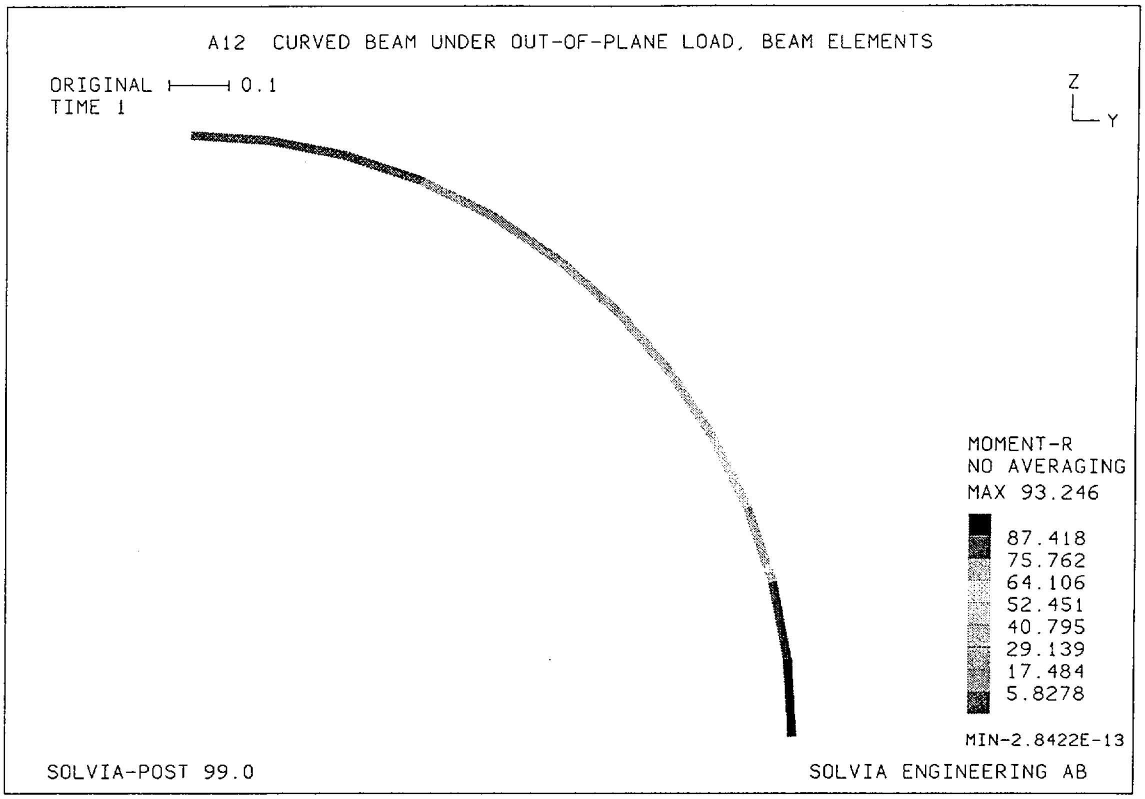
Torsional Moment		Bending Moment	
Theory	SOLVIA	Theory	SOLVIA
100.0	100.0	100.0	100.0

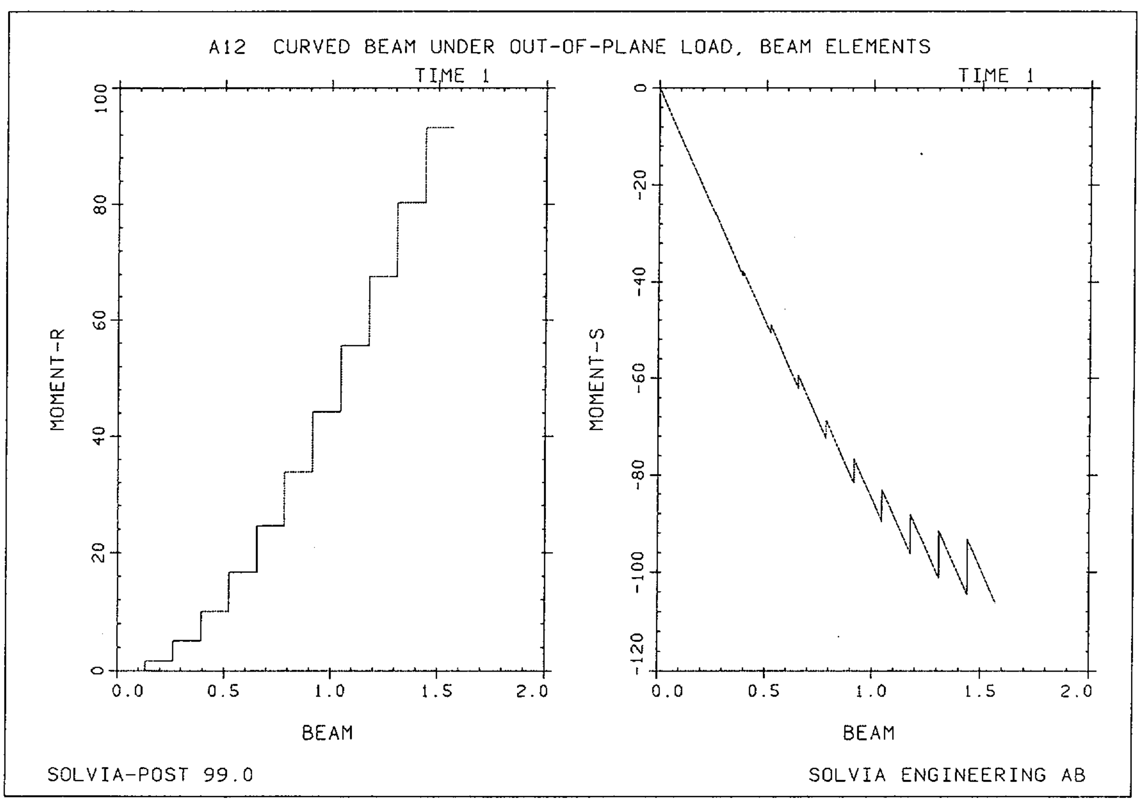
A contour plot of the torsional moment is shown in the top figure on page A12.3. The variation of the torsion and the bending moments are shown in the bottom figure on page A12.3.

User Hints

- The stiffness matrix of the linear BEAM element is evaluated in closed form, which is effective computationally. The element forces/moments or stresses are referred to the element coordinate system. Since the element is straight there are in general discontinuities in the variation of these quantities along BEAM elements modeling a curved structure.
- Note that the torsional moment in each BEAM element is constant. The bending moment distribution is then forced to be of saw tooth form to maintain equilibrium with the moment formed by the applied force.




A12 CURVED BEAM UNDER OUT-OF-PLANE LOAD, BEAM ELEMENTS
ORIGINAL 0.1
TIME 1
Z
Y
MOMENT-R
NO AVERAGING
MAX 93.246
87.418
75.762
64.106
52.451
40.795
29.139
17.484
5.8278
MIN-2.8422E-13
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A12 CURVED BEAM UNDER OUT-OF-PLANE LOAD, BEAM ELEMENTS
TIME 1
MOMENT-R
0
20
40
60
80
100
0.0
0.5
1.0
1.5
2.0
BEAM
TIME 1
MOMENT-S
0
-20
-40
-60
-80
-100
-120
0.0
0.5
1.0
1.5
2.0
BEAM
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A12  CURVED BEAM UNDER OUT-OF-PLANE LOAD, BEAM ELEMENTS'
*
DATABASE  CREATE
*
SYSTEM  1 CYLINDRICAL
COORDINATES  /  ENTRIES  NODE  R  THETA
 1  /  2  1.  /  3  1. 90.
*
MATERIAL  1  ELASTIC  E=2.E11 NU=0.3
*
EGROUP  1  BEAM RESULTS=FORCES
SECTION  1  RECTANGULAR WTOP=0.02 D=0.02
GLINE  2 3 1  EL=12 SYSTEM=1
*
FIXBOUNDARIES  /  1 2
*
LOADS  CONCENTRATED
 3 1 100.
*
SET  NSYMBOLS=YES
SUBFRAME  21
MESH  NNUMBERS=MYNODES VECTOR=LOAD
MESH  ENUMBER=YES BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A12  CURVED BEAM UNDER OUT-OF-PLANE LOAD, BEAM ELEMENTS
*
DATABASE  CREATE
*
WRITE  FILENAME='a12.lis'
*
CONTOUR  AVERAGE=NO
MESH  CONTOUR=MR ORIGINAL=YES DEFORMED=NO VIEW=X
*
EPLINE  NAME=BEAM
 12  2 1  TO  1  2 1
*
ELINE  LINENAME=BEAM KIND=MR OUTPUT=ALL SUBFRAME=21
ELINE  LINENAME=BEAM KIND=MS OUTPUT=ALL
*
NLIST  ZONENAME=N3 DIRECTION=156
NLIST  DIRECTION=156 KIND=REACTION
END
```

EXAMPLE A13

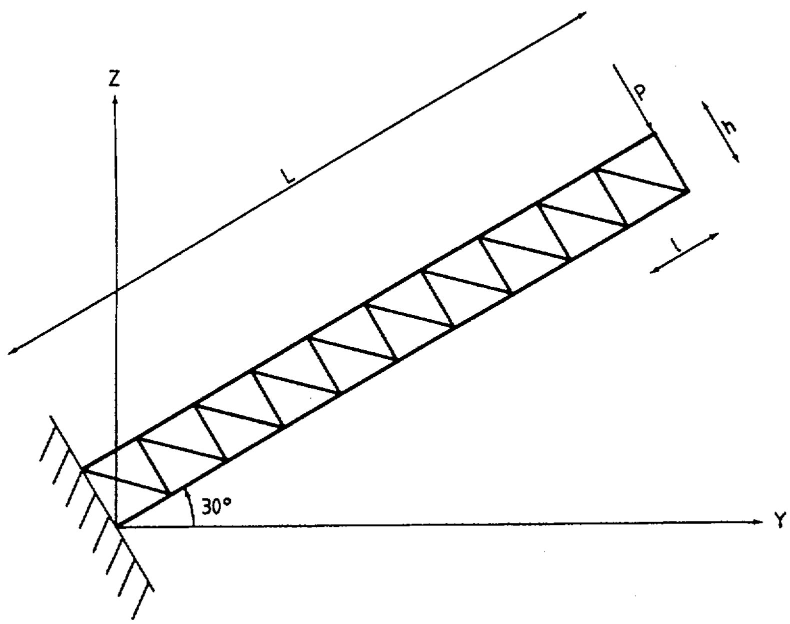
CANTILEVER TRUSS STRUCTURE UNDER CONCENTRATED LOAD

Objective

To verify the TRUSS element when used with SKEW degree-of-freedom directions.

Physical Problem

A cantilever truss structure under concentrated end load is considered as shown in the figure below.



$L = 5.000$ m

$h = 0.500$ m

$l = 0.500$ m

$A = 0.0001\,m^2$

$P = 1000$ N

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

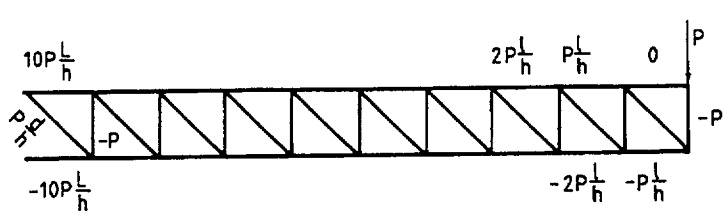
$d = \sqrt{h^2 + l^2}$

Finite Element Model

The truss structure is modelled with 2-node TRUSS elements as shown in the figures on page A13.3. The structure is inclined in the global coordinate system so it is convenient to employ SKEW degree-of-freedom directions.

Solution Results

The structure is statically determinate and the internal forces are as shown in the figure below.



The displacement of the point of load application (node 22) in the direction of the cantilever is:

$$\delta = \sum_{i=1}^{9} \frac{iPl^2}{hAE} = 1.125 \cdot 10^{-3}(m)$$

The SOLVIA numerical solution obtained using input data on pages A13.5 and A13.6 is as follows:

Internal element forces (N):

Element	Theory	SOLVIA
1	-10000.00	-10000.00
11	9000.00	9000.00
21	1414.21	1414.21

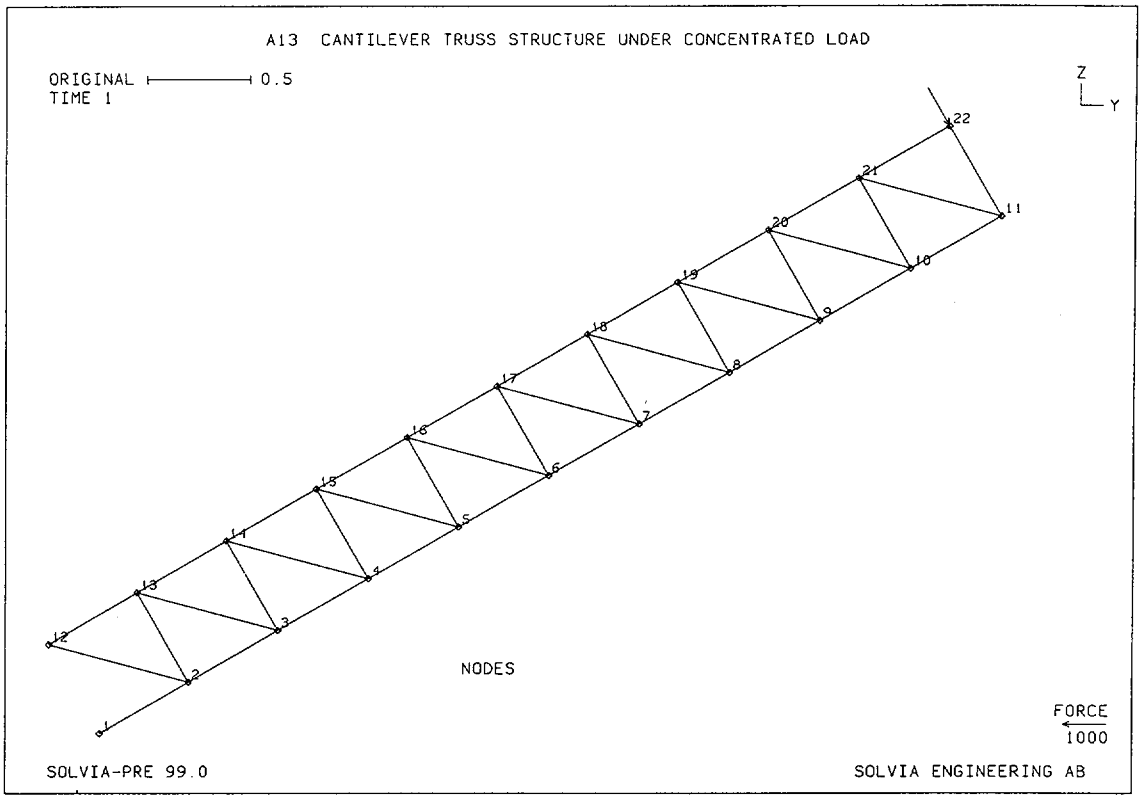
Displacement δ (mm):

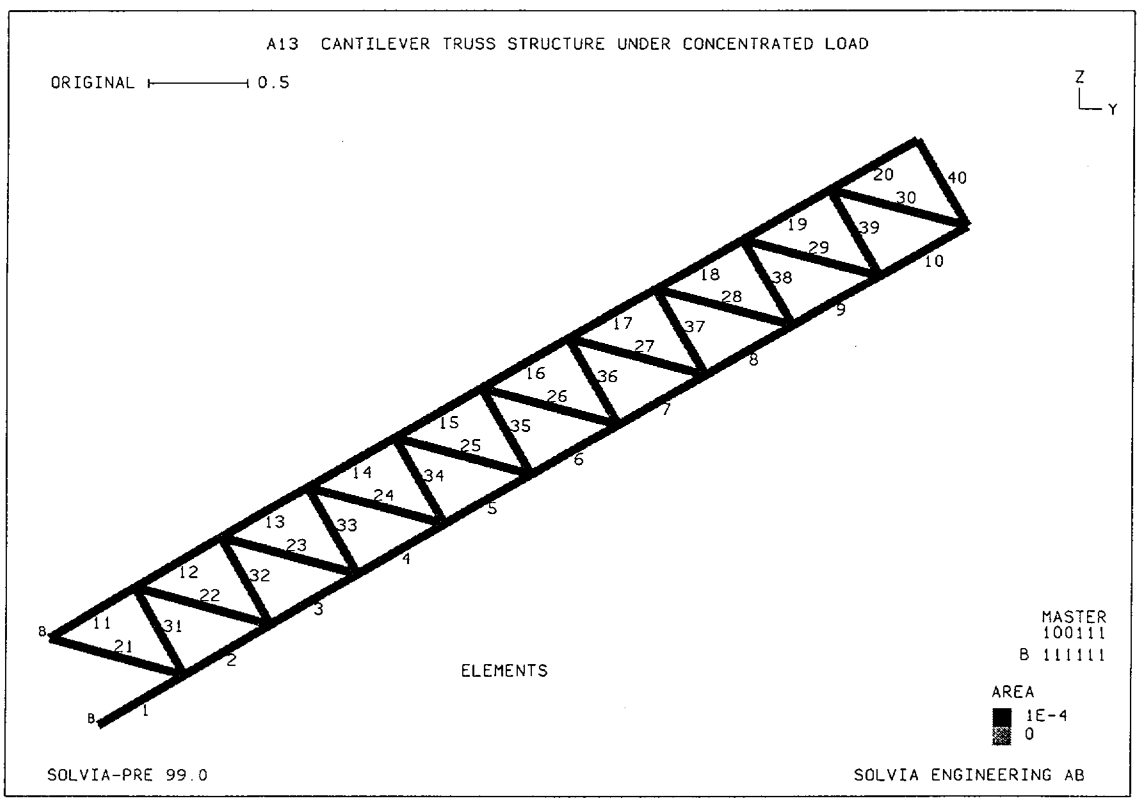
Theory	SOLVIA
1.12500	1.12500

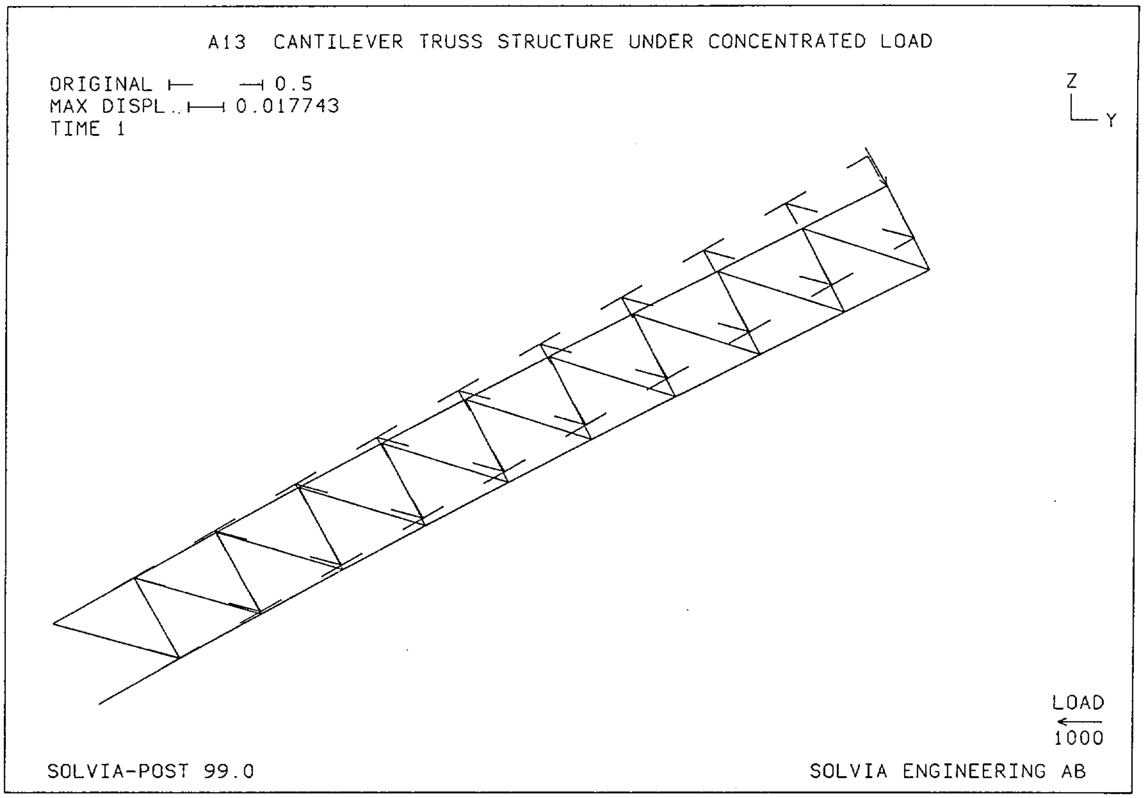
The deformed mesh is shown in the top figure on page A13.4. A contour plot of the internal element forces, R-force, is shown in the bottom figure on page A13.4.

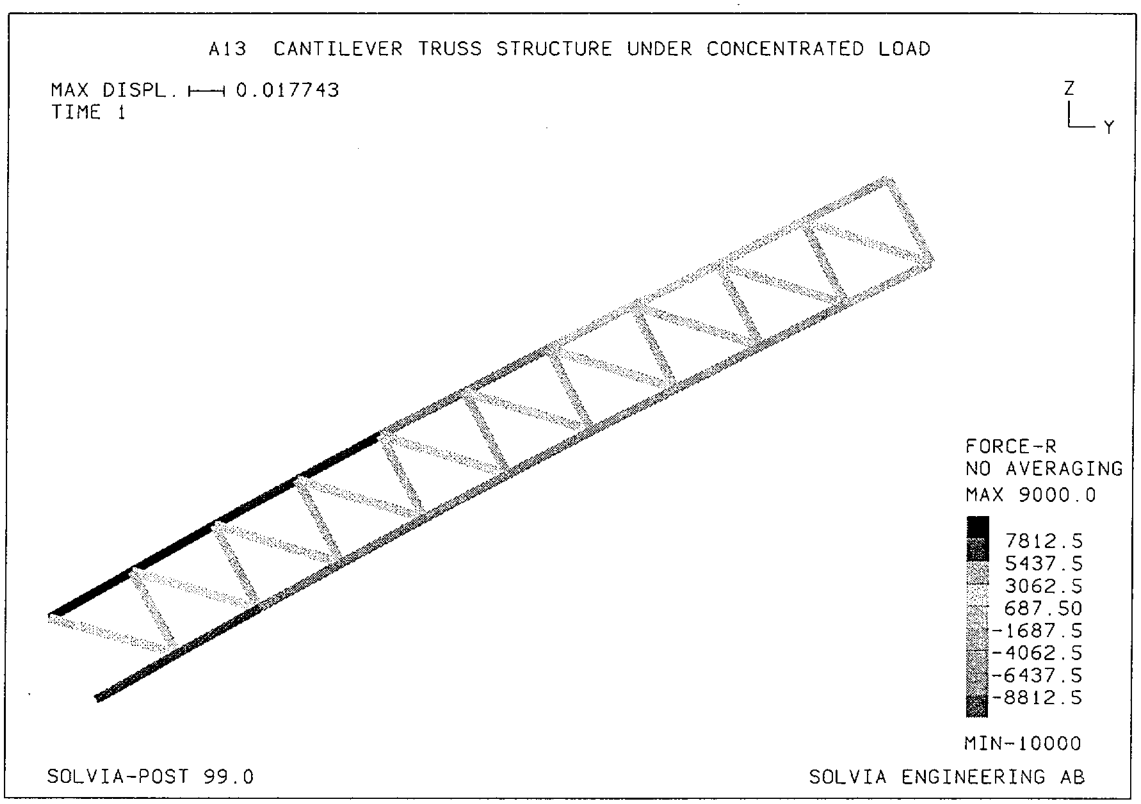
User Hints

- The SKEW degree-of-freedom System used for each node is oriented in the principal directions of the cantilever. The input concentrated force and all nodal results are then referred to this coordinate system.


A13 CANTILEVER TRUSS STRUCTURE UNDER CONCENTRATED LOAD
ORIGINAL 0.5
TIME 1
Z
Y
NODES
FORCE
1000
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A13 CANTILEVER TRUSS STRUCTURE UNDER CONCENTRATED LOAD
ORIGINAL 0.5
Z
Y
ELEMENTS
MASTER
100111
B 111111
AREA
1E-4
0
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A13 CANTILEVER TRUSS STRUCTURE UNDER CONCENTRATED LOAD
ORIGINAL 0.5
MAX DISPL. 0.017743
TIME 1
Z
Y
LOAD
1000
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A13 CANTILEVER TRUSS STRUCTURE UNDER CONCENTRATED LOAD
MAX DISPL. 0.017743
TIME 1
Z
Y
FORCE-R
NO AVERAGING
MAX 9000.0
7812.5
5437.5
3062.5
687.50
-1687.5
-4062.5
-6437.5
-8812.5
MIN-10000
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A13  CANTILEVER TRUSS STRUCTURE UNDER CONCENTRATED LOAD'
*
DATABASE  CREATE
*
MASTER  IDOF=100111
SKEWSYSTEMS  EULERANGLES
 1 30
SYSTEM  1  CARTESIAN  PHI=30
COORDINATES
 ENTRIES NODE   YL   ZL
            1   0.   0.    TO
           11   5.   0.
           12   0.   0.5   TO
           22   5.   0.5
NSKEWS
  1 1  TO  22 1
*
MATERIAL  1  ELASTIC E=2.E11 NU=0.
*
EGROUP  1 TRUSS
ENODES
 ENTRIES  EL   N1   N2
           1    1    2   TO
          10   10   11
          11   12   13   TO
          20   21   22
          21   12    2   TO
          30   21   11
          31   13    2   TO
          40   22   11
EDATA
 ENTRIES EL   AREA
          1  .0001
*
LOADS  CONCENTRATED
  22 3 -1000
*
FIXBOUNDARIES  /  1 12
*
SET  VIEW=X HEIGHT=0.25
MESH  NNUMBERS=YES NSYMBOLS=YES VECTOR=LOAD
TEXT  STRING='NODES' XPT=10. YPT=2.
MESH  ENUMBERS=YES BCODE=ALL CONTOUR=AREA
TEXT  STRING='ELEMENTS' XPT=10. YPT=2.
*
SOLVIA
END
```

SOLVIA-POST input

```
*   A13   CANTILEVER TRUSS STRUCTURE UNDER CONCENTRATED LOAD
*
DATABASE CREATE
*
WRITE   FILENAME='a13.lis'
*
SET   VIEW=X
MESH   ORIGINAL=DASHED VECTOR=LOAD
*
CONTOUR   AVERAGE=NO
MESH   CONTOUR=FR
*
NLIST   ZONENAME=N22
NLIST   KIND=REACTIONS
ELIST
END
```

EXAMPLE A14

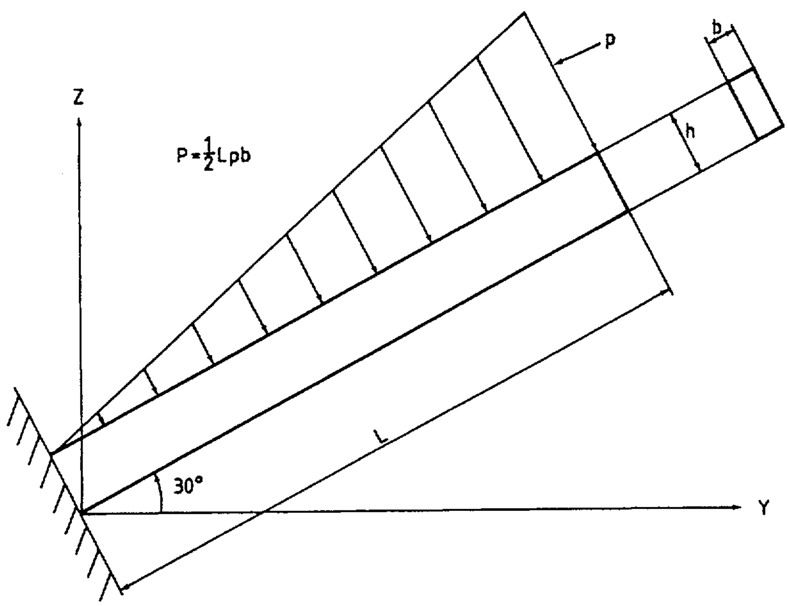
CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, PLANE STRESS

Objective

To verify the PLANE STRESS element subjected to distributed loading and when employing SKEW degree of freedom systems.

Physical Problem:

A cantilever beam of rectangular cross-section is considered when subjected to a triangular distributed load, see figure below.



$L = 1.000$ m $\qquad p = 0.200 \cdot 10^6$ N/m^2

$h = 0.100$ m $E = 2.0 \cdot 10^{11}$ N/m^2

$b = 0.050$ m $\nu = 0.3$

Finite Element Model

The figures on page A14.3 shows the finite element model. The upper face of the elements is acted upon by a linearly varying pressure load. The model is inclined in the Global System.

Solution Results

Using beam theory the theoretical solution for the end displacement is:

$$\delta = -\tfrac{11}{60}\frac{PL^3}{EI} - \tfrac{5}{6}\frac{PL}{A_s G}$$

The input data, as given on pages A14.5 and A14.6 yields the following solution:

End displacement (mm):

Theory	SOLVIA
-1.113	-1.110

The axial stress (N / m^2, in the direction of the beam) for element 10 and node 3 is:

Theory	SOLVIA
$40.00 \cdot 10^6$	$39.77 \cdot 10^6$

User Hints

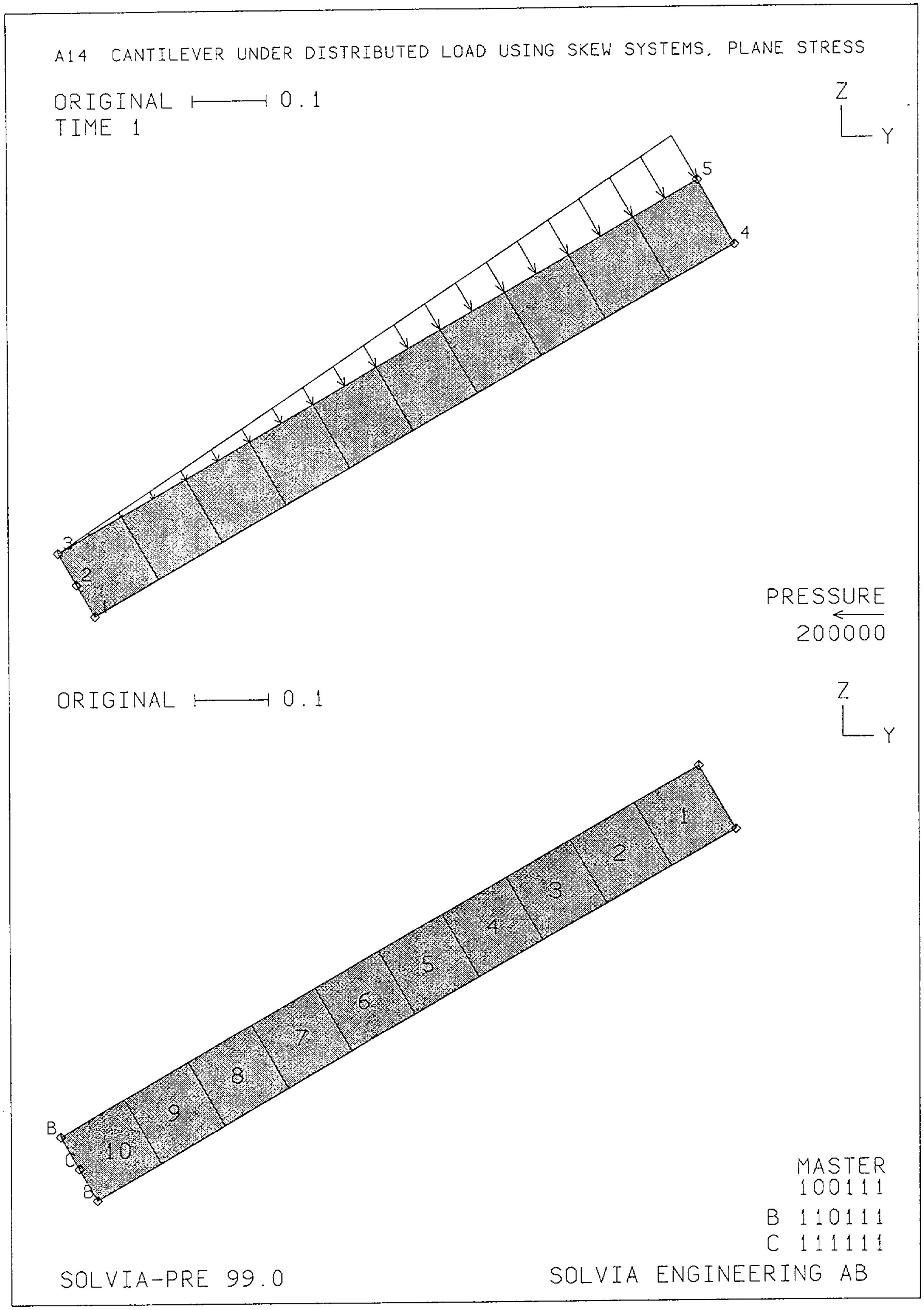
- The exact displacement solution according to beam theory for a linearly varying distributed load contains the 5-th power of the coordinate. Several 8-node (parabolic) PLANE elements are, therefore, used to obtain a good approximation to the analytical solution.

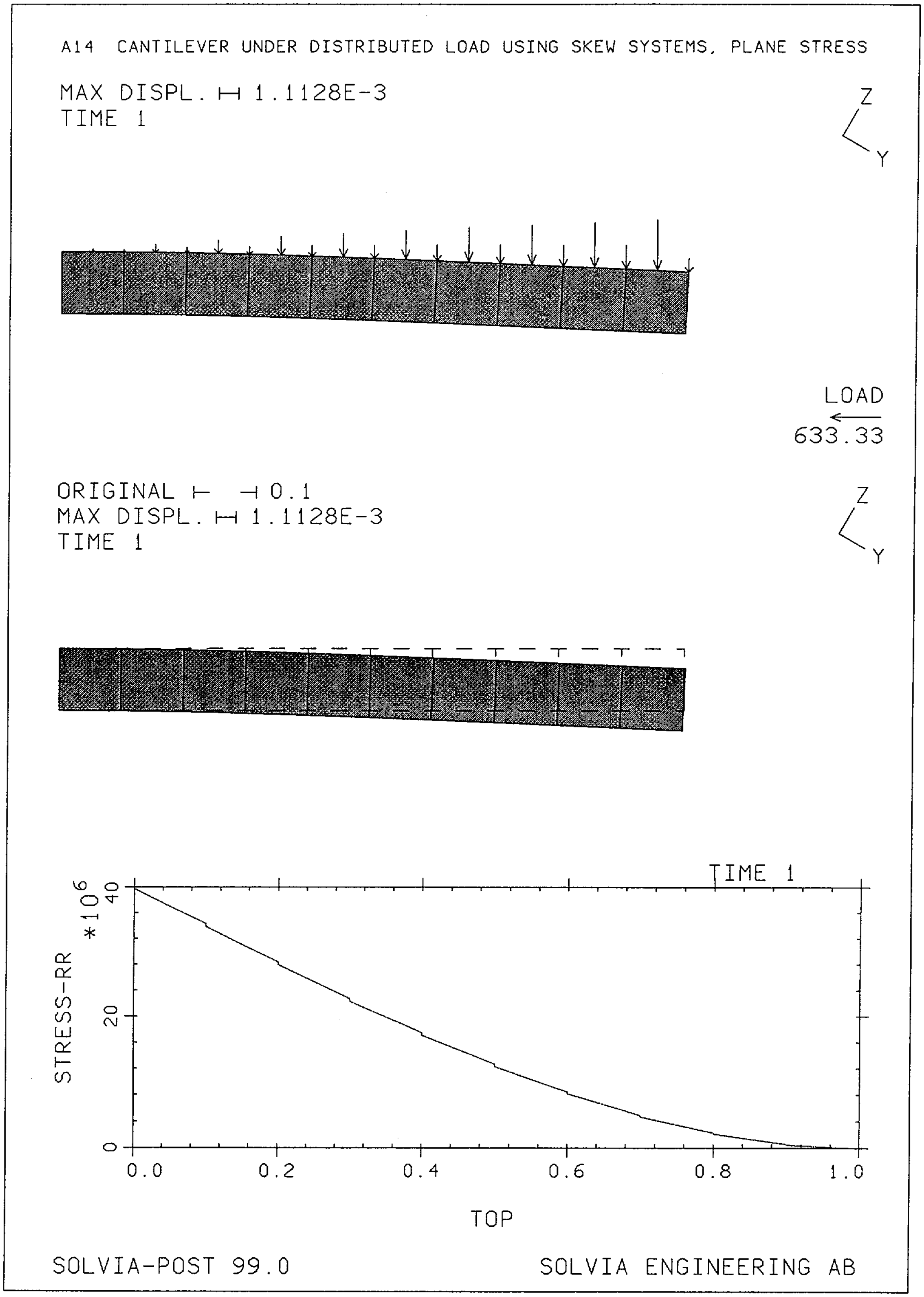
- For the PLANE STRESS 2 element the stress results are calculated in the global Y and Z directions. By specifying STRESSREFERENCE = ELEMENT in the DATABASE command in SOLVIA-POST the program transforms element stress results to the Element Stress System for PLANE and SOLID elements [1]. Alternatively, the stresses could be referenced to a Local System defined by the SYSTEM command, see example A15.

- Note that the summation of reaction forces and applied loads in SOLVIA-POST is made in the global coordinate system, i.e., nodal forces referring to skew systems are transformed to the global coordinate system before summation.

Reference

[1] SOLVIA-POST 99.0 Users Manual, Report SE 99-1, pp. 5.1-5.6.


A14 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, PLANE STRESS
ORIGINAL 0.1
TIME 1
Z
Y
5
4
3
2
1
PRESSURE
200000
ORIGINAL 0.1
Z
Y
1
2
3
4
5
6
7
8
9
10
B
C
B
MASTER
100111
B 110111
C 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A14 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, PLANE STRESS
MAX DISPL. 1.1128E-3
TIME 1
Z
Y
LOAD
633.33
ORIGINAL 0.1
MAX DISPL. 1.1128E-3
TIME 1
Z
Y
TIME 1
STRESS-RR *10^6
0
20
40
0.0
0.2
0.4
0.6
0.8
1.0
TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD 'A14  CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS,
PLANE STRESS'
*
DATABASE  CREATE
*
MASTER  IDOF=100111
*
SET  MYNODES=10
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
*
EGROUP  1  PLANE STRESS2  RESULTS=NSTRESSES
GSURFACE  5 3 1 4  EL1=10 EL2=1 NODES=8
EDATA
 ENTRIES EL THICK
          1  0.05
LOADS ELEMENT  INPUT=LINE
 3 5  0. 2.E5
*
FIXBOUNDARIES  23 /  2
FIXBOUNDARIES  2  /  1 3
*
NSKEWS  INPUT=SURFACE
 5 3 1 4  1
*
SET  PLOTORIENTATION=PORTRAIT NSYMBOLS=MYNODES
SUBFRAME  12
MESH  NNUMBERS=MYNODES VECTOR=LOAD
MESH  ENUMBERS=YES BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A14  CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, PLANE
STRESS
*
DATABASE  CREATE  STRESSREFERENCE=ELEMENT
WRITE  FILENAME='a14.lis'
*
SET  PLOTORIENTATION=PORTRAIT
VIEW  ID=1 XVIEW=1 ROTATION=-30
SET  VIEW=1
*
SUBFRAME  13
MESH  VECTOR=LOAD
MESH  ORIGINAL=DASHED GSCALE=OLD
*
EPLINE  NAME=TOP
 10  2 5 1  TO  1  2 5 1
ELINE  LINENAME=TOP KIND=SRR OUTPUT=ALL
*
SUMMATION  KIND=LOAD
SUMMATION  KIND=REACTION DETAILS=YES
*
NLIST  ZONENAME=EL1
EMAX  SELECT=S-EFFECTIVE
END
```

EXAMPLE A15

CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, PLANE STRAIN

Objective

To verify the PLANE STRAIN element subjected to distributed loading and when employing a SKEW degree-of-freedom System.

Physical Problem

Same as in Example A14, see figure on page A14-1.

Finite Element Model

Same as in Example A14 except that PLANE STRAIN elements are used, see figure on page A15.2.

Solution Results

To obtain the same theoretical solution as for the plane stress case of Example A14 the following material data is used:

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

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

where E and ν are the Young's modulus and Poisson's ratio used in Example A14.

The input data on page A15.4 gives the following solutions:

End displacement (mm):

Theory	SOLVIA
-1.113	-1.110

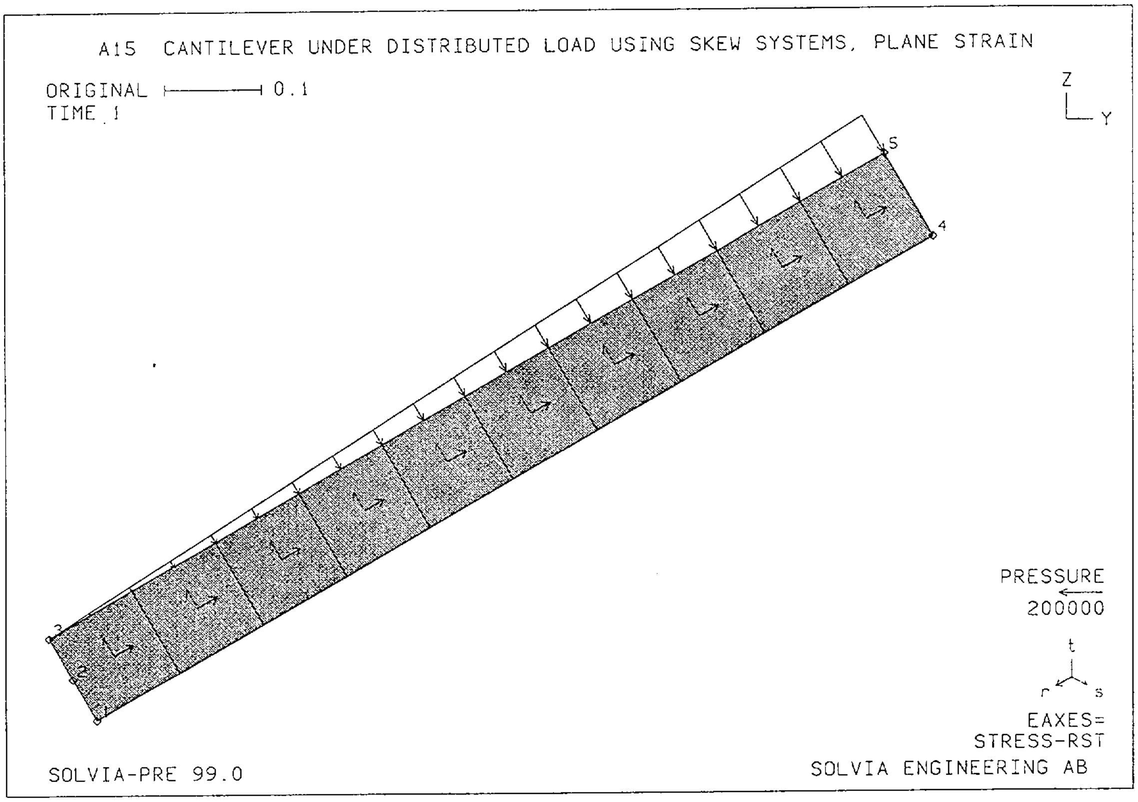
The axial stress (N/m^2 in the direction of the beam) for element 10 at node 3 is:

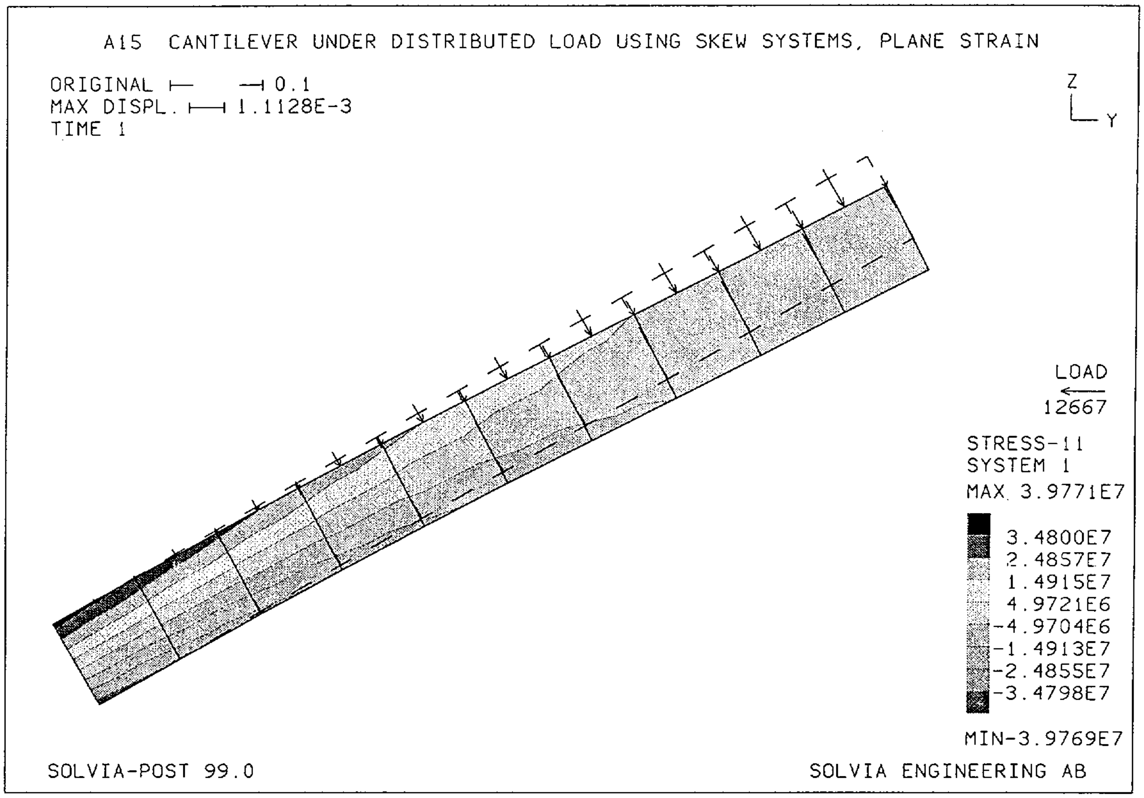
Theory	SOLVIA
$40.00 \cdot 10^6$	$39.77 \cdot 10^6$

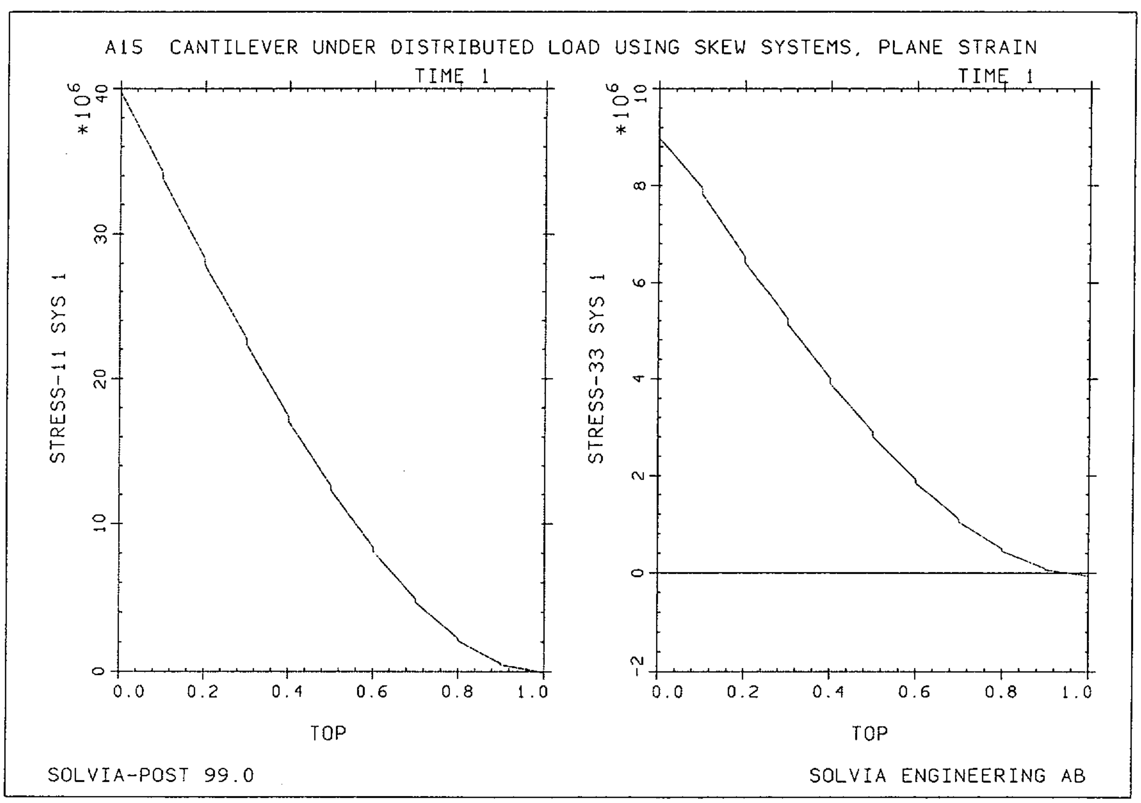
Since the PLANE STRAIN element in SOLVIA gives stress results in the Global System, a Local System with the x_1-direction oriented in the axial direction was defined using the SYSTEM command. Alternatively, SOLVIA-POST could be employed to transform the global stresses to the Element Stress System.

User Hints

- Note that the thickness of the PLANE STRAIN element is always equal to unity. The pressure is, therefore, here acting on a 20 times wider cantilever than in Example A14, giving a factor of 20 times larger reaction forces than for Example A14.




A15 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, PLANE STRAIN
ORIGINAL 0.1
MAX DISPL. 1.1128E-3
TIME 1
Z
Y
LOAD
12667
STRESS-11
SYSTEM 1
MAX 3.9771E7
3.4800E7
2.4857E7
1.4915E7
4.9721E6
-4.9704E6
-1.4913E7
-2.4855E7
-3.4798E7
MIN-3.9769E7
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A15 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, PLANE STRAIN
TIME 1
*10^6
STRESS-11 SYS 1
TOP
TIME 1
*10^6
STRESS-33 SYS 1
TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A15  CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS,
 PLANE STRAIN'
*
DATABASE  CREATE
*
MASTER  IDOF=100111
SKEWSYSTEMS EULERANGLES
 1 30
SYSTEM 1 CARTESIAN  PHI=30
COORDINATES
 1  TO  3  0. 0. 0.1
 4  0. 1.  /  5  0. 1. 0.1
*
MATERIAL  1  ELASTIC E=1.89349E11 NU=0.230769
*
EGROUP  1  PLANE STRAIN  RESULTS=NSTRESSES
GSURFACE  5 3 1 4  EL1=10 EL2=1 NODES=8
LOADS ELEMENT  INPUT=LINE
 3 5  0. 2.E5
*
FIXBOUNDARIES  23 /  2
FIXBOUNDARIES  2  /  1 3
*
NSKEWS  INPUT=SURFACE
 5 3 1 4  1
*
SET  NSYMBOLS=MYNODES
MESH  NNUMBERS=MYNODES EAXES=STRESS-RST VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A15  CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS,
*       PLANE STRAIN
*
DATABASE  CREATE
SYSTEM 1 CARTESIAN NA=4 NB=3
*
WRITE  FILENAME='a15.lis'
*
MESH  ORIGINAL=DASHED VECTOR=LOAD CONTOUR=S11 SYSTEM=1
*
EPLINE  NAME=TOP
 10  2 5 1  TO  1  2 5 1
ELINE  LINENAME=TOP KIND=S11 OUTPUT=ALL SYSTEM=1 SUBFRAME=21
ELINE  LINENAME=TOP KIND=S33 SYSTEM=1
*
SUMMATION  KIND=LOAD
SUMMATION  KIND=REACTION DETAILS=YES
*
NLIST  ZONENAME=EL1
EMAX  SELECT=S-EFFECTIVE
END
```

EXAMPLE A16

CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, SOLID

Objective

To verify the SOLID element subjected to pressure loading and when employing SKEW degree-of-freedom Systems.

Physical Problem

Same problem as shown in Example A14, see figure on page A14-1.

Finite Element Model

The model is shown in the figure on page A16.2 and the top figure on page A16.3. The elements are 20-node SOLID elements which have a quadratic assumption for the displacements, also in the transverse direction of the cantilever.

Solution Results

The theoretical solution for the end displacement is the same as given in Example A14.

The input data shown on pages A16.4 and A16.5 gives the following results:

End displacement (mm):

Theory	SOLVIA
-1.113	-1.111

The axial stress (N/m^2, in the direction of the beam) for element 10 at node 2 is:

Theory	SOLVIA
$40.0 \cdot 10^6$	$39.8 \cdot 10^6$

Since SOLVIA gives the stresses referred to the Global System, SOLVIA-POST is employed to transform the global stress components to the Element Stress System by specifying STRESSREFERENCE = ELEMENT in the DATABASE command. Alternatively, the stresses could be referenced to a local system defined by the SYSTEM command, see example A15.

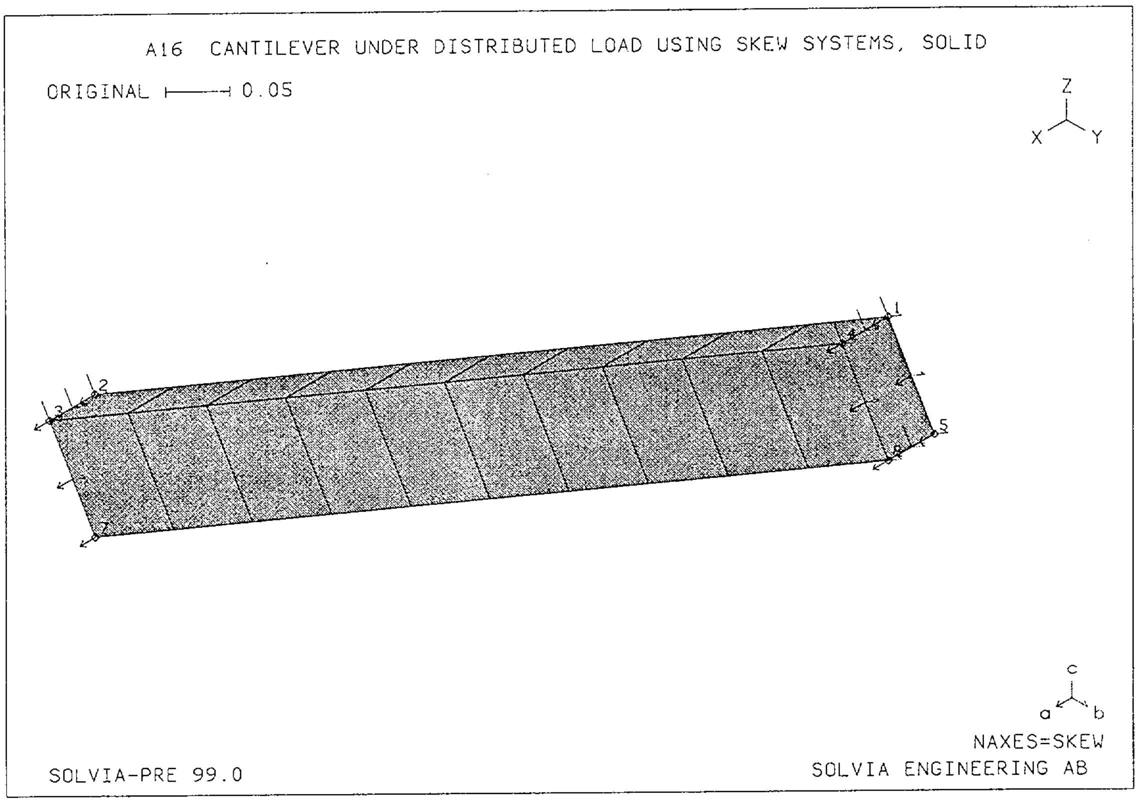
User Hints

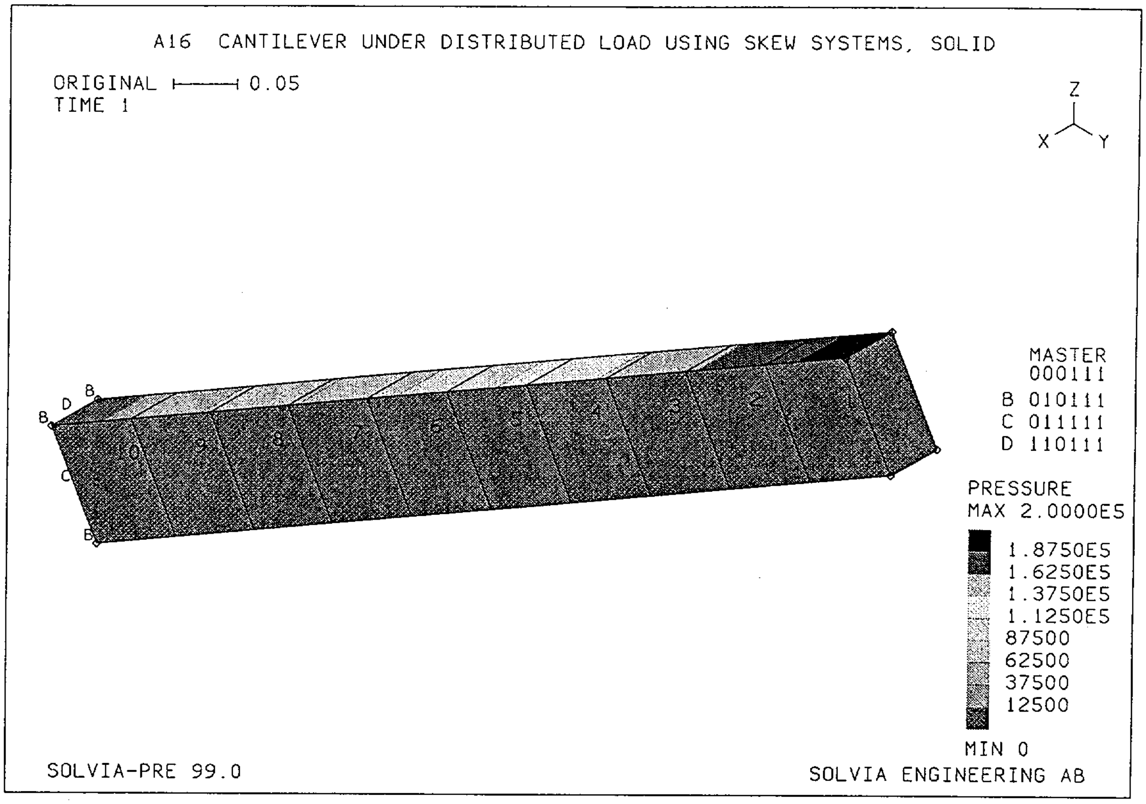
- The midside nodes in the transverse direction of the beam are included in order to model the anticlastic curvature. For a description of the phenomenon of anticlastic bending, see for example [1] p. 175.

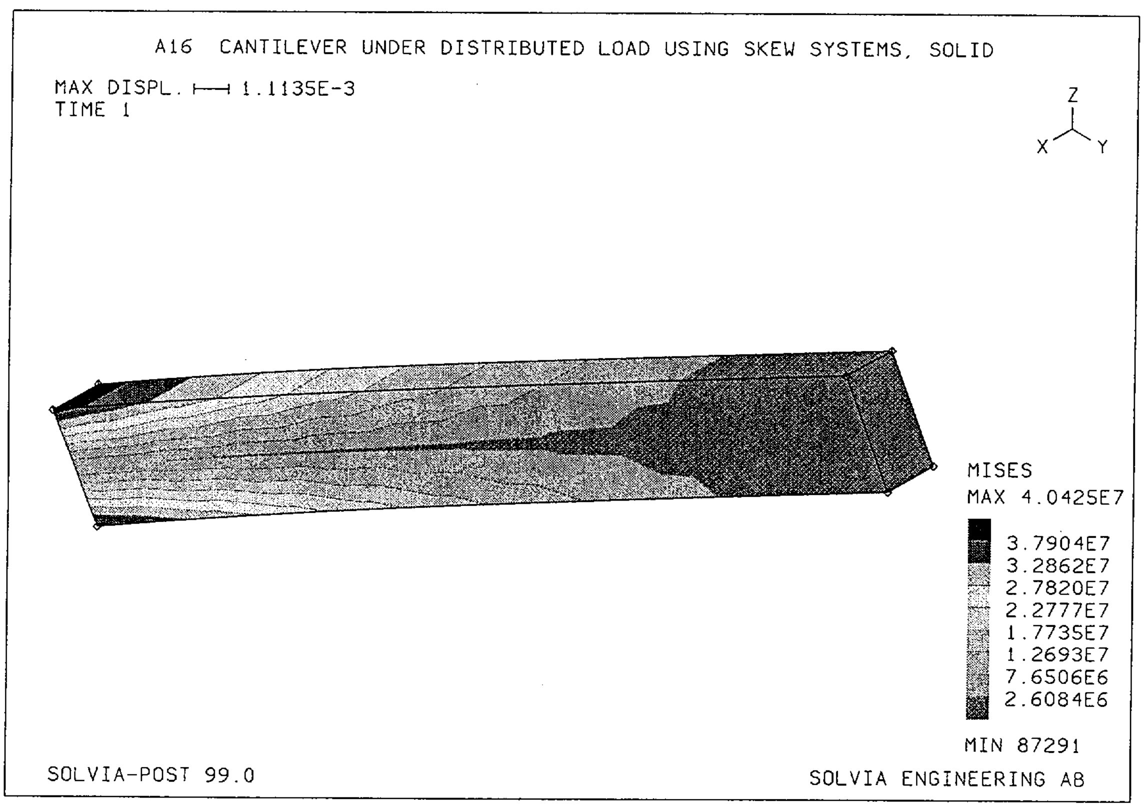
- If 2×2×2 integration is employed spurious (zero energy) modes are excited in the transverse direction. These spurious modes could be avoided by introducing additional fixed boundary conditions for the nodes at the built-in end of the beam. However, this change would also introduce additional stresses in the beam.

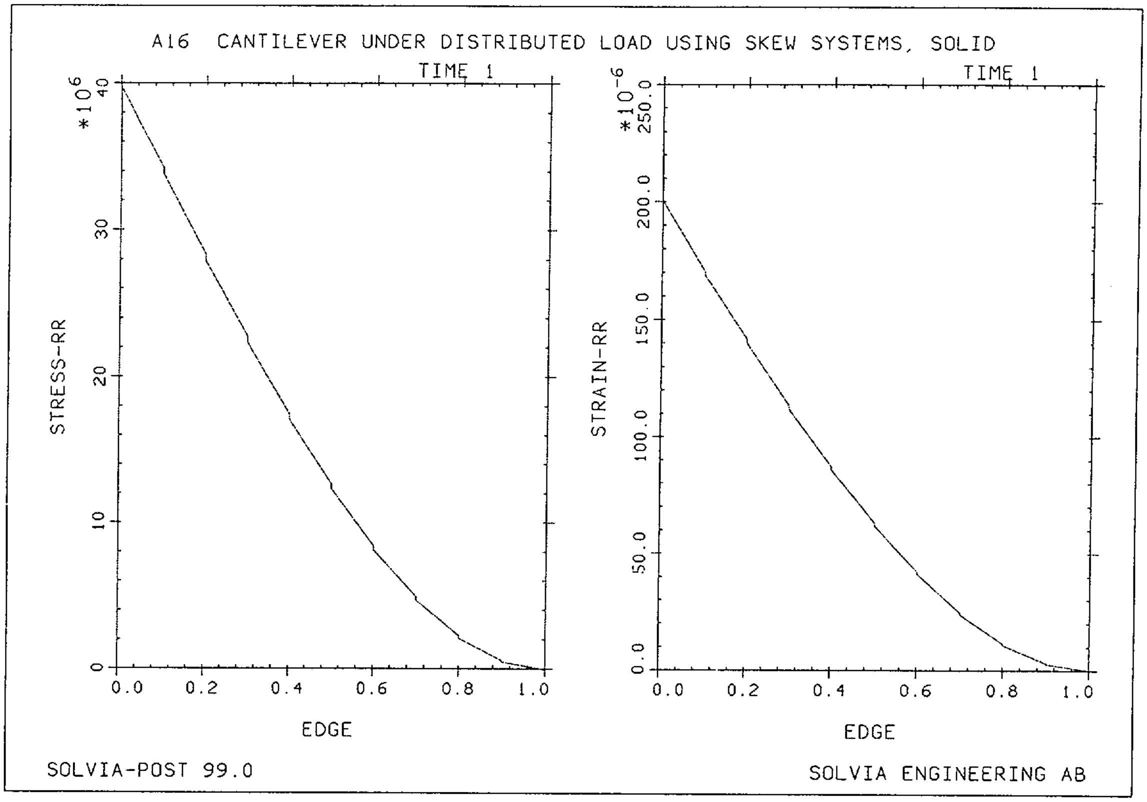
Reference

[1] Oden, J.T., Ripperger, E.A., Mechanics of Elastic Structures, Second Edition, McGraw-Hill, 1981.




A16 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, SOLID
ORIGINAL 0.05
TIME 1
Z
X Y
MASTER
000111
B 010111
C 011111
D 110111
PRESSURE
MAX 2.0000E5
1.8750E5
1.6250E5
1.3750E5
1.1250E5
87500
62500
37500
12500
MIN 0
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A16 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, SOLID
MAX DISPL. 1.1135E-3
TIME 1
Z
X Y
MISES
MAX 4.0425E7
3.7904E7
3.2862E7
2.7820E7
2.2777E7
1.7735E7
1.2693E7
7.6506E6
2.6084E6
MIN 87291
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB



SOLVIA-PRE input

```
HEAD  'A16  CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS,
 SOLID'
*
DATABASE  CREATE
*
MASTER  IDOF=000111
SYSTEM  1  CARTESIAN  PHI=30
COORDINATES
 1  -0.025 1. 0.1
 2  -0.025 0. 0.1
 3   0.025 0. 0.1
 4   0.025 1. 0.1
 5  -0.025 1. 0.
 6  -0.025 0. 0.
 7   0.025 0. 0.
 8   0.025 1. 0.
*
SKEWSYSTEMS  EULERANGLES
 1 30
*
MATERIAL  1  ELASTIC E=2.E11 NU=0.3
*
```

SOLVIA-PRE input (cont.)

```
EGROUP  1  SOLID  RESULTS=NSTRESSES
GVOLUME  1 2 3 4 5 6 7 8  EL1=10 EL2=1 EL3=1 NODES=20
LOADS  ELEMENT  INPUT=SURFACE
 2 3 4 1  0. 0. 2.E5 2.E5
*
NSKEWS  INPUT=SURFACE
 2 3 6 7  1
 1 4 5 8  1
*
FIXBOUNDARIES  123  INPUT=SURFACE  /  2 3 6 7
FREEBOUNDARIES   3  INPUT=LINES    /  2 3  /  6 7
FREEBOUNDARIES   1  INPUT=LINES    /  2 6  /  3 7
*
SET  NSYMBOLS=MYNODES
MESH  NNUMBERS=MYNODES NAXES=SKEW
MESH  CONTOUR=PRESSURE ENUMBERS=YES BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A16  CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, SOLID
*
DATABASE  CREATE STRESSREFERENCE=ELEMENT
*
WRITE  FILENAME='a16.lis'
*
MESH  CONTOUR=MISES NSYMBOLS=MYNODES OUTLINE=YES
*
EPLINE  NAME=EDGE
 10  2 9 1  TO  1  2 9 1
*
SUBFRAME  21
ELINE  LINENAME=EDGE KIND=SRR OUTPUT=ALL
ELINE  LINENAME=EDGE KIND=ERR
*
ZONE  NAME=TIPNODES INPUT=NODES
 1 4 5 8
*
NLIST  ZONENAME=TIPNODES
NLIST  KIND=REACTIONS
EMAX  NUMBER=3
END
```

EXAMPLE A17

CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, BEAM

Objective

To verify the BEAM element subjected to distributed loading and when employing skew coordinate systems.

Physical Problem

Same as in Example A14, see figure on page A14-1.

Finite Element Model

The model is shown in the top figure on page A17.2. Ten BEAM elements are used to model the cantilever.

Solution Results

The theoretical solution for the end displacement is the same as given in Example A14.

The input data on page A17.3 gives the following results:

End displacement (mm):

Theory	SOLVIA
-1.113	-1.110

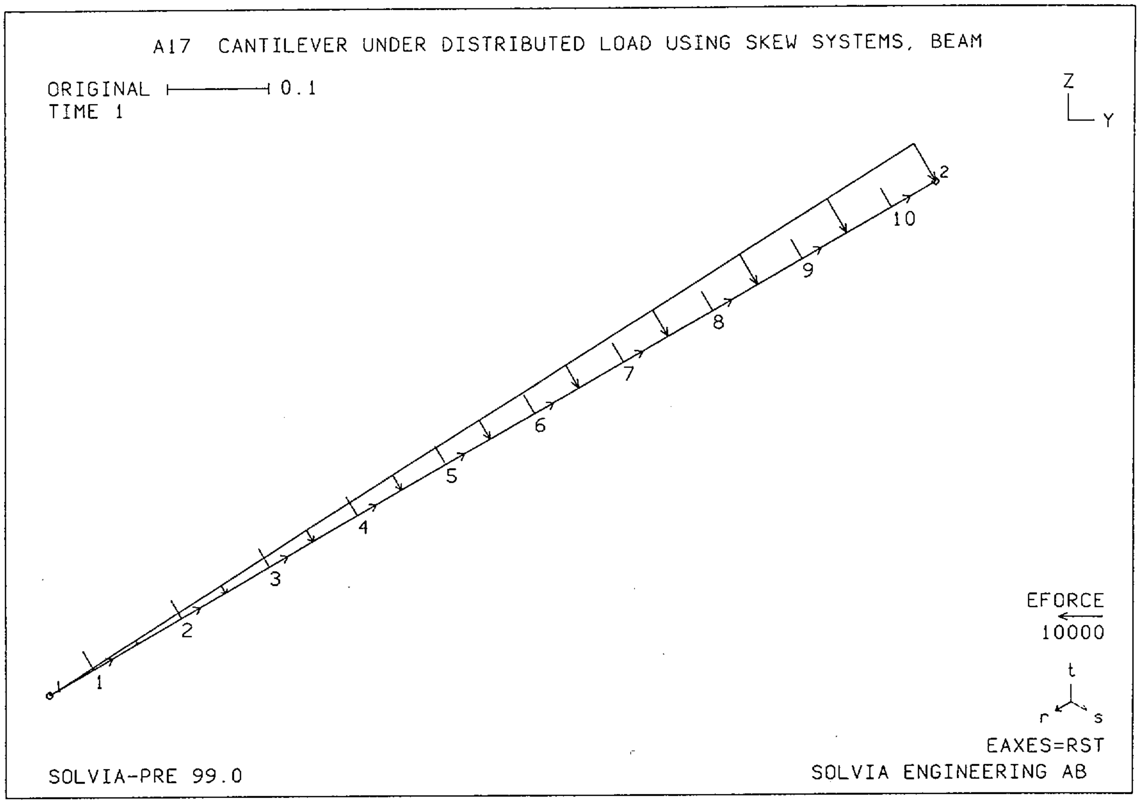
The axial stress (N/m^2, in the direction of the beam) on the top surface, closest to the fixed end is:

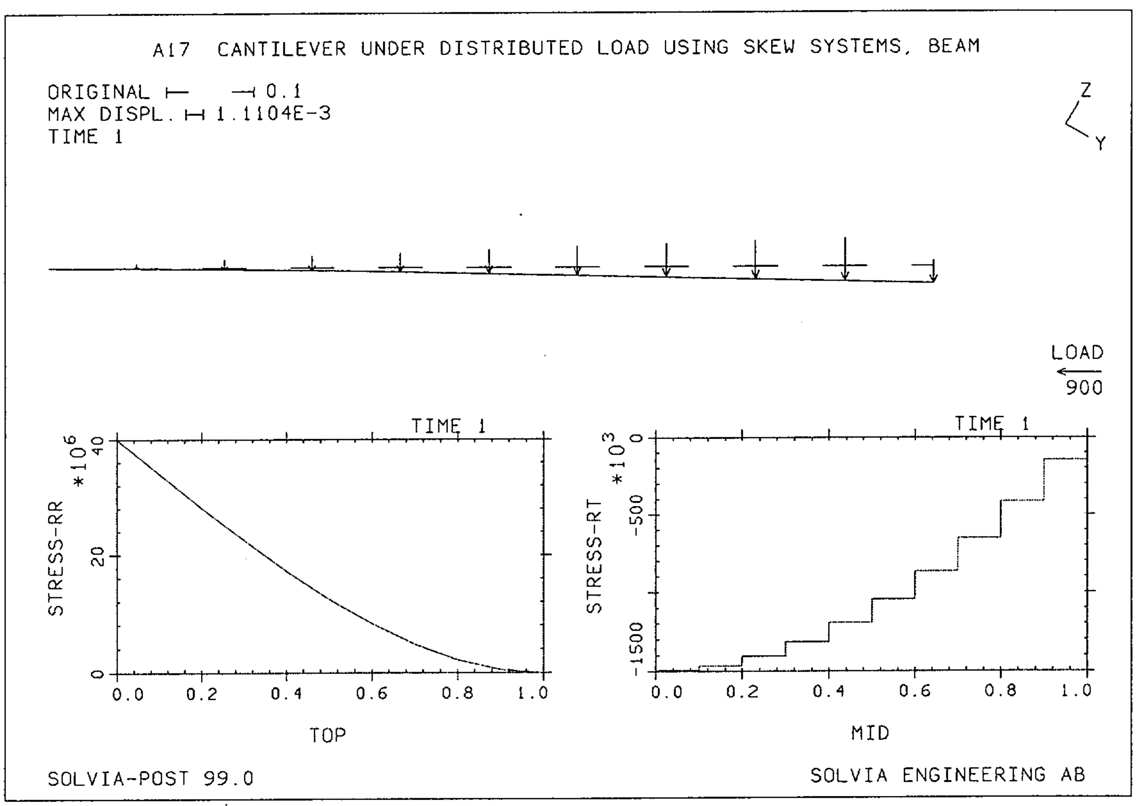
Theory	SOLVIA
$40.00 \cdot 10^6$	$40.00 \cdot 10^6$

User Hints

- The BEAM elements do not predict the exact displacement in this example since its displacement assumption along the beam is a cubic polynomial while the exact displacement varies with the 5th power of the axial coordinate. However, 10 BEAM elements give a very good solution to this example problem.

- Note that the distributed load for the BEAM element is force per unit length.


A17 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, BEAM
ORIGINAL 0.1
TIME 1
Z
Y
1
2
3
4
5
6
7
8
9
10
EFORCE
10000
t
r
s
EAXES=RST
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A17 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, BEAM
ORIGINAL 0.1
MAX DISPL. 1.1104E-3
TIME 1
Z
Y
LOAD
900
TIME 1
STRESS-RR *10^6
0
20
40
0.0 0.2 0.4 0.6 0.8 1.0
TOP
TIME 1
STRESS-RT *10^3
0
-500
-1500
0.0 0.2 0.4 0.6 0.8 1.0
MID
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD 'A17  CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS,
 BEAM'
*
DATABASE  CREATE
*
SKEWSYSTEMS  EULERANGLES
 1 30
SYSTEM  1  CARTESIAN  PHI=30
COORDINATES
 1  /  2  0. 1.
*
MATERIAL  1  ELASTIC E=2.E11 NU=0.3
*
EGROUP  1  BEAM RESULT=STRESSES
SECTION  1  RECTANGULAR WTOP=0.05 D=0.1
BEAMVECTOR
 1  -1.
GLINE  N1=1 N2=2 AUX=-1  EL=10
LOADS  ELEMENT  TYPE=FORCE INPUT=LINE
 1 2  -T  0. 1.E4
*
NSKEWS  INPUT=LINE
 1 2  1
FIXBOUNDARIES  /  1
*
SET  VIEW=X NSYMBOLS=MYNODES
MESH  ENUMBERS=YES NNUMBERS=MYNODES EAXES=RST VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A17  CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, BEAM
*
DATABASE  CREATE
*
WRITE  FILENAME='a17.lis'
*
VIEW  ID=1 XVIEW=1 ROTATION=-30
SET  VIEW=1
MESH  VECTOR=LOAD ORIGINAL=DASHED SUBFRAME=12
*
EPLINE  NAME=TOP  /  1 1 9  TO  10 1 9
EPLINE  NAME=MID  /  1 6 14 TO  10 6 14
*
ELINE  LINENAME=TOP KIND=SRR OUTPUT=ALL SUBFRAME=2121
ELINE  LINENAME=MID KIND=SRT SUBFRAME=2221
*
NLIST  DIRECTION=34
NLIST  KIND=REACTIONS DIRECTION=34
SUMMATION  KIND=LOAD
END
```

EXAMPLE A18

CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, ISOBEAM

Objective

To verify the ISOBEAM element subjected to distributed loading and when employing SKEW degree-of-freedom Systems.

Physical Problem

Same as in Example A14, see figure on page A14-1.

Finite Element Model

The model is shown in the figures on page A18.2. Ten parabolic ISOBEAM elements are used to model the cantilever.

Solution Results

The theoretical solution for the end displacement is the same as given in Example A14.

The input data shown on pages A18.4 and A18.5 gives the following results:

End displacement (mm):

Theory	SOLVIA
-1.113	-1.109

The axial stress (N/m^2, in the direction of the beam) on the top surface, closest to the fixed end is:

Theory	SOLVIA
$40.00 \cdot 10^6$	$40.00 \cdot 10^6$

SOLVIA-POST is employed to calculate bending and shear stresses from element bending moments and shear forces.

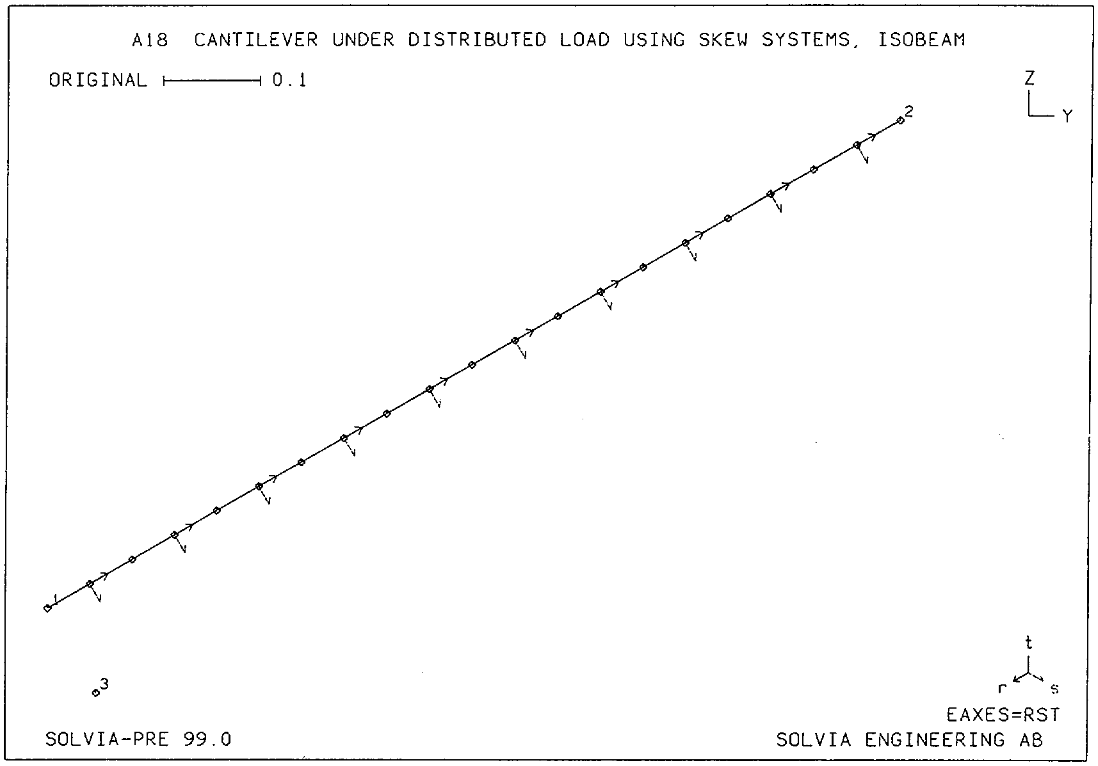
The deformed mesh and a contour plot of the bending moment and the variation of axial stress and shear stress along the top surface of the beam are shown on page A18.3.

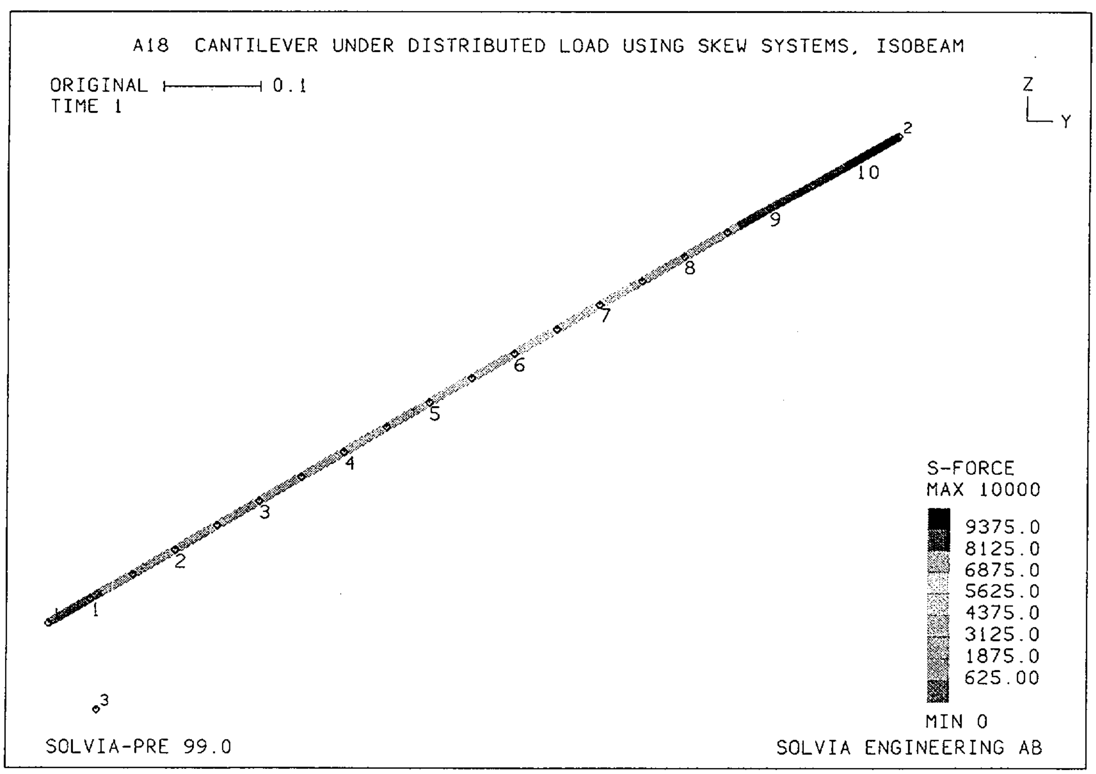
User Hints

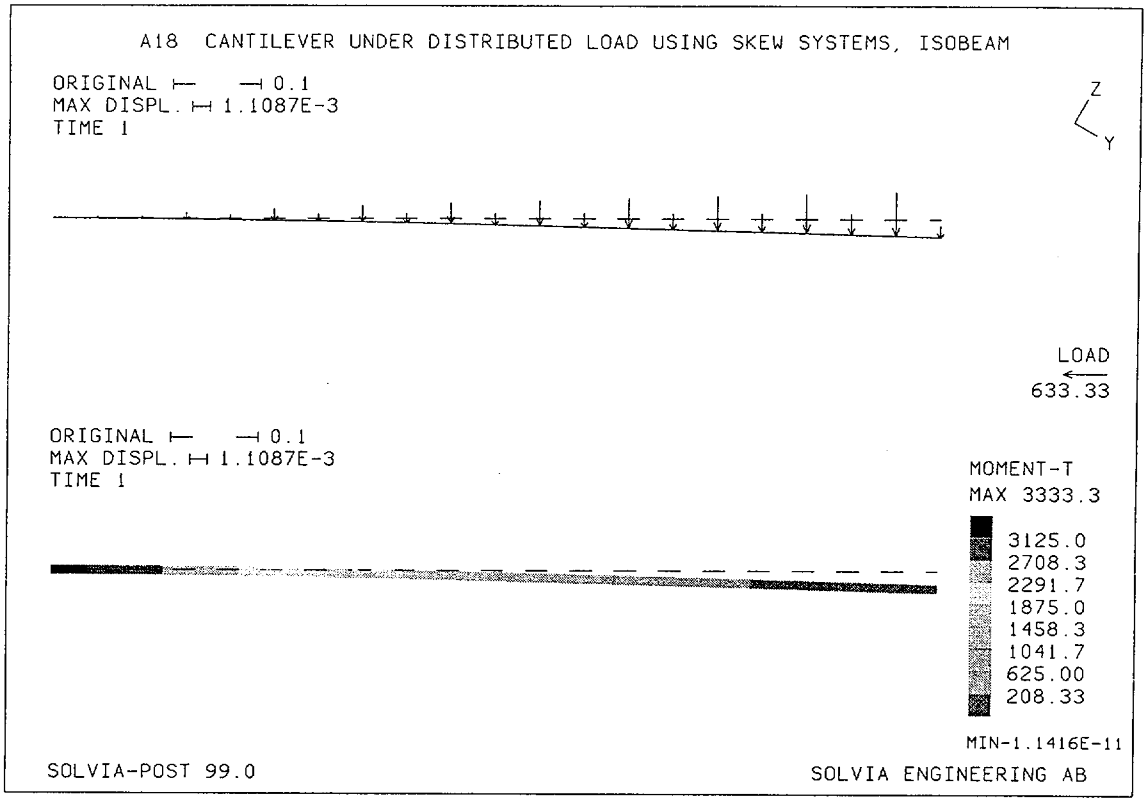
- Note that the SOLVIA predicted displacements agree well with the analytical solution even though 3-node ISOBEAM elements were used (with a parabolic displacement assumption in each beam element).

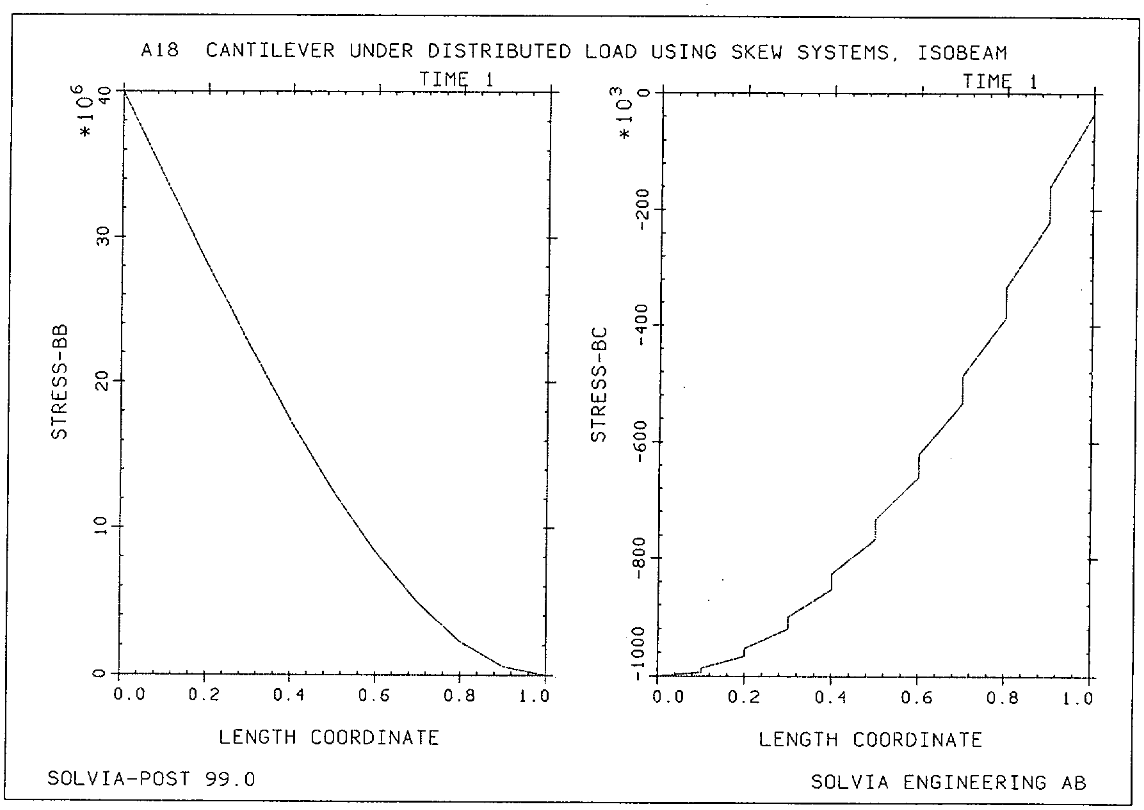
- The forces and moments calculated by SOLVIA for the internal nodes of the elements only balance the external loads applied to these nodes. They are, therefore, not equal to the stress resultants obtained by integrating the stresses through the beam thickness (see Example A8).

The forces and moments calculated for element end nodes are, however, in equilibrium with the externally applied forces/moments plus those supplied by adjoining elements. Note that the exact forces and moments for the node at the fixed end are calculated.






A18 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, ISOBEAM
ORIGINAL 0.1
MAX DISPL. 1.1087E-3
TIME 1
Z
Y
LOAD
633.33
ORIGINAL 0.1
MAX DISPL. 1.1087E-3
TIME 1
MOMENT-T
MAX 3333.3
3125.0
2708.3
2291.7
1875.0
1458.3
1041.7
625.00
208.33
MIN-1.1416E-11
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A18 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, ISOBEAM
TIME 1
*10^6
STRESS-BB
0
10
20
30
40
0.0
0.2
0.4
0.6
0.8
1.0
LENGTH COORDINATE
TIME 1
*10^3
STRESS-BC
0
-200
-400
-600
-800
-1000
0.0
0.2
0.4
0.6
0.8
1.0
LENGTH COORDINATE
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A18  CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS,
 ISOBEAM'
*
DATABASE  CREATE
*
SKEWSYSTEMS  EULERANGLES
 1 30
SYSTEM  1  CARTESIAN  PHI=30
COORDINATES
 1  /  2  0. 1.  /  3  0. 0. -0.1
*
MATERIAL  1  ELASTIC E=2.E11 NU=0.3
*
EGROUP  1  ISOBEAM  RESULTS=FORCES
SECTION  1  SDIM=0.1 TDIM=0.05
GLINE  1 2 3  EL=10 NODES=3
LOADS  ELEMENT  TYPE=FORCE INPUT=LINE
 1 2  S  0. 1.E4
*
NSKEWS  INPUT=LINE
 1 2  1
*
FIXBOUNDARIES  /  1 3
*
SET  VIEW=X NSYMBOLS=YES NNUMBERS=MYNODES
MESH  EAXES=RST
MESH  ENUMBERS=YES CONTOUR=SFORCE
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A18  CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS,
ISOBEAM
*
DATABASE  CREATE
*
WRITE  FILENAME='a18.lis'
*
VIEW  ID=1 XVIEW=1 ROTATION=-30
SET  VIEW=1 ORIGINAL=DASHED
*
MESH  ORIGINAL=DASHED VECTOR=LOAD SUBFRAME=12
MESH  CONTOUR=MT AXES=NO
*
EPLINE  NAME=TOP
 1  1 2  TO  10  1 2
*
EVARIABLE  NAME=TMOMENT TYPE=ISOBEAM KIND=MT
EVARIABLE  NAME=SFORCE  TYPE=ISOBEAM KIND=FS
*
CONSTANT  NAME=W VALUE=83.3333E-6
CONSTANT  NAME=A VALUE=5.0000E-3
*
RESULTANT  NAME=STRESSBB STRING=' ABS( TMOMENT / W )'
RESULTANT  NAME=STRESSBC STRING='-ABS( SFORCE  / A )'
*
AXIS  ID=1 LABEL='LENGTH COORDINATE'
AXIS  ID=2 LABEL='STRESS-BB'
AXIS  ID=3 LABEL='STRESS-BC'
*
SUBFRAME  21
RLINE  LINENAME=TOP RESULTANTNAME=STRESSBB XAXIS=1 YAXIS=2,
       OUTPUT=ALL
RLINE  LINENAME=TOP RESULTANTNAME=STRESSBC XAXIS=1 YAXIS=3,
       OUTPUT=ALL
*
NLIST  ZONENAME=EL10 DIRECTION=34
NLIST  KIND=REACTIONS DIRECTION=34
END
```

EXAMPLE A19

CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, PLATE

Objective

To verify the PLATE element subjected to distributed loading and when employing SKEW degree-of-freedom Systems.

Physical Problem

Same problem as in Example A14, see figure A14-1.

Finite Element Model

The model is shown on page A19.2. Twenty PLATE elements are used to model the cantilever.

Solution Results

The theoretical solution for the end displacement is the same as given in Example A14.

The input data on pages A19.4 and A19.5 give the following results:

The moments at the fixed end are:

Theory (Nm)	SOLVIA (Nm/m)	SOLVIA (Nm)
3333	67528	3376 (element 4, at node 1)
3333	64442	3222 (element 1, at node 1)
3333	64442	3222 (element 1, at node 2)

End displacement (mm):

Theory	SOLVIA
-1.113	-1.101

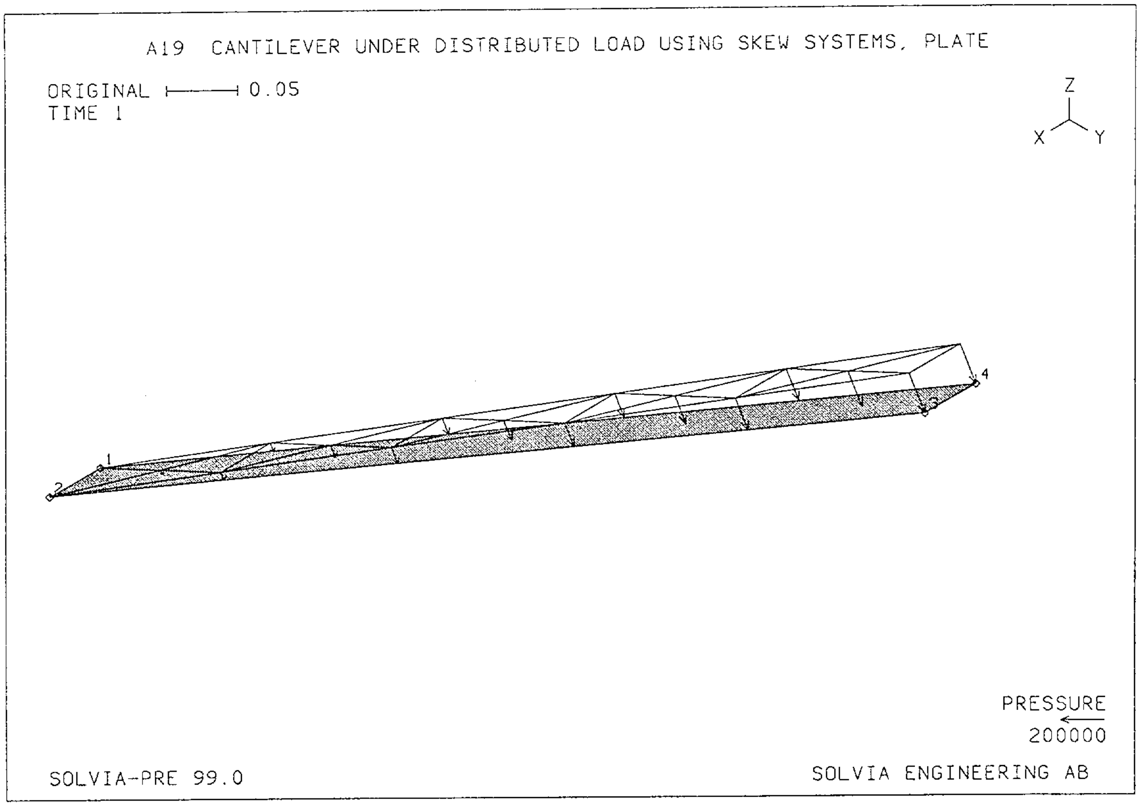
The deformed mesh and the variation of the bending moment along the beam (at stress points at the middle and at the edge) are shown in the figures on page A19.3.

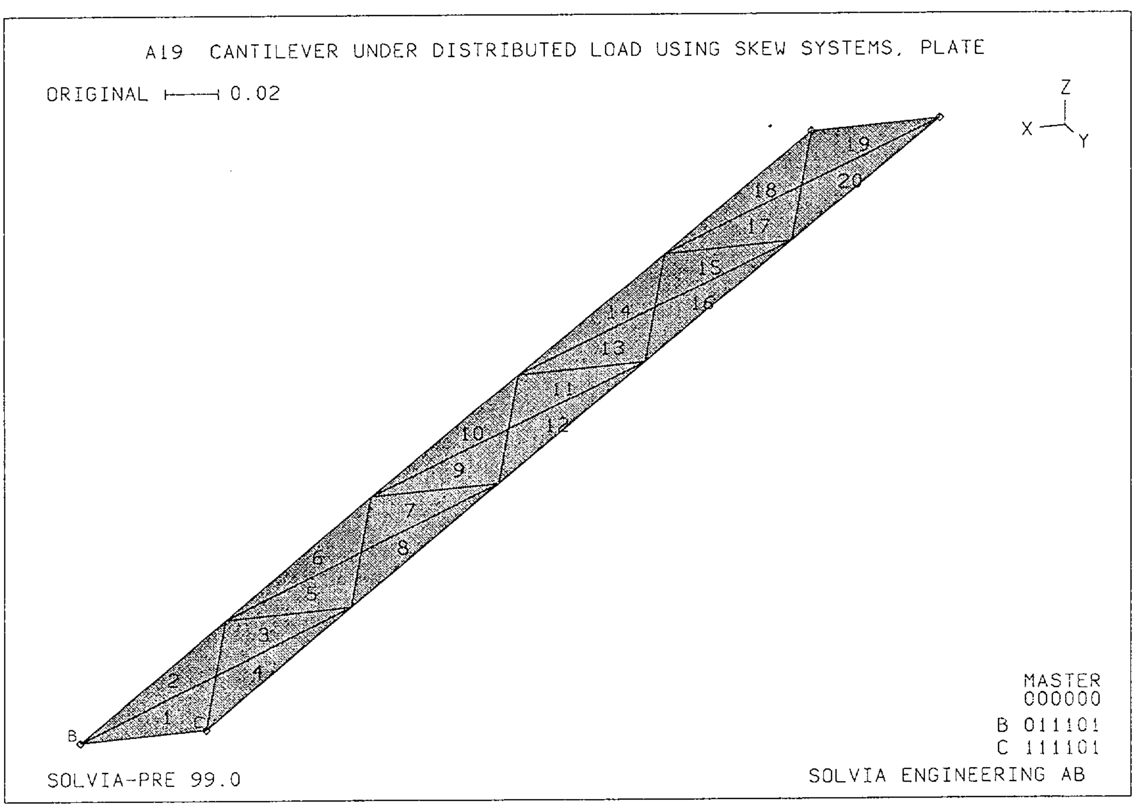
User Hints

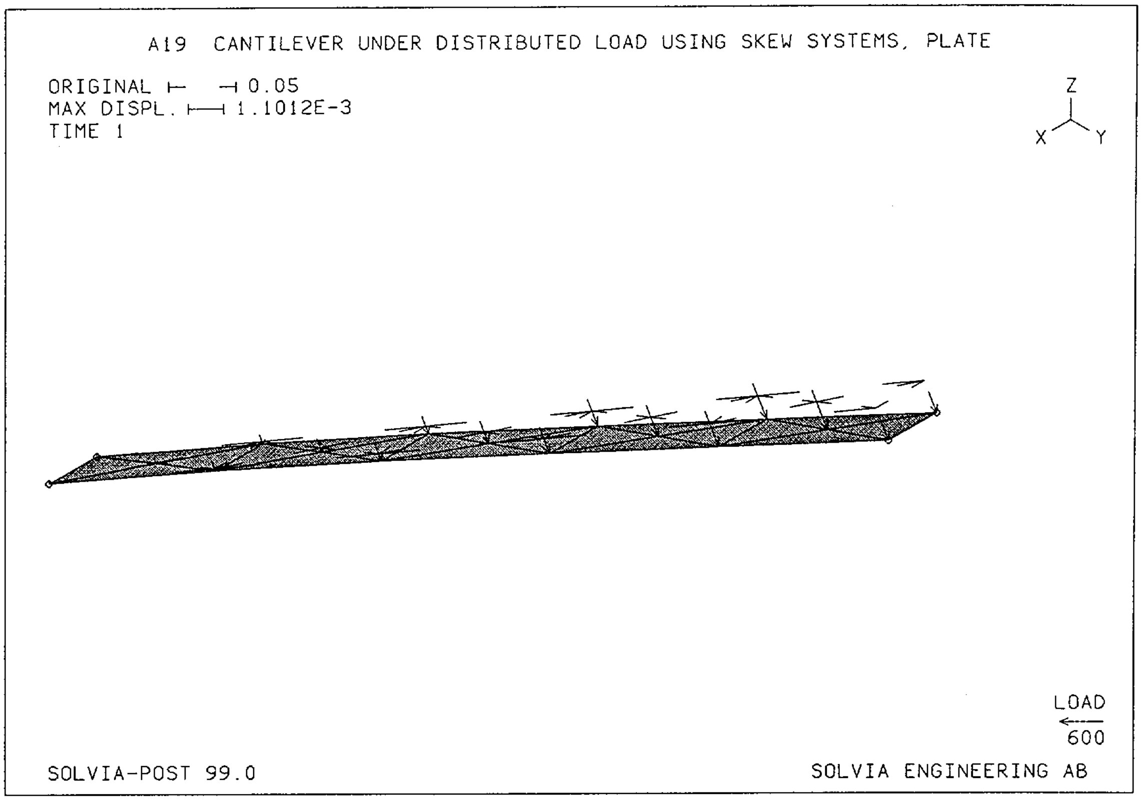
- Since the PLATE element model does not include the strain energy due to transverse shear strains in the formulation and the Kirchhoff hypothesis of zero transverse shear strains is imposed at discrete points, the element is applicable to thin (including very thin) plates and shells.

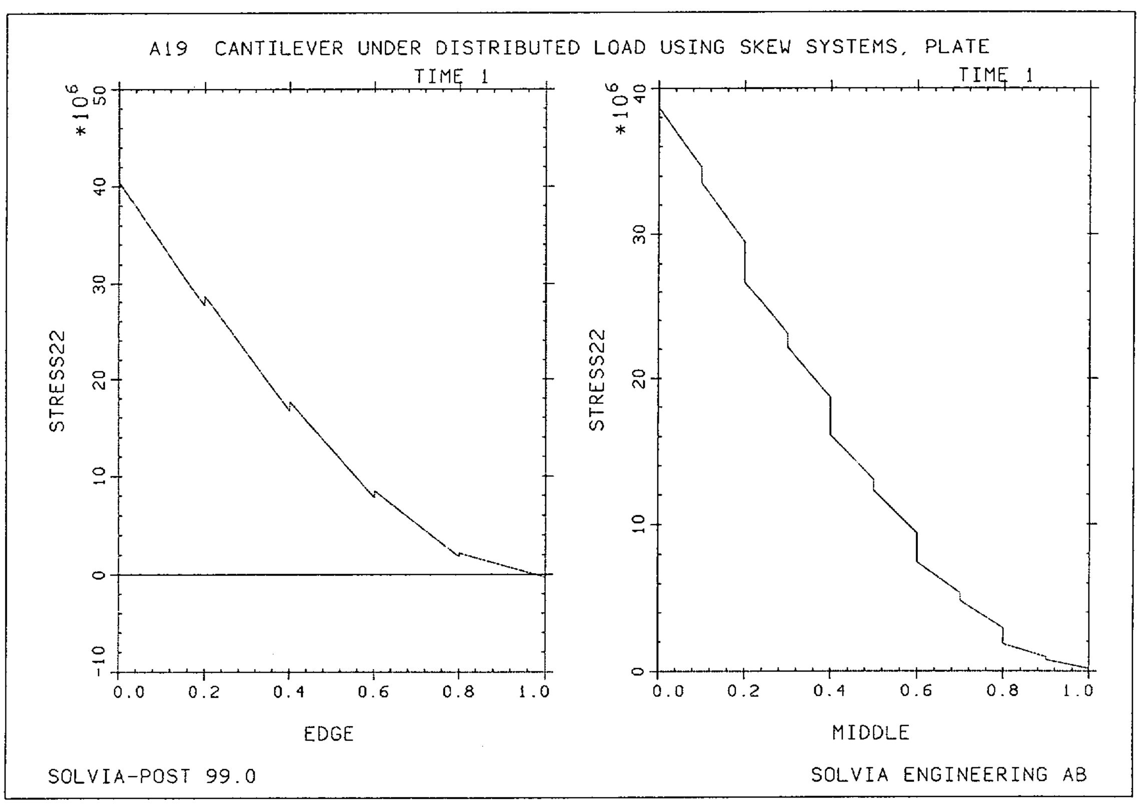
- Note that smaller stress jumps are obtained for the line along the edge than for the line along the middle of the cantilever, see figures on page A19.3. A finer mesh is necessary to reduce the stress jumps.

- Section forces/moments per unit length are obtained as the basic element result for the PLATE element. Using the RESULTANT command a transformation to the axial stress values is achieved.


A19 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, PLATE
ORIGINAL 0.05
TIME 1
Z
X
Y
1
2
3
4
PRESSURE
200000
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A19 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, PLATE
ORIGINAL 0.02
Z
X
Y
1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
B
C
MASTER
000000
B 011101
C 111101
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A19 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, PLATE
ORIGINAL 0.05
MAX DISPL. 1.1012E-3
TIME 1
Z
X
Y
LOAD
600
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A19 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, PLATE
TIME 1
TIME 1
*10^6
STRESS22
EDGE
MIDDLE
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A19  CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS,
 PLATE'
*
DATABASE  CREATE
*
COORDINATES
 1  0.
 2  0.05
 3  0.05   0.8660254  0.5
 4  0.     0.8660254  0.5
*
SKEWSYSTEM  EULERANGLES
 1 30
*
MATERIAL  1  ELASTIC E=2.E11 NU=0.3
*
EGROUP  1  PLATE
GSURFACE  1 2 3 4  EL1=1 EL2=5
STRESSTABLE  1  1 2 3 4 5 6 7
EDATA  /  1 0.1
LOADS  ELEMENT  INPUT=SURFACE
 1 2 3 4  T  0. 0. 2.E5 2.E5
*
NSKEWS  INPUT=SURFACE
 1 2 3 4  1
*
FIXBOUNDARIES  12346  /  1
FIXBOUNDARIES   2346  /  2
*
SET  NSYMBOLS=MYNODES
MESH  VIEW=I OUTLINE=YES VECTOR=LOAD NNUMBERS=MYNODES
VIEW  ID=1 XVIEW=1 YVIEW=3 ZVIEW=1
MESH  VIEW=1 ENUMBER=YES BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A19  CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, PLATE
*
DATABASE  CREATE
SYSTEM 1 CARTESIAN NA=2 NB=4
*
WRITE  FILENAME='a19.lis'
*
SET  NSYMBOLS=MYNODES
MESH  ORIGINAL=DASHED VECTOR=LOAD
*
EPLINE  NAME=EDGE
 4  2 4 1  STEP 4 TO  20  2 4 1
EPLINE  NAME=MIDDLE
 1  4 7 3  /  3  3 7 4
 5  4 7 3  /  7  3 7 4
 9  4 7 3  /  11 3 7 4
 13 4 7 3  /  15 3 7 4
 17 4 7 3  /  19 3 7 4
*
EVARIABLE FORCE PLATE F22
EVARIABLE MOMENT PLATE M22
CONSTANT THICK 0.1
CONSTANT SIX   6.0
RESULTANT STRESS22 'FORCE/THICK+SIX*MOMENT/(THICK*THICK)'
*
RLINE  LINENAME=EDGE   RESULTANT=STRESS22 OUTPUT=ALL SYSTEM=1
SUBFRAME=21
RLINE  LINENAME=MIDDLE RESULTANT=STRESS22 OUTPUT=ALL SYSTEM=1
*
ELIST  ZONENAME=EL1 SYSTEM=1
NLIST  DIRECTION=34
NLIST  KIND=REACTION DIRECTION=34
END
```

EXAMPLE A20

CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, SHELL

Objective

To verify the SHELL element when subjected to distributed loading and when employing SKEW degree-of-freedom Systems.

Physical Problem

Same problem as in Example A14, see figure A14-1.

Finite Element Model

The model is shown in the figure on page A20.2. It consists of five 9-node SHELL elements.

Solution Results

The theoretical solution for the end displacement is the same as given in A14.

The input data on pages A20.4 and A20.5 gives the following results:

Displacement (mm):

Theory	SOLVIA
-1.113	-1.107

The axial stress (N/m^2), in the direction of the beam) at node 3 is:

Theory	SOLVIA
$40.0 \cdot 10^6$	$38.91 \cdot 10^6$

The deformed mesh and the variation of the axial stress and strain along the beam is shown on page A20.3.

User Hints

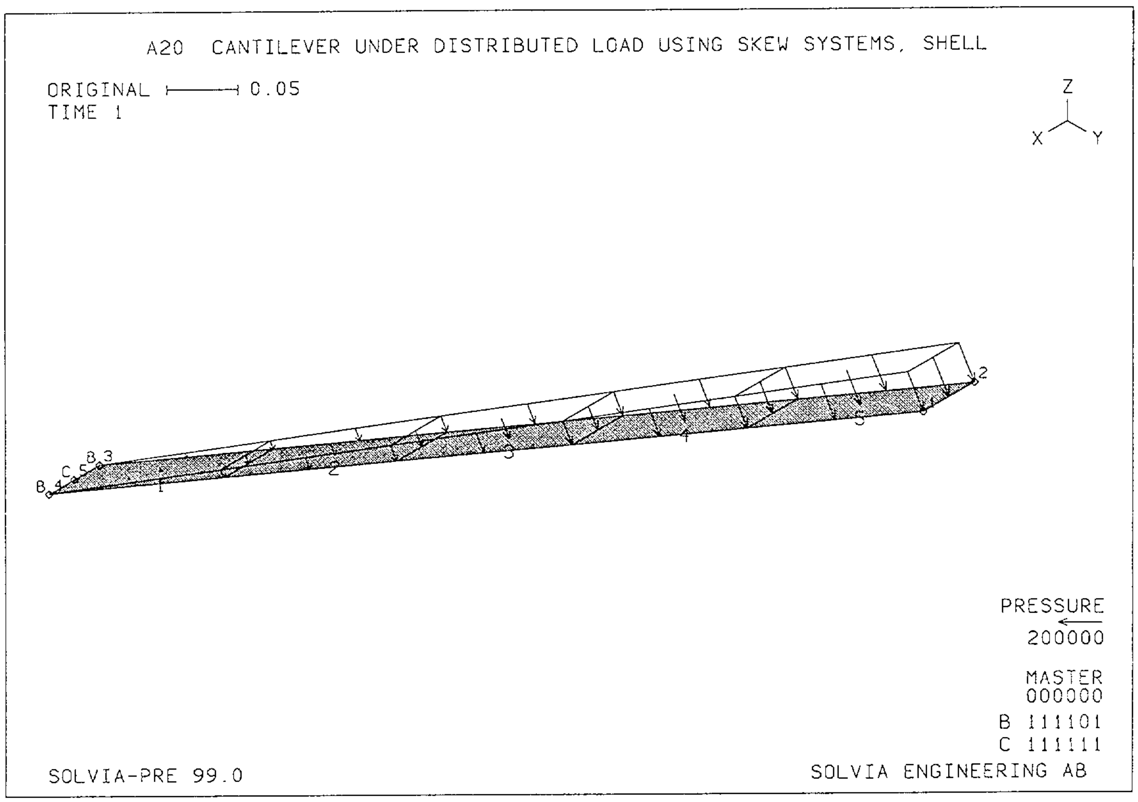
- All shell director vectors are in this example generated automatically by SOLVIA-PRE and the shell rotations of nodes with rotation boundary conditions specified are automatically referenced to the Global (or SKEW) directions [1].

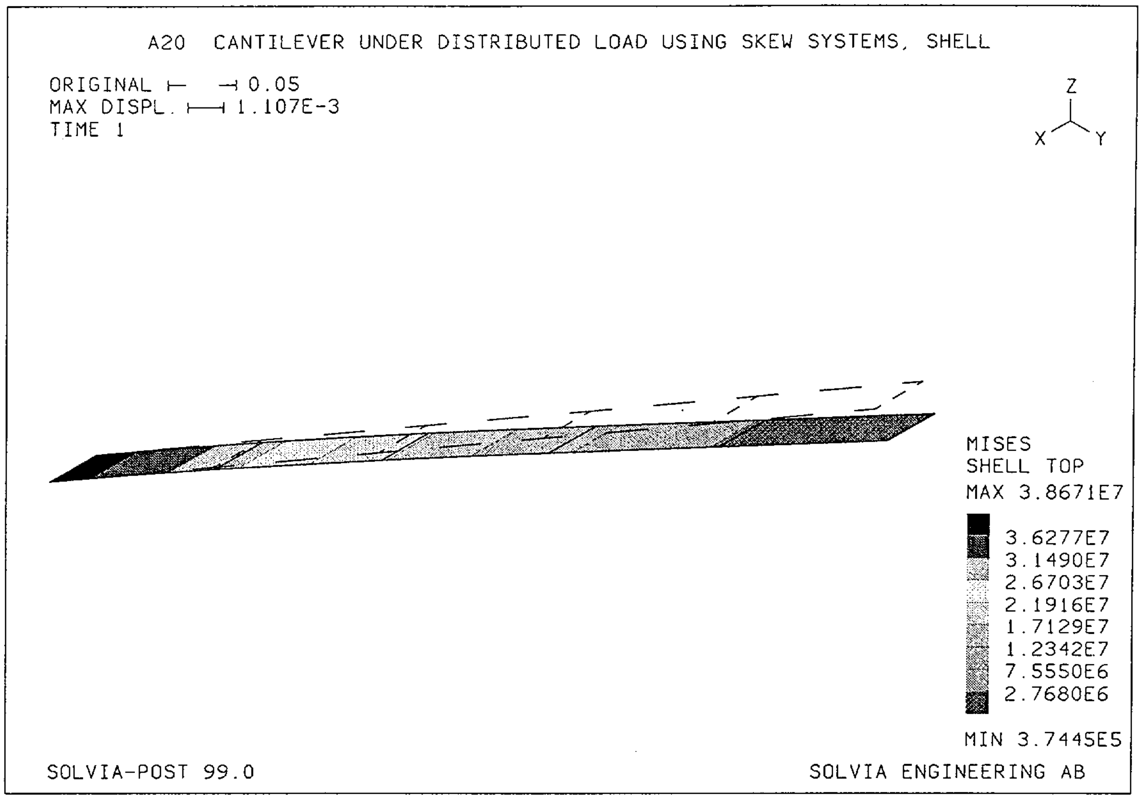
- In this example the Element Stress System is oriented in the principal directions of the beam, since all shell director vectors are orthogonal to the mid-surface. The stresses are, therefore, conveniently requested to be given in the Element Stress System.

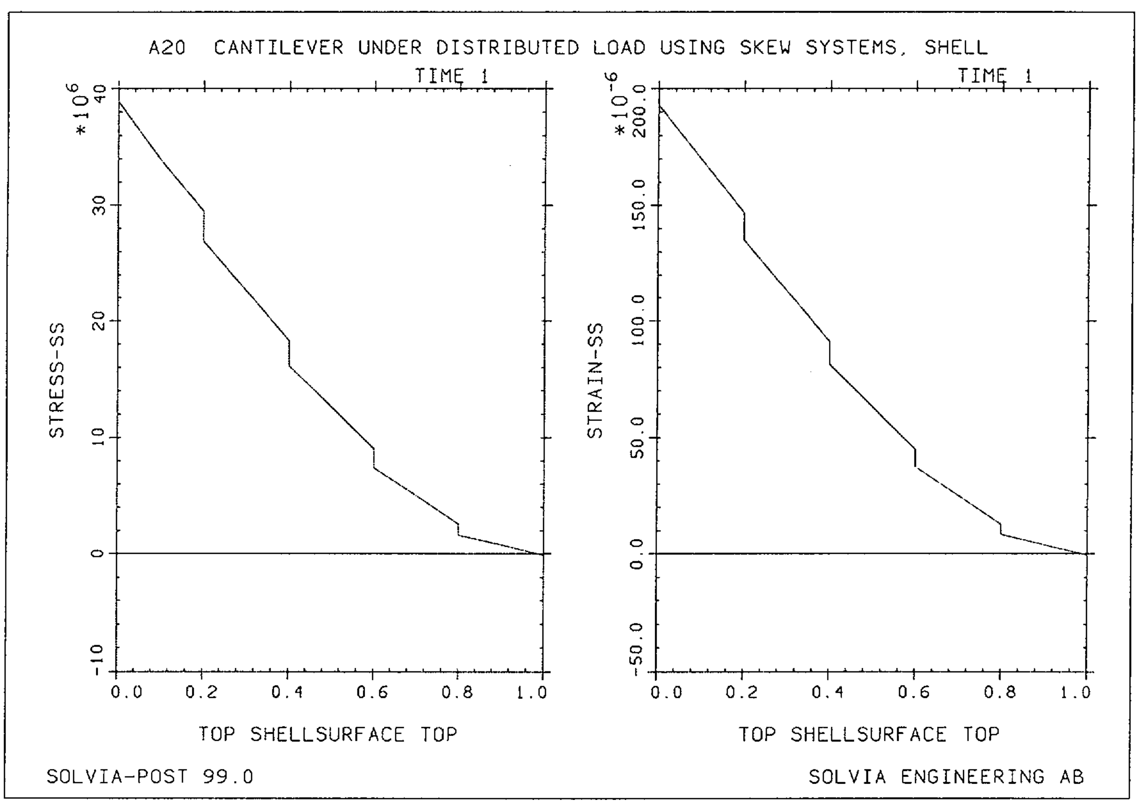
- If 2×2×2 integration is employed, spurious modes are present in the model, compare Example A16. The default integration orders must, therefore, be used in this example and are also recommended for general use.

Reference

[1] SOLVIA-PRE 99.0 Users Manual, Stress Analysis, Report SE 99-1, p. 7.25.




A20 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, SHELL
ORIGINAL 0.05
MAX DISPL. 1.107E-3
TIME 1
Z
X Y
MISES
SHELL TOP
MAX 3.8671E7
3.6277E7
3.1490E7
2.6703E7
2.1916E7
1.7129E7
1.2342E7
7.5550E6
2.7680E6
MIN 3.7445E5
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A20 CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, SHELL
TIME 1
*10^6
STRESS-SS
40
30
20
10
0
-10
0.0 0.2 0.4 0.6 0.8 1.0
TOP SHELLSURFACE TOP
*10^-6
STRAIN-SS
200.0
150.0
100.0
50.0
0.0
-50.0
TOP SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A20  CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS,
SHELL'
*
DATABASE  CREATE
*
SYSTEM  1 CYLINDRICAL
COORDINATES
 ENTRIES NODE  R  THETA  XL
          1    1   30   0.05
          2    1   30   0
          3    0   30   0
          4    0   30   0.05
          5    0   30   0.025
SKEWSYSTEMS  EULER-ANGLES
 1  30
*
MATERIAL  1  ELASTIC  E=2.E11 NU=0.3
*
EGROUP  1  SHELL  STRESSREFERENCE=ELEMENT RESULTS=NSTRESSES
THICKNESS  1  0.1
GSURFACE  3 4 1 2  EL1=1 EL2=5 NODES=9
LOADS  ELEMENT  TYPE=PRESSURE INPUT=SURFACE
 3 4 1 2  T  0. 0. 2.E5 2.E5
*
NSKEWS  INPUT=SURFACE
 1 2 3 4  1
*
FIXBOUNDARIES  12346  /  3 4
FIXBOUNDARIES         /  5
*
SET  NSYMBOLS=MYNODES
MESH  NNUMBER=MYNODES ENUMBERS=YES BCODE=ALL VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A20  CANTILEVER UNDER DISTRIBUTED LOAD USING SKEW SYSTEMS, SHELL
*
DATABASE  CREATE
*
WRITE  FILENAME='a20.lis'
*
MESH  ORIGINAL=DASHED CONTOUR=MISES
*
EPLINE  NAME=TOP
 1  1 8 4  TO  5  1 8 4
ELINE  LINENAME=TOP KIND=SSS OUTPUT=ALL SUBFRAME=21
ELINE  LINENAME=TOP KIND=ESS
*
NLIST  DIRECTION=34
NLIST  KIND=REACTION DIRECTION=34
EMAX  SELECT=S-EFFECTIVE
END
```

EXAMPLE A21

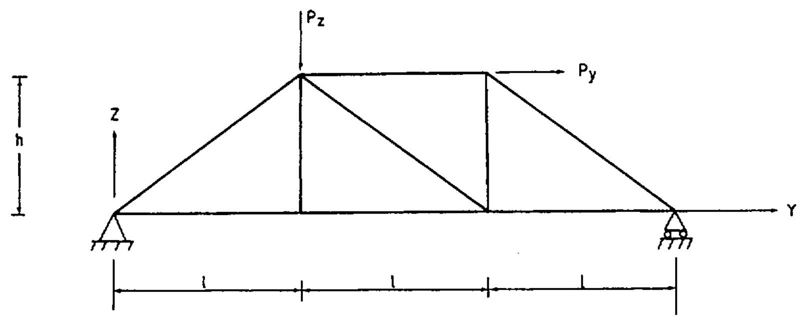
PLANAR TRUSS

Objective

To verify the TRUSS element when applied to a 2-dimensional structure.

Physical Problem

The planar truss structure shown in figure below is considered.



$l = 20$ ft

$h = 15$ ft

$E \cdot A = 30 \cdot 10^4$ kips $- ft^2/ft^2$

$P_z = 10$ kips

$P_y = 20$ kips

Finite Element Model

The model is shown in the top figure on page A21.2. The 2-node TRUSS element is used to model the elements in this problem.

Solution Results

The theoretical displacement solution is given in [1] p. 257. Using the input data on page A21.3 the following results are obtained:

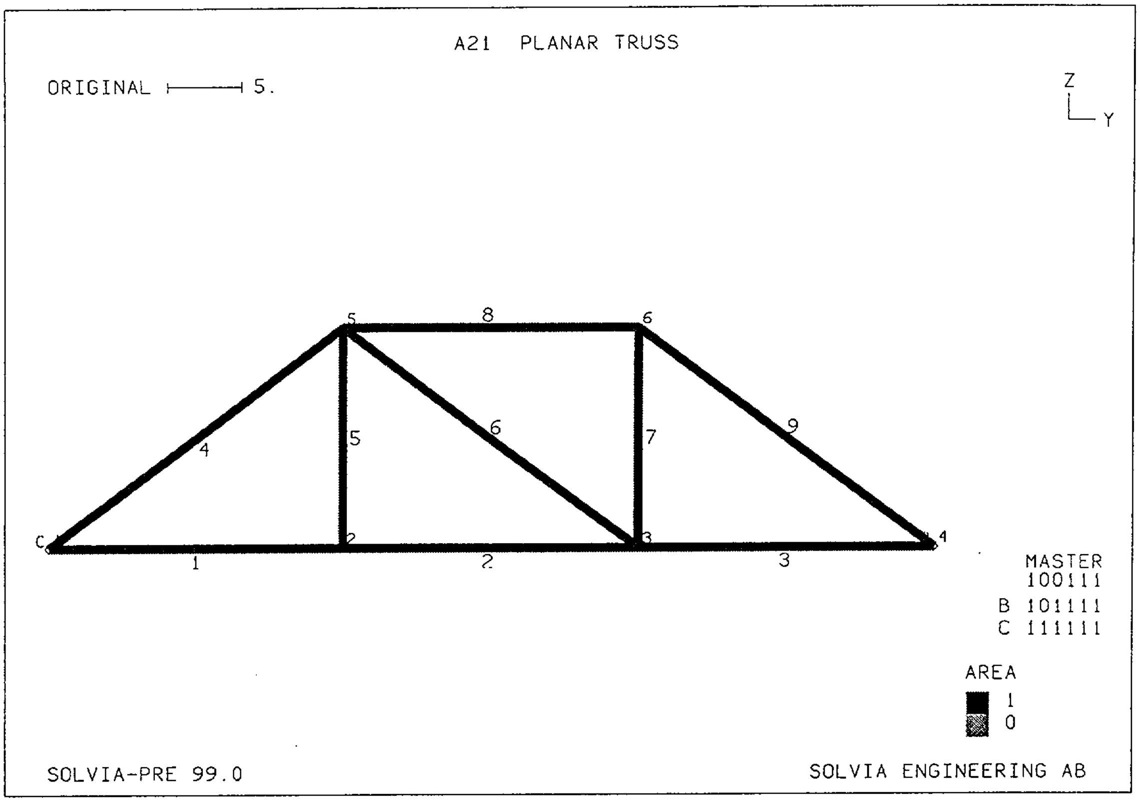
Nodal displacements (10^{-3} feet):

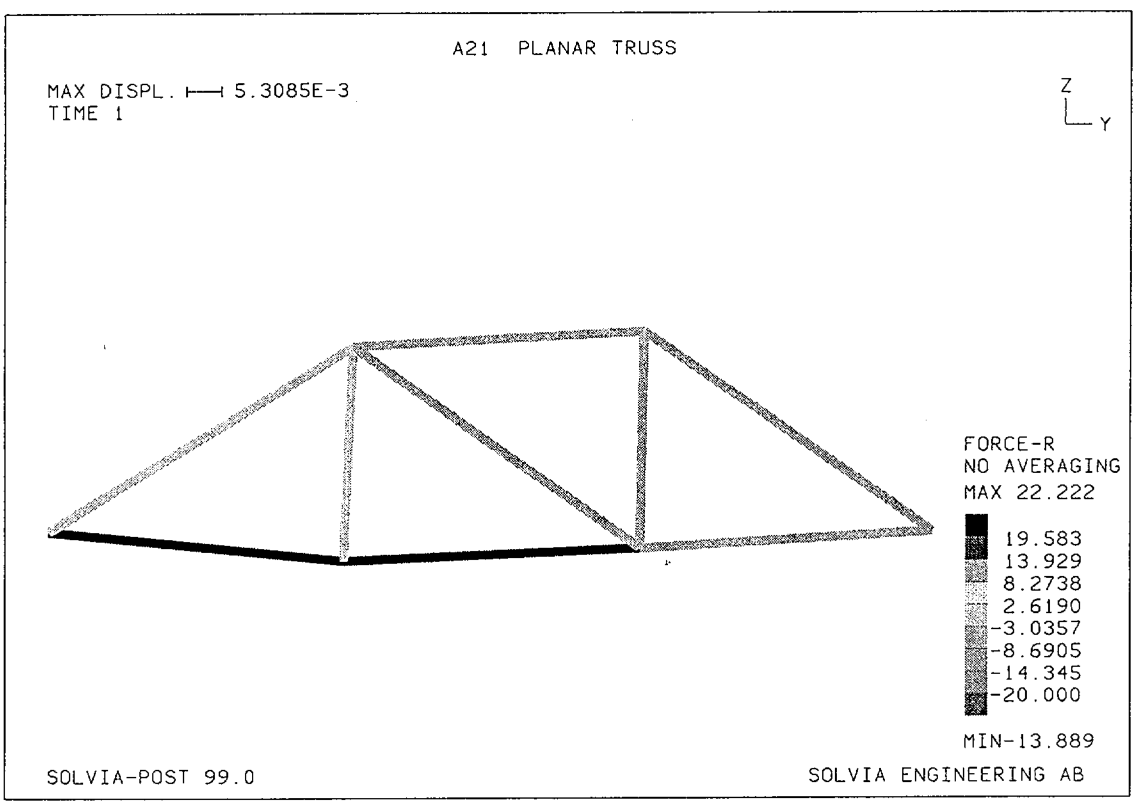
Node	Theory		SOLVIA	
	y-dir.	z-dir.	y-dir.	z-dir.
1	0	0	0	0
2	1.48	-4.38	1.48	-4.38
3	2.96	-2.50	2.96	-2.50
4	3.70	0	3.70	0
5	3.00	-4.38	3.00	-4.38
6	3.59	-2.08	3.59	-2.08

A contour plot of the element forces is shown in the bottom figure on page A21.2.

Reference

[1] Tuma, J.J., Munshi, R.K., Theory and Problems of Advanced Structural Analysis, Schaum's Outline Series, McGraw-Hill.


A21 PLANAR TRUSS
ORIGINAL 5.
Z
Y
8
5
6
4
5
6
7
9
C
2
3
4
1
2
3
MASTER
100111
B 101111
C 111111
AREA
1
0
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A21 PLANAR TRUSS
MAX DISPL. 5.3085E-3
TIME 1
Z
Y
FORCE-R
NO AVERAGING
MAX 22.222
19.583
13.929
8.2738
2.6190
-3.0357
-8.6905
-14.345
-20.000
MIN-13.889
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A21  PLANAR TRUSS'
*
DATABASE  CREATE
*
MASTER  IDOF=100111
COORDINATES
 1  TO  4  0. 60.  /  5  0. 20. 15.  /  6  0. 40. 15.
*
MATERIAL  1  ELASTIC E=30.E4
*
EGROUP  1  TRUSS
ENODES
 1  1 2  TO  3  3 4
 4  5 1  TO  6  5 3
 7  6 3  /   8  6 5  /  9  6 4
EDATA  /  1  1.
*
FIXBOUNDARIES  23  /  1
FIXBOUNDARIES   3  /  4
*
LOADS  CONCENTRATED
 5 3 -10
 6 2  20
*
MESH  VIEW=X NSYMBOLS=YES NNUMBERS=YES ENUMBERS=YES,
      BCODE=ALL CONTOUR=AREA
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A21  PLANAR TRUSS
*
DATABASE  CREATE
WRITE  FILENAME='a21.lis'
*
NLIST
NLIST  KIND=REACTION
*
SET  VIEW=X
CONTOUR  AVERAGE=NO VMIN=-20
MESH  CONTOUR=FR
*
END
```

EXAMPLE A22

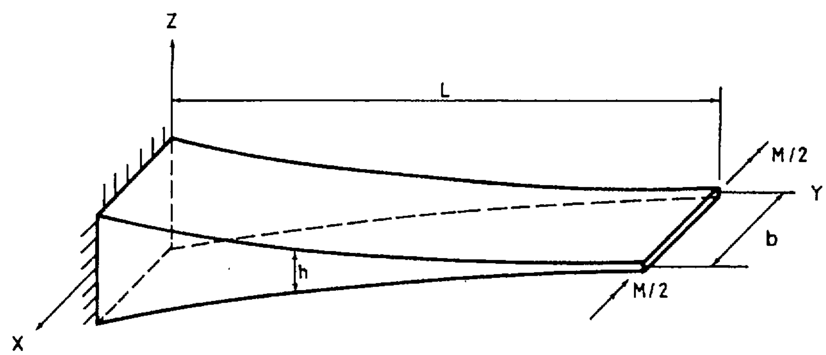
TAPERED CANTILEVER UNDER TIP LOAD

Objective

To verify the SHELL element with variable thickness and when used as a transition element connected to a SOLID element.

Physical Problem

A cantilever of variable thickness and subjected to a moment load at the end, see figure below, is considered.



$L = 100$ in.
$b = 10$ in.
$h = 10/(1+9 \cdot y/L)$ in.
$E = 3.0 \cdot 10^7$ psi
$\nu = 0$
$m = 1000$ in. lbf

Finite Element Model

The model is shown in the figures on page A22.3. It consists of seven SHELL elements and two SOLID elements. The SHELL elements use a quadratic displacement assumption. A linear thickness variation of each SHELL element is assumed. Reduced integration in the rs-plane and the closed Newton-Cotes method in the thickness direction are used.

Solution Results

Beam theory gives the following displacement and top surface bending stress solution along the cantilever:

$$\delta(y) = -\frac{1}{5 \cdot 81 \cdot 10^4}\left(\left(1+\frac{9y}{100}\right)^5 - \frac{45y}{100} - 1\right)$$

$$\sigma_y(y) = \frac{6 \cdot M}{b \cdot h^2}$$

The SOLVIA numerical solution obtained using the input data shown on pages A22.6 and A22.7 is as follows:

End displacement (inch):

Theory	SOLVIA
-0.02468	-0.02424

A vector plot of principal stresses for the top layer of integration points and a contour plot of the von Mises effective stress are also shown on page A22.4. The top figure on page A22.5 shows the deviation of effective stress from the nodal mean value.

Bending and shear stresses along the shell elements using reduced and full integration are shown in the bottom figures on page A22.5. The bending stress is shown together with the theoretical solution.

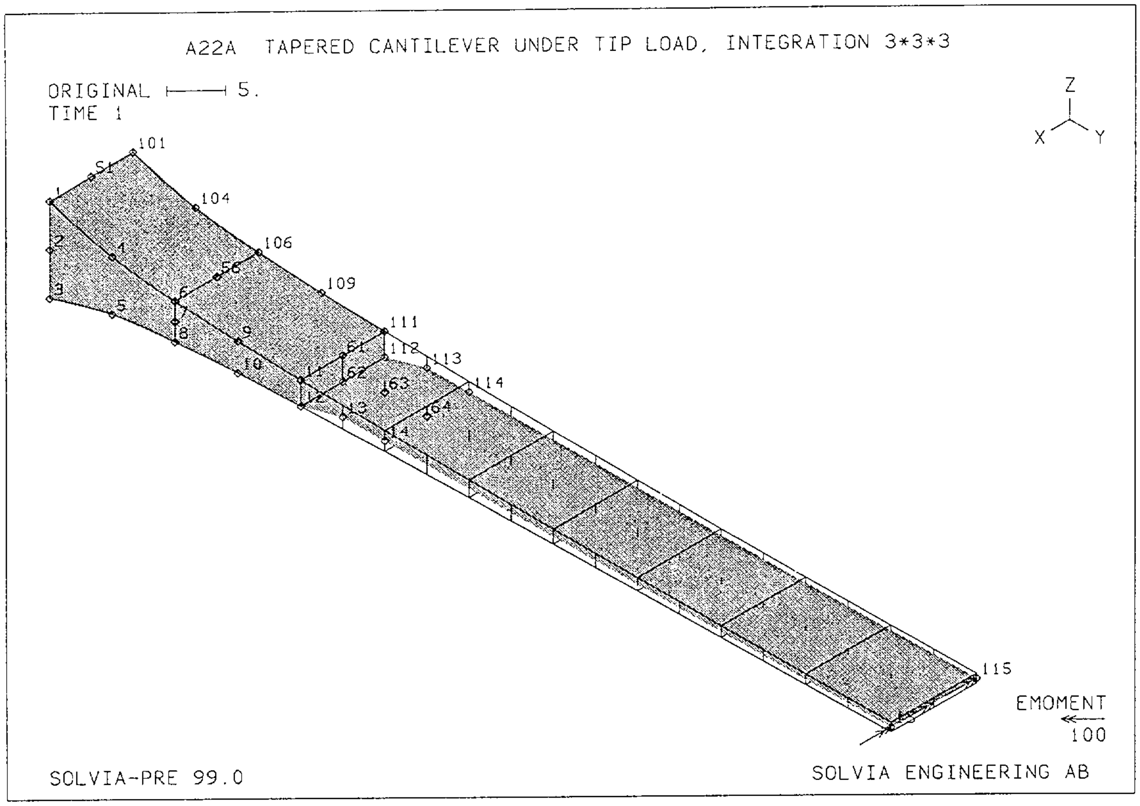
User Hints

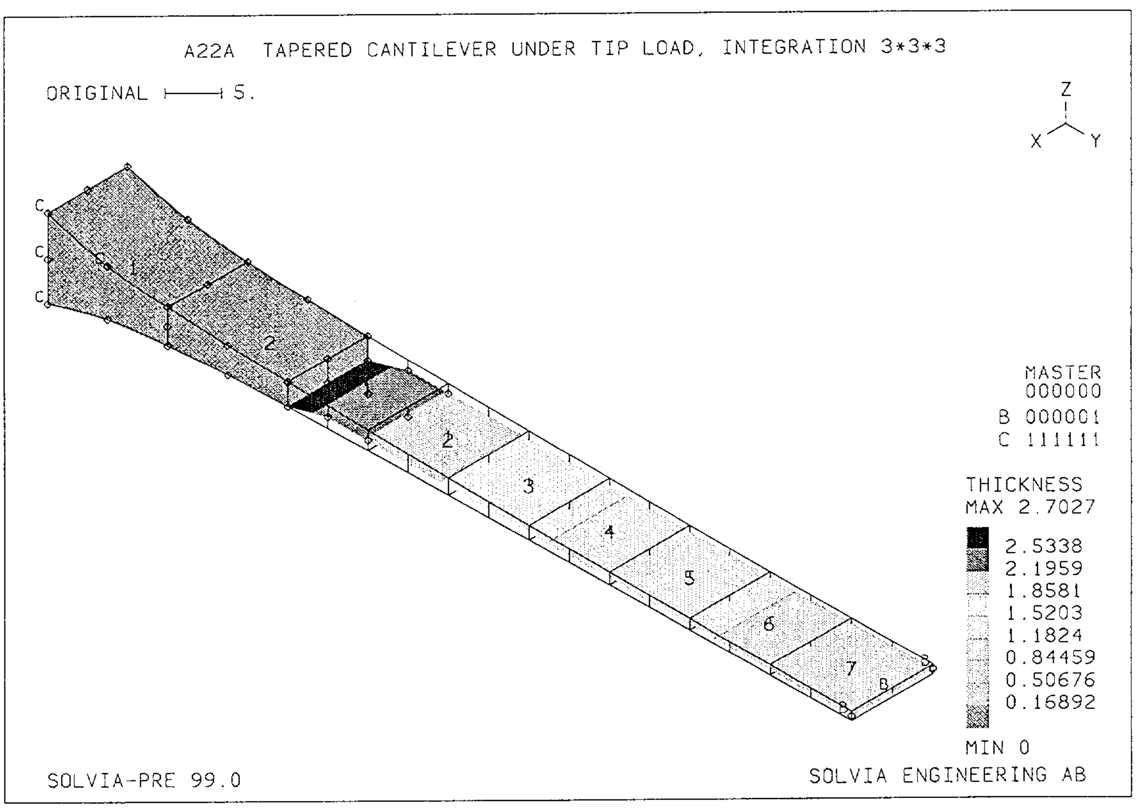
- Poisson's ratio is set to zero in order to simulate the plane stress condition associated with beam theory.
- The material law for the SHELL element is such that the stress in the thickness direction is zero. The stiffness in the thickness direction of a transition SHELL element is, therefore, small at the transition nodes. However, an adjoining SOLID element provides stiffness in the thickness direction.
- SOLVIA-PRE automatically assigns GLOBAL rotations for SHELL midsurface nodes with applied moments specified. The applied moments at the end nodes are then ensured to be acting about the global X-direction [1].
- Since the SHELL element midsurface nodes have no rotational stiffness about the thickness direction, the rotational stiffness about the Z-direction at the end nodes must be fixed [1].
- The plotting of contours for SHELL elements is always performed on a surface formed by the nodes defining the shell elements. When only midsurface nodes are used, contour plots are always located on the SHELL midsurface although the contour results are valid for the TOP, MID or BOTTOM shell surface. For a SHELL transition element, thus an element employing top and bottom nodes at some locations as in this example, then the contours are plotted on the surface formed by the most distant of the nodes defining the SHELL element. For further information, see the commands CONTOUR and SHELLSURFACE in the SOLVIA-POST Users Manual.
- The calculated stresses are less accurate than the calculated displacements. The exact displacements vary as a 5th order polynomial, while the SOLID and SHELL elements are only capable to describe a parabolic variation.

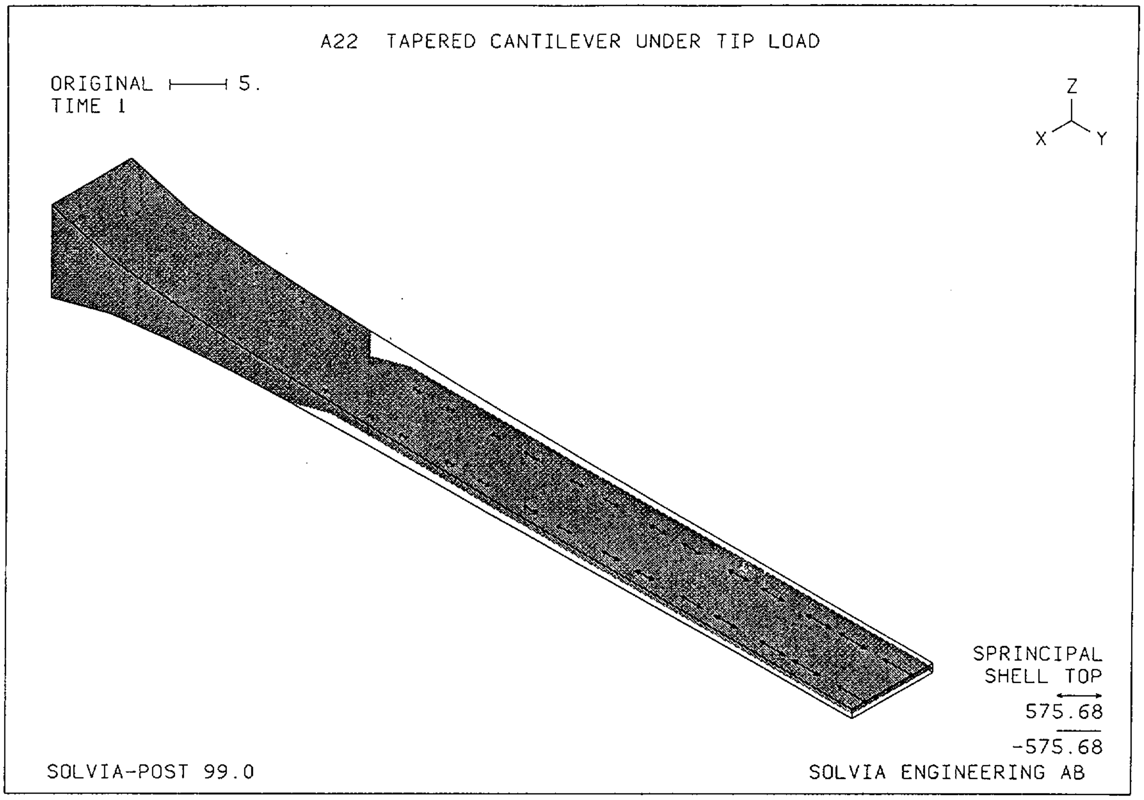
  The approximation in stresses is illustrated by the level of stress jumps (stress deviations) between elements, see the figure on page A22.5. Note, however, that the calculated stresses at the tip of the cantilever are also approximated although no stress jump can be shown there since it is part of the boundary of the structure.
- Linear variation of the SHELL thickness is used in the model to simplify the input data. The use of a complete SHELL thickness table will increase the accuracy of the solution.
- In this example the reduced integration of the SHELL elements improves the stress distribution.

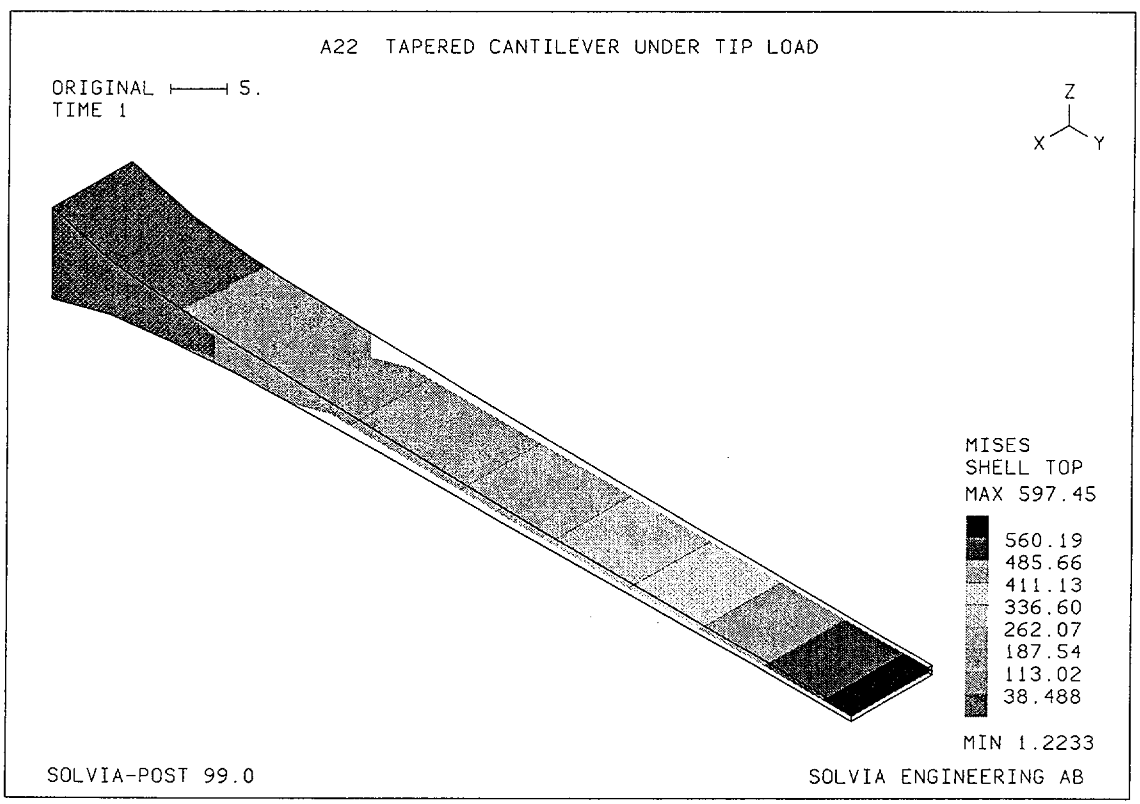
Reference

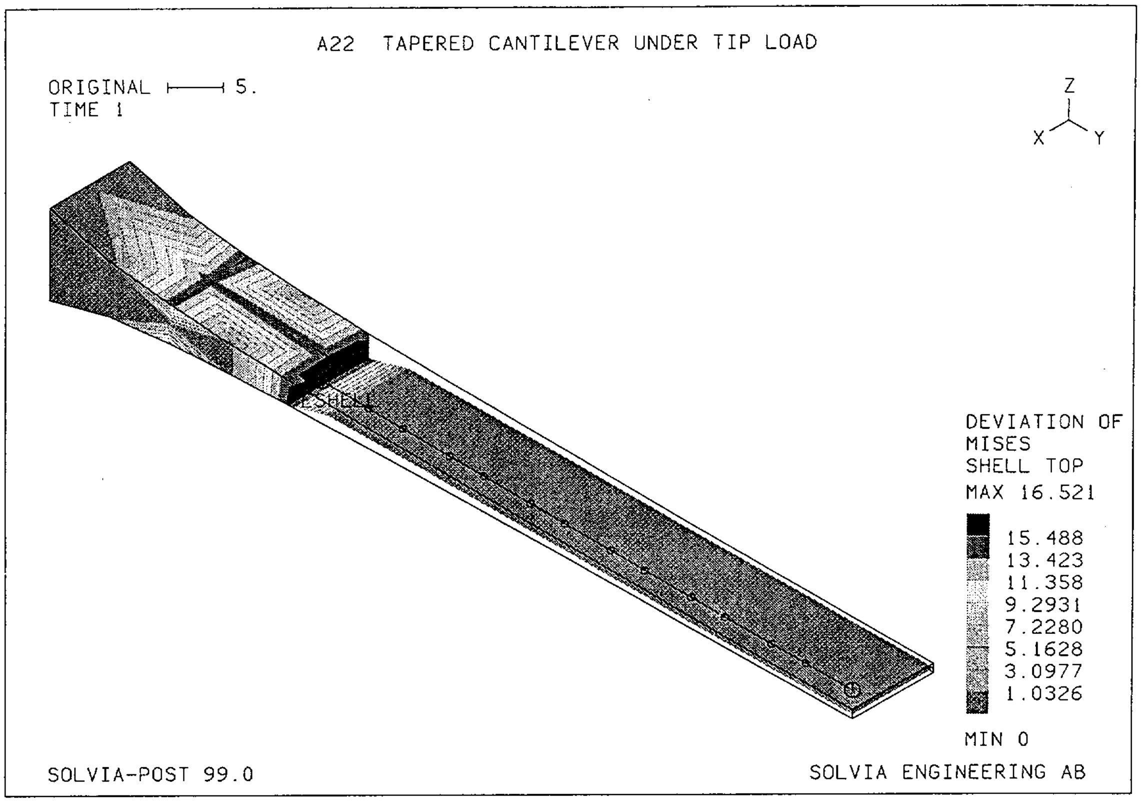
[1] SOLVIA-PRE 99.0 Users Manual, Stress Analysis, Report SE 99-1, p. 7.25.


A22A TAPERED CANTILEVER UNDER TIP LOAD, INTEGRATION 3*3*3
ORIGINAL 5.
TIME 1
Z
X Y
101
104
106
109
111
112
113
114
115
EMOMENT
100
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

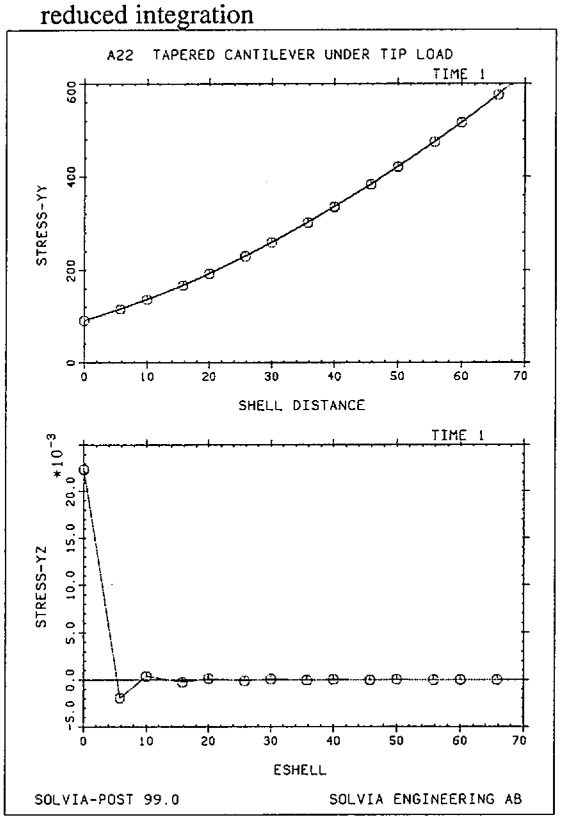

A22A TAPERED CANTILEVER UNDER TIP LOAD, INTEGRATION 3*3*3
ORIGINAL 5.
Z
X Y
MASTER
000000
B 000001
C 111111
THICKNESS
MAX 2.7027
2.5338
2.1959
1.8581
1.5203
1.1824
0.84459
0.50676
0.16892
MIN 0
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A22 TAPERED CANTILEVER UNDER TIP LOAD
ORIGINAL 5.
TIME 1
Z
X Y
SPRINCIPAL
SHELL TOP
575.68
-575.68
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

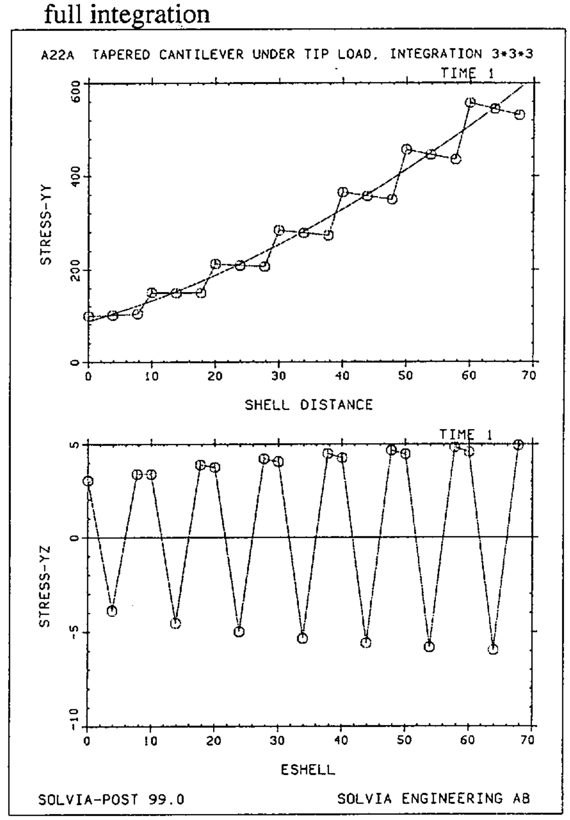

A22 TAPERED CANTILEVER UNDER TIP LOAD
ORIGINAL 5.
TIME 1
Z
X Y
MISES
SHELL TOP
MAX 597.45
560.19
485.66
411.13
336.60
262.07
187.54
113.02
38.488
MIN 1.2233
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB



reduced integration



full integration



SOLVIA-PRE input

```
HEADING  'A22  TAPERED CANTILEVER UNDER TIP LOAD'
*
DATABASE  CREATE
SET MYNODES=200
*
COORDINATES
ENTRIES NODE  Y  Z
  1  0.   5.        /  2  0.    0.        /  3   0.  -5.
  4  7.5  2.98507  /  5  7.5  -2.98507
  6  15.  2.12766  /  7  15.   0.        /  8  15.  -2.12766
  9  22.5 1.65289  / 10  22.5 -1.65289
 11  30.  1.35135  / 12  30.  -1.35135
 13  35.  0.  TO  14  40.  0.
 15 100.
NGENERATION  TIMES=2 NSTEP=50 XSTEP=-5
 1 3 6 8 11 TO 14
NGENERATION  NSTEP=100 XSTEP=-10
 2 4 5 7 9 10 15
*
MATERIAL  1  ELASTIC E=3.E7
*
EGROUP  1  SOLID
ENODES
 ENTRIES  EL  N1  N2  N3  N4  N5  N6  N7  N8 N9   N10 N11  N12 ,
              N13 N14 N15 N16 N17 N18 N19 N20
           1  106 101 1   6   108 103 3   8  104  51  4    56 ,
              105 53  5   58  107 102 2   7
           2  111 106 6   11  112 108 8   12 109  56  9    61 ,
              110 58  10  62  0   107 7
*
EGROUP  2  SHELL  RINT=2 SINT=2 TINT=-3
ENODES
 ENTRIES  EL  N1 N2 N3  N4  N6 N8 N17 N24 N20  N5 N13 N7
          1   11 14 114 111 64 61  12  62  112 13 63  113
GSURFACE  14 15 115 114 EL1=6 NODES=9
THICKNESS  1 T1=2.703 T2=2.174 T3=2.174 T4=2.703
THICKNESS  2 T1=2.174 T2=1.818 T3=1.818 T4=2.174
THICKNESS  3 T1=1.818 T2=1.563 T3=1.563 T4=1.818
THICKNESS  4 T1=1.563 T2=1.370 T3=1.370 T4=1.563
THICKNESS  5 T1=1.370 T2=1.220 T3=1.220 T4=1.370
THICKNESS  6 T1=1.220 T2=1.099 T3=1.099 T4=1.220
THICKNESS  7 T1=1.099 T2=1.000 T3=1.000 T4=1.099
EDATA  / ENTRIES EL NTH
 1 1 TO 7 7
*
FIXBOUNDARIES     /  1  TO  4
FIXBOUNDARIES  6 INPUT=LINE /  15 115
*
LOADS ELEMENT TYPE=MOMENT INPUT=LINE
 15 115 edge -100
*
SET  MIDSURFACE=NO SMOOTHNESS=YES NSYMBOL=MYNODES
MESH  NNUMBERS=MYNODES VECTOR=LOAD
MESH  CONTOUR=THICKNESS BCODE=ALL ENUMBERS=YES
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A22  TAPERED CANTILEVER UNDER TIP LOAD
*
DATABASE  CREATE
*
WRITE  FILENAME='a22.lis'
SET  MIDSURFACE=NO ORIGINAL=YES DEFORMED=NO OUTLINE=YES
EGROUP 2
EPLINE ESHELL / 1 12 6 TO 7 12 6
*
MESH  VECTOR=SPRINCIPAL
MESH  CONTOUR=MISES
MESH  CONTOUR=SDEVIATION PLINES=ESHELL
*
AXIS 1 VMIN=0 VMAX=70  LABEL='SHELL DISTANCE'
AXIS 2 VMIN=0. VMAX=600 LABEL='STRESS-YY'
USERCURVE 1 / READ A22A.DAT
*
SET PLOTORIENTATION=PORTRAIT
ELINE ESHELL KIND=SYY SYMBOL=1 XAXIS=1 YAXIS=2 SUBFRAME=12
PLOT USERCURVE 1 XAXIS=-1 YAXIS=-2 SUNFRAME=OLD
ELINE ESHELL KIND=SYZ SYMBOL=1
*
NLIST  MYNODES DIRECTION=34
NLIST  KIND=REACTION DIRECTION=2
EMAX   SELECT=S-EFFECTIVE
END
```

EXAMPLE A23

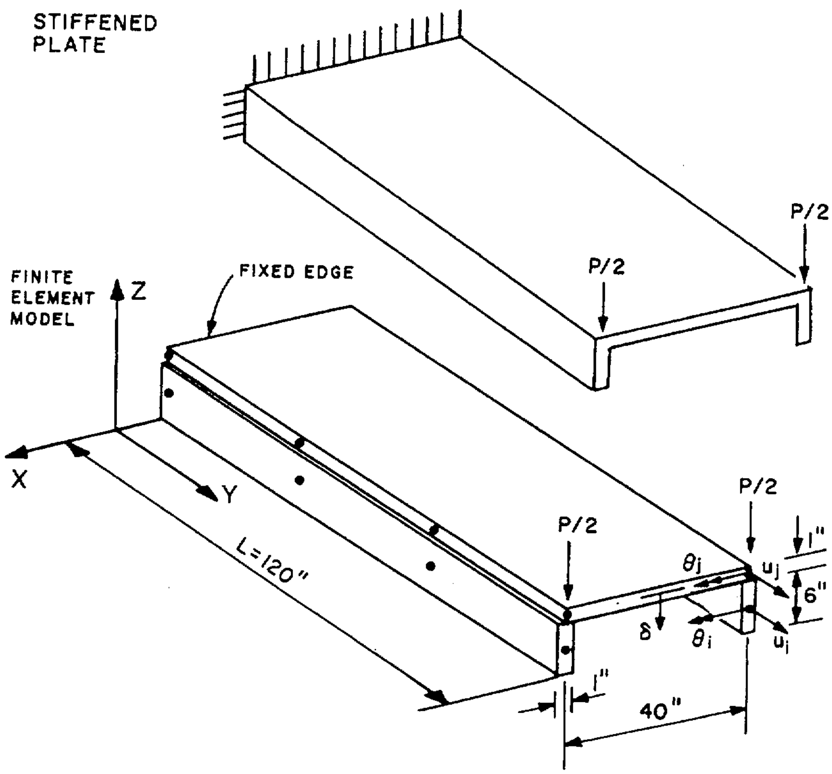
STIFFENED PLATE CANTILEVER UNDER TIP LOAD

Objective

To verify the use of the ISOBEAM element as a stiffener for the shell element using rigid links.

Physical Problem

A cantilever of channel cross-section is loaded by concentrated end loads as shown in the figure below.



$p = 200$ lbf

$E = 3.0 \cdot 10^7$ psi

$\nu = 0$

Finite Element Model

The model is shown in the figure on page A23.2. The nodes 1, 5, 9 and 13 are fixed. Each ISOBEAM node is coupled to the corresponding SHELL node using a rigid link. Using this model with a linear displacement assumption in the X-direction for the SHELL element, no shear-lag effects are modeled, which allows an easy comparison with theoretical results from beam theory. Poisson's ratio is for the same reason set to zero.

Solution Results

The theoretical displacement is

$$\delta = \frac{PL^3}{3EI}$$

The input data on page A23.4 gives the following result:

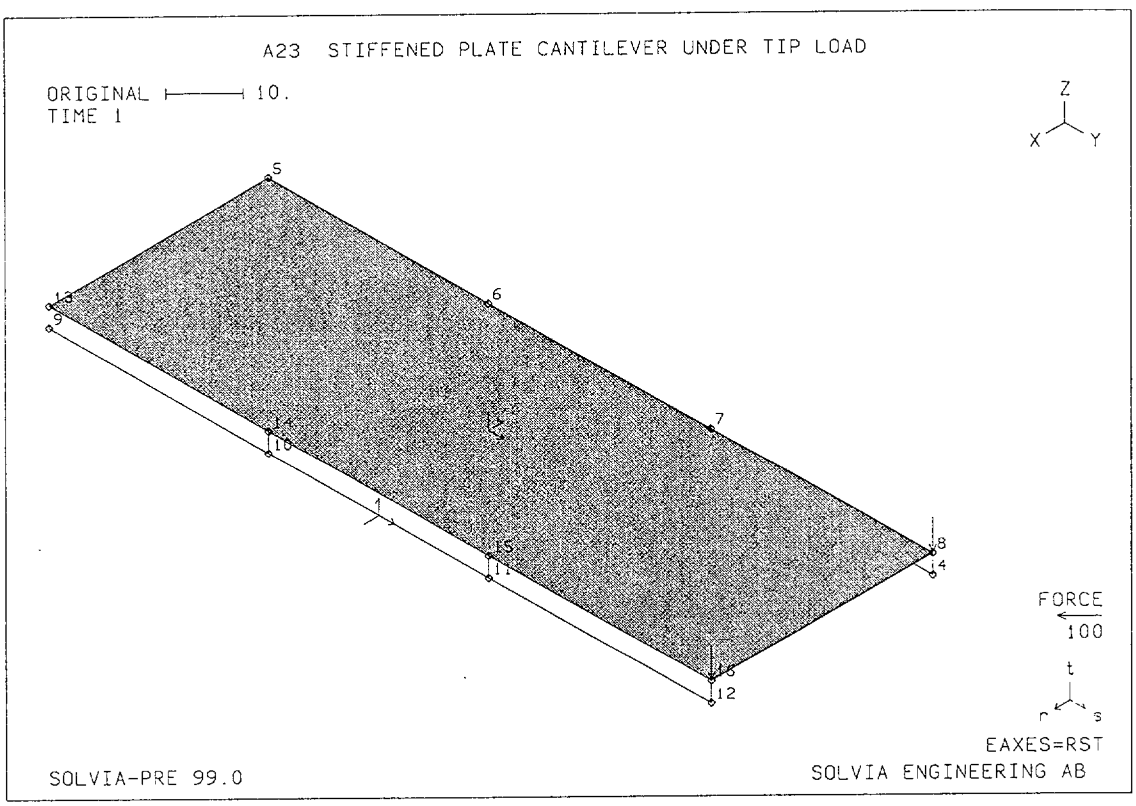
End displacement δ (inch):

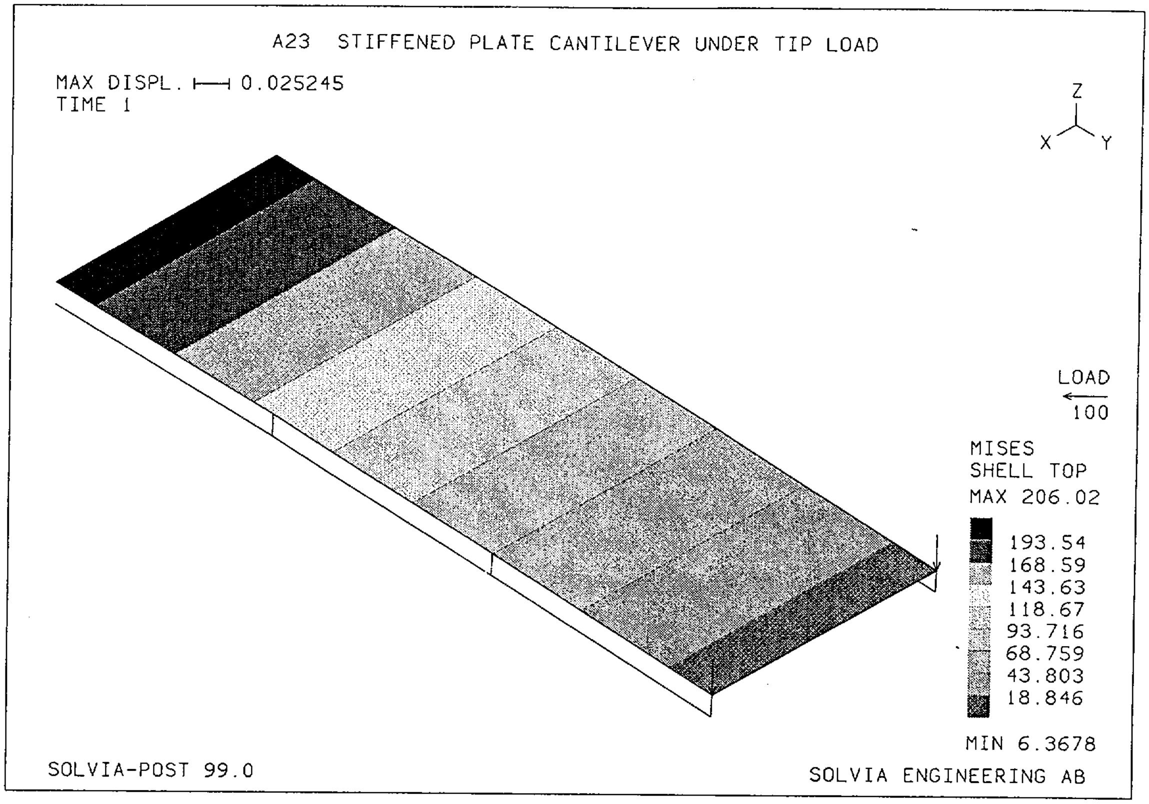
Theory	SOLVIA
0.02520	0.02523

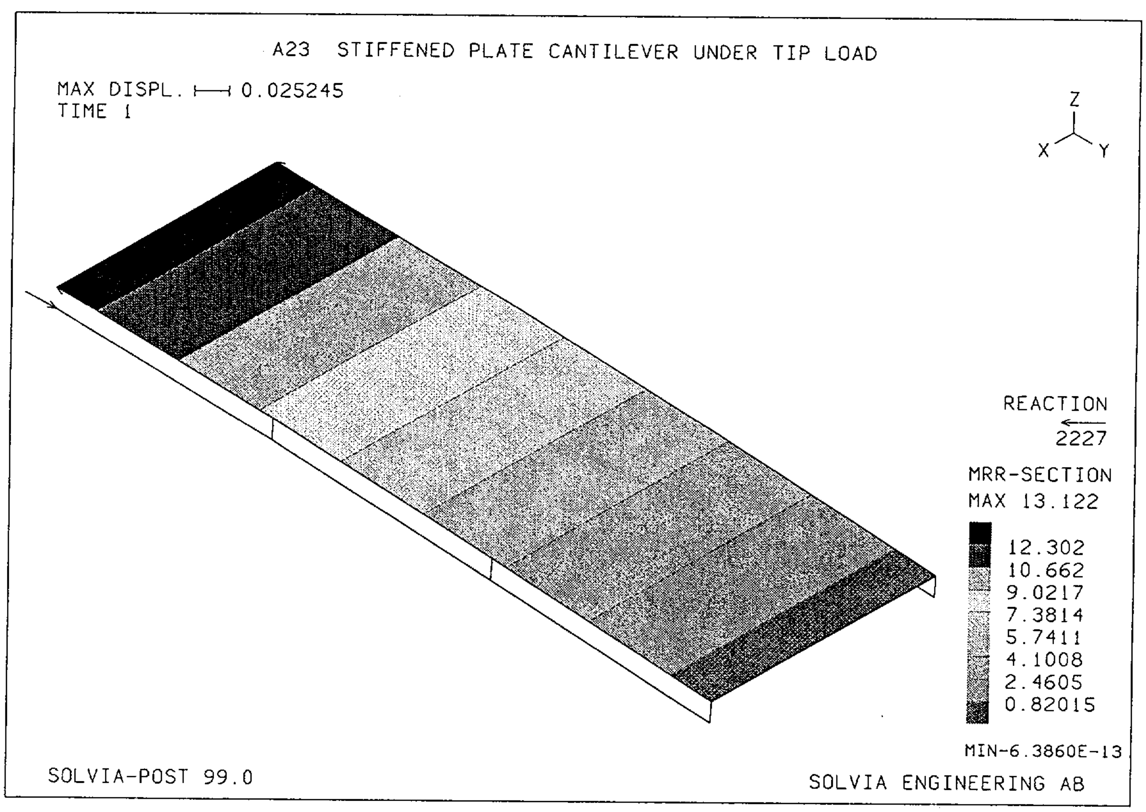
Contour plots of the effective stress and the bending section moment in the SHELL portion are shown in the figures on page A23.3.

User Hints

- Note that SOLVIA-PRE automatically assigns GLOBAL rotations for the shell nodes connected to rigid links and for nodes with a specified boundary condition for rotation. The procedure is further described in the command SHELLNODES of the SOLVIA-PRE 99.0 Users Manual, Stress Analysis, p. 7.23.




A23 STIFFENED PLATE CANTILEVER UNDER TIP LOAD
MAX DISPL. 0.025245
TIME 1
Z
X
Y
LOAD
100
MISES
SHELL TOP
MAX 206.02
193.54
168.59
143.63
118.67
93.716
68.759
43.803
18.846
MIN 6.3678
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A23 STIFFENED PLATE CANTILEVER UNDER TIP LOAD
MAX DISPL. 0.025245
TIME 1
Z
X
Y
REACTION
2227
MRR-SECTION
MAX 13.122
12.302
10.662
9.0217
7.3814
5.7411
4.1008
2.4605
0.82015
MIN-6.3860E-13
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A23  STIFFENED PLATE CANTILEVER UNDER TIP LOAD'
*
DATABASE  CREATE
*
COORDINATES
  1   0. 0. 3.    TO   4    0. 120. 3.
  5   0. 0. 6.5   TO   8    0. 120. 6.5
  9  40. 0. 3.    TO  12   40. 120. 3.
 13  40. 0. 6.5   TO  16   40. 120. 6.5
*
MATERIAL  1  ELASTIC E=3.E7
*
EGROUP  1  SHELL  STRESSREFERENCE=GLOBAL
THICKNESS  1  1.0
ENODES
 ENTRIES EL N1 N2 N3 N4 N5 N7 N9 N11
          1  8  5 13 16  7 14  6  15
*
EGROUP  2  ISOBEAM
SECTION  1  SDIM=6. TDIM=1.
STRESSTABLE  1  111 121 112 122,
                211 221 212 222,
                311 321 312 322
ENODES
 ENTRIES EL AUX N1 N2 N3 N4
          1  5   1  4  2  3
          2 13   9 12 10 11
LOADS  CONCENTRATED
 8 3 -100  /  16 3 -100
*
FIXBOUNDARIES  /  1 5 13 9
RIGIDLINK
 2 6  /  3 7  /  4 8  /  10 14  /  11 15  /  12 16
*
MESH  EAXES=RST NSYMBOLS=YES NNUMBERS=YES VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A23  STIFFENED PLATE CANTILEVER UNDER TIP LOAD
*
DATABASE  CREATE
*
WRITE  FILENAME='a23.lis'
*
MESH  CONTOUR=MISES VECTOR=LOAD
MESH  CONTOUR=MRR VECTOR=REACTION
*
NLIST  DIRECTION=234
NLIST  KIND=REACTION DIRECTION=234
EMAX  SELECT=S-EFFECTIVE
END
```

EXAMPLE A24

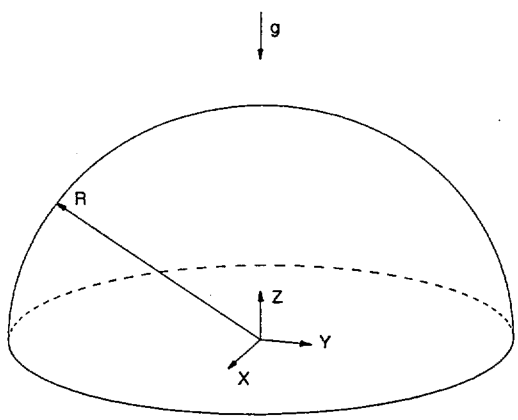
ANALYSIS OF SPHERICAL DOME UNDER SELF WEIGHT

Objective

To verify the PLANE STRESS3 (membrane) element to model a spherical surface when subjected to gravity loading.

Physical Problem

A spherical roof (hemisphere) subjected to gravity loading is considered as shown in the figure below.



$R = 4.5$ m
$h = 0.08$ m
$E = 2.0 \cdot 10^{10}$ N/m^2
$\nu = 0.3$
$\rho = 3000$ kg/m^3
$g = 9.81$ m/s^2

Finite Element Model

A sector of 30 degrees of the spherical surface is modeled using 80 parabolic PLANE STRESS3 elements. SKEW degree of freedom Systems are used to define the boundary conditions in the non-global circumferential direction as seen in the figures on pages A24.2 and A24.3. A consistent mass matrix assumption is used in the analysis.

Solution Results

The analytical solution to this problem can be found in many text books, e.g., [1]. For the membrane solution of this problem it is important that the support reactions are acting in the tangent direction to the meridians.

A Local Spherical System is defined with the x_1 and x_2 axis in the meridional and the tangential directions, respectively. The input data of pages A24.4 and A24.5 gives the following results:

At the top of the spherical dome (Z = 4.5)

	Stress-11	Stress-22 [kPa]
Analytical	-66.2	-66.2
SOLVIA	-66.4	-66.4

At the support of the spherical dome (Z = 0)

	Stress-11	Stress-22 [kPa]
Analytical	-132.4	132.4
SOLVIA	-131.4	132.9

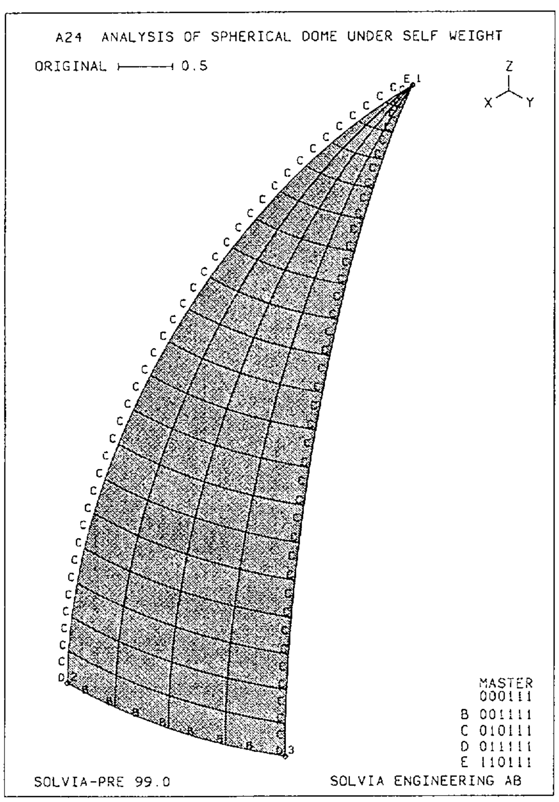
Contour plots and stress distributions along the spherical surface can be seen in the figures on pages A24.3 and A24.4.

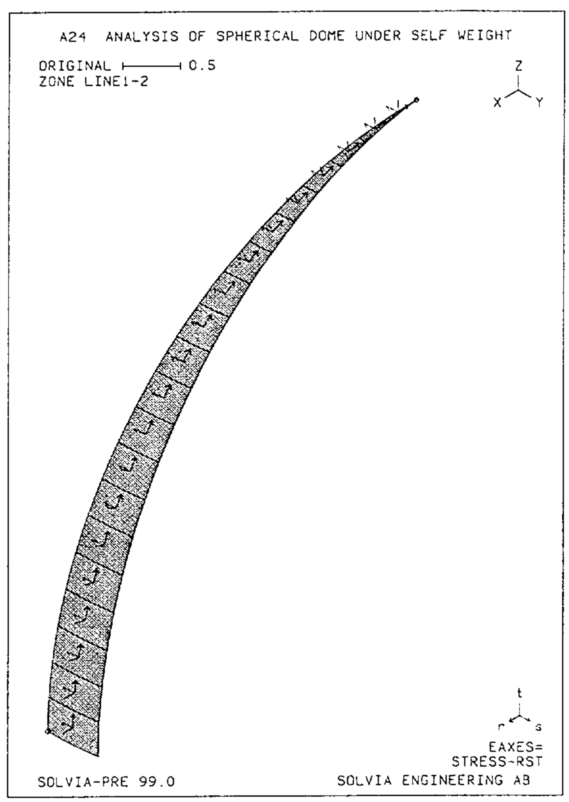
User Hints

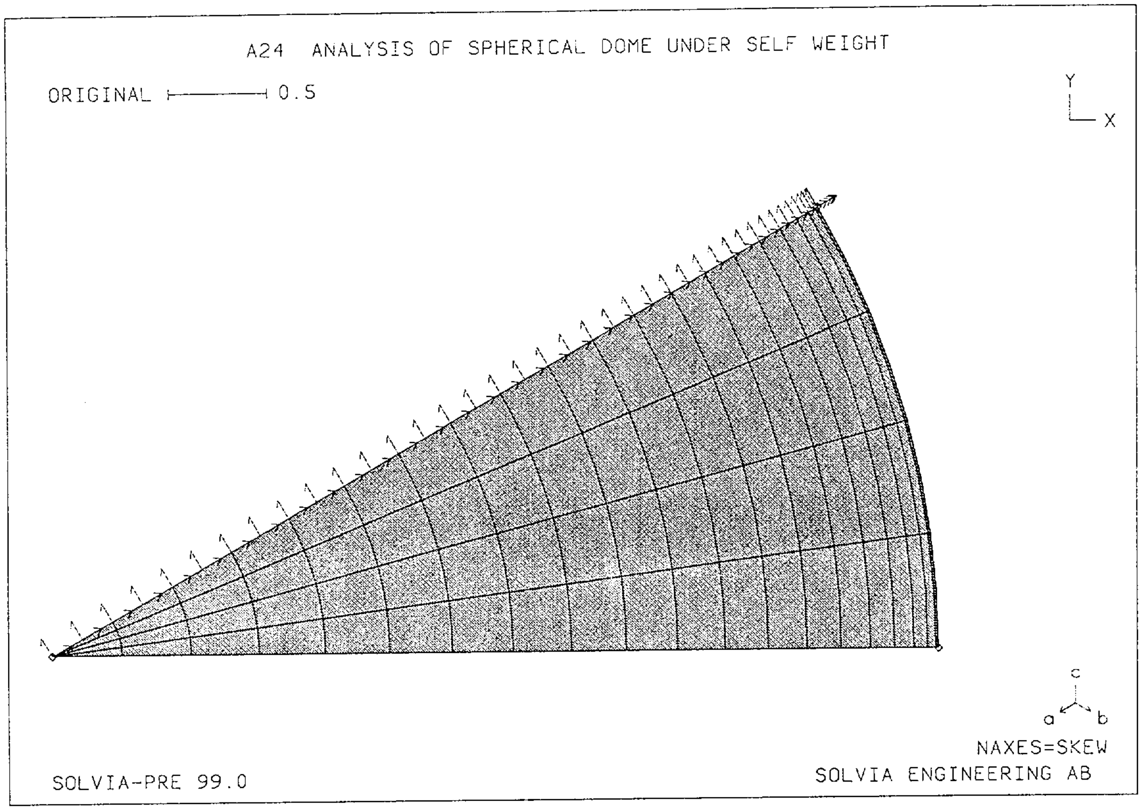
- A relative large number of elements must be used in the meridional direction to describe the stress variation in this example. In the circumferential direction, however, only one element can be used.
- Note that a flat PLANE STRESS3 element has no stiffness in the direction orthogonal to its plane in a small displacement analysis.

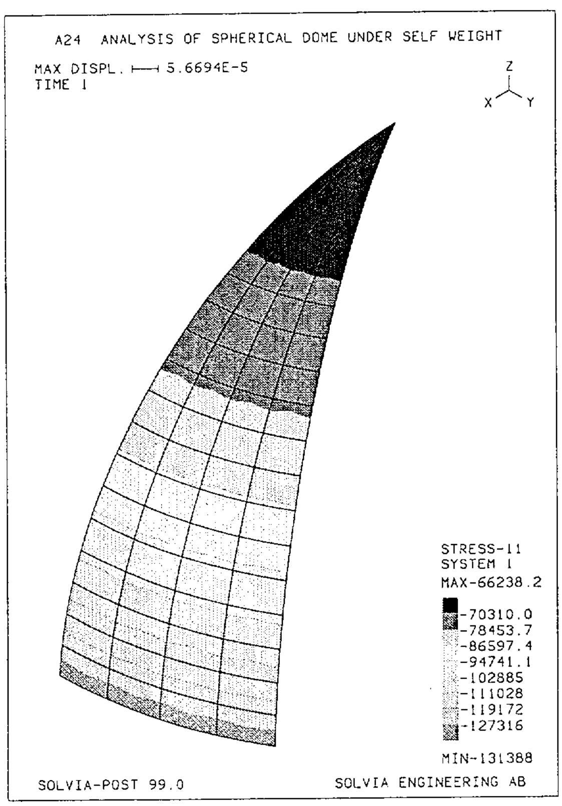
Reference

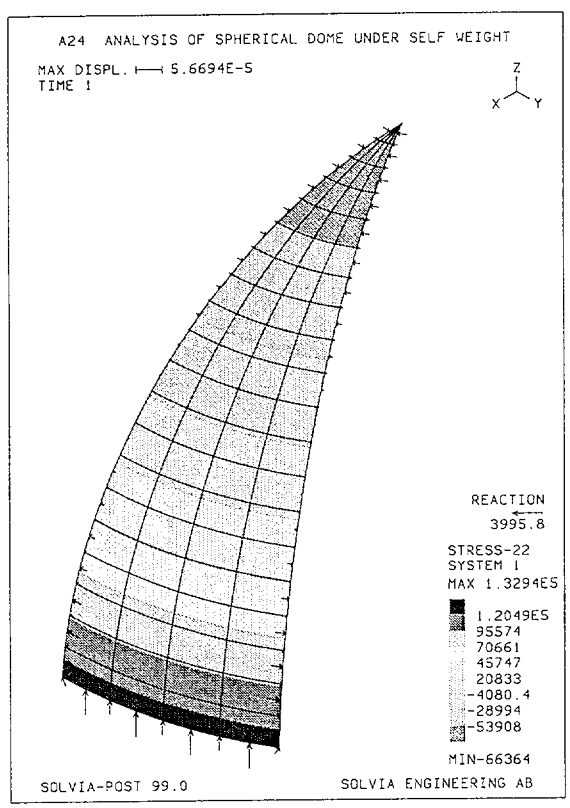
[1] Timoshenko, S.P. and Woinowsky-Krieger, S., Theory of Plates and Shells, Second Edition, McGraw-Hill, 1959, p. 436.

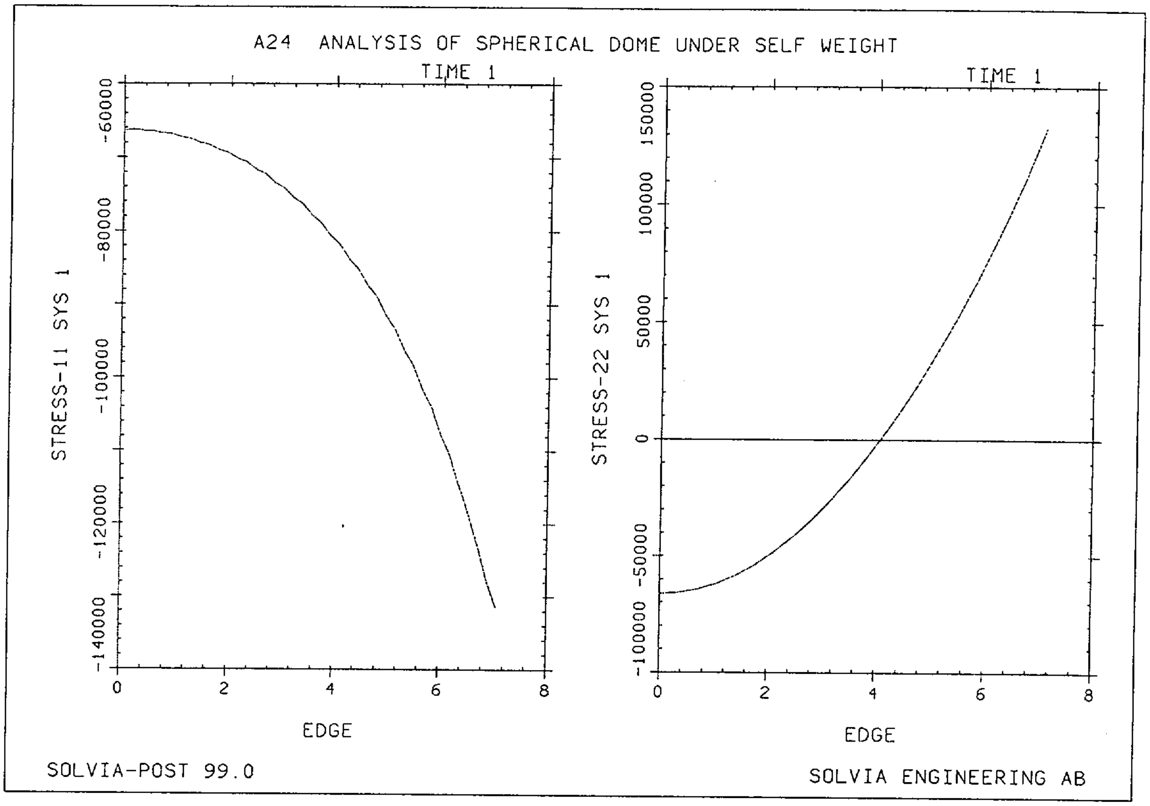





A24 ANALYSIS OF SPHERICAL DOME UNDER SELF WEIGHT
ORIGINAL 0.5
Y
X
c
a
b
NAXES=SKEW
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A24 ANALYSIS OF SPHERICAL DOME UNDER SELF WEIGHT
MAX DISPL. 5.6694E-5
TIME 1
Z
X
Y
STRESS-11
SYSTEM 1
MAX-66238.2
-70310.0
-78453.7
-86597.4
-94741.1
-102885
-111028
-119172
-127316
MIN-131388
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A24 ANALYSIS OF SPHERICAL DOME UNDER SELF WEIGHT
MAX DISPL. 5.6694E-5
TIME 1
Z
X
Y
REACTION
3995.8
STRESS-22
SYSTEM 1
MAX 1.3294E5
1.2049E5
95574
70661
45747
20833
-4080.4
-28994
-53908
MIN-66364
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB



SOLVIA-PRE input

```
HEADING  'A24  ANALYSIS OF SPHERICAL DOME UNDER SELF WEIGHT'
*
DATABASE  CREATE
*
MASTER  IDOF=000111
ANALYSIS  TYPE=STATIC MASSMATRIX=CONSISTENT
*
SYSTEM  1 SPHERICAL
COORDINATES
ENTRIES NODE  R     THETA    PHI
          1   4.5    0.       0.
          2   4.5    0.      90.
          3   4.5   30.      90.
*
SKEWSYSTEMS  VECTORS
 1  0.8660254 0.5 0.  -0.5 0.8660254 0.
*
MATERIAL  1  ELASTIC  E=2.E10 NU=0.3 DENSITY=3000.
*
EGROUP  1  PLANE STRESS3  RESULTS=NSTRESSES
EDATA  /  1  0.08
GSURFACE  1 2 3 1  EL1=20 EL2=4 NODES=8 SYSTEM=1 BLENDING=ANGLES
*
NSKEWS  INPUT=LINE  /  1 3  1
*
```

SOLVIA-PRE input (cont.)

```
FIXBOUNDARIES  2  INPUT=LINES  /  1 2  /  1 3
FIXBOUNDARIES  3  INPUT=LINES  /  2 3
FIXBOUNDARIES  1  INPUT=NODES  /  1
*
LOADS  MASSPROPORTIONAL  ZFACTOR=1. ACCGRA=-9.81
*
SET  NSYMBOLS=MYNODES PLOTORIENTATION=PORTRAIT
MESH  BCODE=ALL NNUMBERS=MYNODES
ZONE  NAME=LINE1-2 INPUT=ELEMENTS  /  1 TO 20
MESH  ZONENAME=LINE1-2 EAXES=STRESS-RST
SET  PLOTORIENTATION=LANDSCAPE
MESH  VIEW=Z NAXES=SKEW
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A24  ANALYSIS OF SPHERICAL DOME UNDER SELF WEIGHT
*
DATABASE  CREATE
SYSTEM 1 SPHERICAL
*
WRITE  FILENAME='a24.lis'
*
EPLINE  NAME=EDGE
 1  1 5 2  TO  20  1 5 2
*
SET  PLOTORIENTATION=PORTRAIT
MESH  CONTOUR=S11 SYSTEM=1
MESH  CONTOUR=S22 VECTOR=REACTION SYSTEM=1
*
SET  PLOTORIENTATION=LANDSCAPE
ELINE  LINENAME=EDGE KIND=S11 OUTPUT=ALL SYSTEM=1 SUBFRAME=21
ELINE  LINENAME=EDGE KIND=S22 OUTPUT=ALL SYSTEM=1
*
NLIST  ZONENAME=N1
*
SUMMATION  KIND=LOAD
MASS-PROPERITES
END
```

EXAMPLE A25

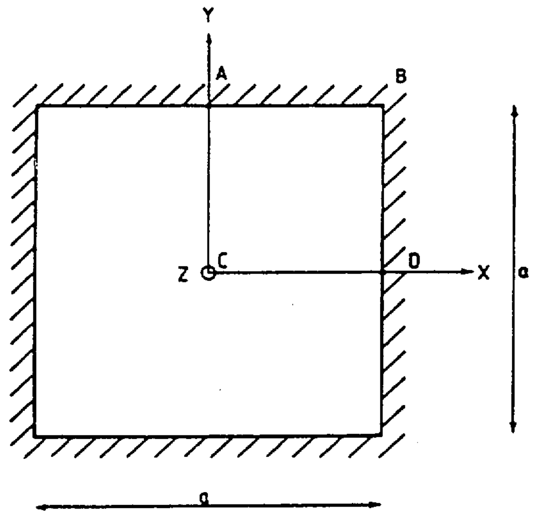
CLAMPED SQUARE PLATE UNDER PRESSURE LOAD

Objective

To verify the behaviour of the SHELL element subjected to pressure load.

Physical Problem

A thin plate with clamped edges under pressure load as shown in the figure below is considered. The load is acting in the negative Z-direction.



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

$\nu = 0.3$

$a = 1.8 \ m$

$h = 0.01 \ m$ (thickness)

$p = 1.0 \cdot 10^3 \ N/m^2$ (pressure)

Finite Element Model

Because of symmetry conditions only a quarter of the plate is considered, A-B-C-D in the figure above. Nine cubic SHELL elements are used for the finite element model shown in figures on page A25.3. The model allows no deformation in the X- and Y-directions and no rotation about the Z-axis.

Solution Results

The theoretical solution of this problem is given in [1] p. 197 and the expression for the central deflection is given as

$$w = 0.00126 \cdot p \cdot a^4 / D$$

where

p = pressure load

a = side length

$$D = \frac{Eh^3}{12(1-\nu^2)}$$

The expression for the top surface stress at points C and D are:

$$\sigma_{xxC} = \sigma_{yyC} = -0.0231 \cdot \frac{6 \cdot p \cdot a^2}{h^2}$$

$$\sigma_{xxD} = 0.0513 \cdot \frac{6 \cdot p \cdot a^2}{h^2}$$

The following numerical solutions have been obtained using the input data shown on page A25.5.

Case	Displ. (m) point C	Stress σ_{xx} (N/m^2) point C	point D
Theory	$-6.88 \cdot 10^{-4}$	$-4.49 \cdot 10^{6}$	$9.97 \cdot 10^{6}$
16-node SHELL, 3×3	$-6.91 \cdot 10^{-4}$	$-4.61 \cdot 10^{6}$	$9.23 \cdot 10^{6}$

In addition the following solutions have been obtained for comparison:

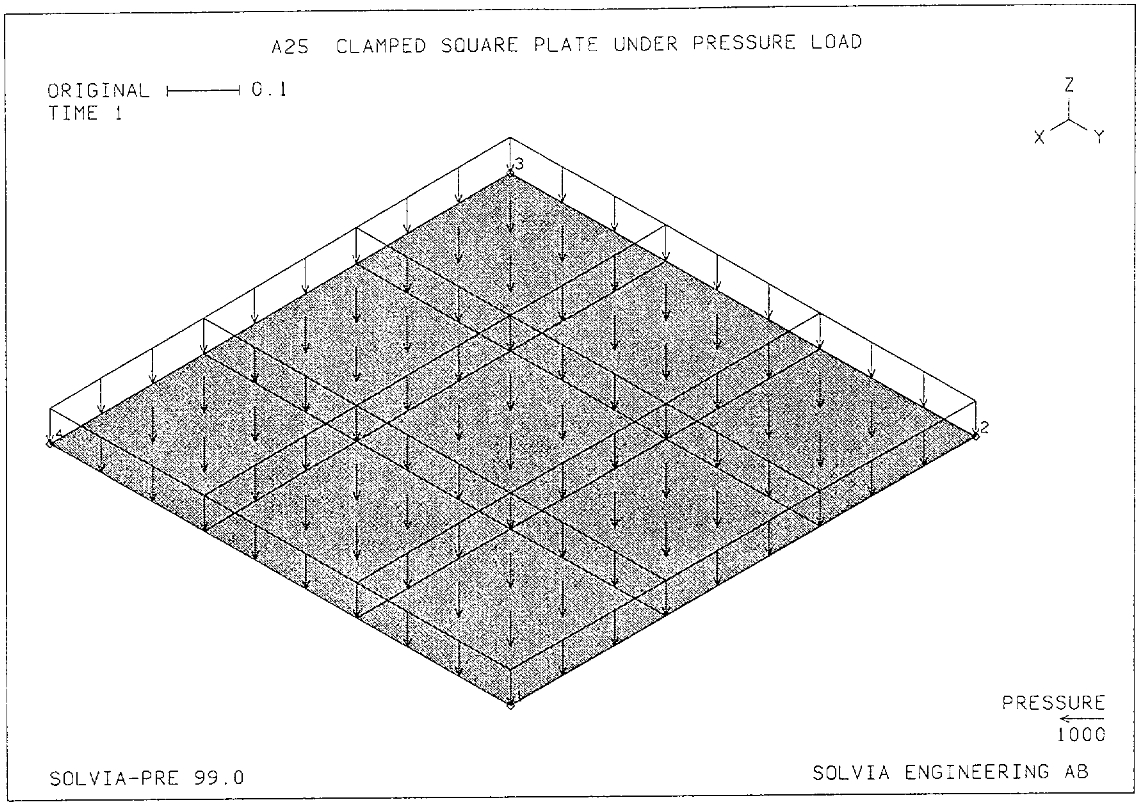
16-node SHELL, 6×6	$-6.91 \cdot 10^{-4}$	$-4.48 \cdot 10^{6}$	$9.79 \cdot 10^{6}$
4-node SHELL, 3×3	$-6.77 \cdot 10^{-4}$	$-4.60 \cdot 10^{6}$	$5.13 \cdot 10^{6}$
4-node SHELL, 6×6	$-6.88 \cdot 10^{-4}$	$-4.49 \cdot 10^{6}$	$7.08 \cdot 10^{6}$
4-node SHELL, 12×12	$-6.90 \cdot 10^{-4}$	$-4.46 \cdot 10^{6}$	$8.38 \cdot 10^{6}$
PLATE, 3×3 (quad.)	$-7.06 \cdot 10^{-4}$	$-4.61 \cdot 10^{6}$	$10.6 \cdot 10^{6}$
PLATE, 6×6 (quad.)	$-6.95 \cdot 10^{-4}$	$-4.49 \cdot 10^{6}$	$10.6 \cdot 10^{6}$

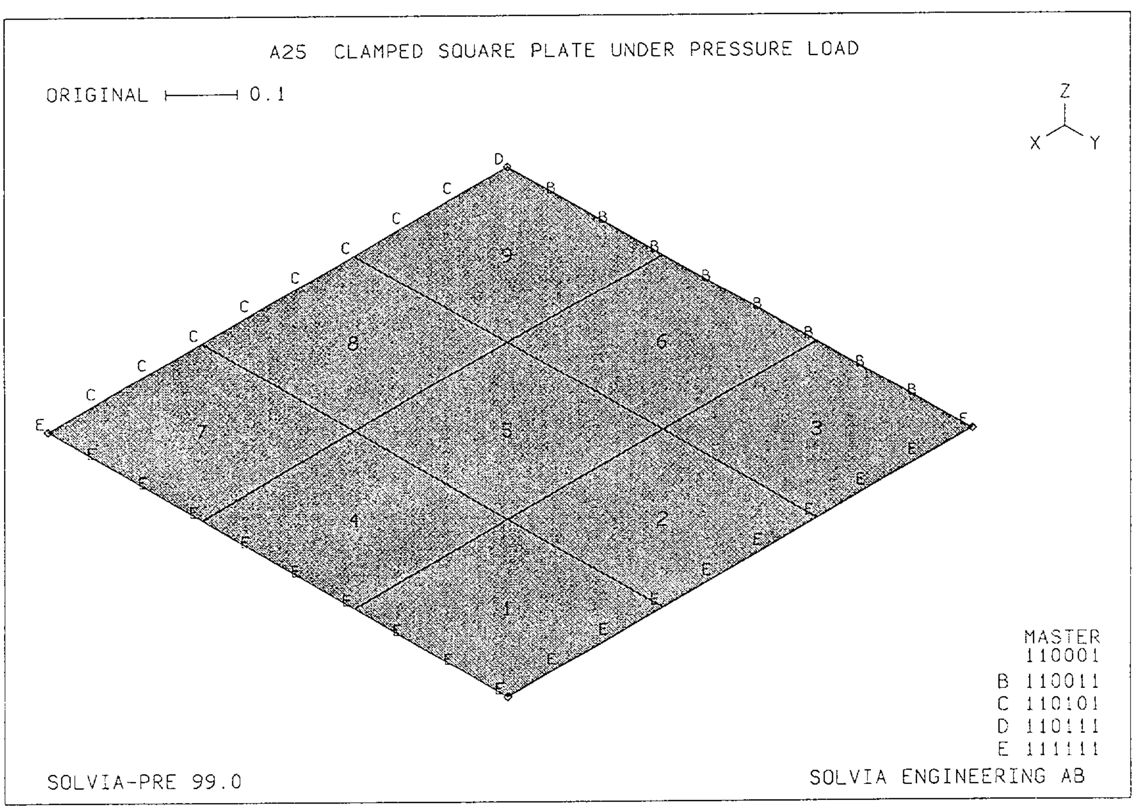
User Hints

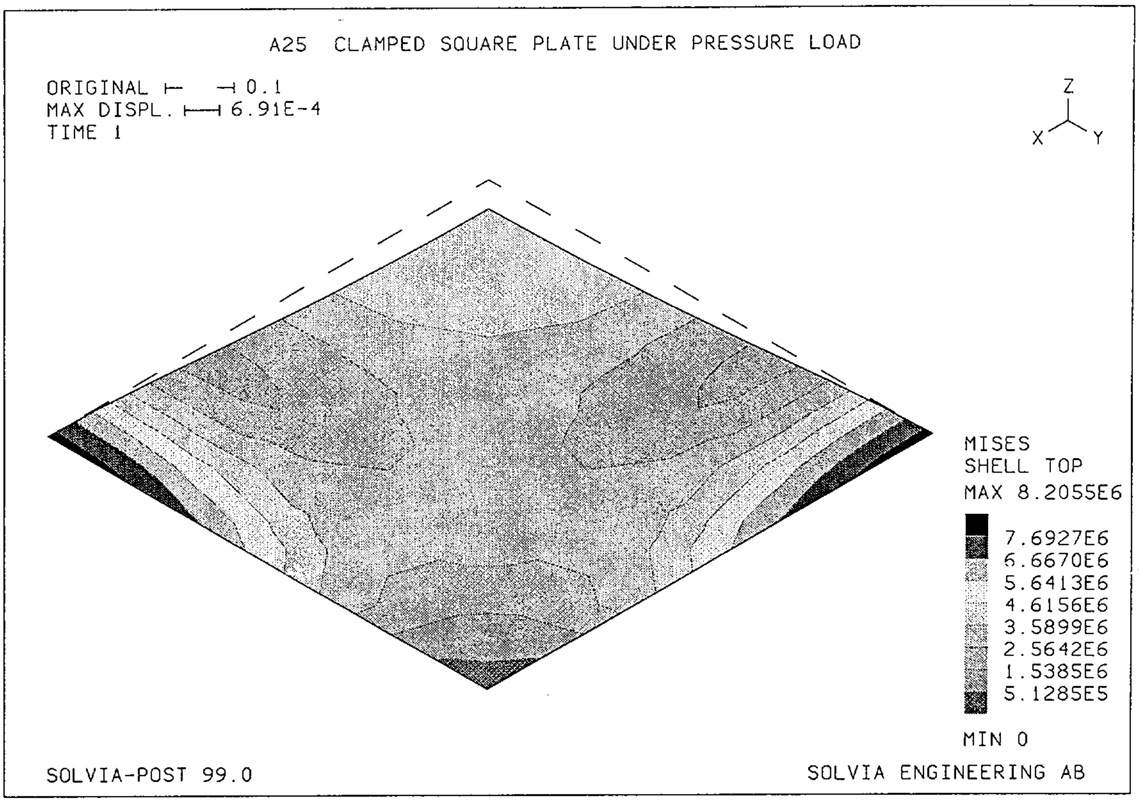
- Note that the 16-node SHELL elements provide a reliable and accurate solution.
- The solutions obtained using the 4-node SHELL element are good regarding displacements but more approximate regarding stresses. Considerable stress jumps occur between neighbouring elements since the stresses are rather constant over each element.
- The solutions obtained using PLATE elements are good but not as accurate as the solutions obtained by the 16-node SHELL elements. The PLATE element behaves well in bending. In membrane action the PLATE is, however, a constant strain element.
- Note that the 16-node SHELL solutions require much more CPU time than the 4-node SHELL solutions and the PLATE solutions.

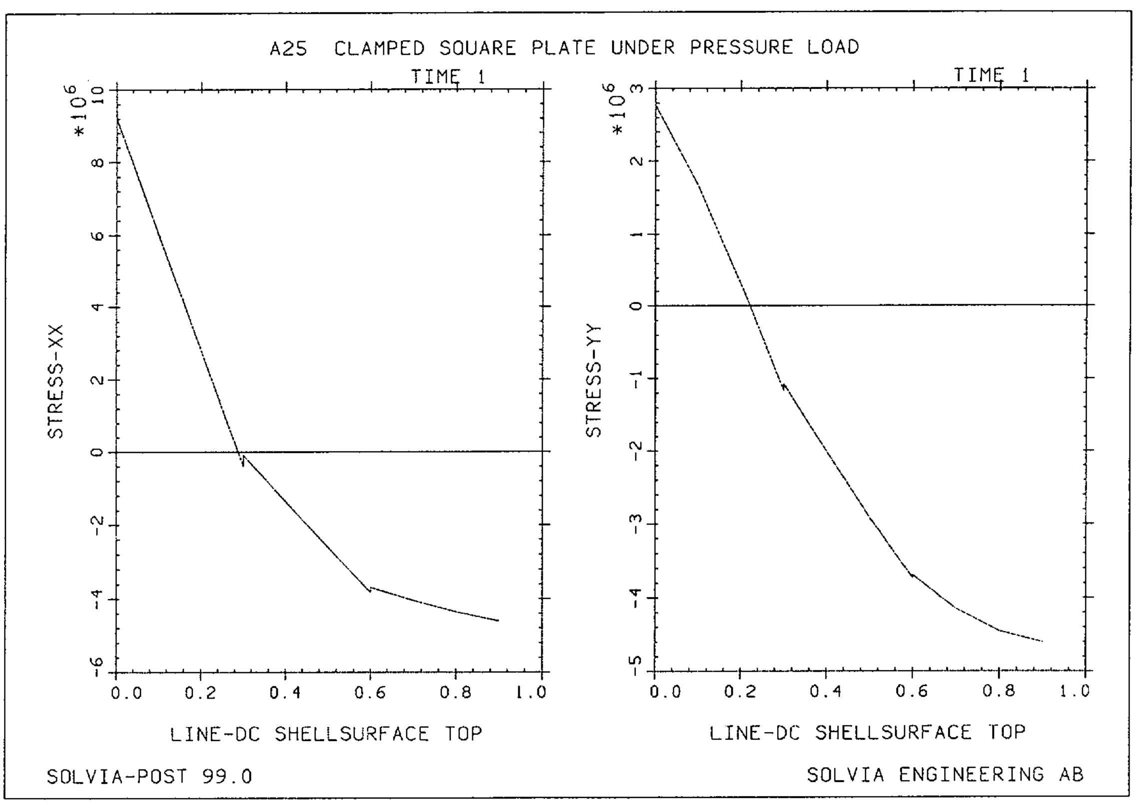
Reference

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


A25 CLAMPED SQUARE PLATE UNDER PRESSURE LOAD
ORIGINAL 0.1
TIME 1
Z
X Y
PRESSURE
1000
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A25 CLAMPED SQUARE PLATE UNDER PRESSURE LOAD
ORIGINAL 0.1
Z
X Y
MASTER
110001
B 110011
C 110101
D 110111
E 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A25 CLAMPED SQUARE PLATE UNDER PRESSURE LOAD
ORIGINAL 0.1
MAX DISPL. 6.91E-4
TIME 1
Z
X
Y
MISES
SHELL TOP
MAX 8.2055E6
7.6927E6
6.6670E6
5.6413E6
4.6156E6
3.5899E6
2.5642E6
1.5385E6
5.1285E5
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

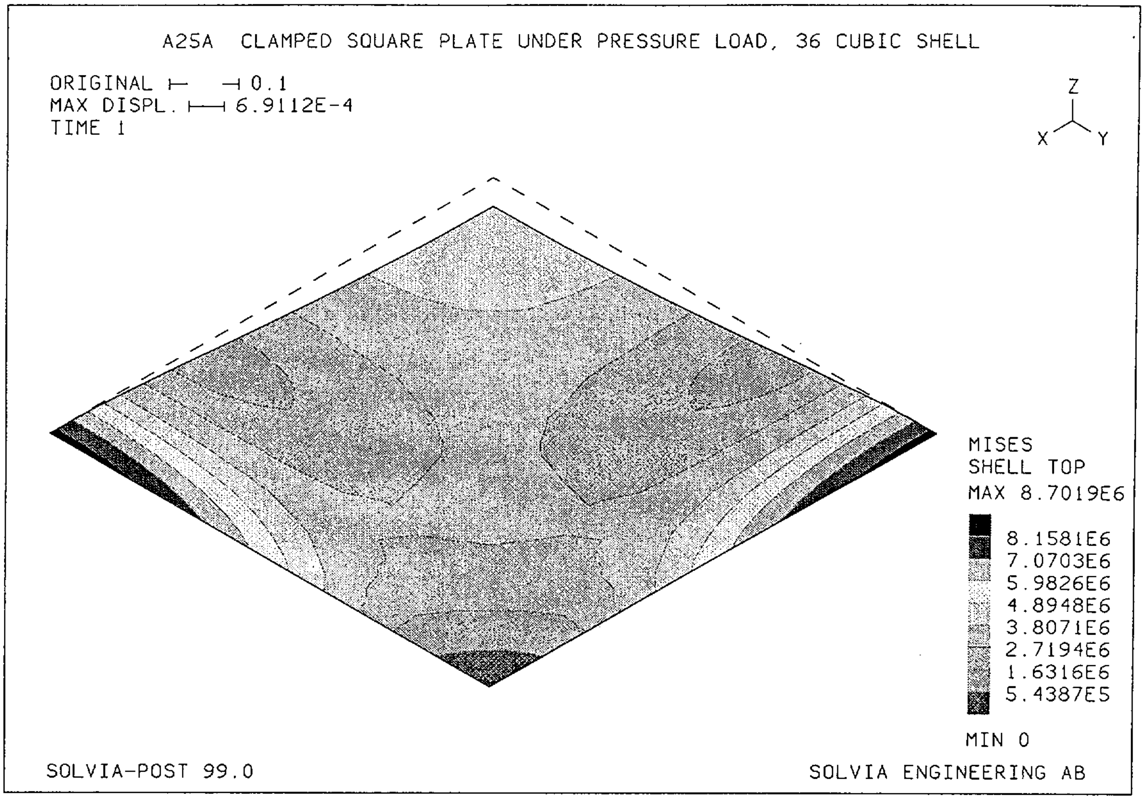

A25 CLAMPED SQUARE PLATE UNDER PRESSURE LOAD
TIME 1
TIME 1
*10^6
STRESS-XX
STRESS-YY
LINE-DC SHELLSURFACE TOP
LINE-DC SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

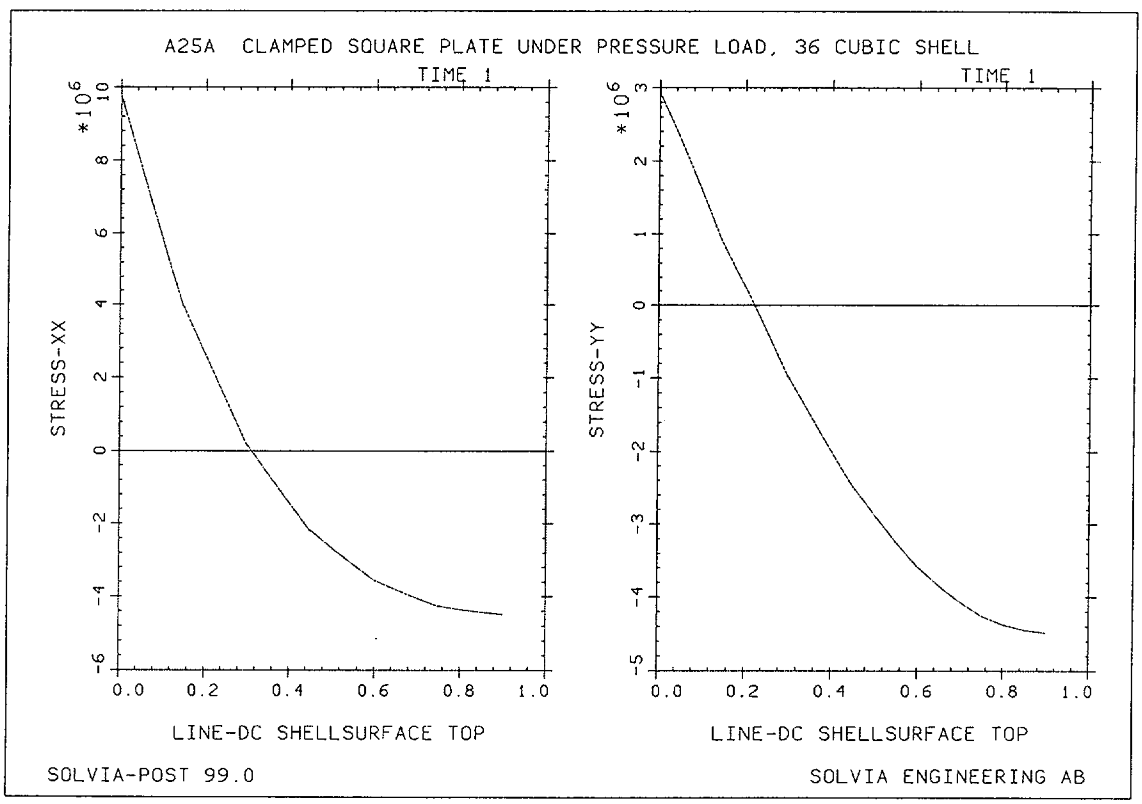
SOLVIA-PRE input

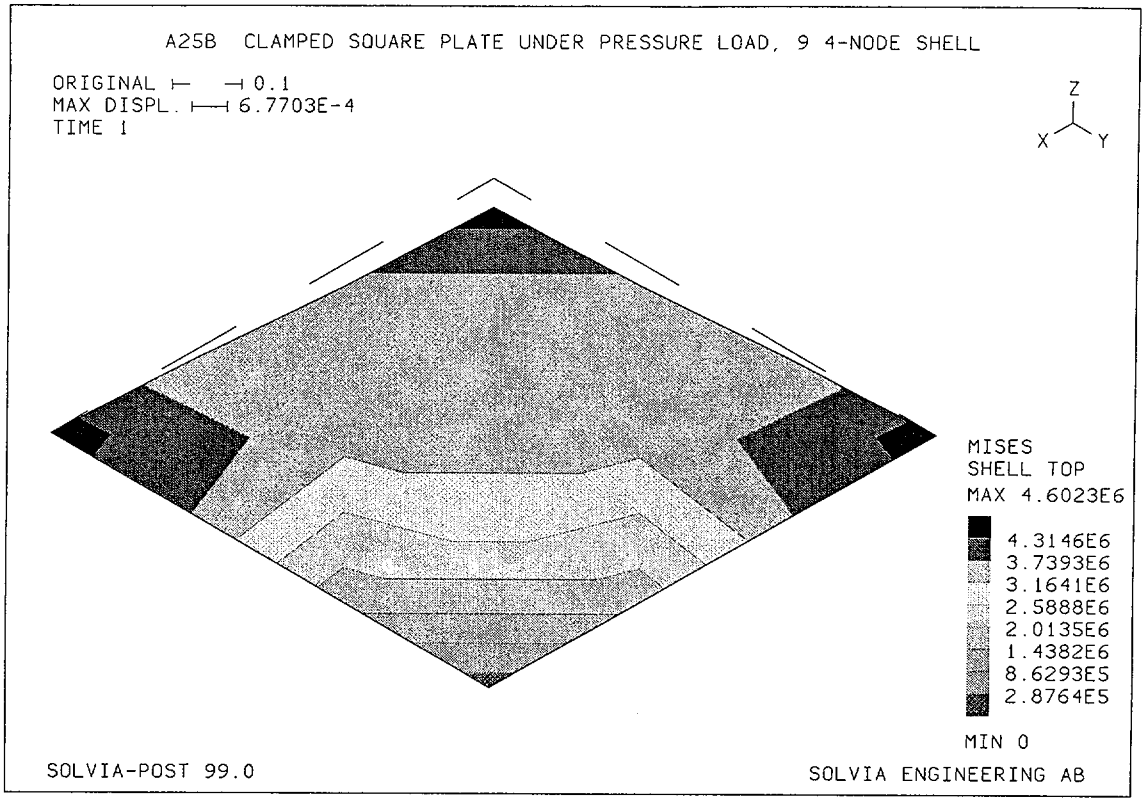
```
HEADING  'A25  CLAMPED SQUARE PLATE UNDER PRESSURE LOAD'
*
DATABASE  CREATE
*
PARAMETER  $ELX=3 $ELY=3 $SHELL=16
MASTER  IDOF=110001
COORDINATES
 1  0.9 0.9  /  2  0. 0.9  /  3  /  4  0.9
*
MATERIAL  1  ELASTIC E=2.1E11 NU=0.3
*
EGROUP  1  SHELL  STRESSREFERENCE=GLOBAL RESULTS=NSTRESSES
THICKNESS  1  0.01
GSURFACE  1 2 3 4  EL1=$ELX EL2=$ELY NODES=$SHELL
*
FIXBOUNDARIES    INPUT=LINES  /  1 2  /  1 4
FIXBOUNDARIES  4 INPUT=LINES  /  3 4
FIXBOUNDARIES  5 INPUT=LINES  /  2 3
*
LOADS  ELEMENT  TYPE=PRESSURE INPUT=SURFACE
 1 2 3 4  T  1000
*
SET  NSYMBOLS=MYNODES
MESH  VECTOR=LOAD NNUMBERS=MYNODES
MESH  ENUMBERS=YES BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

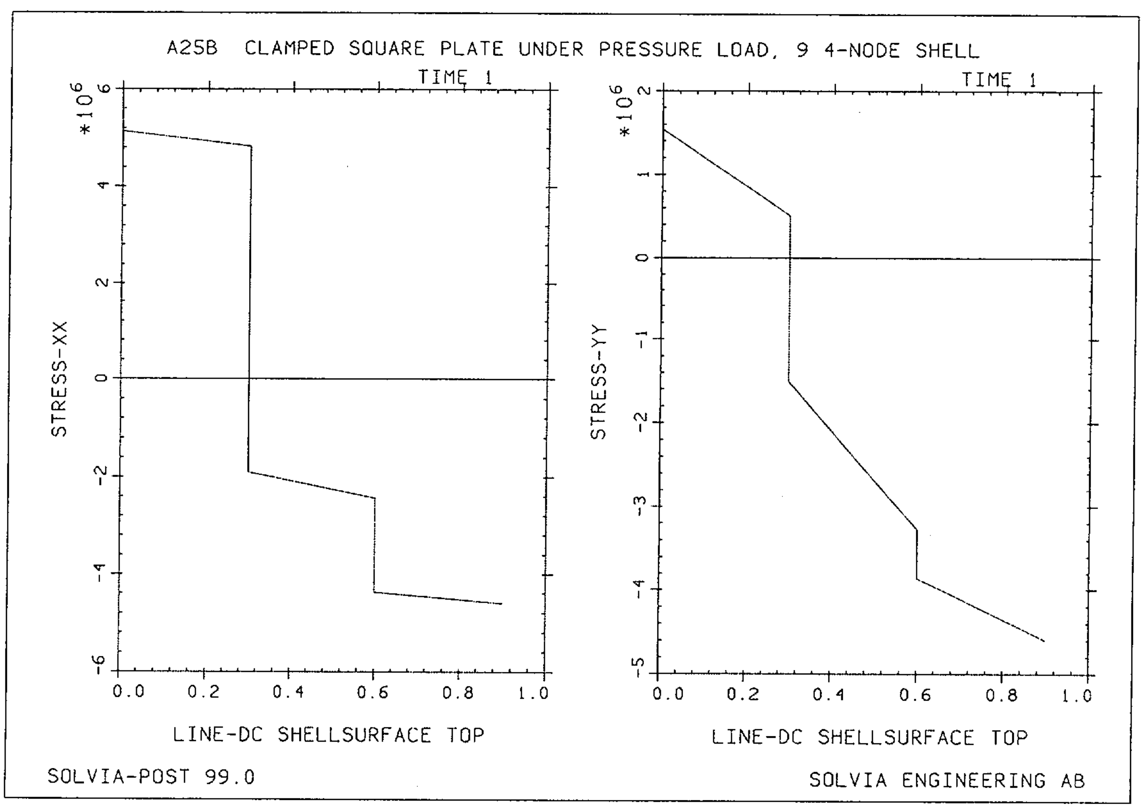
```
*  A25  CLAMPED SQUARE PLATE UNDER PRESSURE LOAD
*
DATABASE  CREATE
*
WRITE  FILENAME='a25.lis'
*
MESH  ORIGINAL=DASHED OUTLINE=YES CONTOUR=MISES
*
EPLINE  NAME=LINE-DC
 7  4 11 7 3  TO  9  4 11 7 3
ELINE  LINENAME=LINE-DC KIND=SXX OUTPUT=ALL SUBFRAME=21
ELINE  LINENAME=LINE-DC KIND=SYY OUTPUT=ALL
*
NMAX  NUMBER=5
END
```


A25A CLAMPED SQUARE PLATE UNDER PRESSURE LOAD, 36 CUBIC SHELL
ORIGINAL 0.1
MAX DISPL. 6.9112E-4
TIME 1
Z
X
Y
MISES
SHELL TOP
MAX 8.7019E6
8.1581E6
7.0703E6
5.9826E6
4.8948E6
3.8071E6
2.7194E6
1.6316E6
5.4387E5
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

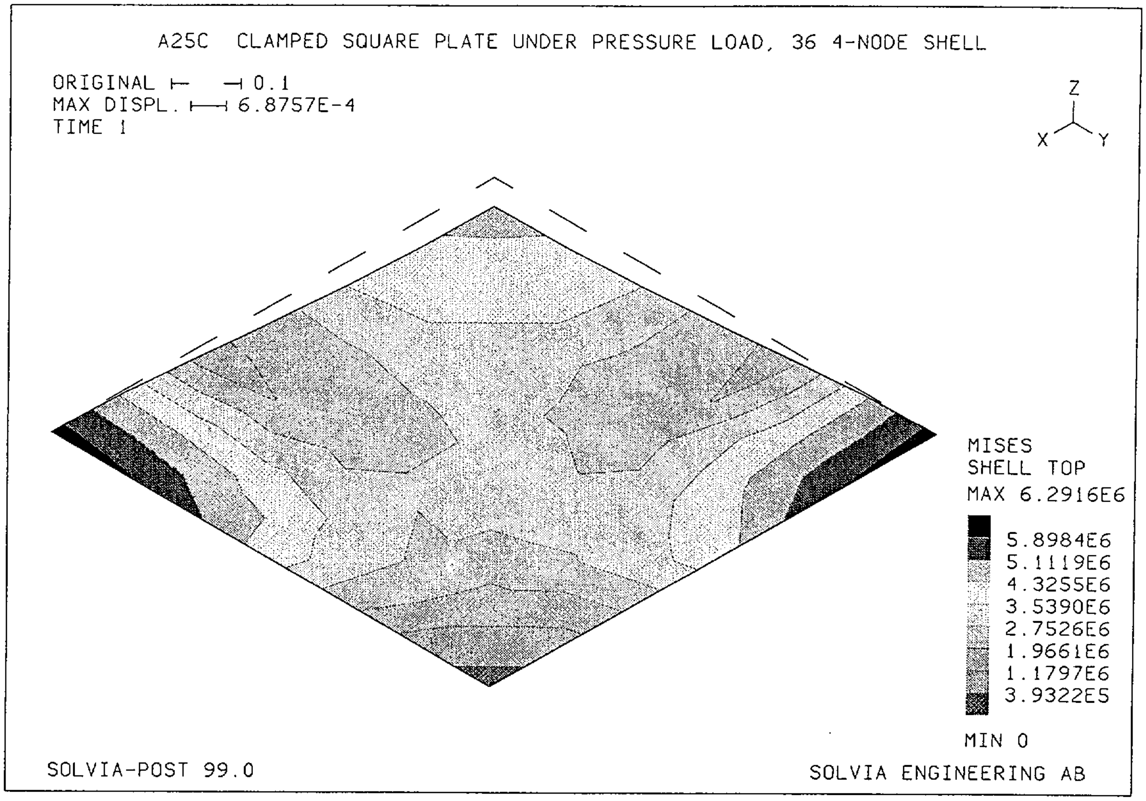

A25A CLAMPED SQUARE PLATE UNDER PRESSURE LOAD, 36 CUBIC SHELL
TIME 1
*10^6
STRESS-XX
10
8
6
4
2
0
-2
-4
-6
0.0
0.2
0.4
0.6
0.8
1.0
LINE-DC SHELLSURFACE TOP
TIME 1
*10^6
STRESS-YY
3
2
1
0
-1
-2
-3
-4
-5
0.0
0.2
0.4
0.6
0.8
1.0
LINE-DC SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

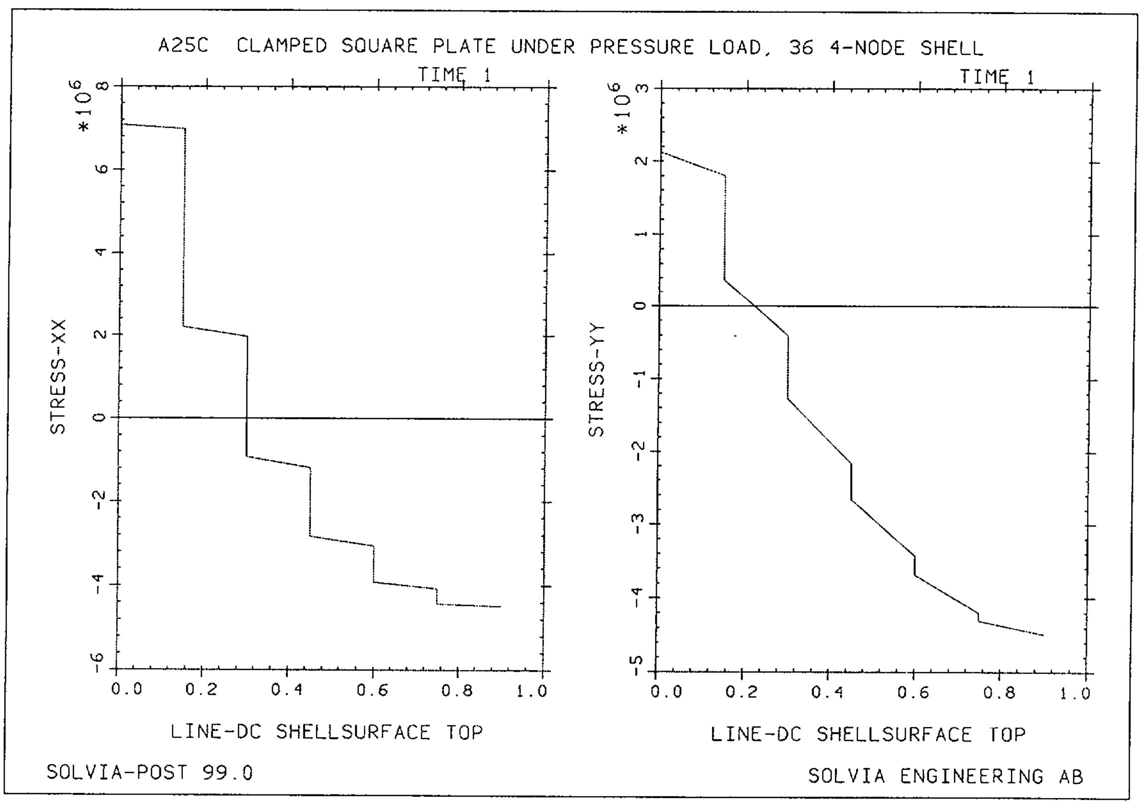


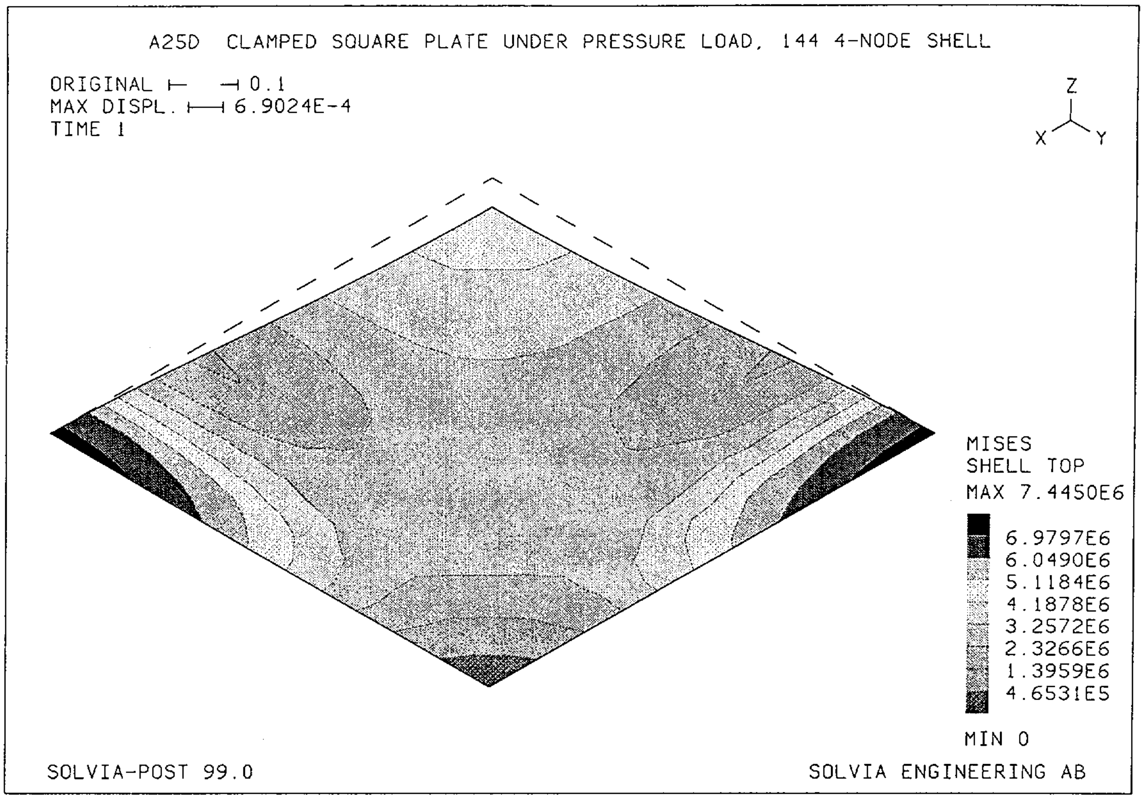
A25B CLAMPED SQUARE PLATE UNDER PRESSURE LOAD, 9 4-NODE SHELL
ORIGINAL 0.1
MAX DISPL. 6.7703E-4
TIME 1
Z
X
Y
MISES
SHELL TOP
MAX 4.6023E6
4.3146E6
3.7393E6
3.1641E6
2.5888E6
2.0135E6
1.4382E6
8.6293E5
2.8764E5
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

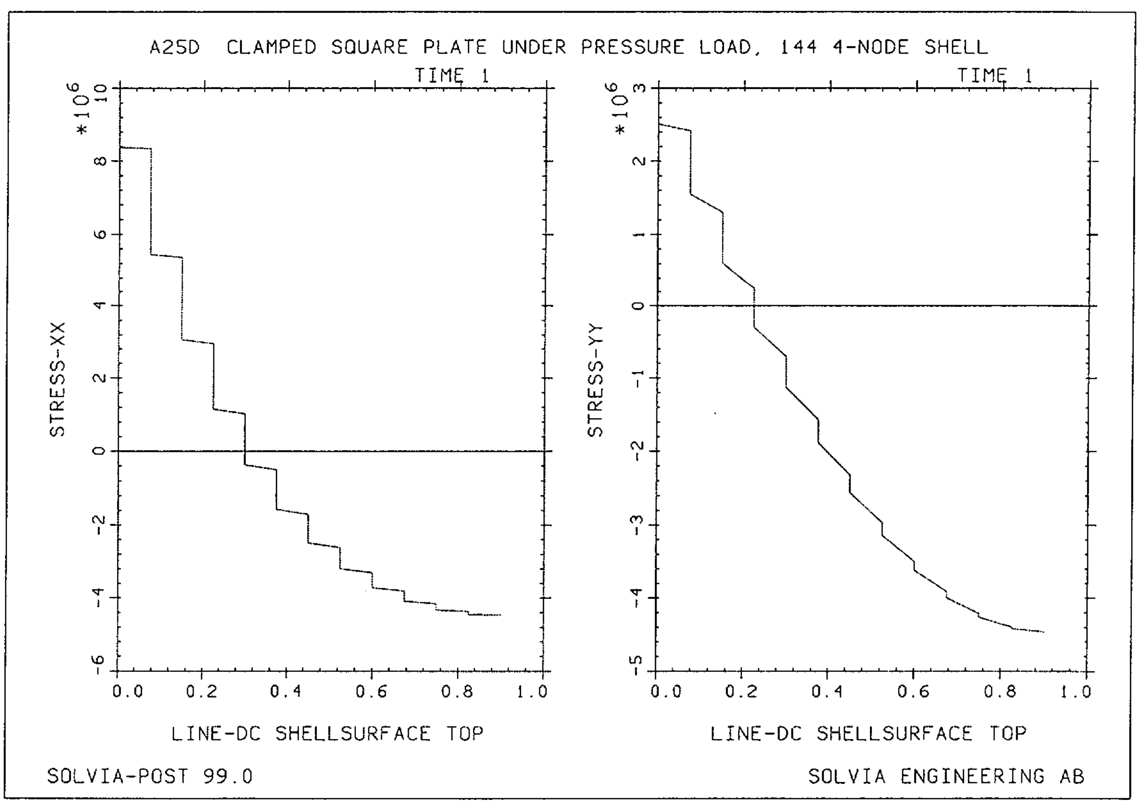


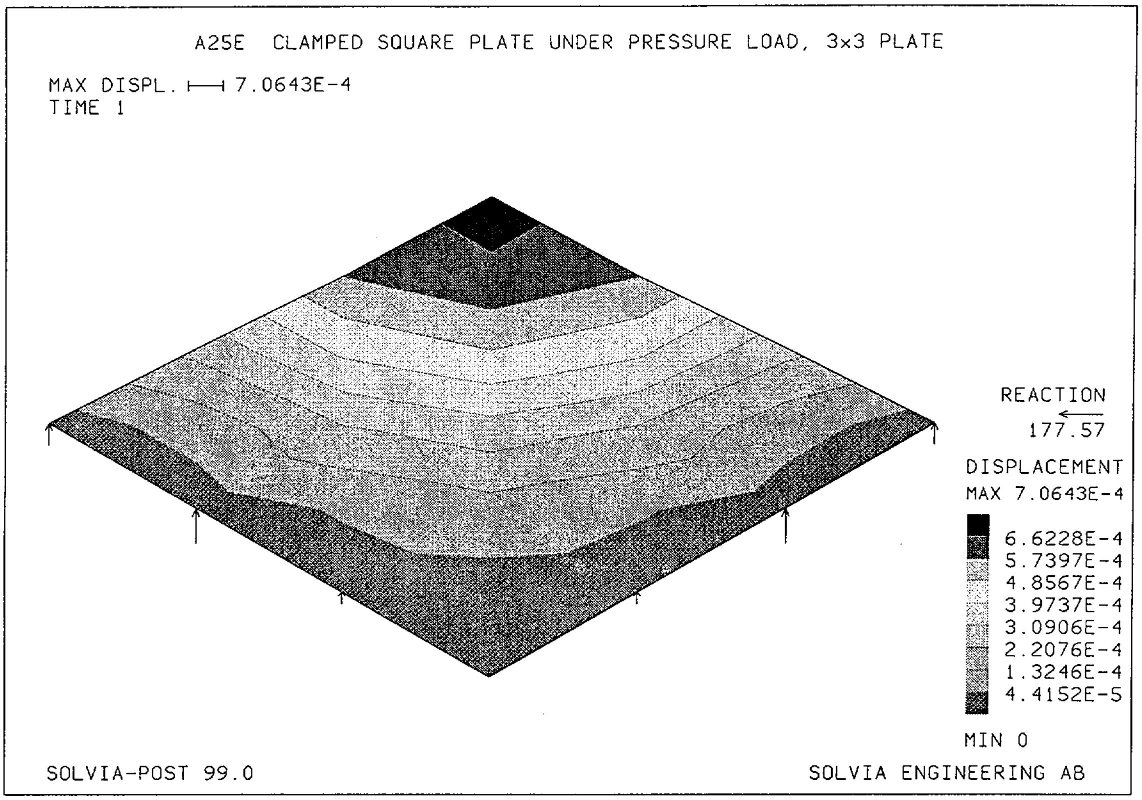
A25B CLAMPED SQUARE PLATE UNDER PRESSURE LOAD, 9 4-NODE SHELL
TIME 1
*10^6
STRESS-XX
LINE-DC SHELLSURFACE TOP
TIME 1
*10^6
STRESS-YY
LINE-DC SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

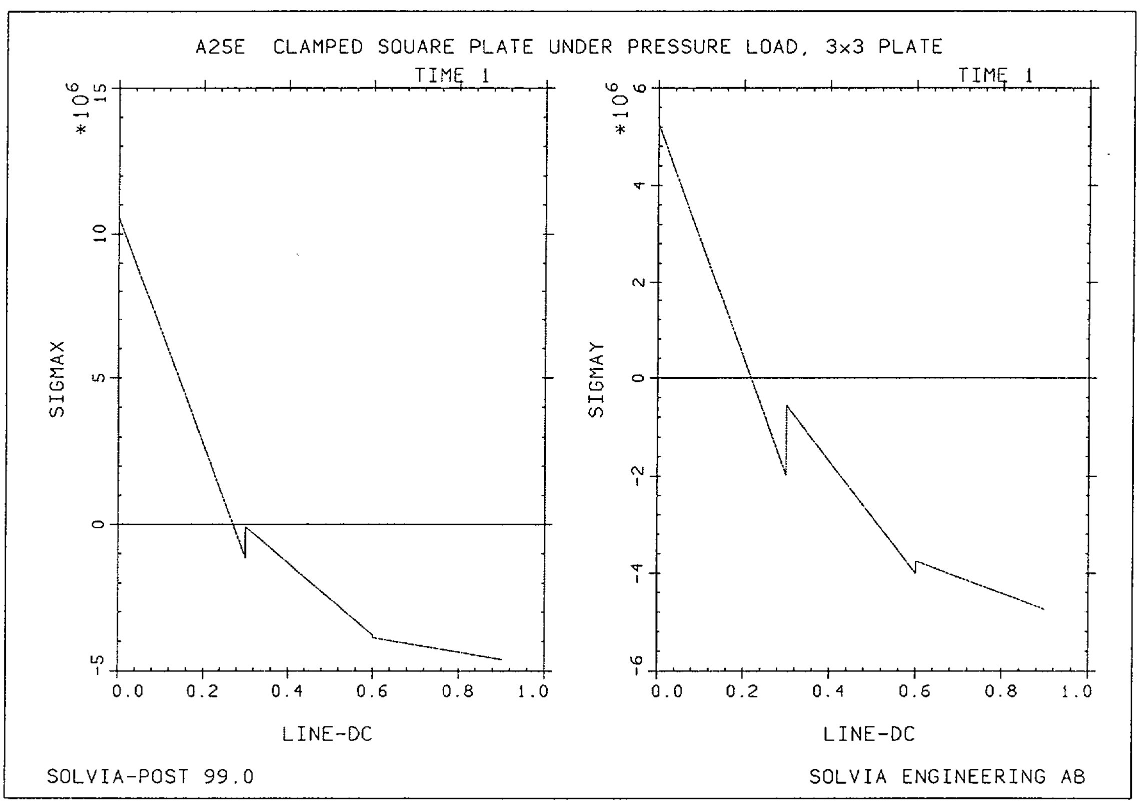

A25C CLAMPED SQUARE PLATE UNDER PRESSURE LOAD, 36 4-NODE SHELL
ORIGINAL 0.1
MAX DISPL. 6.8757E-4
TIME 1
Z
X
Y
MISES
SHELL TOP
MAX 6.2916E6
5.8984E6
5.1119E6
4.3255E6
3.5390E6
2.7526E6
1.9661E6
1.1797E6
3.9322E5
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

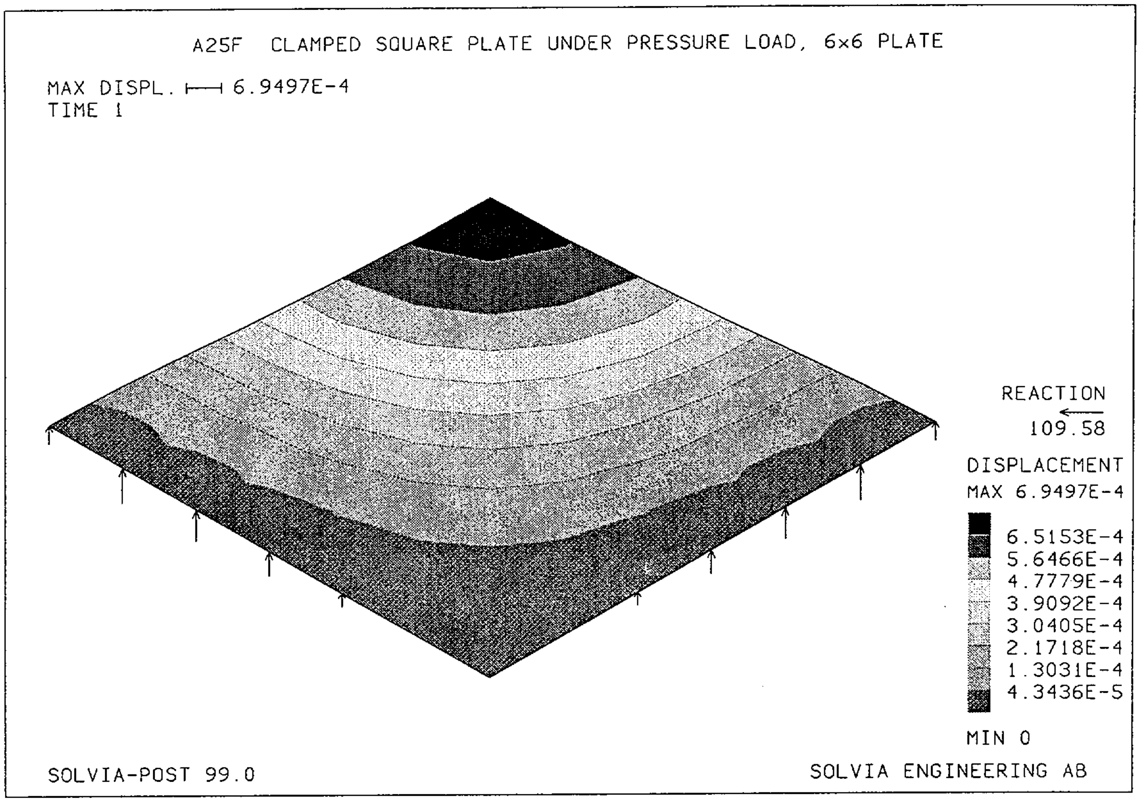

A25C CLAMPED SQUARE PLATE UNDER PRESSURE LOAD, 36 4-NODE SHELL
TIME 1
*10^6
STRESS-XX
LINE-DC SHELLSURFACE TOP
TIME 1
*10^6
STRESS-YY
LINE-DC SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

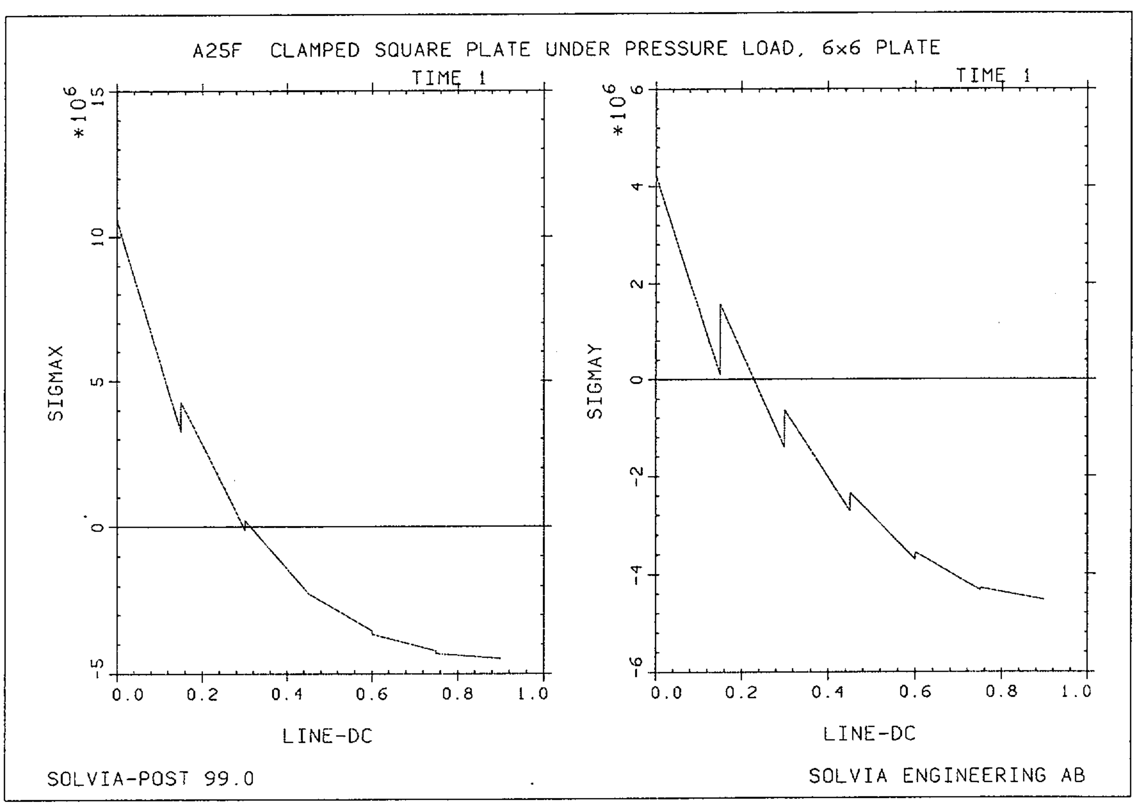

A25D CLAMPED SQUARE PLATE UNDER PRESSURE LOAD, 144 4-NODE SHELL
ORIGINAL 0.1
MAX DISPL. 6.9024E-4
TIME 1
Z
X
Y
MISES
SHELL TOP
MAX 7.4450E6
6.9797E6
6.0490E6
5.1184E6
4.1878E6
3.2572E6
2.3266E6
1.3959E6
4.6531E5
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A25D CLAMPED SQUARE PLATE UNDER PRESSURE LOAD, 144 4-NODE SHELL
TIME 1
*10^6
STRESS-XX
LINE-DC SHELLSURFACE TOP
TIME 1
*10^6
STRESS-YY
LINE-DC SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A25E CLAMPED SQUARE PLATE UNDER PRESSURE LOAD, 3x3 PLATE
MAX DISPL. 7.0643E-4
TIME 1
REACTION
177.57
DISPLACEMENT
MAX 7.0643E-4
6.6228E-4
5.7397E-4
4.8567E-4
3.9737E-4
3.0906E-4
2.2076E-4
1.3246E-4
4.4152E-5
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A25E CLAMPED SQUARE PLATE UNDER PRESSURE LOAD, 3x3 PLATE
TIME 1
*10^6
SIGMAX
LINE-DC
TIME 1
*10^6
SIGMAY
LINE-DC
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A25F CLAMPED SQUARE PLATE UNDER PRESSURE LOAD, 6x6 PLATE
MAX DISPL. 6.9497E-4
TIME 1
REACTION
109.58
DISPLACEMENT
MAX 6.9497E-4
6.5153E-4
5.6466E-4
4.7779E-4
3.9092E-4
3.0405E-4
2.1718E-4
1.3031E-4
4.3436E-5
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A25F CLAMPED SQUARE PLATE UNDER PRESSURE LOAD, 6x6 PLATE
TIME 1
TIME 1
*10^6
*10^6
SIGMAX
SIGMAY
LINE-DC
LINE-DC
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

EXAMPLE A26

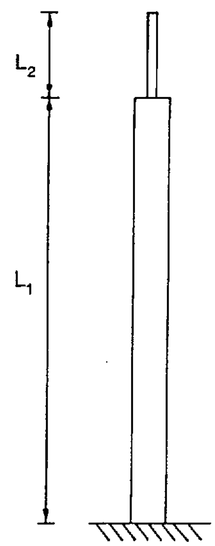
MATERIAL DAMPING IN MODAL SUPERPOSITION

Objective

To demonstrate the calculation of weighted modal damping factors from modal damping factors of materials in a structure.

Physical Problem

A beam structure with a light top part and a heavy bottom part as shown in the figure below is considered in one plane. The natural frequencies and the weighted modal damping factors are calculated.



Top part of the structure
$L_2 = 1$ m

Rectangular section 5×50 mm

$E_2 = 2 \cdot 10^{10}$ N/m^2

$\nu_2 = 0$

$\rho_2 = 1000$ kg/m^3

Modal damping $\zeta_2 = 5\%$

Bottom part of the structure
$L_1 = 5$ m

Rectangular section 50×50 mm

$E_1 = 2 \cdot 10^{11}$ N/m^2

$\nu_1 = 0$

$\rho_1 = 7800$ kg/m^3

Modal damping $\zeta_1 = 1\%$

Finite Element Model

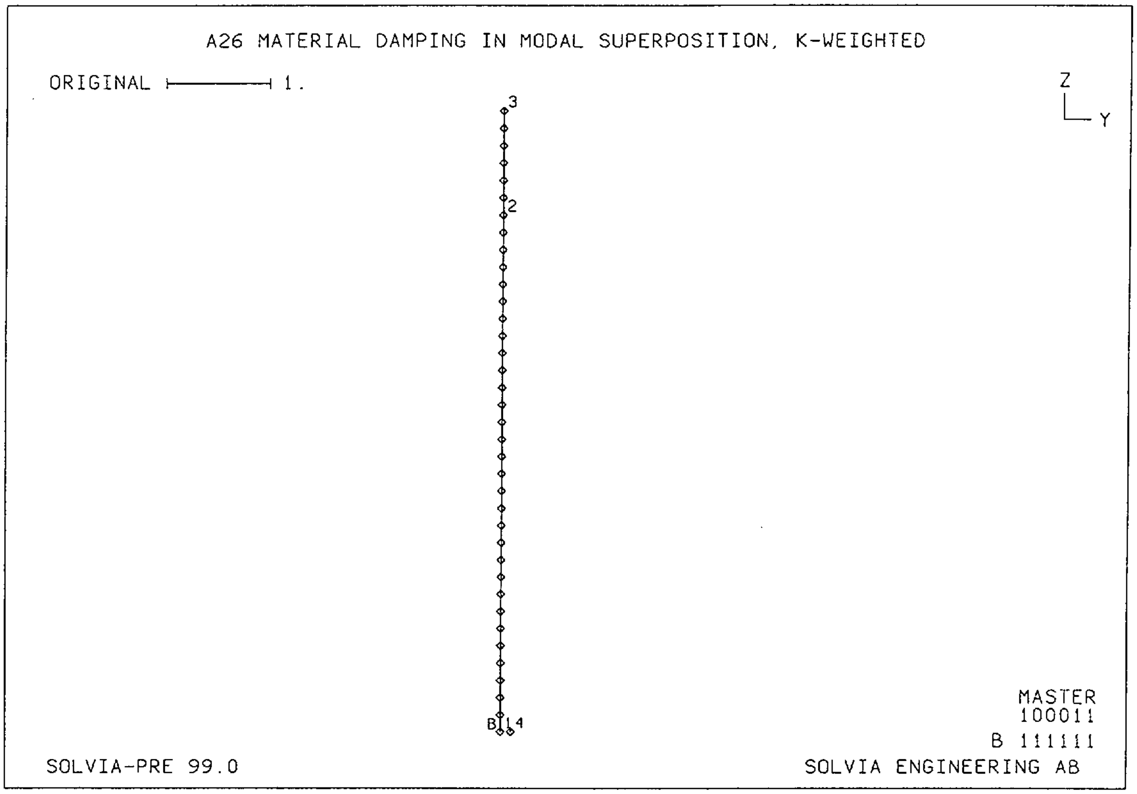
The model consists of two element groups of ISOBEAM elements as shown on page A26.2.

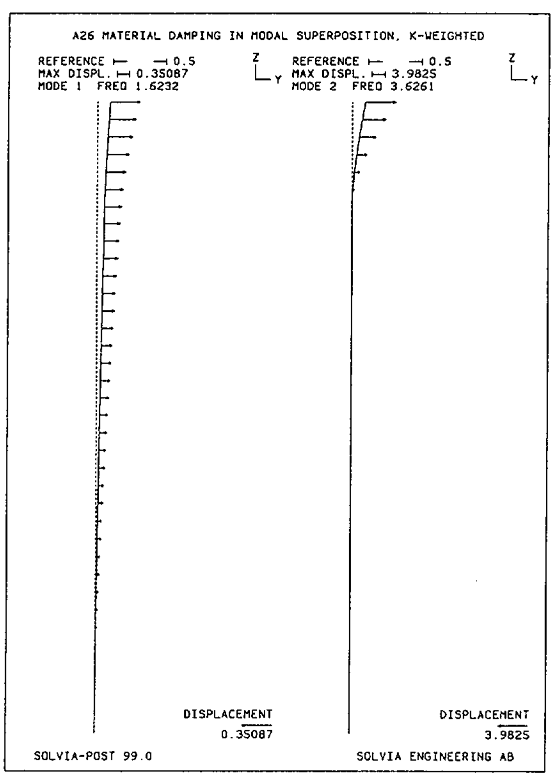
Solution Results

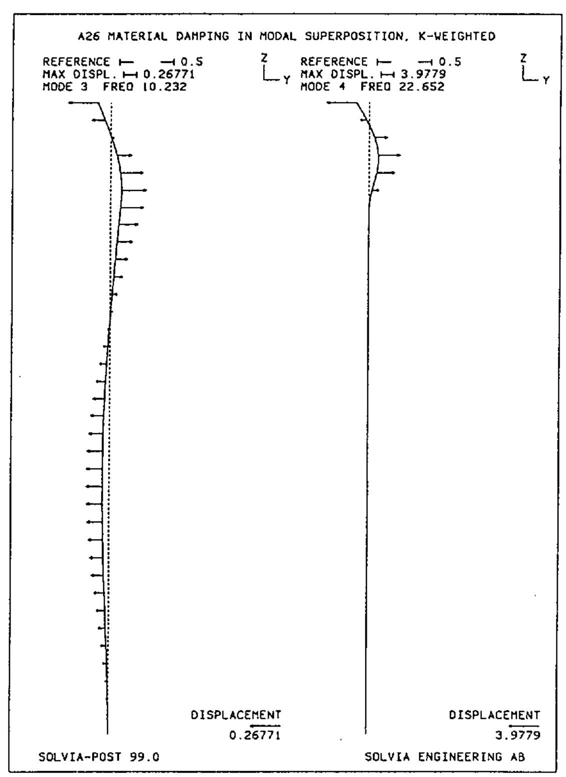
The input data on page A26.3 is used to calculate the natural frequencies and stiffness weighted modal damping factors. The mass weighted modal damping factors are calculated in a separate run.

Mode no.	Frequency Hz	Modal damping K-weighted %	Modal damping M-weighted %
1	1.62	1.01	1.07
2	3.63	4.97	4.94
3	10.23	1.02	1.03
4	22.65	4.93	4.94

The mode shapes are shown below. The bottom part dominates modes 1 and 3 which consequently have a weighted modal damping factor near 1 %. Similarly, the top part dominates modes 2 and 4 which then have damping factors near 5 %.







SOLVIA-PRE input

```
DATABASE  CREATE
HEADING  'A26 MATERIAL DAMPING IN MODAL SUPERPOSITION, K-WEIGHTED'
*
MASTER IDOF=100011
FREQUENCIES  SUBSPACE-ITERATION NEIG=4
MODALDAMPING  INPUT=K-WEIGHTED
ANALYSIS  TYPE=DYNAMIC IMODS=1 NMODES=4
*
COORDINATES
  ENTRIES   NODE Y  Z
             1   0  0
             2   0  5
             3   0  6
             4   .1 0
*
MATERIAL  1  ELASTIC  E=2.E11 NU=0. DENSITY=7800. MODALDAMPING=1.
MATERIAL  2  ELASTIC  E=2.E10 NU=0. DENSITY=1000. MODALDAMPING=5.
*
EGROUP  1  ISOBEAM   MATERIAL=1
SECTION  1  SDIM=0.05 TDIM=0.05
GLINE 1 2 AUX=4 EL=10 NODES=4
*
EGROUP  2  ISOBEAM   MATERIAL=2
SECTION  1  SDIM=0.005 TDIM=0.05
GLINE 2 3 AUX=4 EL=2 NODES=4
*
FIXBOUNDARIES  /  1
*
SET  VIEW=X NSYMBOLS=YES NNUMBER=MYNODES BCODE=ALL
MESH
SOLVIA
END
```

SOLVIA-POST input

```
*  A26 MATERIAL DAMPING IN MODAL SUPERPOSITION, K-WEIGHTED
*
DATABASE CREATE
*
WRITE 'a26.lis'
FREQUENCIES
SET PLOTORIENTATION=PORTRAIT
SET RESPONSETYPE=VIBRATION ORIGINAL=DASHED SMOOTHNESS=YES
SET VECTOR=DISPLACEMENT VIEW=X HEIGHT=0.25
MESH TIME=1 SUBFRAME=21
MESH TIME=2
MESH TIME=3 SUBFRAME=21
MESH TIME=4
END
```

EXAMPLE A27

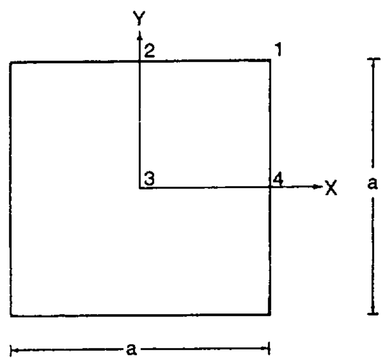
SIMPLY SUPPORTED SQUARE PLATE UNDER PRESSURE LOAD

Objective

To verify the bending and twisting behaviour of the 16-node SHELL element and the PLATE element when modelling a simply supported plate subjected to uniform pressure loading.

Physical Problem

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



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

$\nu = 0.3$

$a = 2.0$ m

$h = 0.05$ m (thickness)

$p = 1.0 \cdot 10^5$ N/m^2 (pressure)

Finite Element Model

Because of the symmetry conditions only one quarter of the plate is considered, 1-2-3-4 in the figure above. Two finite element models are used, one with 10×10 16-node SHELL elements, see page A27.5, and one with 10×10×4 PLATE elements, see page A27.11. The SHELL element model uses a finer mesh towards the boundaries. Two sets of boundary conditions are used with both models:

- Soft boundary conditions, examples A27 and A27B: Both bending and twisting degrees of freedom are free along the simply supported boundary.
- Hard boundary conditions, examples A27A and A27C: The bending degree of freedom is free but the twisting degree of freedom is fixed along the simply supported boundary.

Symmetrical boundary conditions are employed along the symmetry boundaries. All nodes of the models have fixed X- and Y-translations and Z-rotation.

Solution Results

The theoretical solution for a simply supported Kirchhoff plate under the considered loading is given in [1], article no. 30. A Kirchhoff plate has no transverse shear deformation but the effect of transverse shear deformation is discussed at the end of article no. 39.

The following theoretical solution applies to a square Kirchhoff plate:

Node 3 deflection	$w = 0.00406 \cdot pa^4 / D$	(m)
Node 3 bending moment	$M_{xx} = M_{yy} = 0.0479 \cdot pa^2$	(Nm/m)
Node 4 reactive shear force	$F_{xz} = 0.420 \cdot pa$	(N/m)
Node 2 reactive shear force	$F_{yz} = 0.420 \cdot pa$	(N/m)
Node 1 concentrated reaction	$R = 0.065 \cdot pa^2$	(N)

The flexural rigidity of the plate is $D = \dfrac{E h^3}{12\left(1-\nu^2\right)}$

Using the input data shown on pages A27.13 to A27.16 the following results are obtained, which are compared with the corresponding theoretical values:

	Kirchhoff theory	16-node SHELL Soft BC A27	16-node SHELL Hard BC A27A	PLATE Soft BC A27B	PLATE Hard BC A27C
w at node 3 (mm)	-2.70	-2.7707	-2.7127	-2.7001	-2.7001
M_{xx} at node 3, (Nm/m)	-19200	-19473	-19163	-19136	-19136
F_{xz} at node 4 (N/m)	84000	84181	67607	80741 **	80632 **
R corner reaction	-26000	-22558 *	1114 *	-25379	-24926
max M_{xy} (Nm/m)		12814	12994	12933	12968
max σ_{Mises} (MPa)		53.287	54.015		

*) Summation of reactions at the nodes that show downward reaction in case A27.
**) Calculated from reactions. With the used mesh about 4 percent of the pressure goes directly to the degrees of freedom that are fixed in the Z-direction and is not included in the reactions. A finer mesh increases the support reactions.

The distribution of displacements, twisting moment, bending moments, von Mises effective stress and stress deviation, transverse shear forces and force and moment reactions are shown for the 16-node SHELL models on pages A27.5 to A27.10. The r-s-t axes of the Element Stress System are orientated in the same directions as the Global X-Y-Z System, see figure on page A27.5.

Result plots for the PLATE element model with soft boundary conditions are shown on pages A27.11 and A27.12.

The following observations can be made regarding the results:

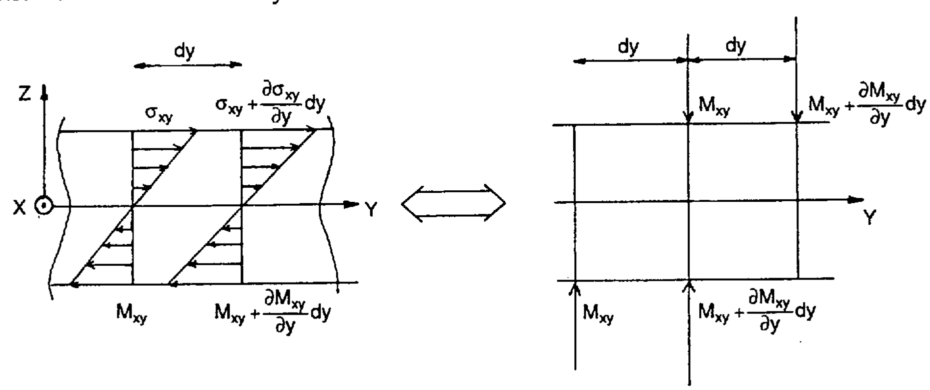
The deflection is slightly larger for the models with 16-node SHELL elements since transverse shear deformation is included.

The bending moment values are almost the same for the studied models.

The transverse shear force per unit length at node 4 is almost the same for the 16-node SHELL model with soft boundary conditions and the PLATE models when the PLATE values are calculated from reactions. Some small differences occur because of the mesh size of the PLATE models adjacent to the simply supported boundaries. The portion of the pressure load which acts on the boundary nodes is not included in the reactions. Only pressure load in the directions of active degrees of freedom is considered in the load and reaction calculations.

The transverse (reactive) shear force per unit length calculated for the 16-node SHELL model with hard boundary conditions is significantly less than the Kirchhoff theoretical value. The reason is that the portion associated with the twisting moment goes directly to ground without being transformed to statically equivalent transverse shear forces.

For a Kirchhoff plate the undeformed normals to the plate midsurface remain normals during deformation. The Kirchhoff plate can be regarded to have an orthotropic material with infinite shear moduli G_{xz} and G_{yz}. A consequence is that the effect on the plate of an applied distributed twisting moment is the same as the statically equivalent distributed force couples, see figure below with a portion of the 4 - 1 boundary.



Twisting moment per unit length and the corresponding shear stress

Statically equivalent force couples acting on two dy portions. The net transverse shear force on one dy portion is $\frac{\partial M_{xy}}{\partial y} dy$

We note that the net transverse shear force per unit length, which is statically equivalent to the twisting moment, is $\partial M_{xy} / \partial y$. Following [1] we obtain at node 4 the theoretical value

$$\frac{\partial M_{xy}}{\partial y} = 0.082 \text{ pa } = 16400 \text{ N/m}$$

If this portion is transferred directly to the fixed twisting support then the transverse shear force reaction remaining at node 4 is

$$F_{xz} = 84000 - 16400 = 67600 \text{ N/m}$$

which is in excellent agreement with the value 67607 N/m computed for the 16-node SHELL model with hard boundary conditions.

The statically equivalent transverse shear force corresponding to the twisting moment is for a Kirchhoff plate balanced at the corner node 1 by a concentrated reaction. In this example half of the reaction is due to the twisting moment along the 4 - 1 boundary and is

$$\tfrac{1}{2}R = -\left(M_{xy}\right)_{\text{at node 1}}$$

There is a good agreement between the concentrated reaction calculated by the PLATE element models and the theoretical value.

The 16-node SHELL model with hard boundary conditions gives no concentrated corner reaction, since the twisting moment is not transformed to statically equivalent transverse shear forces. Using soft boundary conditions there is a distributed shear force close to the corner node acting in the downward direction, but no concentrated corner reaction. The sum of the downward shear forces close to the corner is in reasonable agreement with the concentrated value according to the Kirchhoff theory considering that the shear deformation is a relieving factor.

The maximum value of the twisting moment M_{xy} for the 16-node SHELL and the PLATE models are in close agreement. However, when soft boundary conditions are used for the 16-node SHELL model there is a transition region along the simply supported boundaries extending into the plate of the order one plate thickness. In this transition region the shear stresses σ_{xy} corresponding to the twisting moment are transformed to shear stresses in the thickness direction.

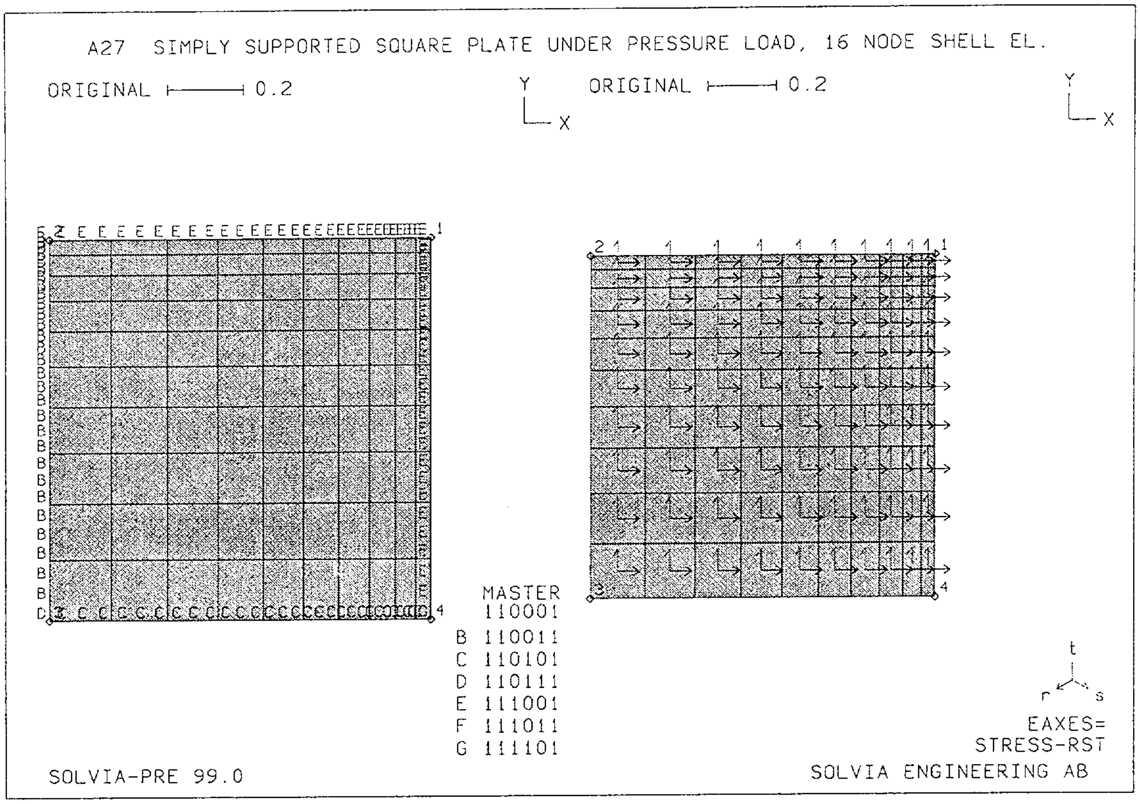
The maximum value of the von Mises effective stress is about the same for the two SHELL models with soft and hard boundary conditions. The stress deviation values indicate that the used mesh is good.

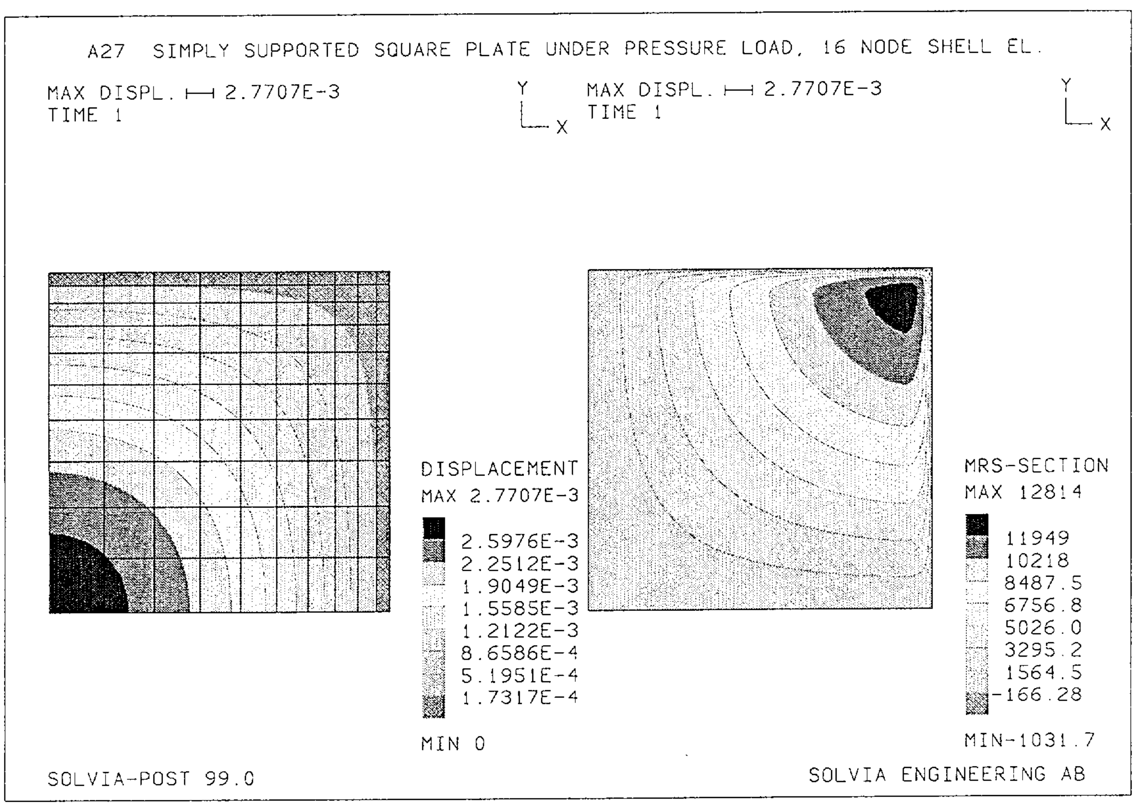
User Hints

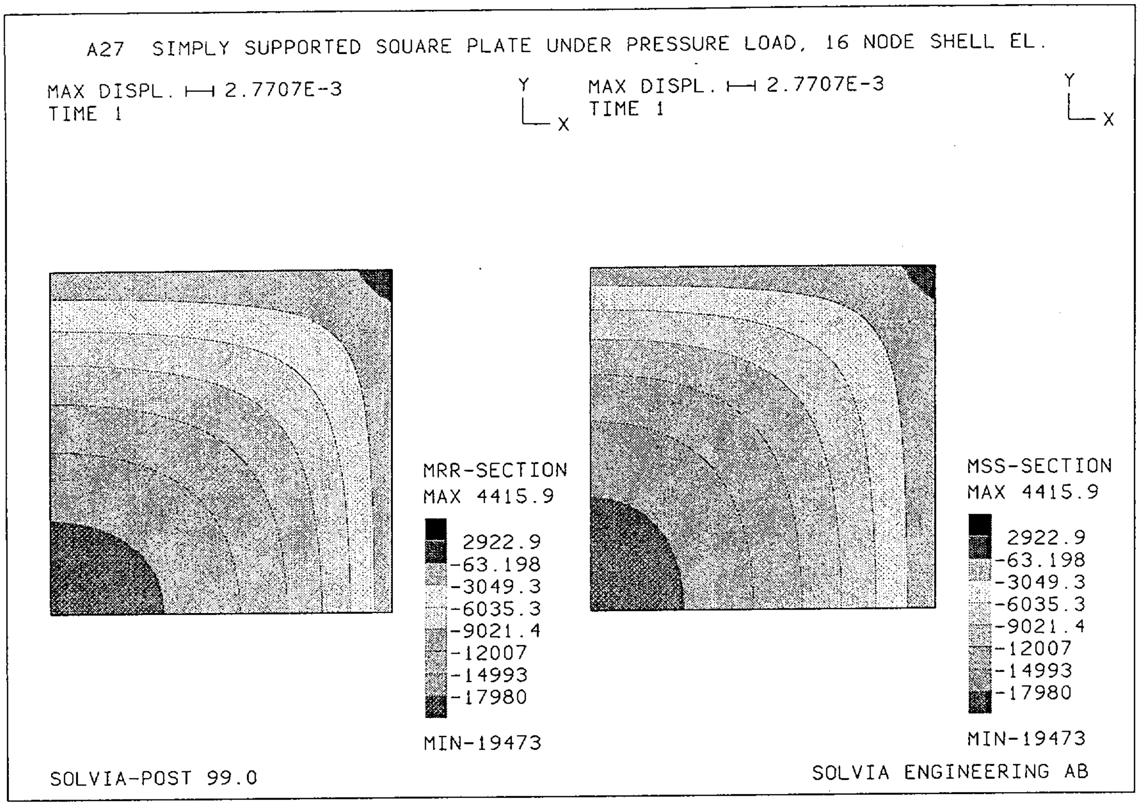
- Soft boundary conditions are in general recommended for a SHELL model with simply supported edges.
- Hard boundary conditions for a SHELL model with simply supported edges allow the twisting moment to be directly resisted by a twisting moment reaction instead of being transformed to statically equivalent transverse shear reactions and concentrated corner reactions. This results in lower values for the vertical reaction forces.
- If only bending moment and deflection results are of interest then both hard and soft boundary conditions can be used along the simply supported boundary.
- For a PLATE model of a simply supported plate then soft and hard boundary conditions give about the same results if a reasonably fine mesh is used.
- The 4-node and the 9-node SHELL elements can also be used successfully for a simply supported plate but the 16-node SHELL element used in this example gives a smoother stress distribution.

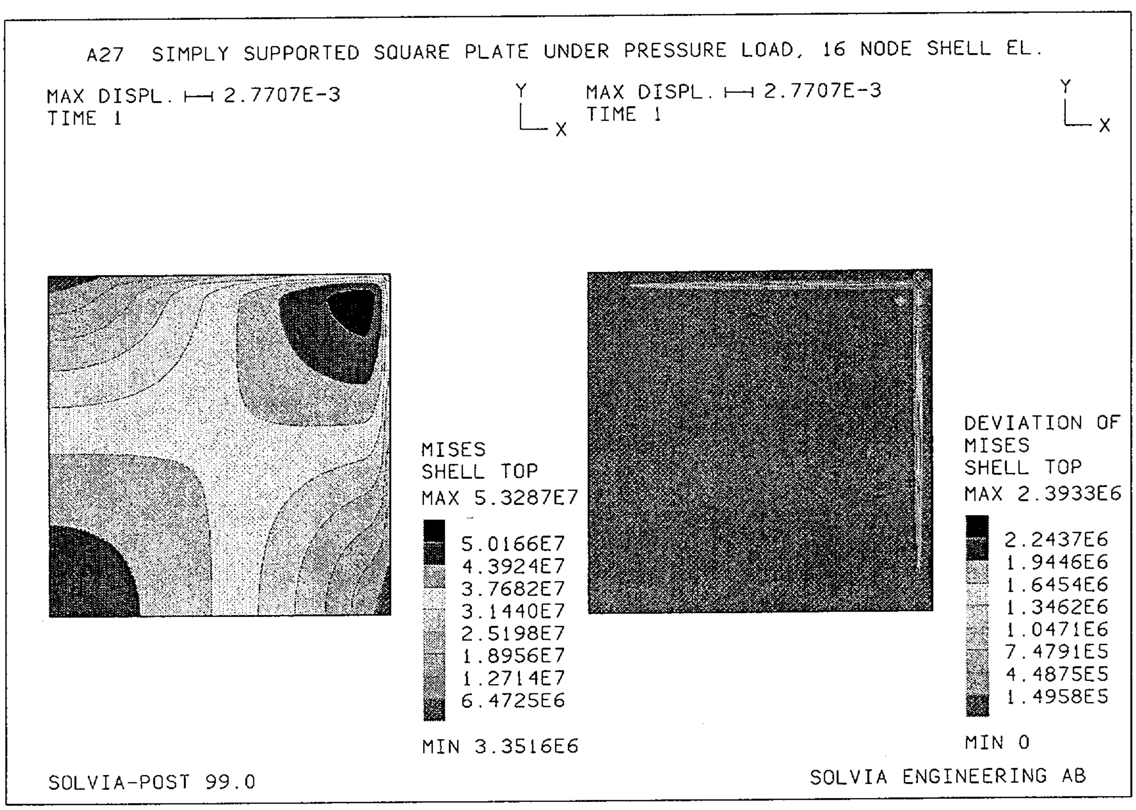
Reference

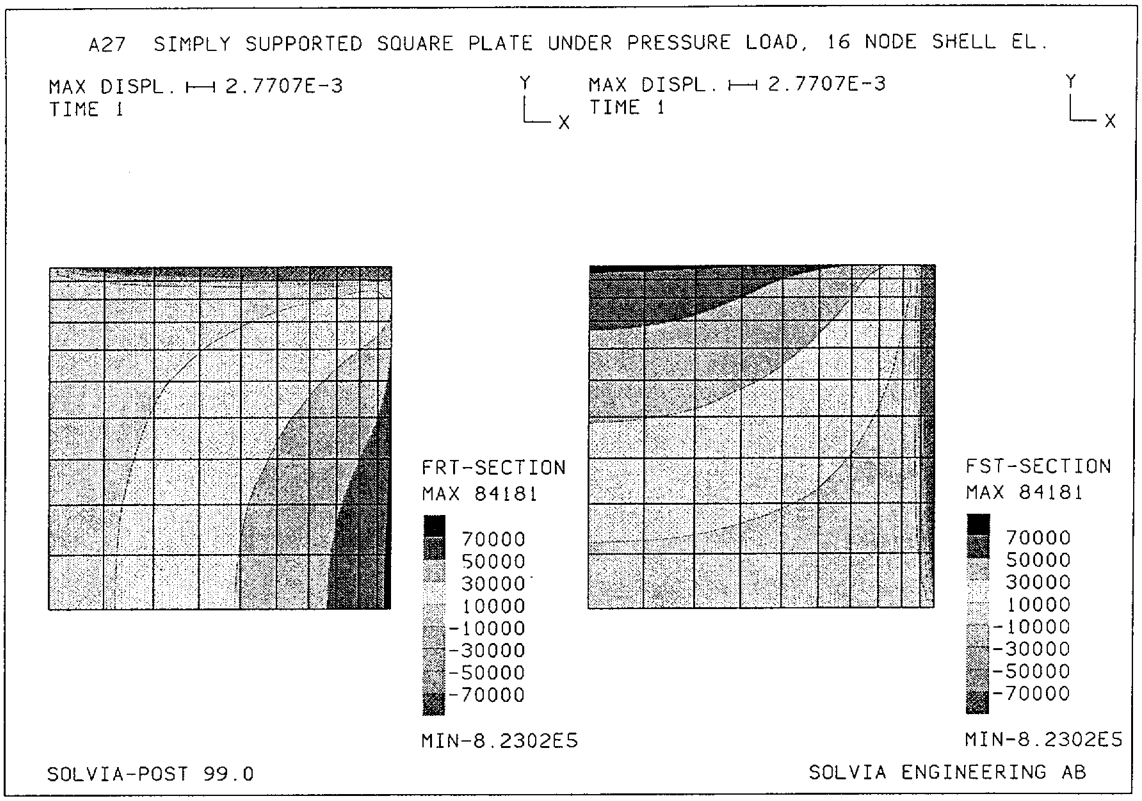
[1] Timoshenko, S.P. and Woinowsky-Krieger, S., Theory of Plates and Shells, Second Edition, McGraw-Hill, 1959.

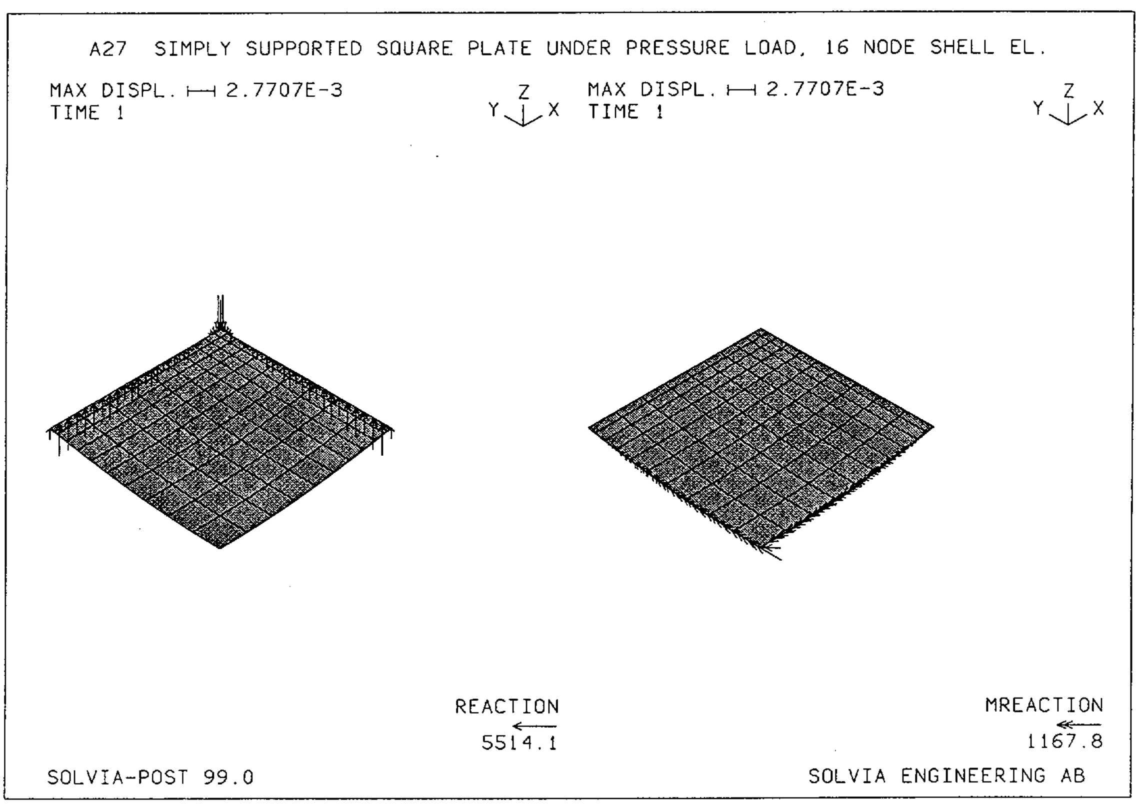

A27 SIMPLY SUPPORTED SQUARE PLATE UNDER PRESSURE LOAD, 16 NODE SHELL EL.
ORIGINAL 0.2
Y
X
ORIGINAL 0.2
Y
X
MASTER
110001
B 110011
C 110101
D 110111
E 111001
F 111011
G 111101
t
r
s
EAXES=
STRESS-RST
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

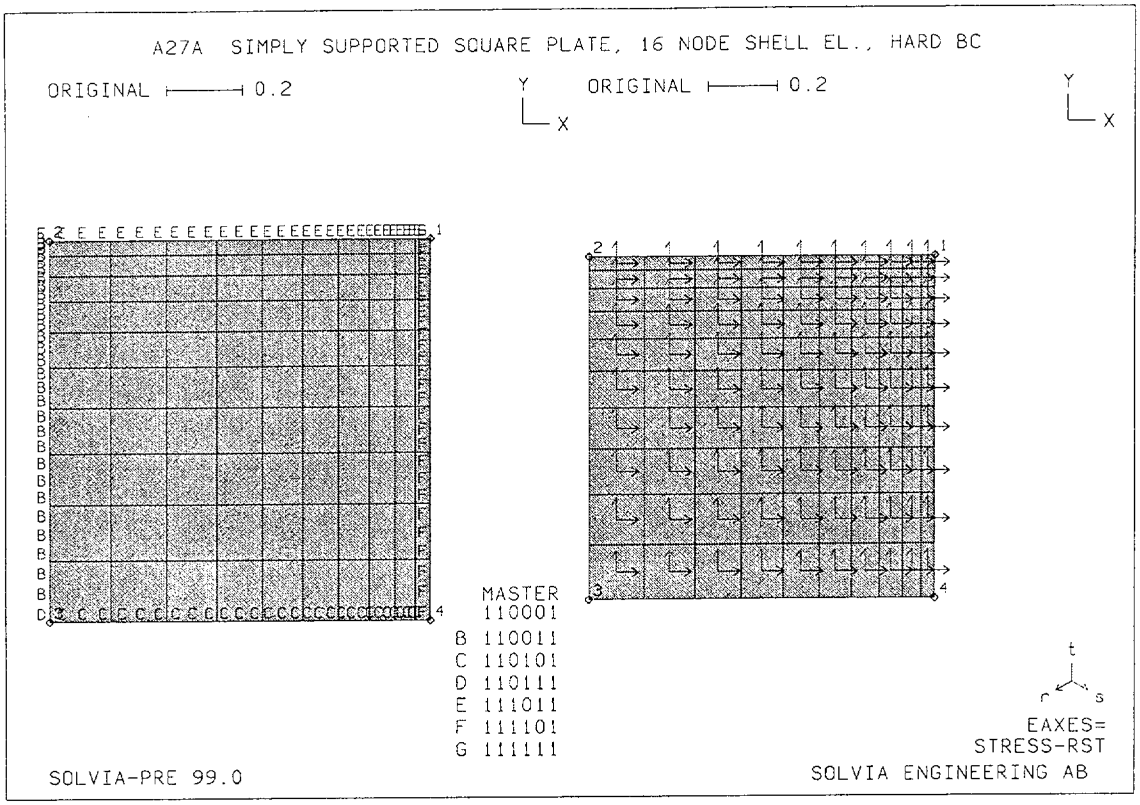

A27 SIMPLY SUPPORTED SQUARE PLATE UNDER PRESSURE LOAD, 16 NODE SHELL EL.
MAX DISPL. 2.7707E-3
TIME 1
Y
X
MAX DISPL. 2.7707E-3
TIME 1
Y
X
DISPLACEMENT
MAX 2.7707E-3
2.5976E-3
2.2512E-3
1.9049E-3
1.5585E-3
1.2122E-3
8.6586E-4
5.1951E-4
1.7317E-4
MIN 0
MRS-SECTION
MAX 12814
11949
10218
8487.5
6756.8
5026.0
3295.2
1564.5
-166.28
MIN-1031.7
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

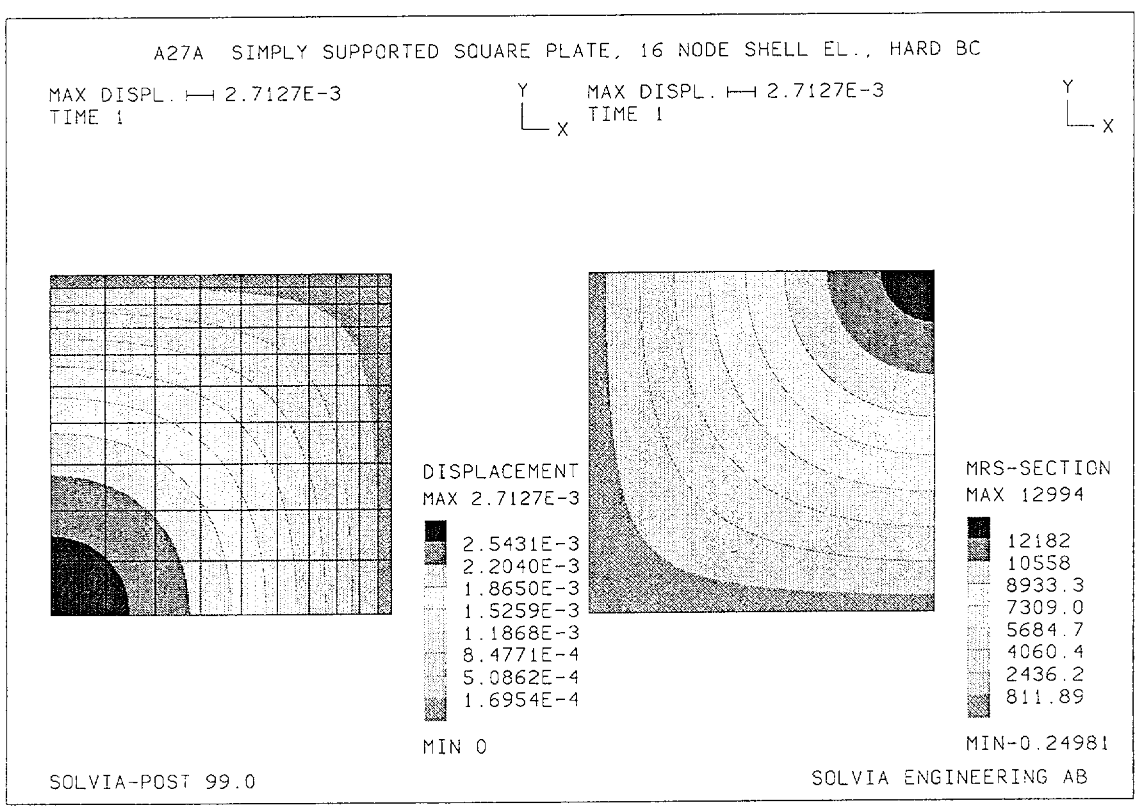

A27 SIMPLY SUPPORTED SQUARE PLATE UNDER PRESSURE LOAD, 16 NODE SHELL EL.
MAX DISPL. 2.7707E-3
TIME 1
Y
X
MAX DISPL. 2.7707E-3
TIME 1
Y
X
MRR-SECTION
MAX 4415.9
2922.9
-63.198
-3049.3
-6035.3
-9021.4
-12007
-14993
-17980
MIN-19473
MSS-SECTION
MAX 4415.9
2922.9
-63.198
-3049.3
-6035.3
-9021.4
-12007
-14993
-17980
MIN-19473
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

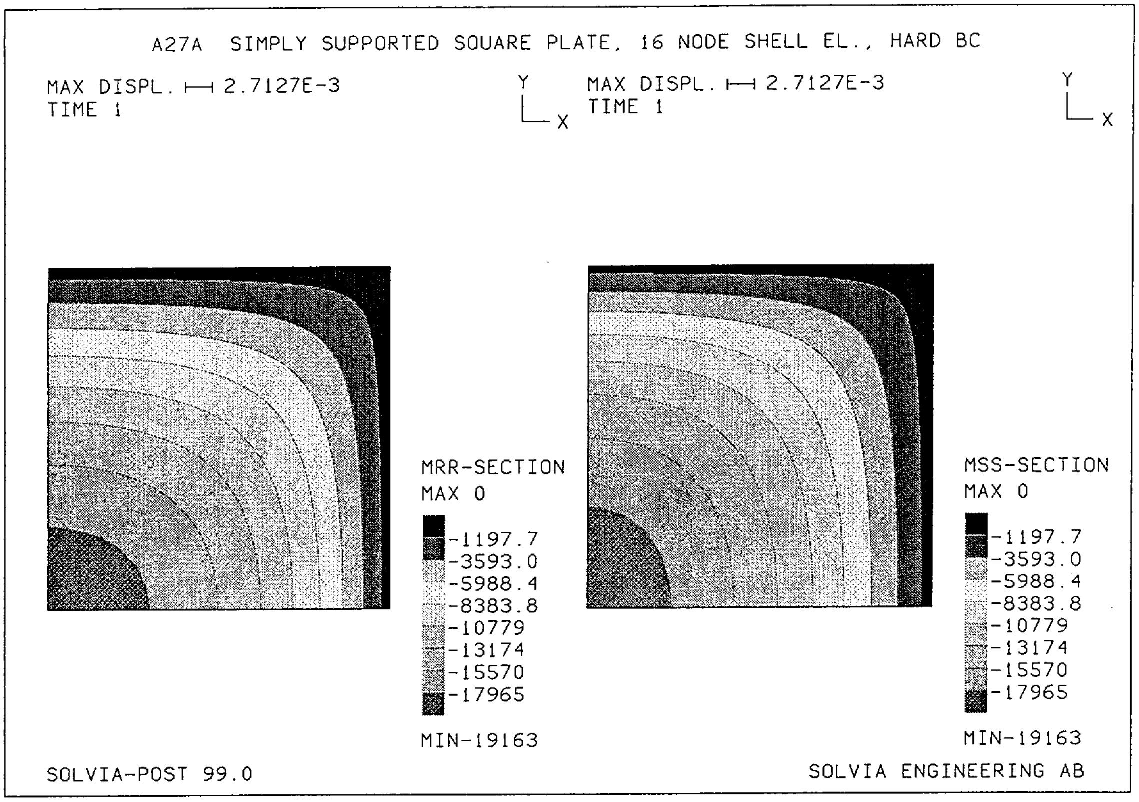

A27 SIMPLY SUPPORTED SQUARE PLATE UNDER PRESSURE LOAD, 16 NODE SHELL EL.
MAX DISPL. 2.7707E-3
TIME 1
Y
X
MAX DISPL. 2.7707E-3
TIME 1
Y
X
MISES
SHELL TOP
MAX 5.3287E7
5.0166E7
4.3924E7
3.7682E7
3.1440E7
2.5198E7
1.8956E7
1.2714E7
6.4725E6
MIN 3.3516E6
DEVIATION OF
MISES
SHELL TOP
MAX 2.3933E6
2.2437E6
1.9446E6
1.6454E6
1.3462E6
1.0471E6
7.4791E5
4.4875E5
1.4958E5
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

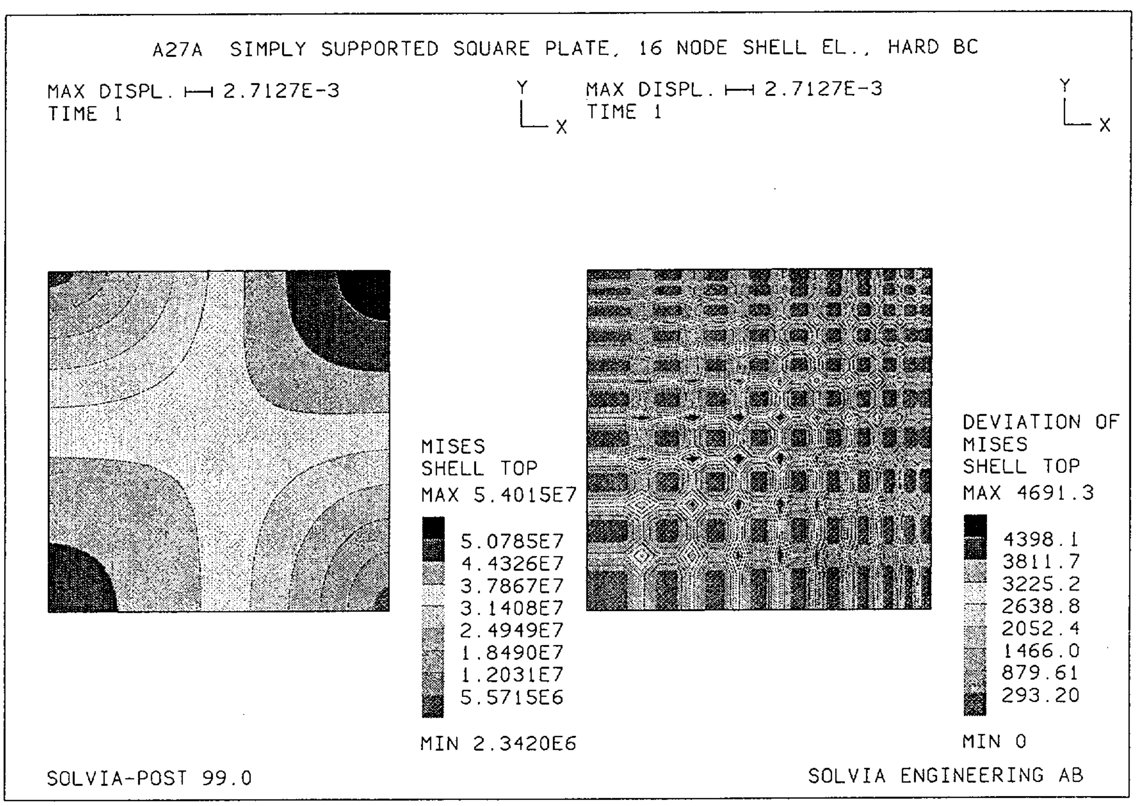

A27 SIMPLY SUPPORTED SQUARE PLATE UNDER PRESSURE LOAD, 16 NODE SHELL EL.
MAX DISPL. 2.7707E-3
TIME 1
Y
X
MAX DISPL. 2.7707E-3
TIME 1
Y
X
FRT-SECTION
MAX 84181
70000
50000
30000
10000
-10000
-30000
-50000
-70000
MIN-8.2302E5
FST-SECTION
MAX 84181
70000
50000
30000
10000
-10000
-30000
-50000
-70000
MIN-8.2302E5
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

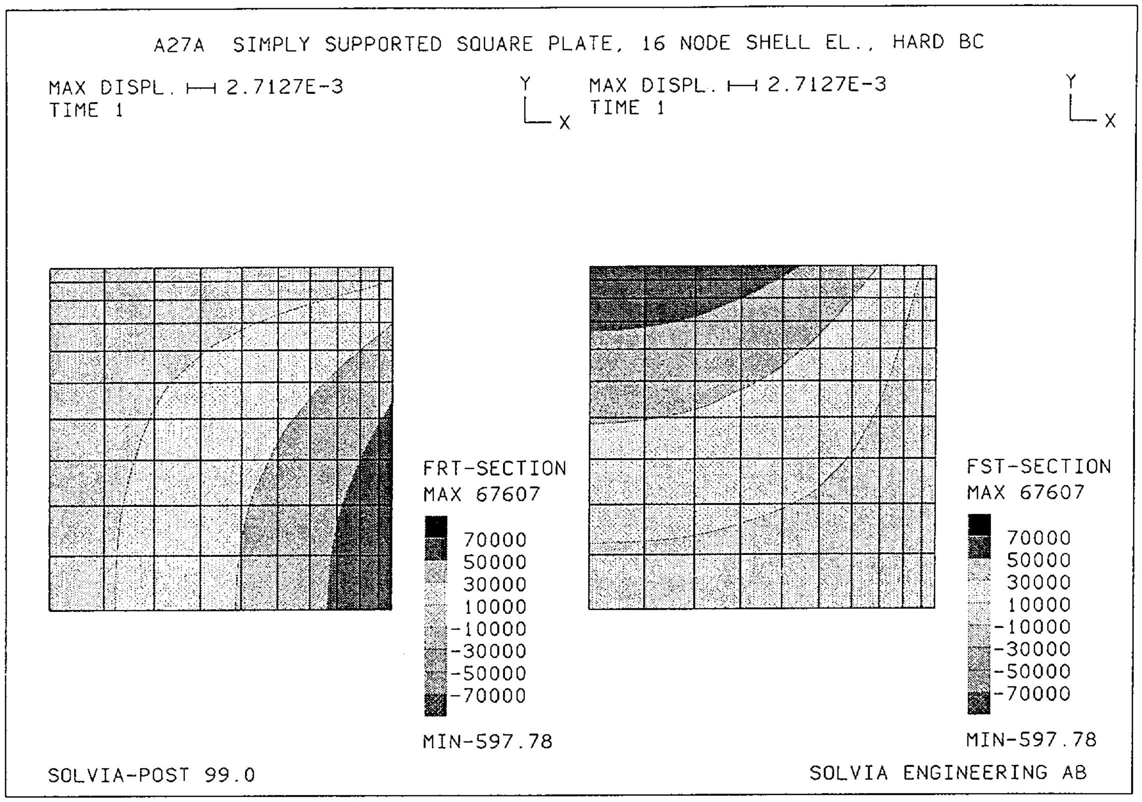

A27 SIMPLY SUPPORTED SQUARE PLATE UNDER PRESSURE LOAD, 16 NODE SHELL EL.
MAX DISPL. 2.7707E-3
TIME 1
Z
Y X
MAX DISPL. 2.7707E-3
TIME 1
Z
Y X
REACTION
5514.1
MREACTION
1167.8
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

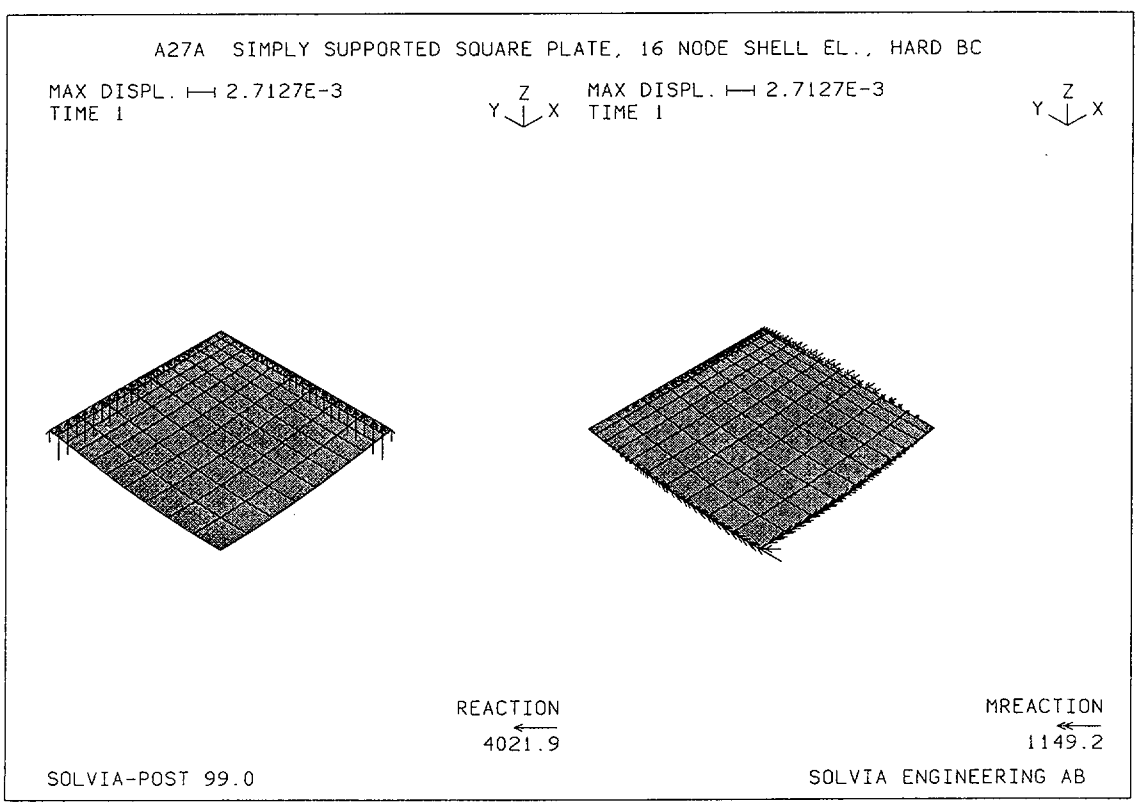

A27A SIMPLY SUPPORTED SQUARE PLATE, 16 NODE SHELL EL., HARD BC
ORIGINAL 0.2
ORIGINAL 0.2
MASTER
110001
B 110011
C 110101
D 110111
E 111011
F 111101
G 111111
EAXES=
STRESS-RST
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

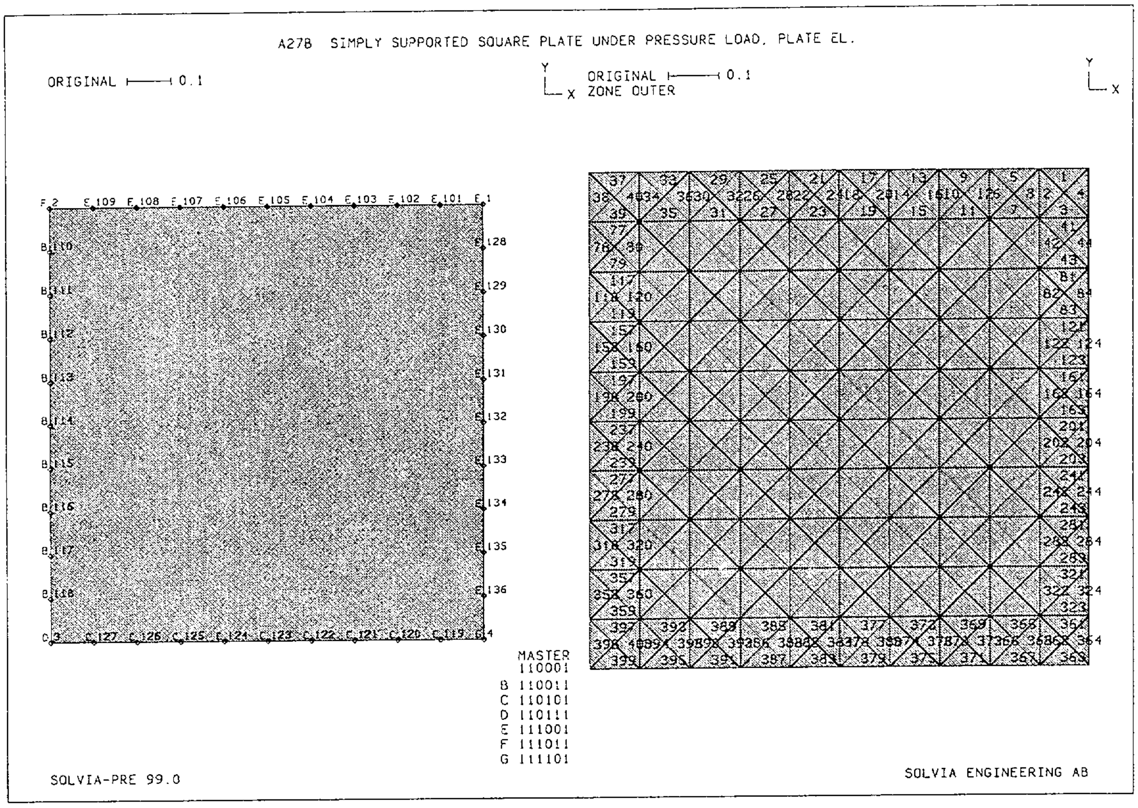

A27A SIMPLY SUPPORTED SQUARE PLATE, 16 NODE SHELL EL., HARD BC
MAX DISPL. 2.7127E-3
TIME 1
MAX DISPL. 2.7127E-3
TIME 1
DISPLACEMENT
MAX 2.7127E-3
2.5431E-3
2.2040E-3
1.8650E-3
1.5259E-3
1.1868E-3
8.4771E-4
5.0862E-4
1.6954E-4
MIN 0
MRS-SECTION
MAX 12994
12182
10558
8933.3
7309.0
5684.7
4060.4
2436.2
811.89
MIN-0.24981
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

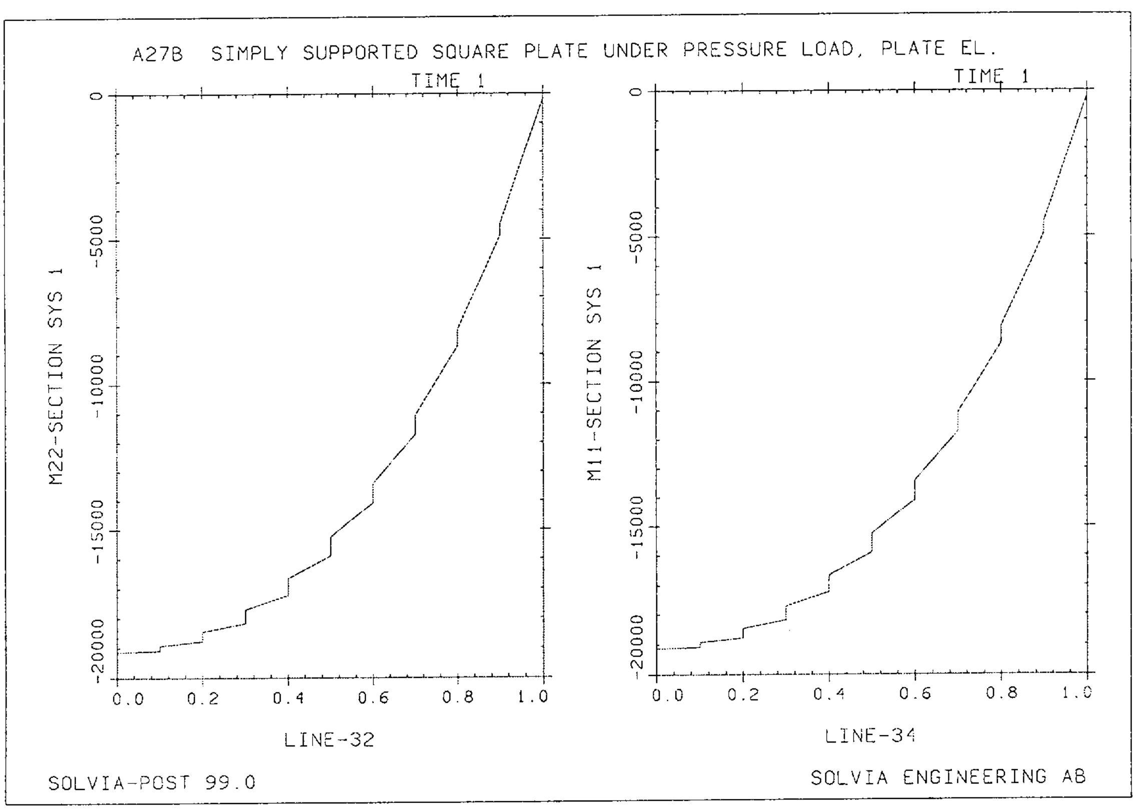

A27A SIMPLY SUPPORTED SQUARE PLATE, 16 NODE SHELL EL., HARD BC
MAX DISPL. ⊢⊣ 2.7127E-3
TIME 1
Y
X
MAX DISPL. ⊢⊣ 2.7127E-3
TIME 1
Y
X
MRR-SECTION
MAX 0
-1197.7
-3593.0
-5988.4
-8383.8
-10779
-13174
-15570
-17965
MIN-19163
MSS-SECTION
MAX 0
-1197.7
-3593.0
-5988.4
-8383.8
-10779
-13174
-15570
-17965
MIN-19163
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

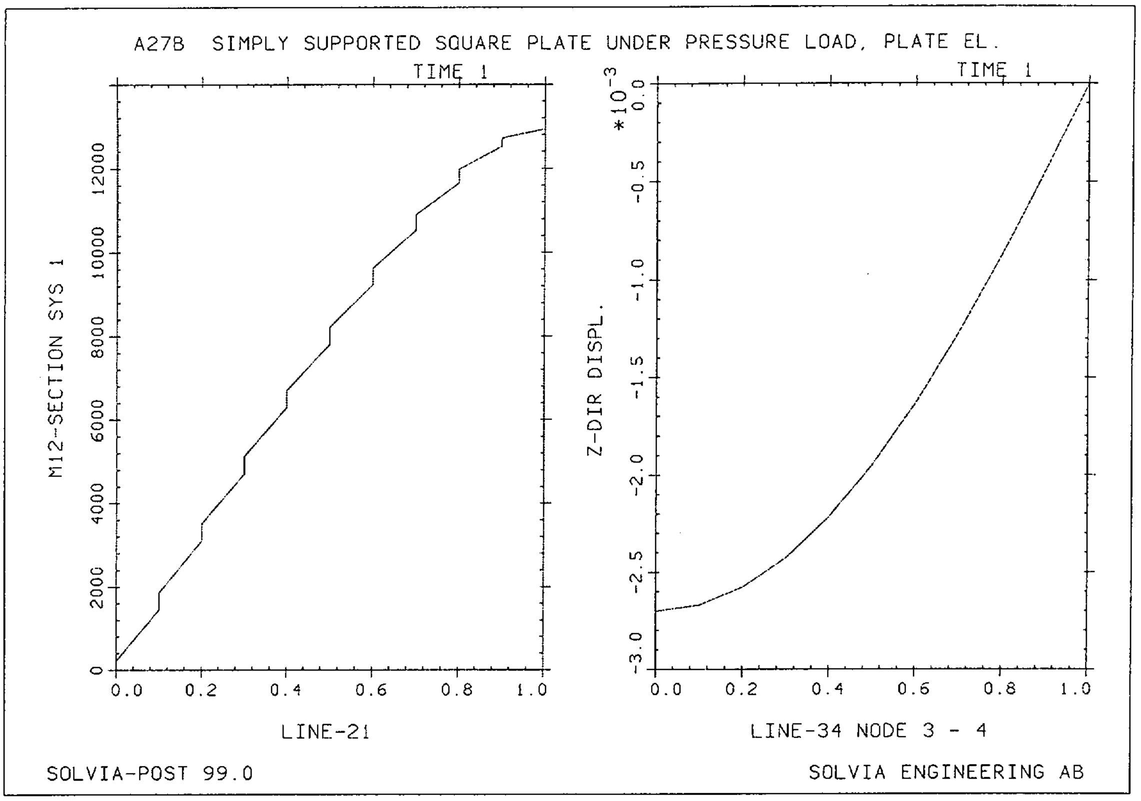

A27A SIMPLY SUPPORTED SQUARE PLATE, 16 NODE SHELL EL., HARD BC
MAX DISPL. ⊢⊣ 2.7127E-3
TIME 1
Y
X
MAX DISPL. ⊢⊣ 2.7127E-3
TIME 1
Y
X
MISES
SHELL TOP
MAX 5.4015E7
5.0785E7
4.4326E7
3.7867E7
3.1408E7
2.4949E7
1.8490E7
1.2031E7
5.5715E6
MIN 2.3420E6
DEVIATION OF
MISES
SHELL TOP
MAX 4691.3
4398.1
3811.7
3225.2
2638.8
2052.4
1466.0
879.61
293.20
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

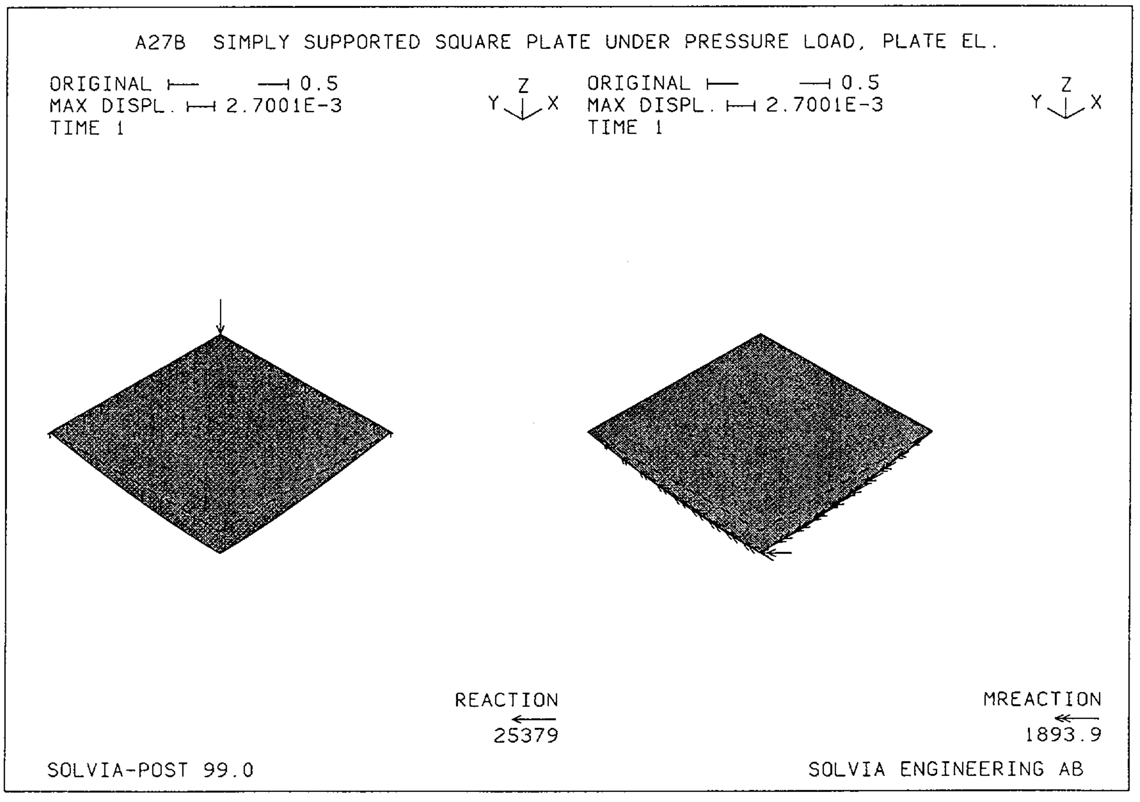

A27A SIMPLY SUPPORTED SQUARE PLATE, 16 NODE SHELL EL., HARD BC
MAX DISPL. 2.7127E-3
TIME 1
Y
X
MAX DISPL. 2.7127E-3
TIME 1
Y
X
FRT-SECTION
MAX 67607
70000
50000
30000
10000
-10000
-30000
-50000
-70000
MIN-597.78
FST-SECTION
MAX 67607
70000
50000
30000
10000
-10000
-30000
-50000
-70000
MIN-597.78
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A27A SIMPLY SUPPORTED SQUARE PLATE, 16 NODE SHELL EL., HARD BC
MAX DISPL. 2.7127E-3
TIME 1
Z
Y X
MAX DISPL. 2.7127E-3
TIME 1
Z
Y X
REACTION
4021.9
MREACTION
1149.2
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A27B SIMPLY SUPPORTED SQUARE PLATE UNDER PRESSURE LOAD, PLATE EL.
ORIGINAL 0.1
ORIGINAL 0.1
ZONE OUTER
MASTER
110001
B 110011
C 110101
D 110111
E 111001
F 111011
G 111101
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A27B SIMPLY SUPPORTED SQUARE PLATE UNDER PRESSURE LOAD, PLATE EL.
TIME 1
TIME 1
M22-SECTION SYS 1
M11-SECTION SYS 1
LINE-32
LINE-34
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A27B SIMPLY SUPPORTED SQUARE PLATE UNDER PRESSURE LOAD, PLATE EL.
TIME 1
M12-SECTION SYS 1
0
2000
4000
6000
8000
10000
12000
0.0
0.2
0.4
0.6
0.8
1.0
LINE-21
TIME 1
Z-DIR DISPL.
$*10^{-3}$
0.0
-0.5
-1.0
-1.5
-2.0
-2.5
-3.0
LINE-34 NODE 3 - 4
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A27B SIMPLY SUPPORTED SQUARE PLATE UNDER PRESSURE LOAD, PLATE EL.
ORIGINAL 0.5
MAX DISPL. 2.7001E-3
TIME 1
Z
Y X
ORIGINAL 0.5
MAX DISPL. 2.7001E-3
TIME 1
Z
Y X
REACTION
25379
MREACTION
1893.9
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A27  SIMPLY SUPPORTED SQUARE PLATE UNDER PRESSURE LOAD,
 16 NODE SHELL EL.'
*
DATABASE  CREATE
*
MASTER  IDOF=110001
COORDINATES
 1  1. 1. /  2  0. 1.  /  3  0. 0.  /  4  1. 0.
*
MATERIAL  1  ELASTIC E=2.1E11 NU=0.3
*
EGROUP  1  SHELL RESULT=NSTRESS
SET MIDNODES=2
LINE STRAIGHT 1 2 EL=10 RATIO=5
LINE STRAIGHT 2 3 EL=10 RATIO=5
GSURFACE  1 2 3 4
THICK 1 0.05
LOADS  ELEMENT  INPUT=SURFACE
 1 2 3 4  T  1.0E5
*
FIXBOUNDARIES  3  INPUT=LINE  /  1 2 / 4 1
FIXBOUNDARIES  4  INPUT=LINE  /  3 4
FIXBOUNDARIES  5  INPUT=LINE  /  2 3
*
* HARD BC USED IN A27A:
*FIXBOUNDARIES  34 INPUT=LINE  /  4 1
*FIXBOUNDARIES  35 INPUT=LINE  /  1 2
*FIXBOUNDARIES  4  INPUT=LINE  /  3 4
*FIXBOUNDARIES  5  INPUT=LINE  /  2 3
*
SET  VIEW=Z NNUMBER=MY NSYMBOL=MY
SUBFRAME  21
MESH BCODE=ALL
MESH EAXES=STRESS
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A27  SIMPLY SUPPORTED SQUARE PLATE UNDER PRESSURE LOAD
*       16 NODE SHELL EL.
*
DATABASE  CREATE
SET VIEW=Z
*
SUBFRAME 21
MESH CONTOUR=DISP
SET OUTLINE=YES
MESH CONTOUR=MRS
*
SUBFRAME 21
MESH CONTOUR=MRR
MESH CONTOUR=MSS
*
SUBFRAME 21
MESH CONTOUR=MISES
MESH CONTOUR=SDEVIATION
*
SET OUTLINE=NO
CONTOUR VMAX=70000 VMIN=-70000
SUBFRAME 21
MESH CONTOUR=FRT
MESH CONTOUR=FST
*
VIEW 1 -1 -1 1
SUBFRAME 21
MESH VECTOR=REACTION
MESH VECTOR=MREACTION
*
WRITE a27.lis
NLIST KIND=REACTIONS DIR=345
*
END
```

SOLVIA-PRE input

```
HEADING  'A27B  SIMPLY SUPPORTED SQUARE PLATE UNDER PRESSURE LOAD,
 PLATE EL.'
*
DATABASE  CREATE
*
MASTER  IDOF=110001
COORDINATES
 1 1. 1. / 2 0. 1.  / 3 0. 0.  / 4 1. 0.
*
MATERIAL  1  ELASTIC E=2.1E11 NU=0.3
*
EGROUP  1  PLATE
STRESSTABLE  1  1 2 3 4 5 6 7
GSURFACE  1 2 3 4  EL1=10 EL2=10
EDATA  /  1 0.05
LOADS  ELEMENT  INPUT=SURFACE
 1 2 3 4  T  1.E5
*
FIXBOUNDARIES  3  INPUT=LINE  /  1 2  / 4 1
FIXBOUNDARIES  4  INPUT=LINE  /  3 4
FIXBOUNDARIES  5  INPUT=LINE  /  2 3
*
* HARD BC USED IN A27C:
*FIXBOUNDARIES  34 INPUT=LINE  /  4 1
*FIXBOUNDARIES  35 INPUT=LINE  /  1 2
*FIXBOUNDARIES  4  INPUT=LINE  /  3 4
*FIXBOUNDARIES  5  INPUT=LINE  /  2 3
*
SET  VIEW=Z HEIGHT=0.20
SUBFRAME  21
MESH  NNUMBER=YES NSYMBOL=YES OUTLINE=YES BCODE=ALL
ZONE OUTER GLOBAL
ZONE OUTER GLOBAL DELETE XMIN=0.1 XMAX=0.9 YMIN=0.1 YMAX=0.9
MESH
MESH OUTER ENUMBER=YES GSCALE=OLD SUBFRAME=OLD
*
SOLVIA
END
```

SOLVIA-POST input

```
*   A27B   SIMPLY SUPPORTED SQUARE PLATE UNDER PRESSURE LOAD,
*          PLATE EL.
*
DATABASE  CREATE
SYSTEM 1 CARTESIAN
*
WRITE  FILENAME='a27b.lis'
*
EPLINE  LINE-34
 399 1 4 2 STEP -4 TO 363 1 4 2
EPLINE  LINE-32
 398 2 4 1 STEP -40 TO 38 2 4 1
EPLINE  LINE-21
 37 2 4 1 STEP -4 TO 1 2 4 1
NPLINE  LINE-34
 3 127 STEP -1 TO 119 4
*
ELINE LINE-32 KIND=M22 OUTPUT=ALL SYSTEM=1 SUBFRAME=21
ELINE LINE-34 KIND=M11 OUTPUT=ALL SYSTEM=1
ELINE LINE-21 KIND=M12 OUTPUT=ALL SYSTEM=1 SUBFRAME=21
NLINE LINE-34 DIR=3 OUTPUT=ALL
*
VIEW 1 -1 -1 1
SET OUTLINE=YES ORIGINAL=DASHED
MESH VECTOR=REACTIONS SUBFRAME=21
MESH VECTOR=MREACTIONS
*
NLIST  KIND=REACTIONS DIRECTION=345
END
```

EXAMPLE A28

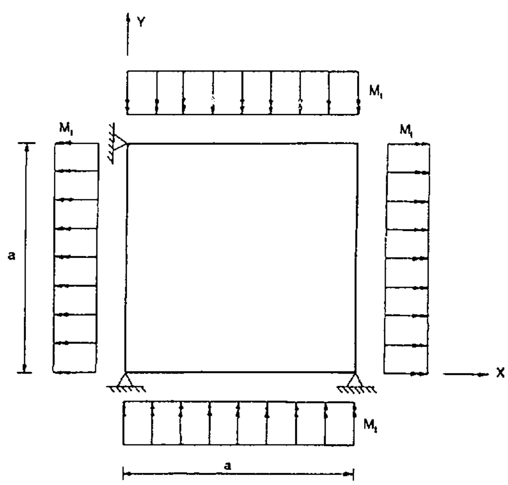
PLATE UNDER UNIFORM TWISTING

Objective

To verify the twisting behaviour of the SHELL and PLATE elements.

Physical Problem

A square plate of side lengths a, which is supported at three of its corners as shown in the figure below is considered. Consistent twisting moments along the plate sides are used in the SHELL element models. A concentrated load acting in the negative Z-direction is applied at the free corner on the PLATE element model.



$E = 4 \cdot 10^{10}\ \mathrm{N/m^2}$
$\nu = 0.2$
$a = 1\ \mathrm{m}$
$t = 0.1\ \mathrm{m}$ (thickness)
$M_t = 5000\ \mathrm{Nm/m}$

Finite Element Model

The figure on page A28.3 shows the irregular mesh used with the 16-node SHELL element model. The irregular mesh of the PLATE element model is shown in the right bottom figure of page A28.4.

Solution Results

The theoretical solution for this problem is a constant twist and the central deflection is

$$u = \frac{M_t\, a^2}{4GK} \quad \text{where} \quad G = \frac{E}{2(1+\nu)} \quad \text{and} \quad K = \frac{at^3}{3}$$

The input data on pages A28.5 and A28.6 gives the following results:

Central deflection, u (mm):

Theory	16-node SHELL	9-node SHELL	4-node SHELL	PLATE
0.450	0.450	0.450	0.450	0.450

Bending and twisting moments (Nm/m) in the whole model referenced to the global X-Y-Z system:

Theory	PLATE element
$M_{xx} = M_{yy} = 0$	$M_{xx} = M_{yy} = 0$
$M_{xy} = -5000$	$M_{xy} = -5000$

The bending stresses are zero for the SHELL element models.

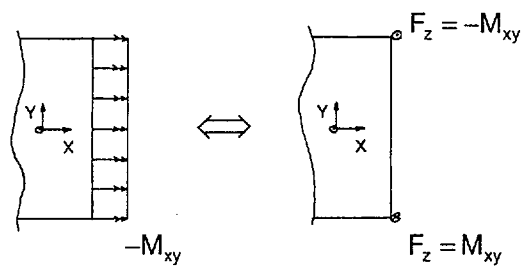
The torsion stresses σ_{xy} in MPa at the TOP surface for the SHELL element models are:

Theory	16-node SHELL	9-node SHELL	4-node SHELL
-3.00	-3.00	-3.00	-3.00

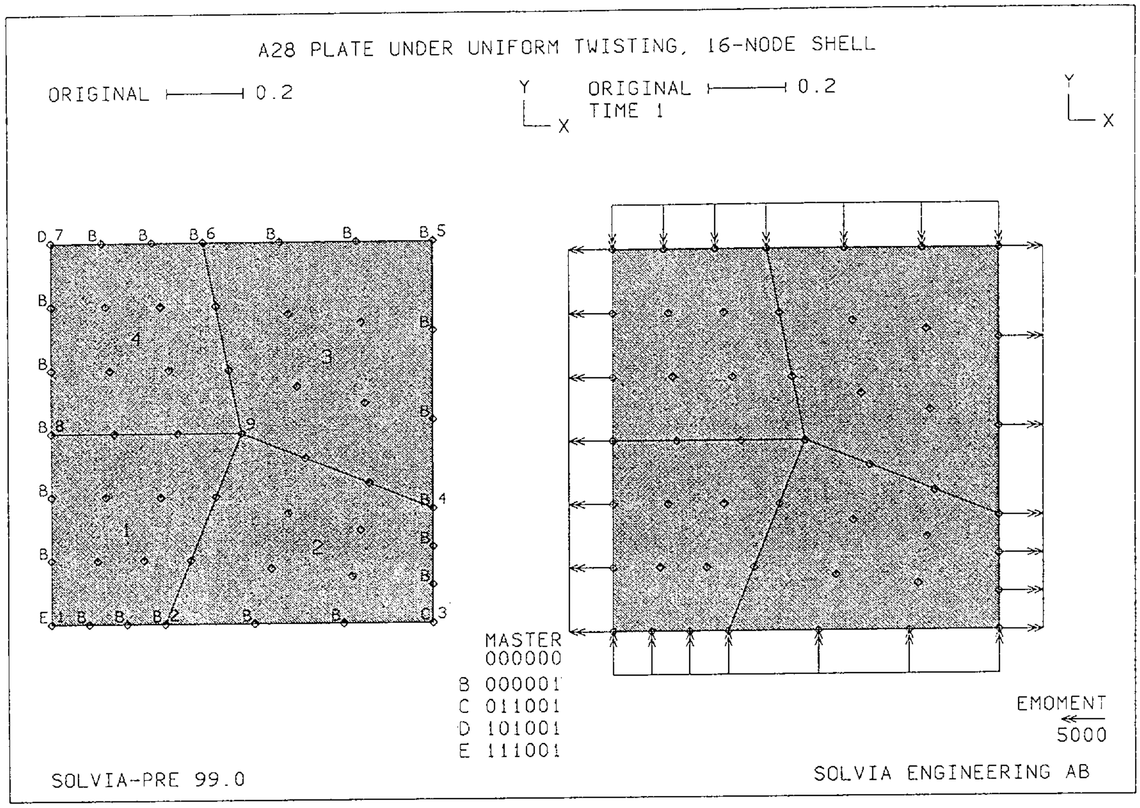
User Hints

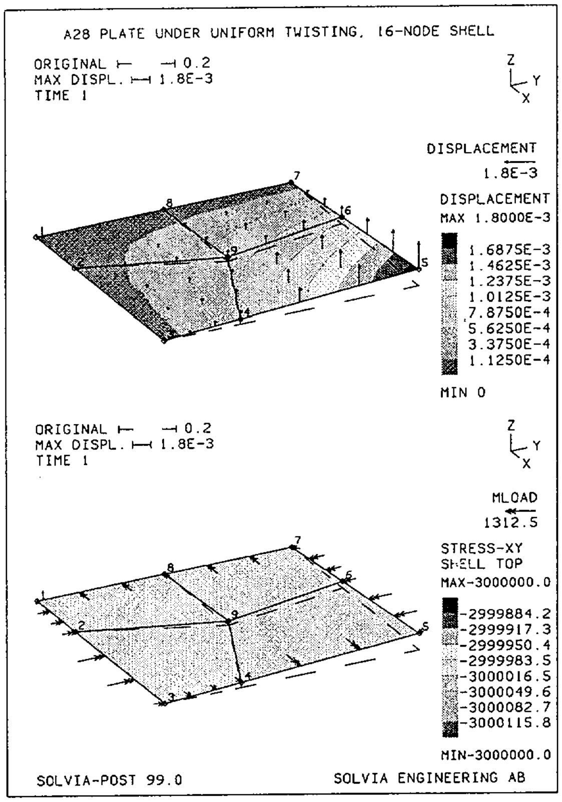
- This example may be regarded as a patch test problem since the theoretical solution is a constant twist. The SOLVIA solution obtained for the irregular mesh is exactly equal to the theoretical solution and the SHELL and PLATE elements have, therefore, the ability to represent stresses due to a constant twist.

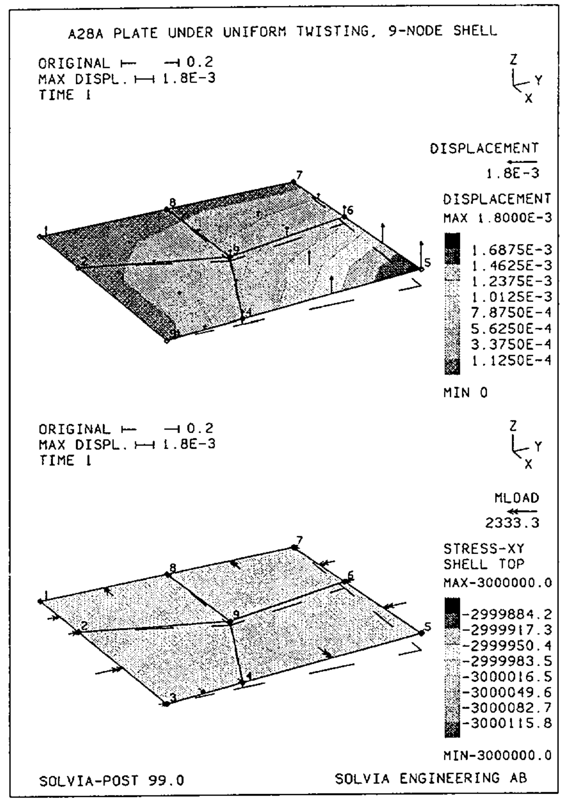
- For a plate without transverse shear deformation (Kirchhoff plate) there is a correspondence between the transverse shear force and the twisting moment so that a constant twisting moment along an edge can be replaced by corner forces, see figure.

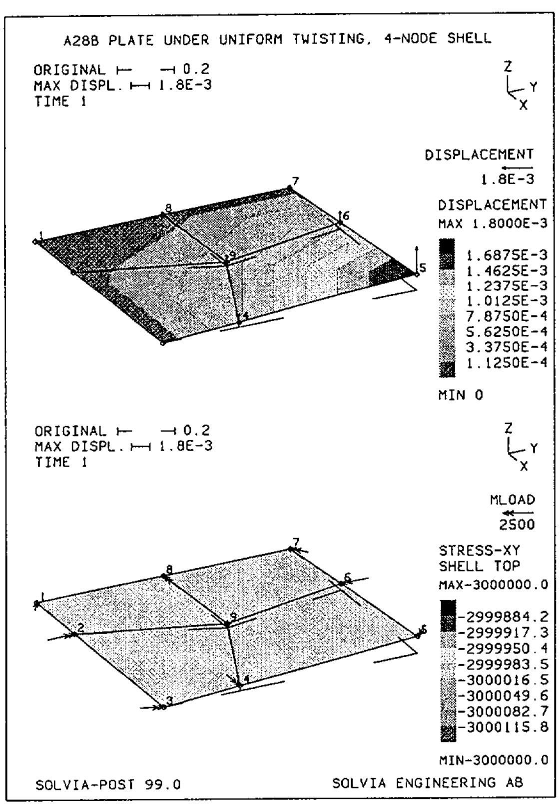


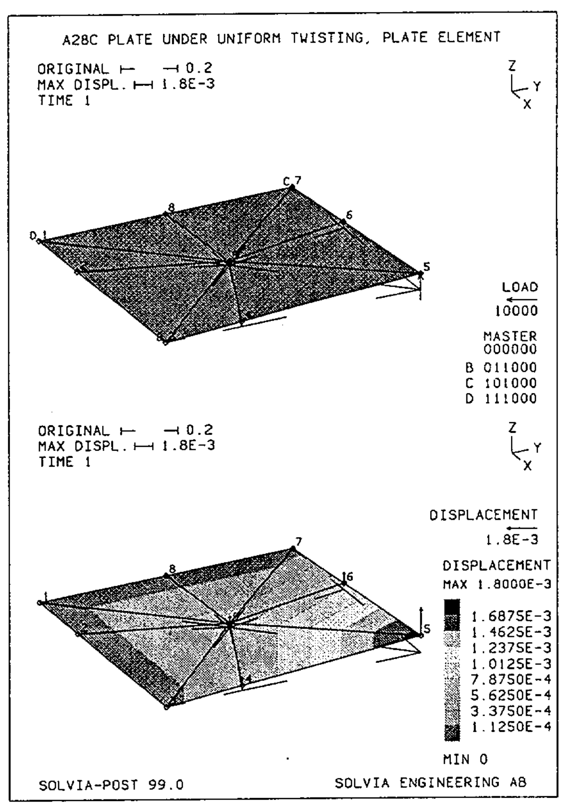
The SHELL element is capable of transverse shear deformation. The above correspondence between transverse shear force and twisting moment is then not valid exactly. If the twisting moment is applied as transverse shear force there will be a transition region along each boundary where transverse shear stresses consistent with the transverse shear force are transformed to in-plane shear stresses consistent with a twisting moment.


A28 PLATE UNDER UNIFORM TWISTING, 16-NODE SHELL
ORIGINAL 0.2
ORIGINAL 0.2
TIME 1
MASTER
000000
B 000001
C 011001
D 101001
E 111001
EMOMENT
5000
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A28 PLATE UNDER UNIFORM TWISTING, 16-NODE SHELL
ORIGINAL ⊢ ⊣ 0.2
MAX DISPL. ⊢⊣ 1.8E-3
TIME 1
DISPLACEMENT
1.8E-3
DISPLACEMENT
MAX 1.8000E-3
1.6875E-3
1.4625E-3
1.2375E-3
1.0125E-3
7.8750E-4
5.6250E-4
3.3750E-4
1.1250E-4
MIN 0
MLOAD
1312.5
STRESS-XY
SHELL TOP
MAX-3000000.0
-2999884.2
-2999917.3
-2999950.4
-2999983.5
-3000016.5
-3000049.6
-3000082.7
-3000115.8
MIN-3000000.0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A28A PLATE UNDER UNIFORM TWISTING, 9-NODE SHELL
ORIGINAL ⊢ ⊣ 0.2
MAX DISPL. ⊢⊣ 1.8E-3
TIME 1
DISPLACEMENT
1.8E-3
DISPLACEMENT
MAX 1.8000E-3
1.6875E-3
1.4625E-3
1.2375E-3
1.0125E-3
7.8750E-4
5.6250E-4
3.3750E-4
1.1250E-4
MIN 0
MLOAD
2333.3
STRESS-XY
SHELL TOP
MAX-3000000.0
-2999884.2
-2999917.3
-2999950.4
-2999983.5
-3000016.5
-3000049.6
-3000082.7
-3000115.8
MIN-3000000.0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A28B PLATE UNDER UNIFORM TWISTING, 4-NODE SHELL
ORIGINAL ⊢ ⊣ 0.2
MAX DISPL. ⊢⊣ 1.8E-3
TIME 1
DISPLACEMENT
1.8E-3
DISPLACEMENT
MAX 1.8000E-3
1.6875E-3
1.4625E-3
1.2375E-3
1.0125E-3
7.8750E-4
5.6250E-4
3.3750E-4
1.1250E-4
MIN 0
MLOAD
2500
STRESS-XY
SHELL TOP
MAX-3000000.0
-2999884.2
-2999917.3
-2999950.4
-2999983.5
-3000016.5
-3000049.6
-3000082.7
-3000115.8
MIN-3000000.0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A28C PLATE UNDER UNIFORM TWISTING, PLATE ELEMENT
ORIGINAL ⊢ ⊣ 0.2
MAX DISPL. ⊢⊣ 1.8E-3
TIME 1
LOAD
10000
MASTER
000000
B 011000
C 101000
D 111000
DISPLACEMENT
1.8E-3
DISPLACEMENT
MAX 1.8000E-3
1.6875E-3
1.4625E-3
1.2375E-3
1.0125E-3
7.8750E-4
5.6250E-4
3.3750E-4
1.1250E-4
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A28 PLATE UNDER UNIFORM TWISTING, 16-NODE SHELL'
DATABASE CREATE
*
COORDINATES
 1  0.  0.  /  2  .3 .0  /  3  1. .0  /  4  1. 0.3  /  5  1. 1.
 6  0.4 1.  /  7  .0 1.  /  8  0. .5  /  9  .5 .5
*
MATERIAL  1 ELASTIC E=4.E10 NU=0.2
*
SET NODES=16
* 9-node SHELL: SET NODES=9
* 4-node SHELL: SET NODES=4
EGROUP  1 SHELL RESULT=NSTRESS
GSURFACE 1 2 9 8 / GSURFACE 2 3 4 9
GSURFACE 4 5 6 9 / GSURFACE 8 9 6 7
THICKNESS 1 0.1
LOAD ELEMENT TYPE=MOMENT INPUT=LINES
 1 3 out -5.E3 / 3 5 out  5.E3 /  5 7 out -5.E3 /  7 1 out  5.E3
*
FIXBOUNDARIES  3  /  1 3 7
FIXBOUNDARIES  1  /  1 7
FIXBOUNDARIES  2  /  1 3
FIXBOUNDARIES  6 LINES / 1 3 / 3 5 / 5 7 / 7 1
*
SET  VIEW=Z NSYMBOL=YES
MESH  NNUMBER=MYNODES ENUMBER=YES BCODE=ALL SUBFRAME=21
MESH  VECTOR=LOAD X=0.5
SOLVIA
END
```

SOLVIA-POST input

```
*  A28  PLATE UNDER TWISTING, 16-NODE SHELL
*
DATABASE  CREATE
WRITE  FILENAME='a28.lis'
*
VIEW  ID=1 XVIEW=1. YVIEW=-0.5 ZVIEW=0.5
SET NSYMBOL=MYNODES NNUMBERS=MYNODES  ORIGINAL=DASHED
SET PLOTORIENTATION=PORTRAIT VIEW=1
SUBFRAME 12
MESH  CONTOUR=DISPLACEMENT VECTOR=DISPLACEMENT
MESH  CONTOUR=SXY VECTOR=MLOAD
*
SHELLSURFACE LIST=TOP
NLIST  MYNODES
ELIST
EMAX
END
```

SOLVIA-PRE input

```
HEADING  'A28C PLATE UNDER UNIFORM TWISTING, PLATE ELEMENT'
DATABASE CREATE
*
COORDINATES
 1  0.  0.  /  2  .3 .0  /  3  1. .0  /  4  1. 0.3  /  5  1. 1.
 6  0.4 1.  /  7  .0 1.  /  8  0. .5  /  9  .5 .5
*
MATERIAL  1 ELASTIC E=4.E10 NU=0.2
*
EGROUP  1 PLATE
ENODES
1 1 2 9 / 2 2 3 9 / 3 3 4 9 / 4 4 5 9
5 5 6 9 / 6 6 7 9 / 8 7 8 9 / 9 8 1 9
EDATA / 1 0.1
*
LOAD CONCENTRATED
5 3 1E4
*
FIXBOUNDARIES  3  /  1 3 7
FIXBOUNDARIES  1  /  1 7
FIXBOUNDARIES  2  /  1 3
*
SET  VIEW=Z NSYMBOL=YES PLOTORIENTATION=PORTRAIT
MESH  NNUMBER=MYNODES ENUMBER=YES BCODE=ALL SUBFRAME=12
MESH  VECTOR=LOAD GSCALE=8
SOLVIA
END
```

SOLVIA-POST input

```
*  A28  PLATE UNDER TWISTING, 16-NODE SHELL
*
DATABASE  CREATE
WRITE  FILENAME='a28.lis'
*
VIEW  ID=1 XVIEW=1. YVIEW=-0.5 ZVIEW=0.5
SET NSYMBOL=MYNODES NNUMBERS=MYNODES  ORIGINAL=DASHED
SET PLOTORIENTATION=PORTRAIT VIEW=1
SUBFRAME 12
MESH  CONTOUR=DISPLACEMENT VECTOR=DISPLACEMENT
MESH  CONTOUR=SXY VECTOR=MLOAD
*
SHELLSURFACE LIST=TOP
NLIST  MYNODES
ELIST
EMAX
END
```

EXAMPLE A29

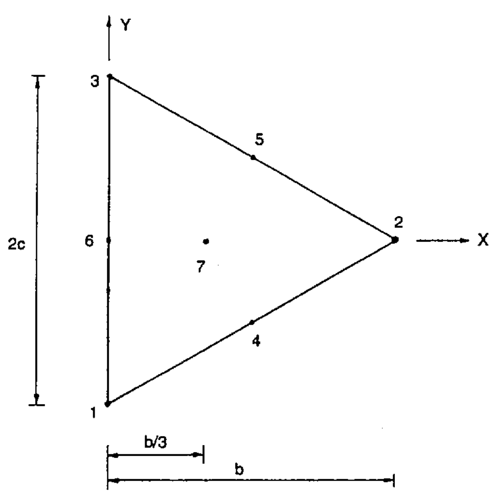
EDGE BENDING AND TWISTING OF A TRIANGULAR PLATE ON CORNER SUPPORTS

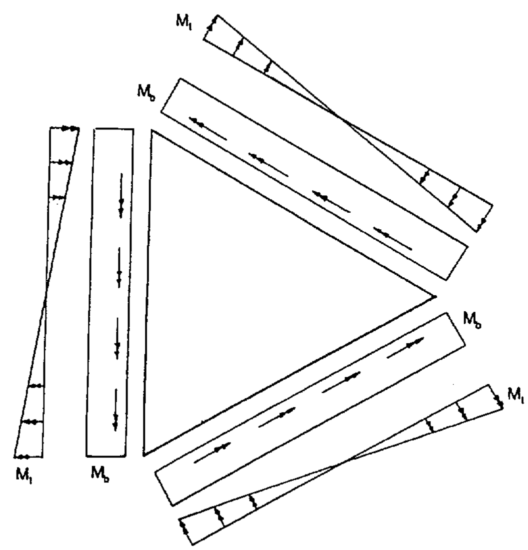
Objective

To verify the behaviour of the SHELL element subjected to edge moment loading.

Physical Problem

A triangular plate on three corner point supports as shown in the figure below is considered. The displacement in the Z-direction is constrained to zero at each corner. The plate is loaded along its boundaries with a constant bending moment and a linearly varying twisting moment.





$E = 207 \cdot 10^9$ N/m^2

$\nu = 0.25$

$t = 2.54 \cdot 10^{-3}$ m

$b = 0.24$ m

$c = 0.138564$ m

$M_b = 300$ Nm/m

$M_t = 194.85$ Nm/m

Finite Element Model

Three 16-node SHELL elements are used to model the plate. The element mesh with the boundary condition and the edge moments is shown in the figure on page A29.3.

Solution Results

Theoretical solutions of this problem are presented in references [1] and [2]. Different FEM-solutions can be found in references [3]. The SOLVIA solution presented here has also been presented in [4].

Centroidal displacement (node 7)

	δ_x (m)
SOLVIA	$2.1228 \cdot 10^{-3}$
Reference [1]	$2.1226 \cdot 10^{-3}$

The internal element moment can be formed from the TOP surface stresses:

$$M_{xx} = \frac{t^2}{6}\,\sigma_{xx} \qquad M_{yy} = \frac{t^2}{6}\,\sigma_{yy} \qquad M_{xy} = \frac{t^2}{6}\,\tau_{xy}$$

Calculated moments from SOLVIA analysis (Nm/m)

	Node 6	Node 7	Node 2	Node 1	Node 3
M_{xx}	300.000	187.503	-37.4903	300.000	300.000
M_{yy}	75.0066	187.503	412.500	75.0066	75.0066
M_{xy}	$1 \cdot 10^{-6}$	$3 \cdot 10^{-7}$	$9 \cdot 10^{-11}$	-194.851	194.851

Theoretical moments from reference [1] (Nm/m)

	Node 6	Node 7	Node 2	Node 1	Node 3
M_{xx}	300.000	187.500	-37.5000	300.000	300.000
M_{yy}	75.0000	187.500	412.500	75.0000	75.0000
M_{xy}	0	0	0	-194.856	194.856

An excellent agreement can be observed.

The stress distributions are shown on page A29.4.

User Hints

- The loading consists of two parts, namely a constant bending moment and a linearly varying twisting moment. The constant bending moment load produces constant curvatures in the plate and hence a deformed shape that is spherical. The stress field consists only of bending stresses and the bending stress components are equal and constant over the faces of the plate. It is a basic requirement for any plate element that constant bending stresses can be modeled exactly.

  The difficulty in the triangular plate problem lies instead in obtaining a good solution to the loading by the linearly varying twisting moment. The magnitude of this applied twisting moment is such that the edges of the plate, which are deformed by the constant bending moment to lie on a spherical surface, are deformed back to their original straight line shape. This magnitude of the twisting moment makes it easy to check the resulting edge displacements since these should be zero.

- The two bending stress components and the in-plane shear stress component are varying linearly due to the applied twisting moment. The shear stress components in the thickness direction are zero. The case of the linearly varying twisting moment can be regarded as a patch test, which illuminates the ability of a plate element to model a linearly varying state of stress.

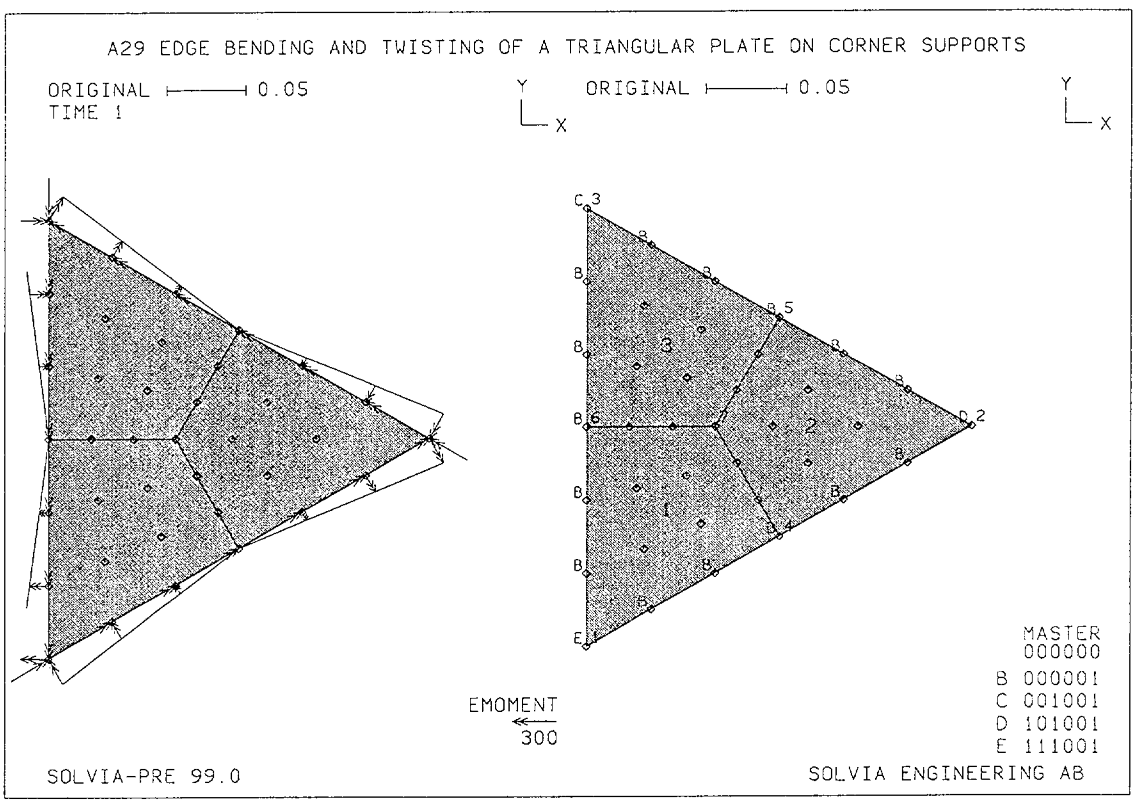
References

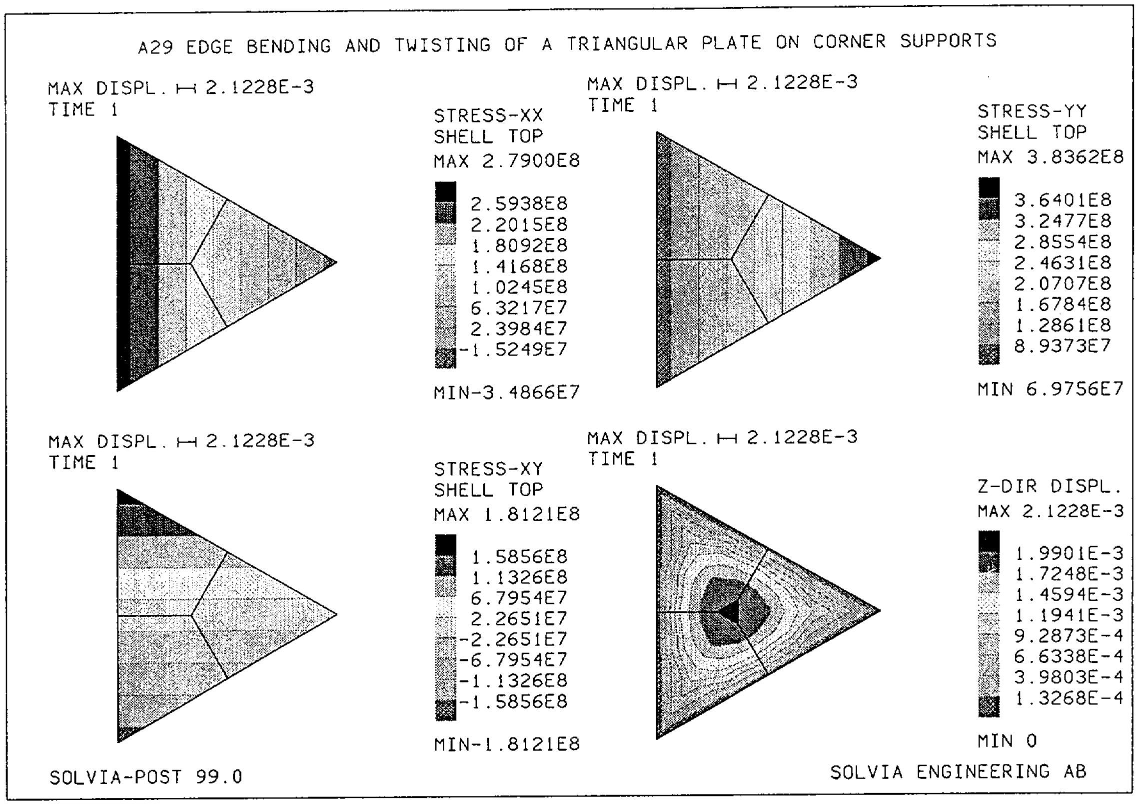
[1] Robinson, J., "A FEN Project - Triangular Plate-Bending Continuum on Three Point Supports", Finite Element News, 1992 Issue no. 1.

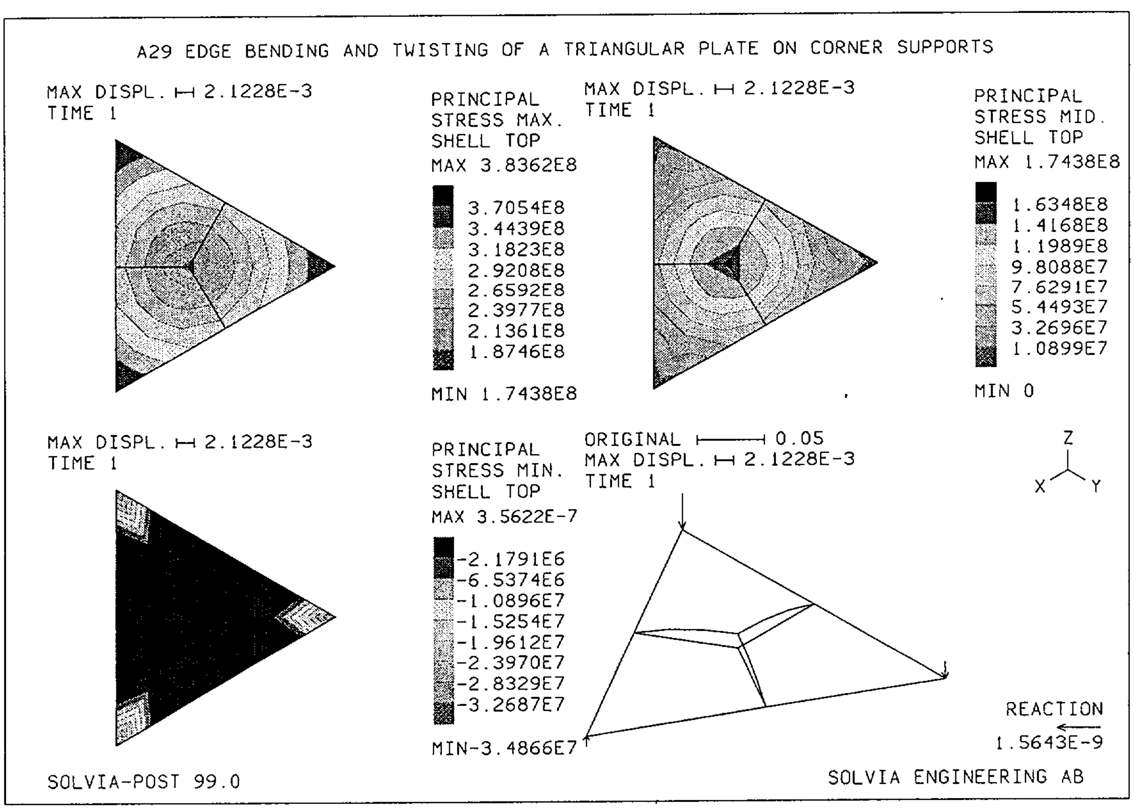
[2] Robinson, J., "An Elasticity Solution for a Triangular Plate-Bending Continuum on Three Point Supports", Finite Element News, 1995 Issue no. 2.

[3] Finite Element News, 1992 Issues no. 2, 4, 5, 6 and 1993 Issue no. 1.

[4] Larsson, G., "A Note on the FEN Project Triangular Plate-Bending on Continuum Supports", Finite Element News, 1994 Issue No. 3.




A29 EDGE BENDING AND TWISTING OF A TRIANGULAR PLATE ON CORNER SUPPORTS
MAX DISPL. 2.1228E-3
TIME 1
STRESS-XX
SHELL TOP
MAX 2.7900E8
2.5938E8
2.2015E8
1.8092E8
1.4168E8
1.0245E8
6.3217E7
2.3984E7
-1.5249E7
MIN-3.4866E7
MAX DISPL. 2.1228E-3
TIME 1
STRESS-YY
SHELL TOP
MAX 3.8362E8
3.6401E8
3.2477E8
2.8554E8
2.4631E8
2.0707E8
1.6784E8
1.2861E8
8.9373E7
MIN 6.9756E7
MAX DISPL. 2.1228E-3
TIME 1
STRESS-XY
SHELL TOP
MAX 1.8121E8
1.5856E8
1.1326E8
6.7954E7
2.2651E7
-2.2651E7
-6.7954E7
-1.1326E8
-1.5856E8
MIN-1.8121E8
MAX DISPL. 2.1228E-3
TIME 1
Z-DIR DISPL.
MAX 2.1228E-3
1.9901E-3
1.7248E-3
1.4594E-3
1.1941E-3
9.2873E-4
6.6338E-4
3.9803E-4
1.3268E-4
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A29 EDGE BENDING AND TWISTING OF A TRIANGULAR PLATE ON CORNER SUPPORTS
MAX DISPL. 2.1228E-3
TIME 1
PRINCIPAL
STRESS MAX.
SHELL TOP
MAX 3.8362E8
3.7054E8
3.4439E8
3.1823E8
2.9208E8
2.6592E8
2.3977E8
2.1361E8
1.8746E8
MIN 1.7438E8
MAX DISPL. 2.1228E-3
TIME 1
PRINCIPAL
STRESS MID.
SHELL TOP
MAX 1.7438E8
1.6348E8
1.4168E8
1.1989E8
9.8088E7
7.6291E7
5.4493E7
3.2696E7
1.0899E7
MIN 0
MAX DISPL. 2.1228E-3
TIME 1
PRINCIPAL
STRESS MIN.
SHELL TOP
MAX 3.5622E-7
-2.1791E6
-6.5374E6
-1.0896E7
-1.5254E7
-1.9612E7
-2.3970E7
-2.8329E7
-3.2687E7
MIN-3.4866E7
ORIGINAL 0.05
MAX DISPL. 2.1228E-3
TIME 1
Z
X
Y
REACTION
1.5643E-9
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
DATABASE CREATE
HEADING 'A29 EDGE BENDING AND TWISTING OF A TRIANGULAR PLATE ON
 CORNER SUPPORTS'
*
COORDINATES
1  0.    -0.138564
2  0.24   0
3  0      0.138564
4  0.12 -0.069282
5  0.12  0.069282
6  0      0
7  0.08   0
*
MATERIAL 1 ELASTIC E=207E9 NU=0.25
*
EGROUP 1 SHELL RESULT=NSTRESS
THICKNESS 1 0.00254
GSURFACE 1 4 7 6
GSURFACE 2 5 7 4
GSURFACE 3 6 7 5
*
LOADS ELEMENT MOMENTINTENSITY INPUT=LINES
1 2 edge -300    -300
2 3 edge -300    -300
3 1 edge -300    -300
1 2 out  -194.85 194.85
2 3 out  -194.85 194.85
3 1 out  -194.85 194.85
*
FIXBOUNDARIES 123  / 1
FIXBOUNDARIES 13   / 2
FIXBOUNDARIES 3    / 3
FIXBOUNDARIES 6 INPUT=LINE / 2 3 / 3 1 / 1 2
*
SET NSYMBOL=YES VIEW=Z
MESH VECTOR=LOAD SUBFRAME=21
MESH BCODE=ALL NNUMBER=MYNODES ENUMBERS=YES
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A29 EDGE BENDING AND TWISTING OF A TRIANGULAR PLATE ON CORNER
SUPPORTS
*
DATABASE CREATE
WRITE 'a29.lis'
NLIST MYNODES
ELIST
ELIST SELECT=SEFFECTIVE
*
SET VIEW=Z AXES=NO HEIGHT=0.28
SUBFRAME 22
MESH CONTOUR=SXX
MESH CONTOUR=SYY
MESH CONTOUR=SXY
MESH CONTOUR=DZISPLACEMENT
*
SUBFRAME 22
MESH CONTOUR=SPMAX
MESH CONTOUR=SPMID
MESH CONTOUR=SPMIN
COLOR LINE
SET VIEW=I ORIGINAL=YES AXES=YES
MESH VECTOR=REACTION
END
```