EXAMPLE A81

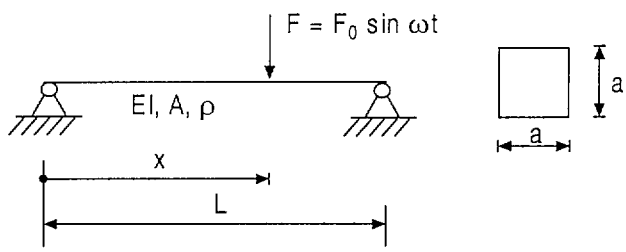
HARMONIC RESPONSE OF A SIMPLY SUPPORTED BEAM

Objective

To verify the harmonic response method when different boundary conditions of a model are used.

Physical Problem

A simply supported beam is exposed to a sinusoidal force as shown in the figure below.



$E = 2.0 \cdot 10^{11}$ N/m^2
$a = 0.1$ m
$L = 10$ m
$x = 7.5$ m
$\rho = 7800$ kg/m^3
$F_o = 100$ N

Finite Element Model

The model consists of eight BEAM elements. Due to symmetry of the structure it is enough to model one half of the beam. The sinusoidal load is non-symmetric and, therefore, two runs with different boundary conditions need to be done. This analysis procedure may be of interest if the structure is very large. The procedure is used here for demonstration and verification reasons only.

Solution Results

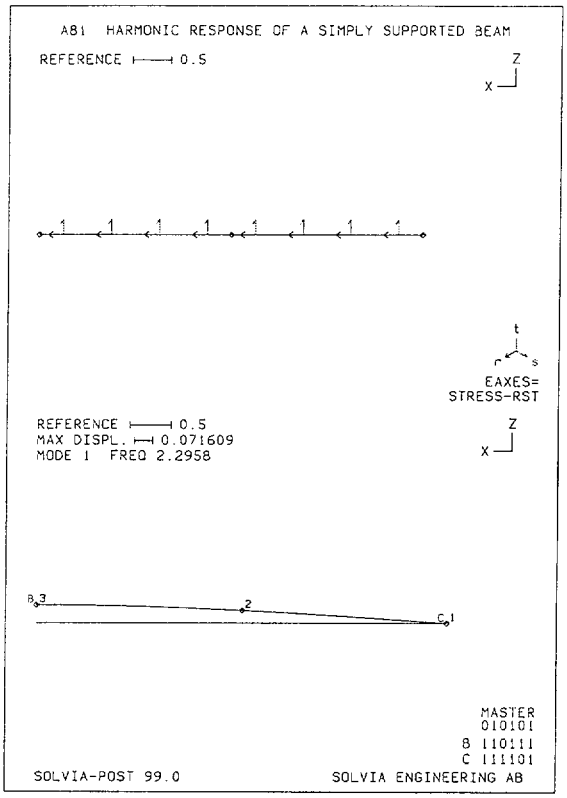
The input data used in the two analyses are shown on pages A81.3 and A81.4.

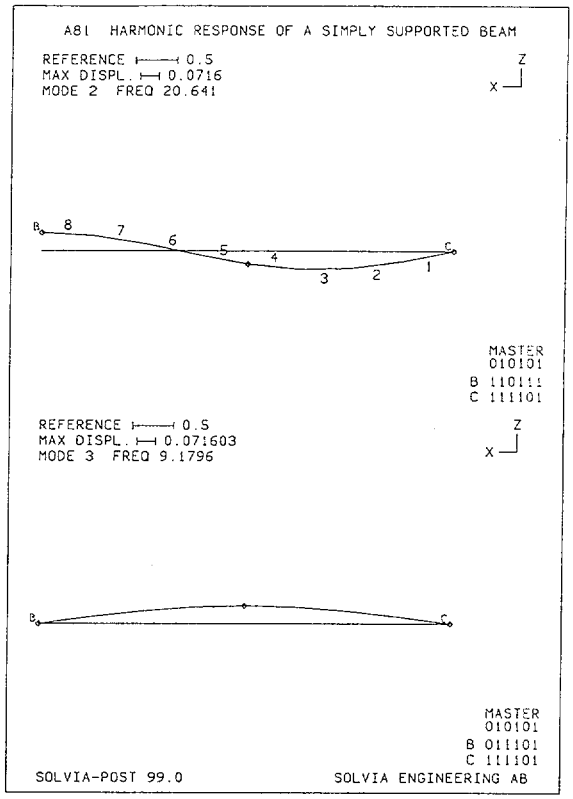
The nodal results from symmetric and anti-symmetric boundary conditions are saved in the SOLVIA-POST database. Mode number 1 and 2 are calculated with symmetric boundary and mode 3 is calculated with anti-symmetric boundary. The mode shapes with their boundary conditions are shown in the top figures on page A81.2.

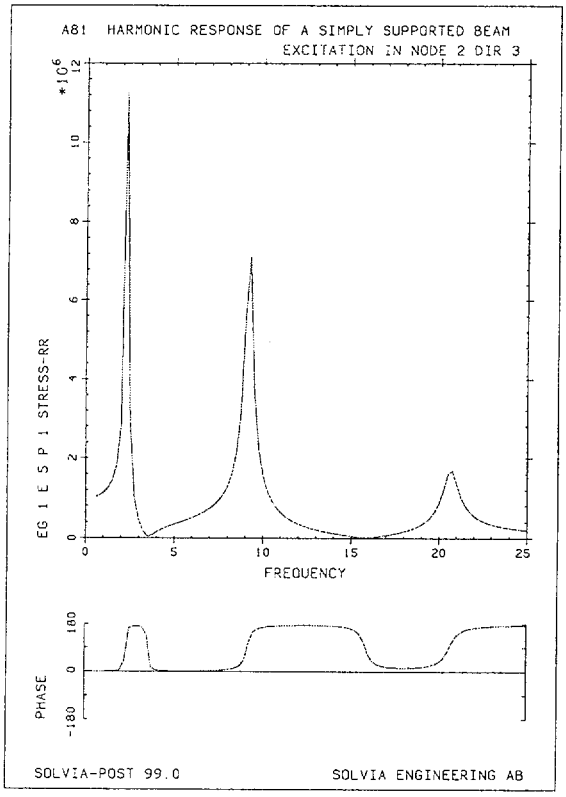
The harmonic response for the Z-displacement, node 2 and the stress component stress-rr in element 5, point 1 have been calculated and are shown in the bottom figures on page A81.2.

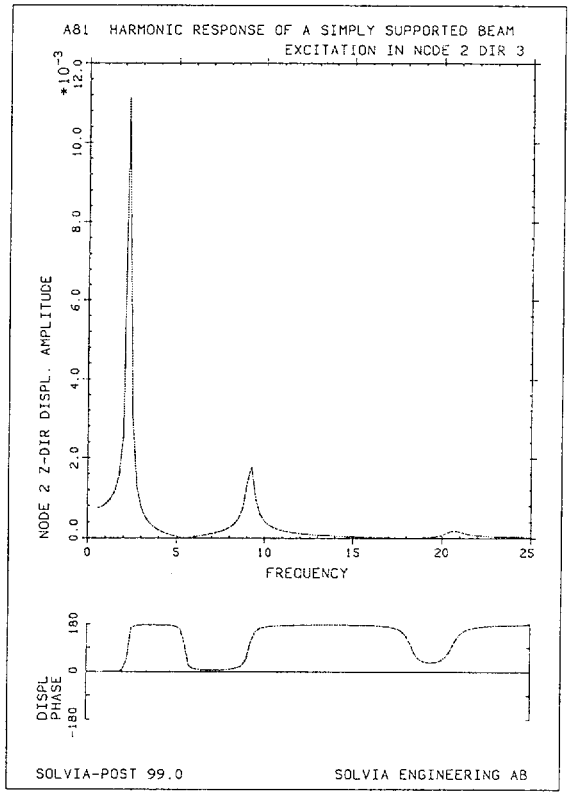
Numerical results for 9 Hz including corresponding theoretical values based on three mode shapes of an Bernoulli-Euler continuous beam:

	SOLVIA-POST	Theory
Z-disp, node 2 [m]	$1.375 \cdot 10^{-3}$	$1.355 \cdot 10^{-3}$
Stress-rr, element 5 point 1 $[N/m^2]$	$5.616 \cdot 10^{6}$	$5.473 \cdot 10^{6}$


A81 HARMONIC RESPONSE OF A SIMPLY SUPPORTED BEAM
REFERENCE 0.5
EAXES=
STRESS-RST
REFERENCE 0.5
MAX DISPL. 0.071609
MODE 1 FREQ 2.2958
MASTER
010101
B 110111
C 111101
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A81 HARMONIC RESPONSE OF A SIMPLY SUPPORTED BEAM
REFERENCE 0.5
MAX DISPL. 0.0716
MODE 2 FREQ 20.641
MASTER
010101
B 110111
C 111101
REFERENCE 0.5
MAX DISPL. 0.071603
MODE 3 FREQ 9.1796
MASTER
010101
B 011101
C 111101
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A81 HARMONIC RESPONSE OF A SIMPLY SUPPORTED BEAM
EXCITATION IN NODE 2 DIR 3
EG 1 E S P 1 STRESS-RR
FREQUENCY
PHASE
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A81 HARMONIC RESPONSE OF A SIMPLY SUPPORTED BEAM
EXCITATION IN NODE 2 DIR 3
NODE 2 Z-DIR DISPL. AMPLITUDE
FREQUENCY
DISPL PHASE
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A81  HARMONIC RESPONSE OF A SIMPLY SUPPORTED BEAM'
*
DATABASE  CREATE
MASTER  IDOF=010101
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=CONSISTENT
FREQUENCIES  SUBSPACE-ITERATION NEIG=2 SSTOL=1.E-10,
             MODALSTRESS=YES
*
COORDINATES
 1  TO  3  5.
*
MATERIAL  1  ELASTIC E=2.E11 NU=0. DENSITY=7800.
*
EGROUP   1  BEAM  RESULTS=STRESSES
SECTION  1  RECTANGULAR  WTOP=0.1 D=0.1
BEAMVECTOR  /  1  0. 0. 1.
GLINE  N1=1 N2=3 AUX=-1  EL=8
*
*   First run, symmetric
*
FIXBOUNDARIES  13  /  1
FIXBOUNDARIES  15  /  3
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A81  HARMONIC RESPONSE OF A SIMPLY SUPPORTED BEAM
*
*   First run, symmetric
*
DATABASE  CREATE
END
```

SOLVIA-PRE input

```
DATABASE  OPEN
FREQUENCIES  SUBSPACE-ITERATION NEIG=1 SSTOL=1.E-10,
             MODALSTRESS=YES
*
*   Second run, anti-symmetric
*
DELETE  BOUNDARIES
FIXBOUNDARIES  13  /  1
FIXBOUNDARIES   3  /  3
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A81  HARMONIC RESPONSE OF A SIMPLY SUPPORTED BEAM
*
*   Second run, anti-symmetric
*
DATABASE  ADD
WRITE  FILENAME='a81b.lis'
*
FREQUENCIES
MASS-PROPERTIES
*
SET  PLOTORIENTATION=PORTRAIT
SET  RESPONSETYPE=VIBRATIONMODE VIEW=Y NSYMBOLS=MYNODES,
     ORIGINAL=YES
MESH  DEFORMED=NO EAXES=STRESS-RST SUBFRAME=12
MESH  TIME=1 BCODE=ALL NNUMBERS=MYNODES
MESH  TIME=2 BCODE=ALL ENUMBER=YES SUBFRAME=12
MESH  TIME=3 BCODE=ALL
*
DAMPING  1  /  1 2  TO  3 2
*
EHARMONIC-RESPONSE  ELEMENT=5 POINT=1 KIND=SRR FSTART=0.5
FSTEP=0.25,
                    FEND=25 AXES=1 OUTPUT=ALL MEND=3
 2 3 50. 0.
END DATA
*
NHARMONIC-RESPONSE  NODE=2 DIRECTION=3 KIND=DISPLACEMENT FSTART=0.5,
                    FSTEP=0.25 FEND=25 AXES=1 OUTPUT=ALL MEND=3
 2 3 50. 0.
END DATA
*
ZONE  NAME=EL45 INPUT=ELEMENTS  /  4 5
ELIST  ZONENAME=EL45 TSTART=1 TEND=3
NLIST  TSTART=1 TEND=3
END
```

EXAMPLE A82

STIFFENED PLATE CANTILEVER UNDER TIP MOMENT

Objective

To verify the use of the linear BEAM element offset capability.

Physical Problem

A cantilever of channel cross-section is loaded by concentrated end moments as shown in the figure on page A23.1.

Finite Element Model

The model is shown in the top figure on page A82.2. The BEAM element nodes lie in the same plane (Z=0) as the PLATE element nodes. By using the offset option in the local t-direction (TOFFSET) a translation of the BEAM elements from the plane of the nodes is made.

Solution Results

The theoretical displacement is

$$\delta = \frac{ML^2}{2EI}$$

The input data on page A82.3 gives the following result:

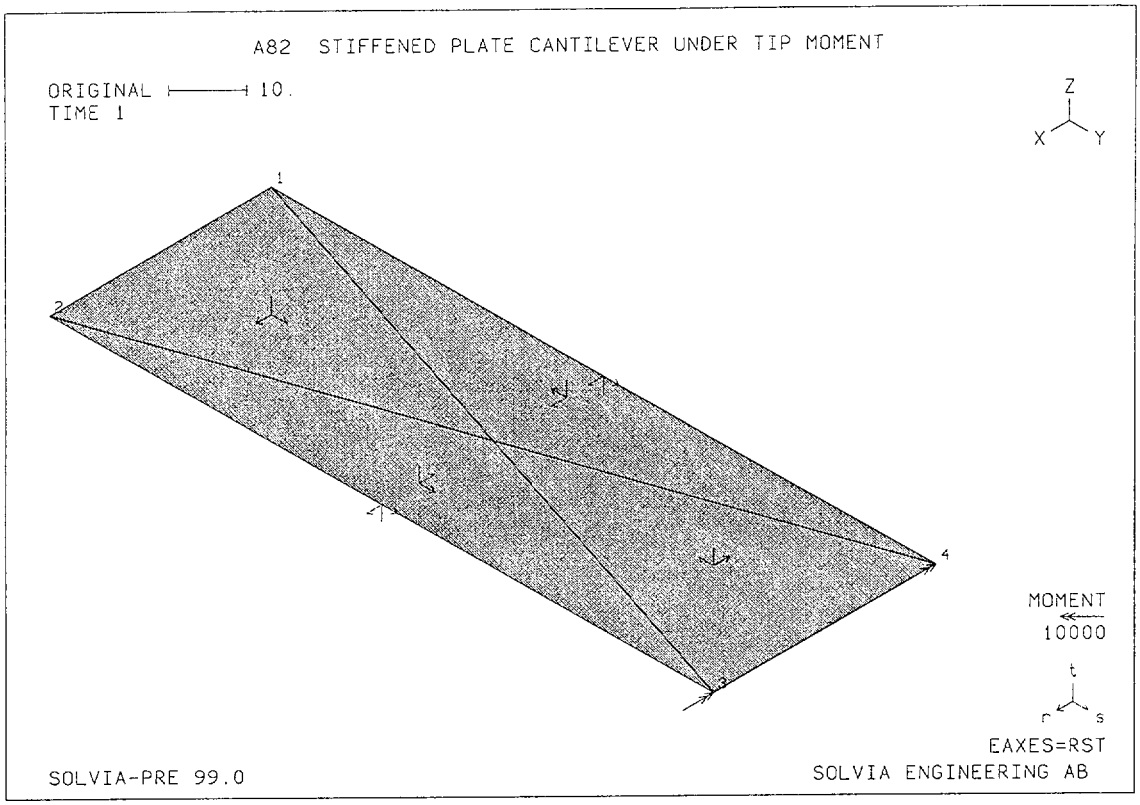
End displacement δ (inch):

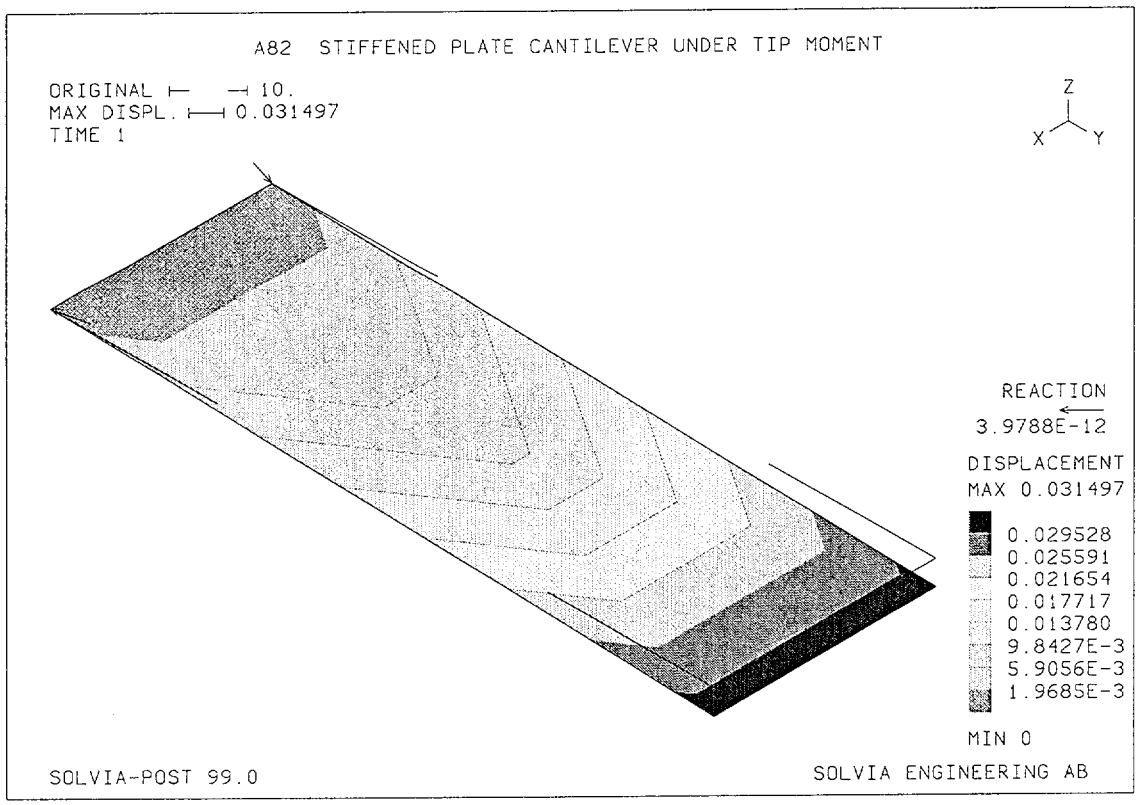
Theory	SOLVIA
0.03149	0.03149

User Hints

- Note that if the plate cantilever is subjected to concentrated force end loads as described in Example A23 a finer mesh is required due to the constant membrane action of the PLATE element.

- Note that the specification of the BEAM section offset data refer to the local beam coordinate system.


A82 STIFFENED PLATE CANTILEVER UNDER TIP MOMENT
ORIGINAL 10.
TIME 1
Z
X Y
1
2
3
4
MOMENT
10000
t
r s
EAXES=RST
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A82 STIFFENED PLATE CANTILEVER UNDER TIP MOMENT
ORIGINAL 10.
MAX DISPL. 0.031497
TIME 1
Z
X Y
REACTION
3.9788E-12
DISPLACEMENT
MAX 0.031497
0.029528
0.025591
0.021654
0.017717
0.013780
9.8427E-3
5.9056E-3
1.9685E-3
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A82  STIFFENED PLATE CANTILEVER UNDER TIP MOMENT'
*
DATABASE  CREATE
COORDINATES
  1  /  2  40.  /  3  40. 120.  /  4  0. 120.
*
MATERIAL  1  ELASTIC E=3.E7 NU=0.
*
EGROUP  1  PLATE
GSURFACE  1 2 3 4  EL1=1 EL2=1
EDATA  /  1  1.
*
EGROUP  2  BEAM
SECTION  1 RECTANGULAR WTOP=1. D=6. TOFFSET=3.5
BEAMVECTOR  /  1  1.
ENODES
 1  -1 2 3
 2  -1 1 4
*
FIXBOUNDARIES  /  1 2
LOADS  CONCENTRATED
 3 4 -10000.
 4 4 -10000.
*
MESH  NNUMBER=MYNODES EAXES=RST VECTOR=MOMENT
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A82  STIFFENED PLATE CANTILEVER UNDER TIP MOMENT
*
DATABASE  CREATE
*
WRITE  FILENAME='a82.lis'
*
MESH ORIGINAL=DASHED OUTLINE=YES CONTOUR=DISPLACEMENTS
VECTOR=REACTION
*
NLIST
NLIST  KIND=REACTION
*
SYSTEM 1 CARTESIAN
ELIST SYSTEM=1
END
```

EXAMPLE A83

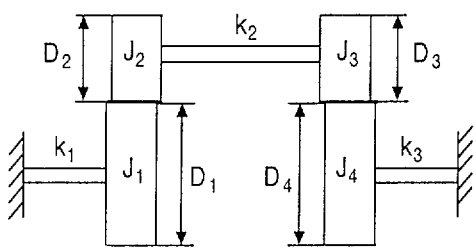
FREQUENCY ANALYSIS OF A TORSION SPRING SYSTEM

Objective

To verify the use of constraint equations in frequency analysis using consistent mass matrix.

Physical Problem

A torsion spring system as shown in the figure below is considered.



Gears

$$n_1 = \frac{D_1}{D_2} = 1.5$$

$$n_4 = \frac{D_4}{D_3} = 1.5$$

Torsion spring constant	Mass moment of inertia
$k_1 = 100$ Nm/rad	$J_1 = 10$ kgm^2
$k_2 = 150$ Nm/rad	$J_2 = 5$ kgm^2
$k_3 = 200$ Nm/rad	$J_3 = 7.5$ kgm^2
	$J_4 = 15$ kgm^2

Finite Element Model

The spring system is modeled using three SPRING AXIALROTATION elements. The massive gears are modeled with concentrated rotational masses and the speed ratios between different shafts are simulated using constraint equations.

Solution Results

The theoretical solution can be found in [1] p. 68. The input data shown on page A83.2 agrees with the theoretical solution:

$f_1 = 0.3775$ Hz

$f_2 = 0.8979$ Hz

User Hints

- Note that a consistent mass matrix must, in general, be used when using RIGIDLINKS or CONSTRAINTS in a dynamic analysis or when massproportional or centrifugal loading is employed.

Reference

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

SOLVIA-PRE input

```
HEADING  'A83  FREQUENCY ANALYSIS OF A TORSION SPRING SYSTEM'
*
DATABASE  CREATE
MASTER  IDOF=111011
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=CONSISTENT
FREQUENCY  DETERMINANT-SEARCH NEIG=2 MODALSTRESSES=YES
*
COORDINATES
 1  /  2  5.  /  3  5. 5.  /  4  10. 5.
 5  10.  /  6  15.
*
MASSES
 2  0 0 0 10.   /  3  0 0 0 5
 4  0 0 0  7.5  /  5  0 0 0 15
*
EGROUP  1  SPRING DIRECTION=AXIALROTATION
PROPERTYSET  1 K=100
PROPERTYSET  2 K=150
PROPERTYSET  3 K=200
ENODES
 1  1 2  /  2  3 4  /  3  5 6
EDATA  /  1 1  /  2 2  /  3 3
*
FIXBOUNDARIES  /  1 6
CONSTRAINTS
 3 4 2 4 -1.5
 4 4 5 4 -1.5
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A83  FREQUENCY ANALYSIS OF A TORSION SPRING SYSTEM
*
DATABASE  CREATE
*
WRITE  FILENAME='a83.lis'
*
FREQUENCIES
SET  RESPONSETYPE=VIBRATIONMODE
NLIST  TSTART=1 TEND=2
ELIST  TSTART=1 TEND=2
END
```

EXAMPLE A84

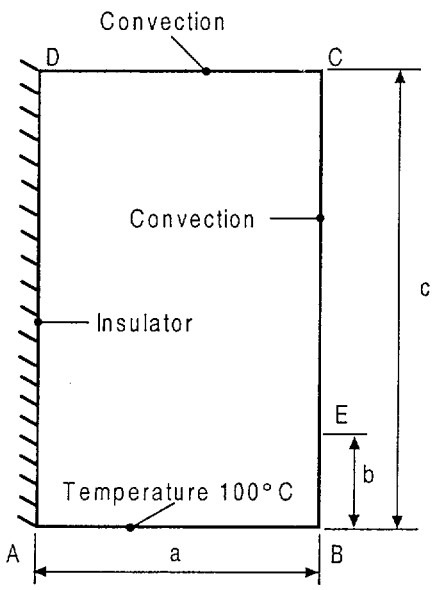
TWO-DIMENSIONAL HEAT TRANSFER WITH CONVECTION

Objective

To verify the PLANE conduction element with convective boundary conditions in a steady-state heat transfer analysis.

Physical Problem

A rectangular area, as shown in the figure below, is subjected to a fixed temperature of 100°C at the bottom edge. The left edge is insulated. The top and right edges have a convective boundary condition with an environmental temperature of 0°C specified. This problem is described in [1].



Uniform thickness

a = 0.6 m

b = 0.2 m

c = 1.0 m

Material properties

Conductivity = 52.0 W/m °C

Convection coefficient = 750 W/m^2 °C

(convection to ambient temperature at 0° C along B-C-D)

Finite Element Model

Fifteen 8-node PLANE conduction elements are used to model the rectangular area as shown in the top figure on page A84.3. The convective boundary condition is specified by convection boundary segments along the top and the right edge of the plate. The mesh spacing is performed as prescribed in [1].

Solution Results

The input data shown on page A84.4 is used in the SOLVIA-TEMP analysis. The analytical solution to this problem is derived in [2] and is presented in [1]. The solution to this problem is a highly variable temperature field as shown in the figure on page A84.3. The analytical value for the temperature at point E is 18.3°C. The temperature calculated by SOLVIA-TEMP at point E (Node 9) is 18.3°C.

A contour plot of temperature distribution and a vector plot of element fluxes and the temperature distribution along the right edge can be seen in the figure on page A84.3.

User Hints

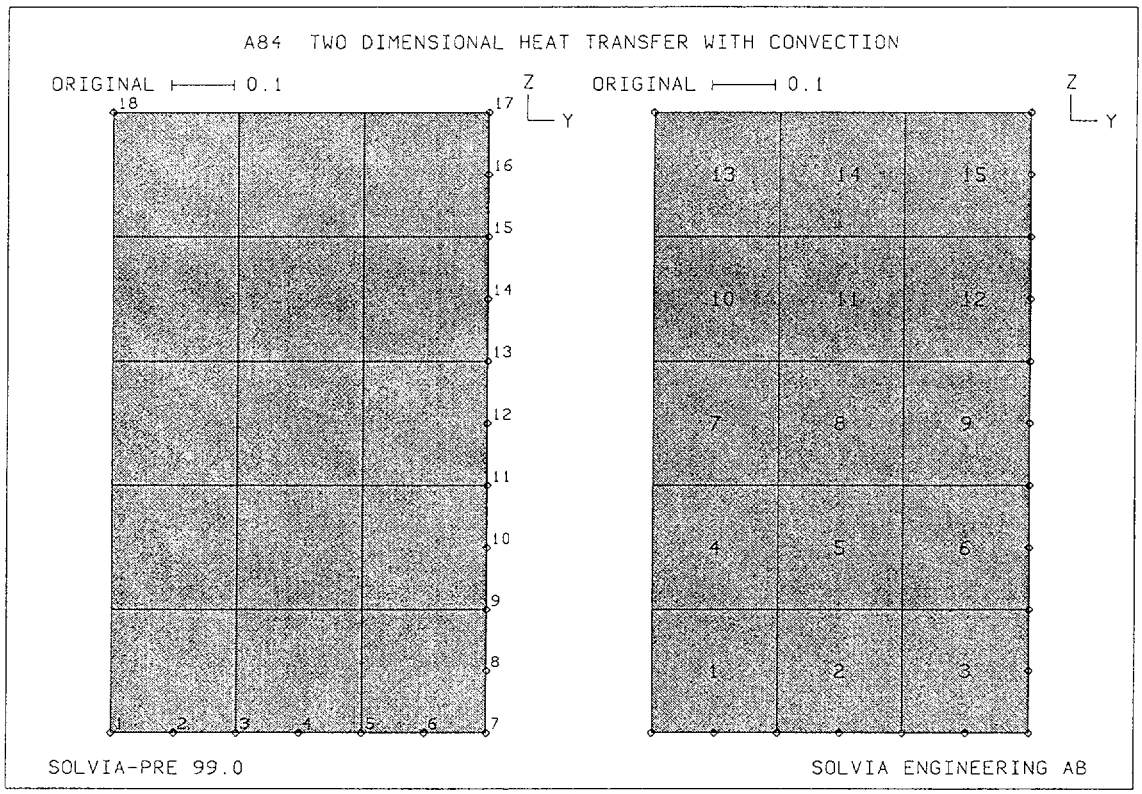
- This problem is modelled by a relative coarse mesh. A much finer mesh should be used in practice near the large temperature gradient at the right lower corner of the model.

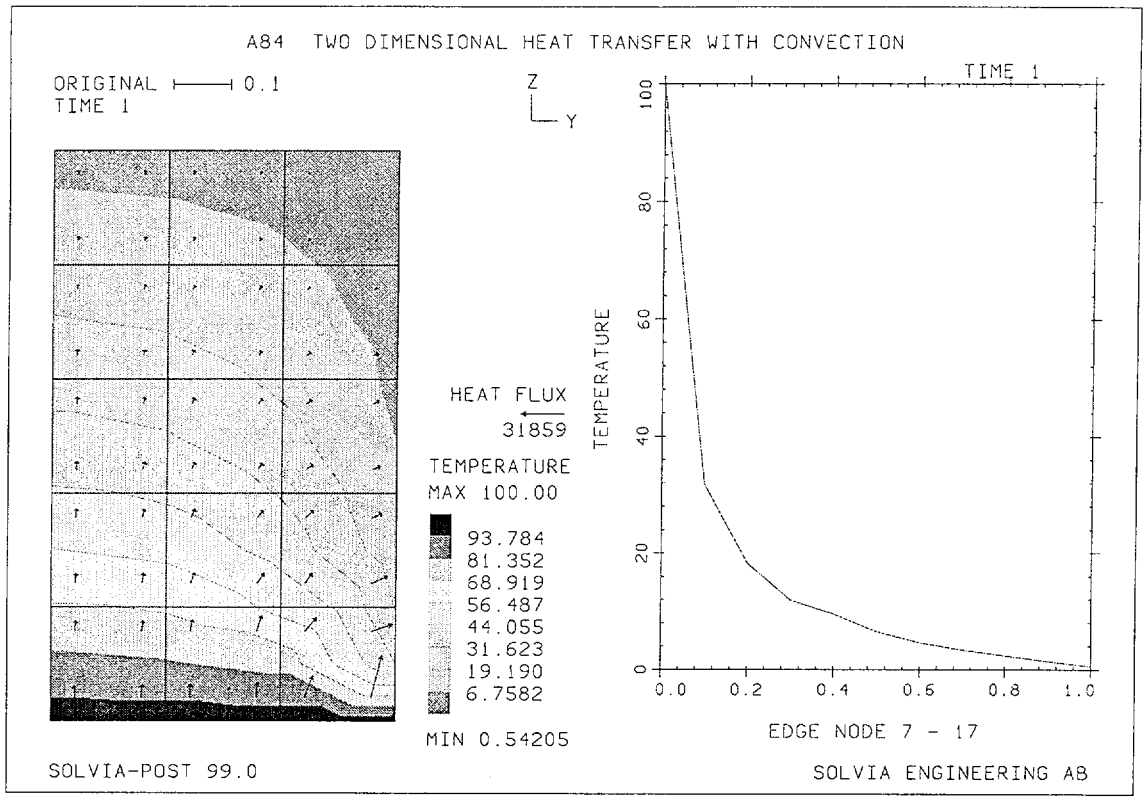
- Note that a contour plot of the flux deviation gives valuable information regarding the quality of the mesh in a SOLVIA-TEMP analysis, see bottom figure of page A84.3.

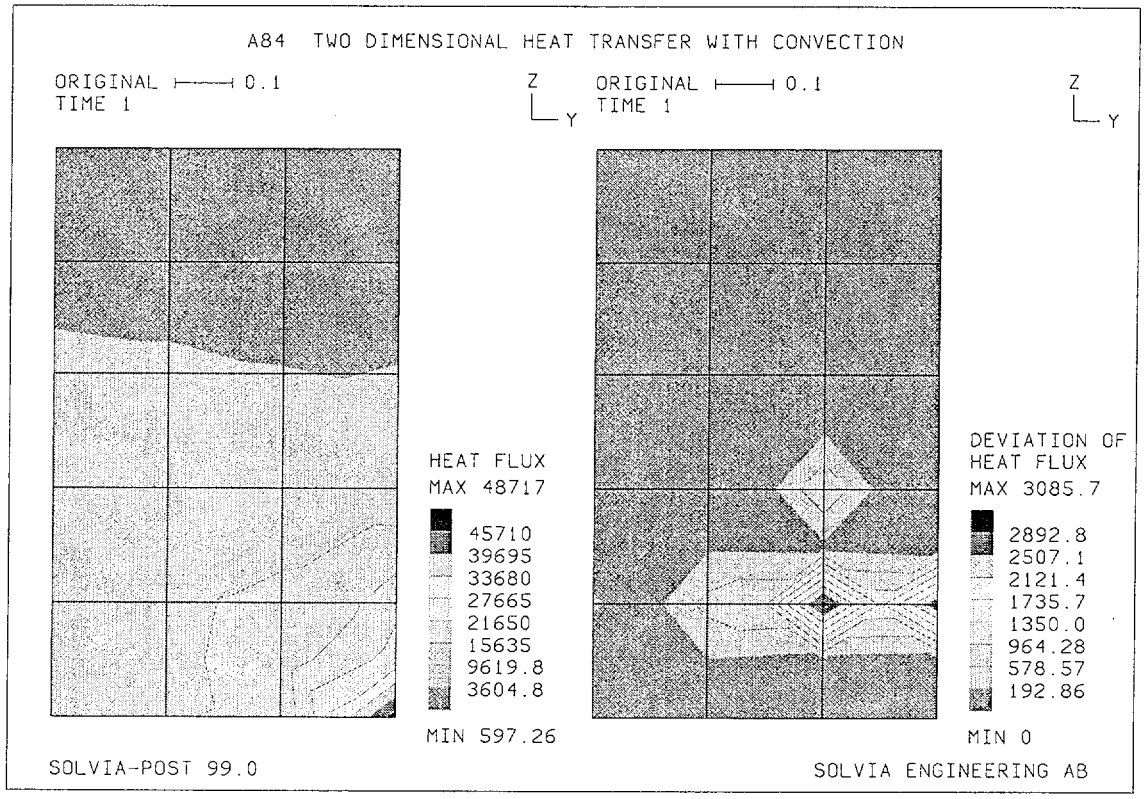
References

[1] NAFEMS, Background to Benchmarks, 1993

[2] Carslaw, H.S. and Jaeger, J.C., Conduction of Heat in Solids, Second Ed., Oxford University Press, 1959.




A84 TWO DIMENSIONAL HEAT TRANSFER WITH CONVECTION
ORIGINAL 0.1
TIME 1
Z
Y
HEAT FLUX
31859
TEMPERATURE
MAX 100.00
93.784
81.352
68.919
56.487
44.055
31.623
19.190
6.7582
MIN 0.54205
TIME 1
TEMPERATURE
0
20
40
60
80
100
0.0
0.2
0.4
0.6
0.8
1.0
EDGE NODE 7 - 17
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A84 TWO DIMENSIONAL HEAT TRANSFER WITH CONVECTION
ORIGINAL 0.1
TIME 1
Z
Y
HEAT FLUX
MAX 48717
45710
39695
33680
27665
21650
15635
9619.8
3604.8
MIN 597.26
ORIGINAL 0.1
TIME 1
Z
Y
DEVIATION OF
HEAT FLUX
MAX 3085.7
2892.8
2507.1
2121.4
1735.7
1350.0
964.28
578.57
192.86
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A84  TWO DIMENSIONAL HEAT TRANSFER WITH CONVECTION'
*
DATABASE  CREATE
*
COORDINATES
 ENTRIES  NODE   Y   Z
           1    0.  0.  TO  7  0.6  TO
          17    0.6 1.
          18    0.  1.
*
T-MATERIAL  1 CONDUCTION K=52.0
T-MATERIAL  2 CONVECTION H=750.
*
EGROUP  1  PLANE STRAIN  MATERIAL=1
GSURFACE  1 7 17 18  EL1=3 EL2=5 NODES=8
*
T-BOUNDARIES  SEGMENTS CON-MATERIAL=2 INPUT=LINES
  7 17
 17 18
*
T-LOADS  ENVIRONMENT CONVECTION=YES INPUT=LINES
  7 17 0. 0.
 17 18 0. 0.
T-LOADS  TEMPERATURE
  1 100.  TO  7 100.
*
SET  NSYMBOLS=MYNODES
MESH  NNUMBERS=MYNODES SUBFRAME=21
MESH  ENUMBER=YES
*
SOLVIA-TEMP
END
```

SOLVIA-POST input

```
*  A84  TWO DIMENSIONAL HEAT TRANSFER WITH CONVECTION
*
T-DATABASE  CREATE
*
WRITE  FILENAME='a84.lis'
*
MESH  CONTOUR=TEMPERATURE VECTOR=TFLUX SUBFRAME=21
*
NPLINE  NAME=EDGE
 7  TO  17
NLINE  LINENAME=EDGE KIND=TEMPERATURE OUTPUT=ALL
*
MESH CONTOUR=TFLUX SUBFRAME=21
MESH CONTOUR=FDEVIATION
*
EMAX  NUMBER=3
END
```

EXAMPLE A85

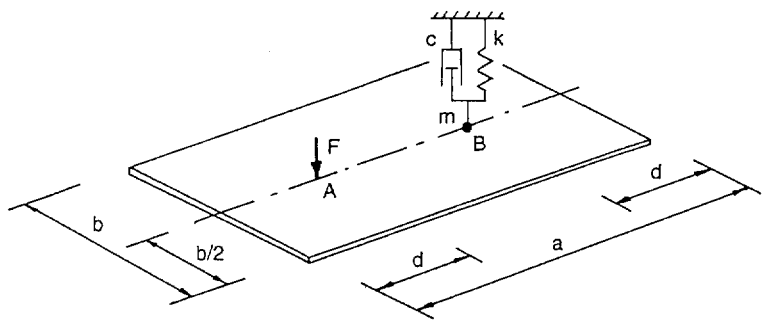
HARMONIC VIBRATION OF A DAMPED PLATE

Objective

To demonstrate a COMPLEX-HARMONIC analysis with SHELL elements.

Physical Problem

A simply supported rectangular plate is subjected to a harmonic transverse point load (at point A in the figure below). At point B the plate has a concentrated mass and spring/damper elements connected to the ground.



$a = 1.5$ m

$b = 1.0$ m

$d = 0.4$ m

$E = 210$ GPa

$\nu = 0.3$

$\rho = 7800$ kg/m^3

$F = F_o \sin \omega t$

$F_o = 1$ N

thickness = 3.0 mm

Rayleigh-damping		Spring to ground	Damper to ground	Point mass
$\alpha = 0.3$ 1/s	$\beta = 0$	$k = 10$ kN/m	$c = 15$ Ns/m	$m = 2.5$ kg

Finite Element Model

The top figure on page A85.2 shows the finite element model. It consists of 150 4-node SHELL elements and two SPRING elements.

Solution Results

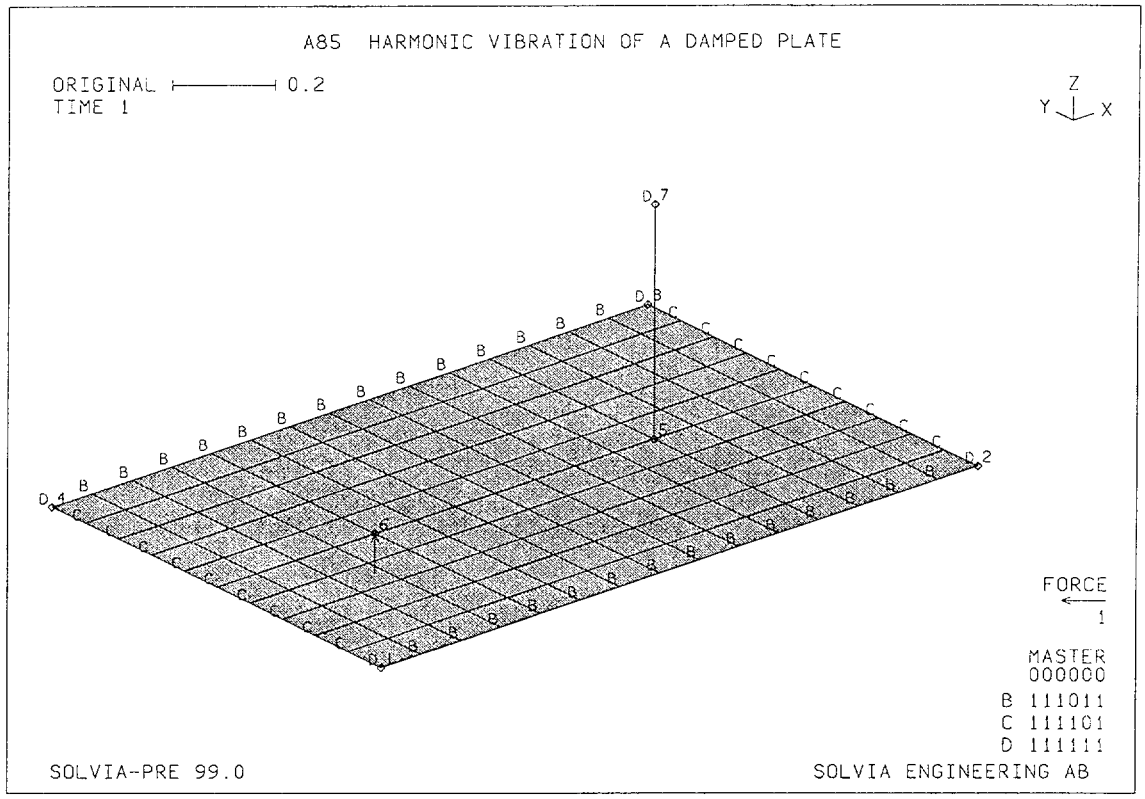
The input data on pages A85.5 and A85.6 has been used. The time average of the power flow in the plate is shown on pages A85.2 and A85.3 at frequencies 9, 11 and 13 Hz. The vector plots of the active power flow show a transport from the load to the damper. A detailed discussion of this example is found in [1] and further examples are found in [2].

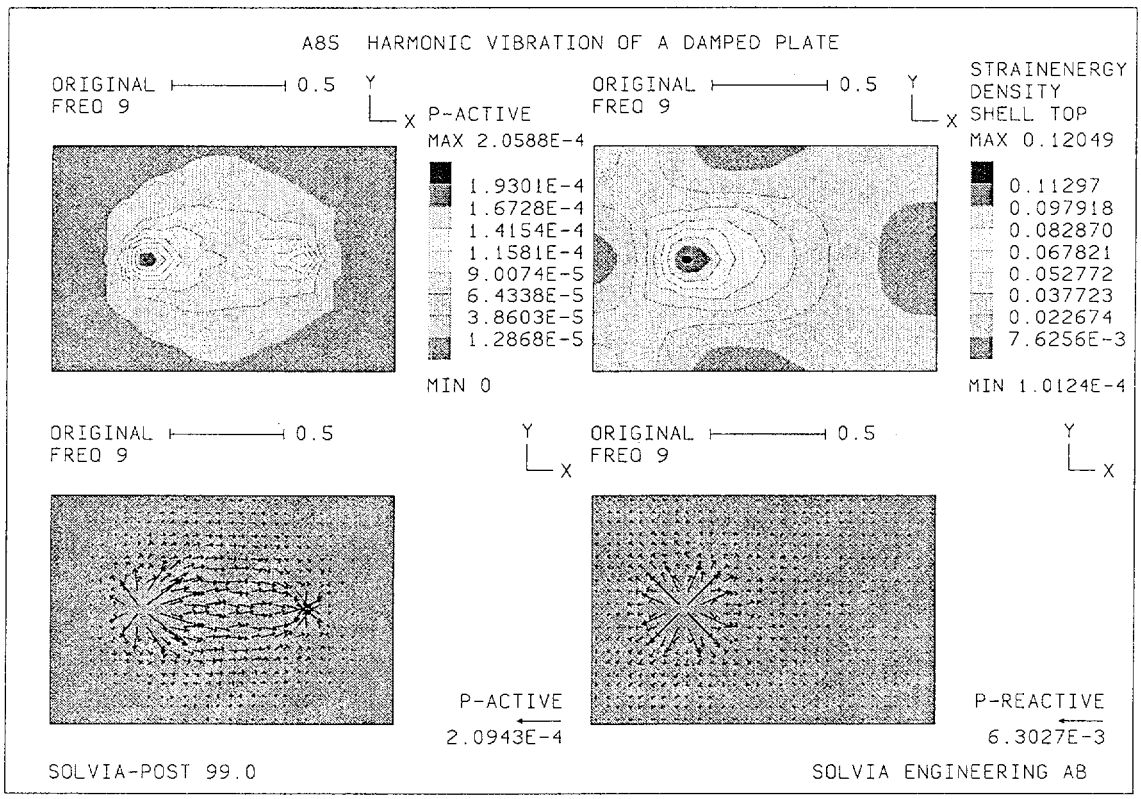
The time averages of the input active and reactive power are shown on page A85.4 as well as the displacement amplitude at point A and the element force amplitude in the damper.

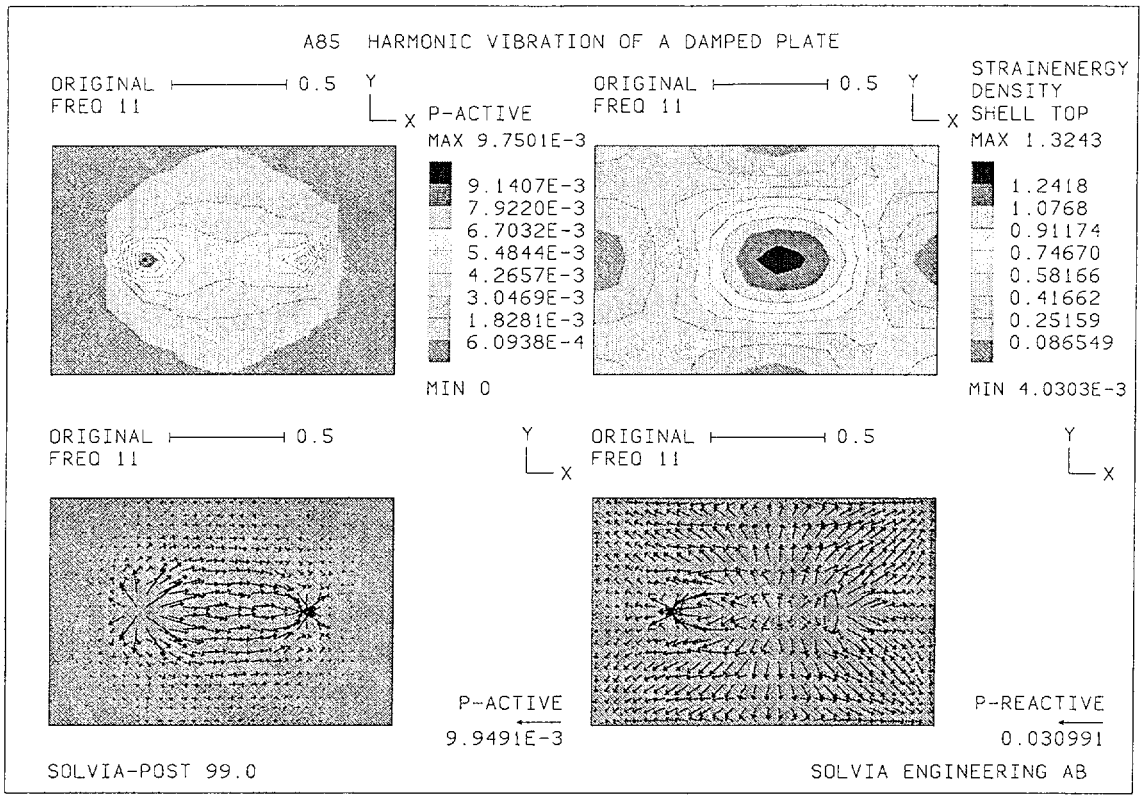
Reference

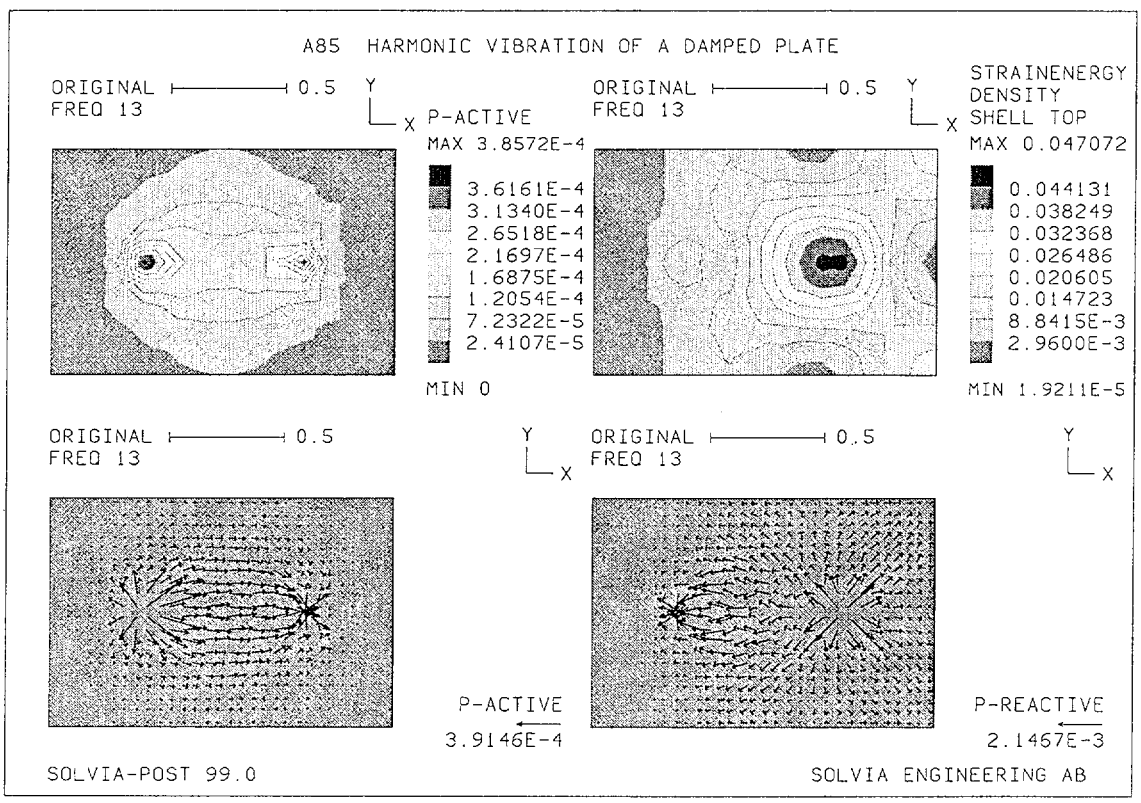
[1] Alfredsson, K.S., "Influence of Local Damping on Active and Reactive Mechanical Power Flow", 4-th International Congress on Intensity Techniques, Senlis, France, 31 Aug - 2 Sept, 1993.

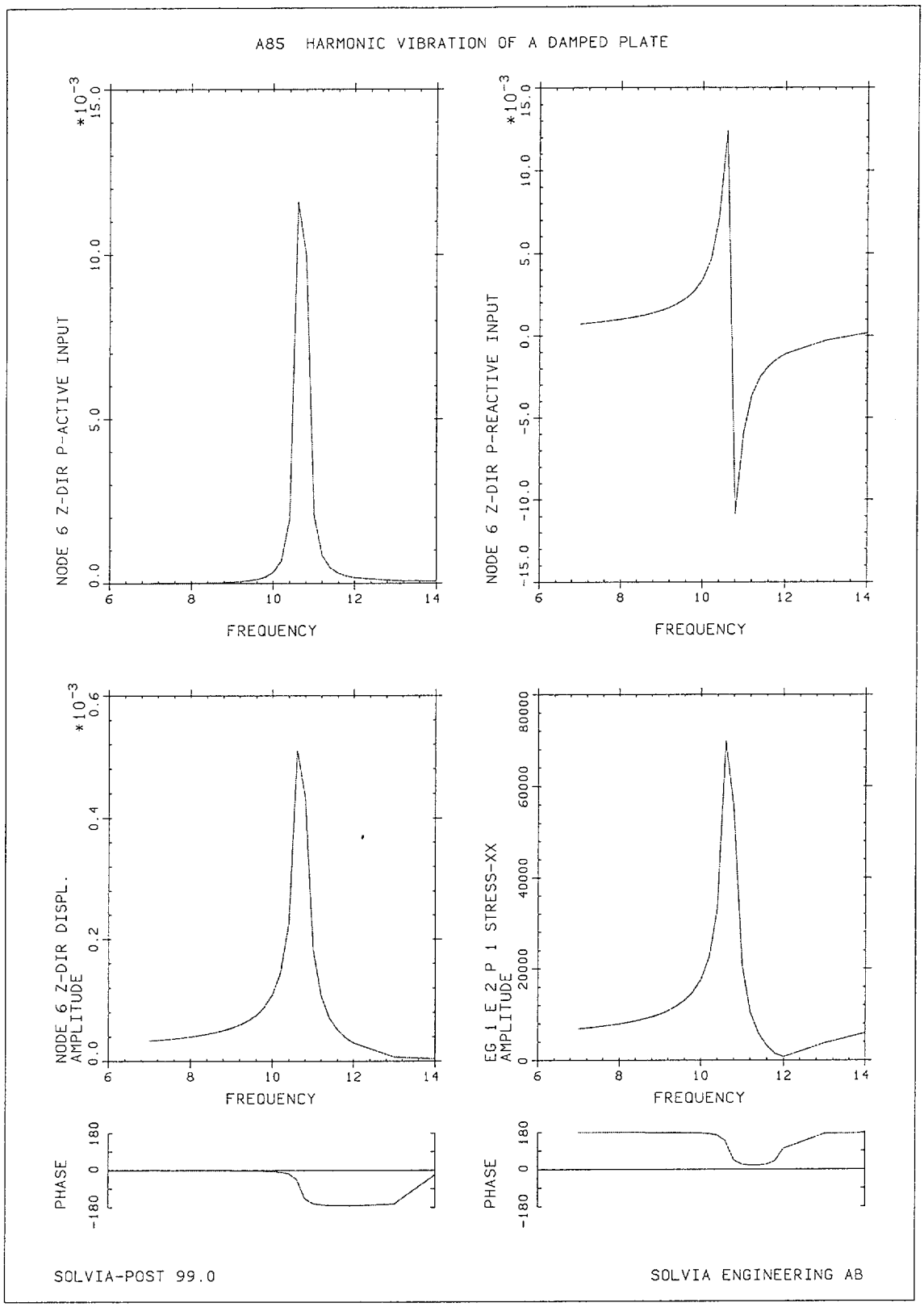
[2] Alfredsson, K.S., Josefson, B.L., Wilson, M.A., "Use of the Energy Flow Concept in Vibration Design", 36th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics and Materials Conference in New Orleans, LA, April 1995.


A85 HARMONIC VIBRATION OF A DAMPED PLATE
ORIGINAL 0.2
TIME 1
Z
Y X
FORCE
1
MASTER
000000
B 111011
C 111101
D 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A85 HARMONIC VIBRATION OF A DAMPED PLATE
ORIGINAL 0.5
FREQ 9
P-ACTIVE
MAX 2.0588E-4
1.9301E-4
1.6728E-4
1.4154E-4
1.1581E-4
9.0074E-5
6.4338E-5
3.8603E-5
1.2868E-5
MIN 0
ORIGINAL 0.5
FREQ 9
STRAINENERGY
DENSITY
SHELL TOP
MAX 0.12049
0.11297
0.097918
0.082870
0.067821
0.052772
0.037723
0.022674
7.6256E-3
MIN 1.0124E-4
ORIGINAL 0.5
FREQ 9
P-ACTIVE
2.0943E-4
ORIGINAL 0.5
FREQ 9
P-REACTIVE
6.3027E-3
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A85 HARMONIC VIBRATION OF A DAMPED PLATE
ORIGINAL
FREQ 11
0.5
Y
X
P-ACTIVE
MAX 9.7501E-3
9.1407E-3
7.9220E-3
6.7032E-3
5.4844E-3
4.2657E-3
3.0469E-3
1.8281E-3
6.0938E-4
MIN 0
ORIGINAL
FREQ 11
0.5
Y
X
STRAINENERGY
DENSITY
SHELL TOP
MAX 1.3243
1.2418
1.0768
0.91174
0.74670
0.58166
0.41662
0.25159
0.086549
MIN 4.0303E-3
ORIGINAL
FREQ 11
0.5
Y
X
P-ACTIVE
9.9491E-3
ORIGINAL
FREQ 11
0.5
Y
X
P-REACTIVE
0.030991
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A85 HARMONIC VIBRATION OF A DAMPED PLATE
ORIGINAL
FREQ 13
0.5
Y
X
P-ACTIVE
MAX 3.8572E-4
3.6161E-4
3.1340E-4
2.6518E-4
2.1697E-4
1.6875E-4
1.2054E-4
7.2322E-5
2.4107E-5
MIN 0
ORIGINAL
FREQ 13
0.5
Y
X
STRAINENERGY
DENSITY
SHELL TOP
MAX 0.047072
0.044131
0.038249
0.032368
0.026486
0.020605
0.014723
8.8415E-3
2.9600E-3
MIN 1.9211E-5
ORIGINAL
FREQ 13
0.5
Y
X
P-ACTIVE
3.9146E-4
ORIGINAL
FREQ 13
0.5
Y
X
P-REACTIVE
2.1467E-3
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A85 HARMONIC VIBRATION OF A DAMPED PLATE
NODE 6 Z-DIR P-ACTIVE INPUT
*10^-3
FREQUENCY
NODE 6 Z-DIR P-REACTIVE INPUT
*10^-3
FREQUENCY
NODE 6 Z-DIR DISPL. AMPLITUDE
*10^-3
FREQUENCY
EG 1 E 2 P 1 STRESS-XX AMPLITUDE
FREQUENCY
PHASE
PHASE
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A85  HARMONIC VIBRATION OF A DAMPED PLATE'
*
DATABASE  CREATE
*
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=CONSISTENT
COMPLEX-HARMONIC  INCREMENTATION=LINEAR
7 2
8 20
12 2 14
RAYLEIGH-DAMPING  ALPHA=0.3 BETA=0.
*
TIMEFUNCTION  1
 0. 1.  /  100. 1.
*
COORDINATES
   1  /  2  1.5  /  3  1.5 1.  /  4  0. 1.
   5  1.1 0.5  /  6  0.4 0.5  /  7  1.1 0.5 0.5
*
MASSES
 5  2.5 2.5 2.5 0. 0. 0.
*
MATERIAL  1  ELASTIC  E=2.1E11 NU=0.3 DENSITY=7800. K=0.
*
EGROUP  1  SHELL
THICKNESS  1  3.E-3
GSURFACE  1 2 3 4  EL1=15 EL2=10 NODES=4
*
EGROUP  2  SPRING DIRECTION=AXIALTRANSLATION
PROPERTYSET  1  K=10E3
PROPERTYSET  2  C=15.
ENODES
 1  5 7  /  2  5 7
EDATA
 1 1  /  2 2
*
FIXBOUNDARIES  12356  INPUT=LINES  /  1 2  /  3 4
FIXBOUNDARIES  12346  INPUT=LINES  /  2 3  /  4 1
FIXBOUNDARIES         INPUT=NODES  /  7
*
LOADS CONCENTRATED
 6 3 1.
*
VIEW  ID=1 XVIEW=-1. YVIEW=-1.2 ZVIEW=0.7
SET  NNUMBERS=MYNODES NSYMBOLS=MYNODES VIEW=1
MESH  BCODE=ALL VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A85  HARMONIC VIBRATION OF A DAMPED PLATE
*
DATABASE  CREATE
WRITE  FILENAME='a85.lis'
*
CONTOUR AVERAGING=YES
SET  RESPONSETYPE=REAL
SET  VIEW=Z ORIGINAL=YES DEFORMED=NO OUTLINE=YES
SET  TIME=9
MESH  CONTOUR=P-ACTIVE SUBFRAME=22
MESH  CONTOUR=ENERGY
MESH  VECTOR=P-ACTIVE
MESH  VECTOR=P-REACTIVE
*
SET  TIME=11
MESH  CONTOUR=P-ACTIVE SUBFRAME=22
MESH  CONTOUR=ENERGY
MESH  VECTOR=P-ACTIVE
MESH  VECTOR=P-REACTIVE
*
SET  TIME=13
MESH  CONTOUR=P-ACTIVE SUBFRAME=22
MESH  CONTOUR=ENERGY
MESH  VECTOR=P-ACTIVE
MESH  VECTOR=P-REACTIVE
*
SET  PLOTORIENTATION=PORTRAIT HEIGHT=0.20
NCOMPLEX-HARMONIC  NODE=6 DIRECTION=3 KIND=P-ACTIVE OUTPUT=ALL
SUBFRAME=22
NCOMPLEX-HARMONIC  NODE=6 DIRECTION=3 KIND=P-REACTIVE
NCOMPLEX-HARMONIC  NODE=6 DIRECTION=3 KIND=DISPLACEMENT
ECOMPLEX-HARMONIC  ELEMENT=2 POINT=1 KIND=SXX
END
```

EXAMPLE A86

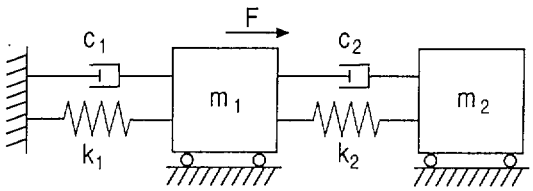
HARMONIC RESPONSE OF A DAMPED TWO-DEGREE-OF-FREEDOM SYSTEM

Objective

To verify the COMPLEX-HARMONIC method with SPRING elements.

Physical Problem

A sinusoidal force is applied to one mass of the damped two-degree-of-freedom system shown in the figure below. This case has been analyzed without damping in Example A77.



$F = F_o \sin \omega t$ $\quad k_1 = 1.0 \cdot 10^4$ N/m

$F_o = 1$ N $\quad k_2 = 1.5 \cdot 10^4$ N/m

$m_1 = 1$ kg $\quad c_1 = 70.0$ Ns/m

$m_2 = 1$ kg $\quad c_2 = 2.0$ Ns/m

Finite Element Model

The model consists of two SPRING elements (DIRECTION = AXIALTRANSLATION) and two concentrated masses. The model is shown on page A86.2.

Solution Results

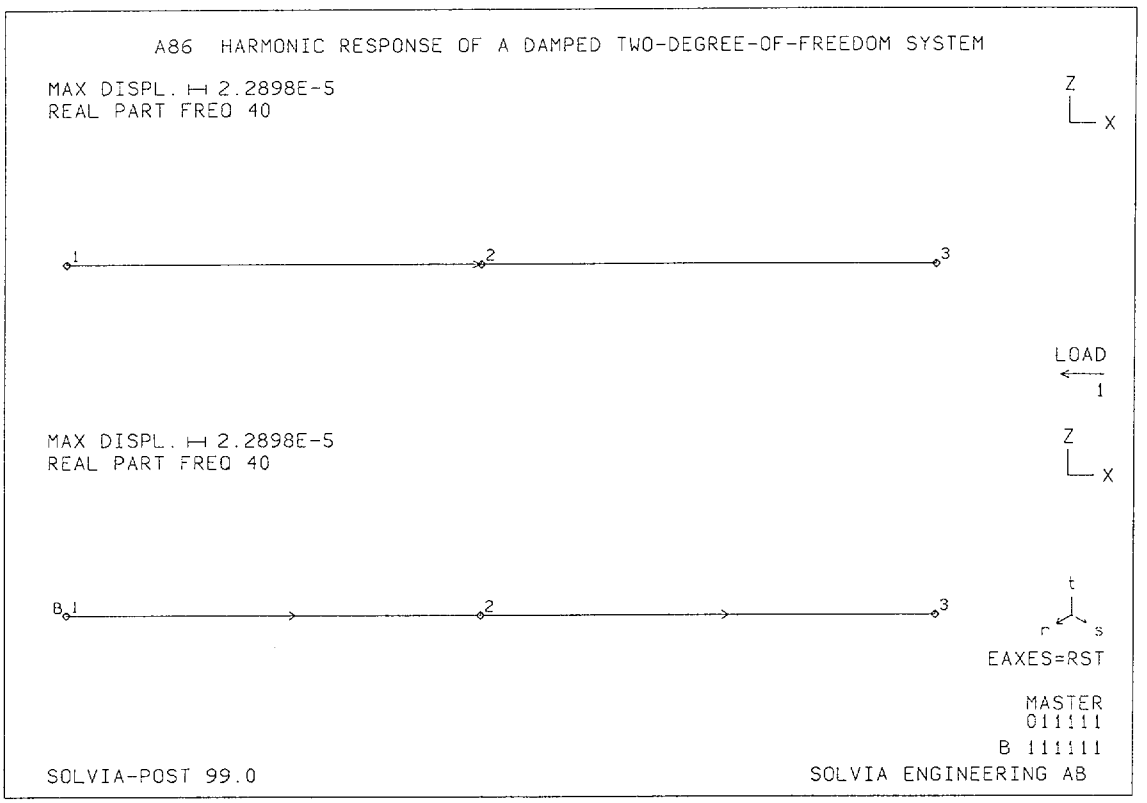
The input data shown on pages A86.5 and A86.6 is used. The displacement amplitude of node 3 has a peak value at frequency 9.4 Hz, see page A86.2. The natural frequencies of an undamped system are 10.3 Hz and 30.1 Hz. For this heavy damped system the second natural frequency has a very slight influence.

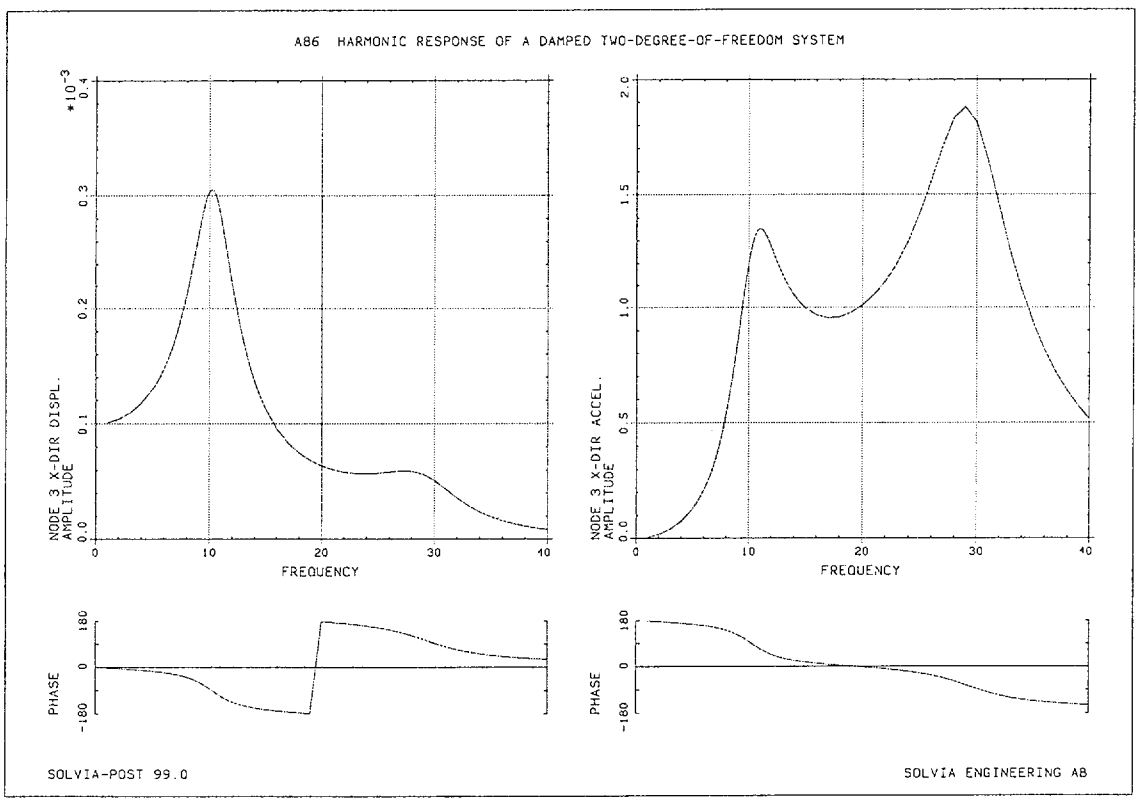
The input power and element forces as a function of frequency are shown on page A86.4.

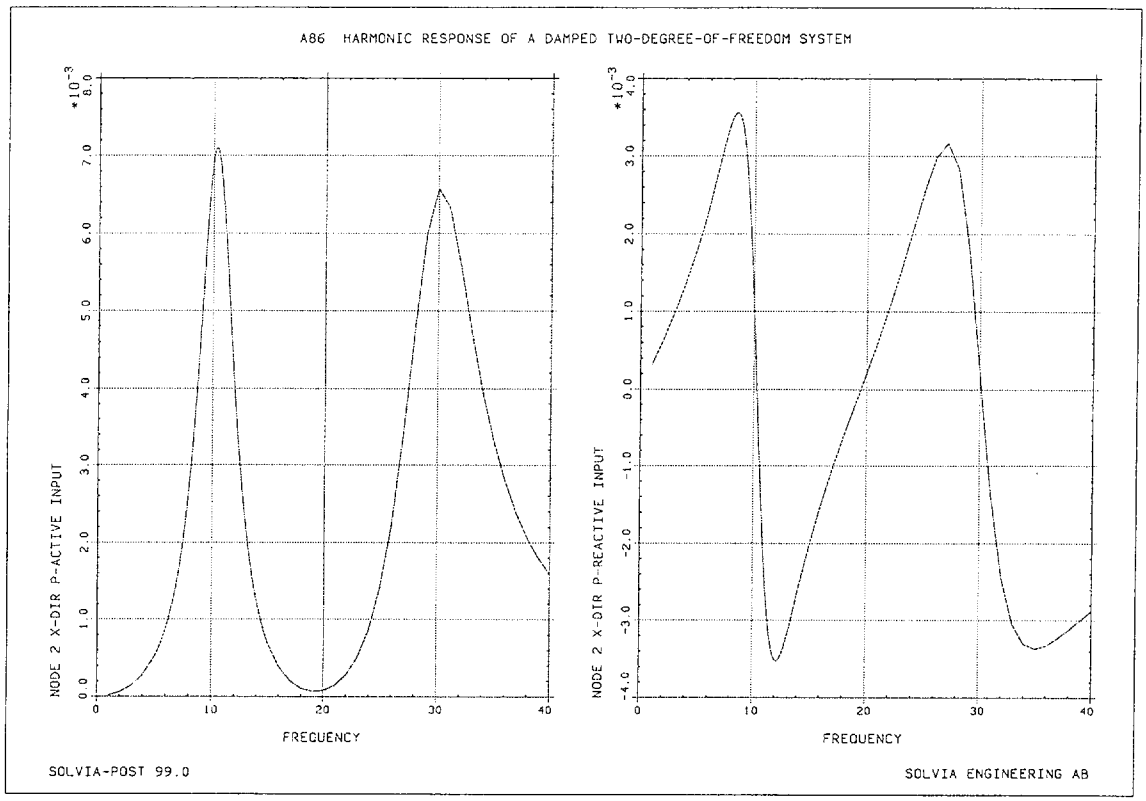
A comparison with the analytical solution for characteristic responses at the frequency 10 Hz is made in the table below.

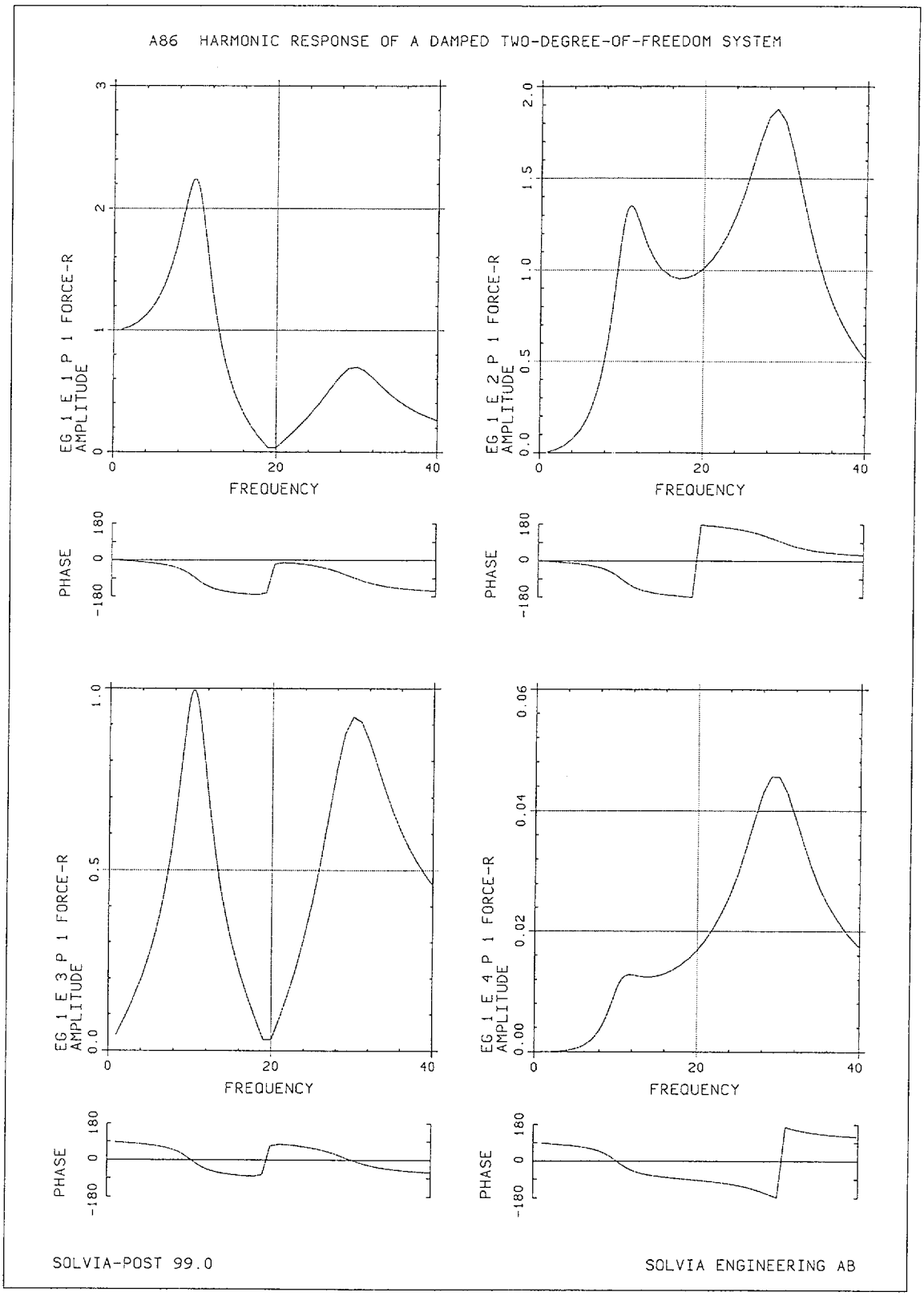
		Analytical	SOLVIA
Displacement, amplitude, node 3	[m]	$3.04 \cdot 10^{-4}$	$3.04 \cdot 10^{-4}$
Velocity, amplitude, node 3	[m/s]	$1.91 \cdot 10^{-2}$	$1.91 \cdot 10^{-2}$
Acceleration, amplitude, node 3	[m/s²]	$1.20 \cdot 10^{-1}$	$1.20 \cdot 10^{-1}$
Element force, amplitude, el. no. 4	[N]	$0.99 \cdot 10^{-2}$	$1.00 \cdot 10^{-2}$
Active powerflow, element no. 4 **)	[Nm/s]	$-8.32 \cdot 10^{-5}$	$-8.32 \cdot 10^{-5}$ *)
Input active powerflow, node 2 **)	[Nm/s]	$6.95 \cdot 10^{-3}$	$6.95 \cdot 10^{-3}$
Strain energy (whole model) **)	[Nm]	$1.49 \cdot 10^{-4}$	$1.49 \cdot 10^{-4}$

*) Power flow in opposite element axial direction. **) Time average values.


A86 HARMONIC RESPONSE OF A DAMPED TWO-DEGREE-OF-FREEDOM SYSTEM
MAX DISPL. = 2.2898E-5
REAL PART FREQ 40
Z
X
1
2
3
LOAD
1
MAX DISPL. = 2.2898E-5
REAL PART FREQ 40
Z
X
B
1
2
3
t
r
s
EAXES=RST
MASTER
011111
B 111111
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A86 HARMONIC RESPONSE OF A DAMPED TWO-DEGREE-OF-FREEDOM SYSTEM
*10^-3
0.4
0.3
0.2
0.1
0.0
NODE 3 X-DIR DISPL. AMPLITUDE
0
10
20
30
40
FREQUENCY
2.0
1.5
1.0
0.5
0.0
NODE 3 X-DIR ACCEL. AMPLITUDE
0
10
20
30
40
FREQUENCY
180
0
-180
PHASE
180
0
-180
PHASE
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A86 HARMONIC RESPONSE OF A DAMPED TWO-DEGREE-OF-FREEDOM SYSTEM
*10^-3
NODE 2 X-DIR P-ACTIVE INPUT
FREQUENCY
*10^-3
NODE 2 X-DIR P-REACTIVE INPUT
FREQUENCY
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A86 HARMONIC RESPONSE OF A DAMPED TWO-DEGREE-OF-FREEDOM SYSTEM
EG 1 E 1 P 1 FORCE-R AMPLITUDE
EG 1 E 2 P 1 FORCE-R AMPLITUDE
EG 1 E 3 P 1 FORCE-R AMPLITUDE
EG 1 E 4 P 1 FORCE-R AMPLITUDE
FREQUENCY
PHASE
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
DATABASE CREATE
*
HEADING 'A86  HARMONIC RESPONSE OF A DAMPED TWO-DEGREE-OF-FREEDOM
SYSTEM'
MASTER IDOF=011111 NSTEP=1
COMPLEX-HARMONIC INTERVAL=LIN
 1  4
 5  50
 15 25 40
ANALYSIS MASSMATRIX=CONSISTENT
*
COORDINATES
ENTRIES NODE X
 1 1 TO 3 3
MASS
2 1 1 1 0 0 0
3 1 1 1 0 0 0
*
EGROUP 1 SPRING DIRECTION=AXIALTRANSLATION
PROPERTYSET 1 K=1.0E4
PROPERTYSET 2 K=1.5E4
PROPERTYSET 3 C=7.0E1
PROPERTYSET 4 C=2.0E0
ENODES
1 1 2 / 2 2 3 / 3 1 2 / 4 2 3
EDATA
1 1 / 2 2 / 3 3 / 4 4
*
FIXBOUNDARIES / 1
LOAD CONC
2 1 1.
TIMEFUNCTION 1
0 1 / 1E3 1
*
SOLVIA MODELSUM=DET
END
```

SOLVIA-POST input

```
*  A86  HARMONIC RESPONSE OF A DAMPED TWO-DEGREE-OF-FREEDOM SYSTEM
*
DATABASE CREATE
WRITE FILENAME='a86.lis'
SET DIAGRAM=GRID
SET RESPONSE=REAL
*
MESH NNUMBERS=YES NSYMBOL=YES VECTOR=LOAD VIEW=-Y SUBFRAME=12
MESH NNUMBERS=YES NSYMBOL=YES BCODE=ALL EAXES=RST
*
SET HEIGHT=0.20
SUBFRAME 21
NCOMPLEX-HARMONIC NODE=3 DIRECTION=1 KIND=DISPLACEMENT OUTPUT=ALL
NCOMPLEX-HARMONIC NODE=3 DIRECTION=1 KIND=ACCELERATION OUTPUT=ALL
SUBFRAME 21
NCOMPLEX-HARMONIC NODE=2 DIRECTION=1 KIND=P-ACTIVE OUTPUT=ALL
NCOMPLEX-HARMONIC NODE=2 DIRECTION=1 KIND=P-REACTIVE OUTPUT=PLOT
NCOMPLEX-HARMONIC NODE=3 DIRECTION=1 KIND=VELOCITY OUTPUT=LIST,
                  FSTART=9 FEND=10
*
SET PLOTORIENTATION=PORTRAIT
ECOMPLEX ELEMENT=1 POINT=1 KIND=FR SUBFRAME=22
ECOMPLEX ELEMENT=2 POINT=1 KIND=FR
ECOMPLEX ELEMENT=3 POINT=1 KIND=FR
ECOMPLEX ELEMENT=4 POINT=1 KIND=FR
ELIST SELECT=POWERFLOW TSTART=10
*
SUMMATION KIND=ENERGY TSTART=10
*
CASECOMBINATION 1 STANDARD FUNCTION=SRSS
 10. 1. REAL
 10. 1. IMAGINARY
SET RESPONSETYPE=CASECOMBINATION
ELIST
NLIST
NLIST KIND=REAC
*
END
```

EXAMPLE A87

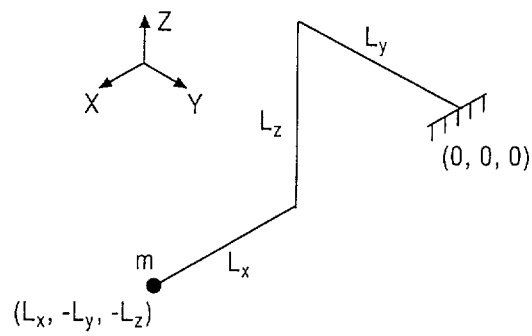
MODAL COMBINATION METHODS IN RESPONSE SPECTRUM ANALYSIS

Objective

To verify the methods of combining modal responses in the response spectrum method.

Physical Problem

A beam structure shown in the figure below is subjected to an earthquake base acceleration in the X-direction. The 3-D structure is asymmetrical. This example has been analyzed in reference [1], ex. E26-8.



$m = 1 \cdot 10^{-2}$ kg

$L_x = L_y = L_z = 1$ m

$E = 1$ N/m^2

$\nu = 0$

Bending moment of inertia

$I_s = I_t = 1$ m^4

Torsional moment of inertia

$I_r = 4/3$ m^4

The numerical values are only selected so that the relation between stiffness and mass is

$\frac{EI}{mL^3} = 100$ 1/sec^2 which is in accordance with reference [1].

The acceleration response spectrum values for the three natural frequencies are specified as

$$S_a = \begin{bmatrix} 2.2700 \\ 2.4500 \\ 6.9800 \end{bmatrix} \text{m/sec}^2 \qquad \text{with 5\% of critical damping}$$

Solution Results

Using the input data on pages A87.4 and A87.5 the following natural frequencies were obtained:

$$f = \begin{bmatrix} 0.73068 \\ 0.76897 \\ 2.3172 \end{bmatrix} \text{Hz} \qquad \text{or} \qquad \omega = \begin{bmatrix} 4.5910 \\ 4.8316 \\ 14.559 \end{bmatrix} \text{rad/sec}$$

The two lowest frequencies are closely spaced with significant cross-correlation factors in the CQC method. The eigenmodes are shown on page A87.3 and the modal displacements in the X-, Y- and Z-directions at node 4 are:

$$\varphi_1 = \begin{bmatrix} 5.79770 \\ -1.84954 \\ 7.93510 \end{bmatrix} \qquad \varphi_2 = \begin{bmatrix} 2.60511 \\ 9.64852 \\ 0.345505 \end{bmatrix} \qquad \varphi_3 = \begin{bmatrix} 7.72011 \\ -1.86687 \\ -6.07575 \end{bmatrix}$$

The calculated modal participation factors for base excitation in the X-direction are:

$$\Gamma_1 = 57.9770 \cdot 10^{-3} \qquad \Gamma_2 = 26.0511 \cdot 10^{-3} \qquad \Gamma_3 = 77.2011 \cdot 10^{-3}$$

The maximum relative displacement for each mode i is $u_{max}^{(i)} = \Gamma_i S_{a_i} \varphi_i / \omega_i^2$ giving

$$u_{max}^{(1)} = \begin{bmatrix} 36.202 \\ -11.549 \\ 49.548 \end{bmatrix} \text{mm} \qquad u_{max}^{(2)} = \begin{bmatrix} 7.123 \\ 26.380 \\ 0.945 \end{bmatrix} \text{mm} \qquad u_{max}^{(3)} = \begin{bmatrix} 19.625 \\ -4.746 \\ -15.445 \end{bmatrix} \text{mm}$$

The relative displacement response at node 4 shown in the table below was calculated by SOLVIA-POST and the values agree with the results of a separate analysis using the formulas given in [2], command MODALCOMBINATION.

Modal Comb. Method	Rel. displ. at node 4 in mm		
	X	Y	Z
SRSS	41.79	29.19	51.91
CQC	46.53	19.18	52.53
Ten Percent	47.56	38.22	52.80
Double Sum (using 100 sec duration)	46.75	36.80	52.79
Algebraic	62.95	10.09	35.05
Absolute Sum	62.95	42.67	65.94

The cross-correlation factors in the CQC method are $\rho_{12} = \rho_{21} = 0.79280$, $\rho_{23} = \rho_{32} = 0.006383$, $\rho_{13} = \rho_{31} = 0.005705$.

The response in the X-direction is under-estimated using the SRSS method compared to the CQC, Ten Percent and Double Sum methods.

The response in the Y-direction is calculated to be much lower using the CQC method compared to the SRSS, Ten Percent and Double Sum methods.

The response in the Z-direction is about the same for the SRSS, CQC, Ten Percent and Double Sum methods.

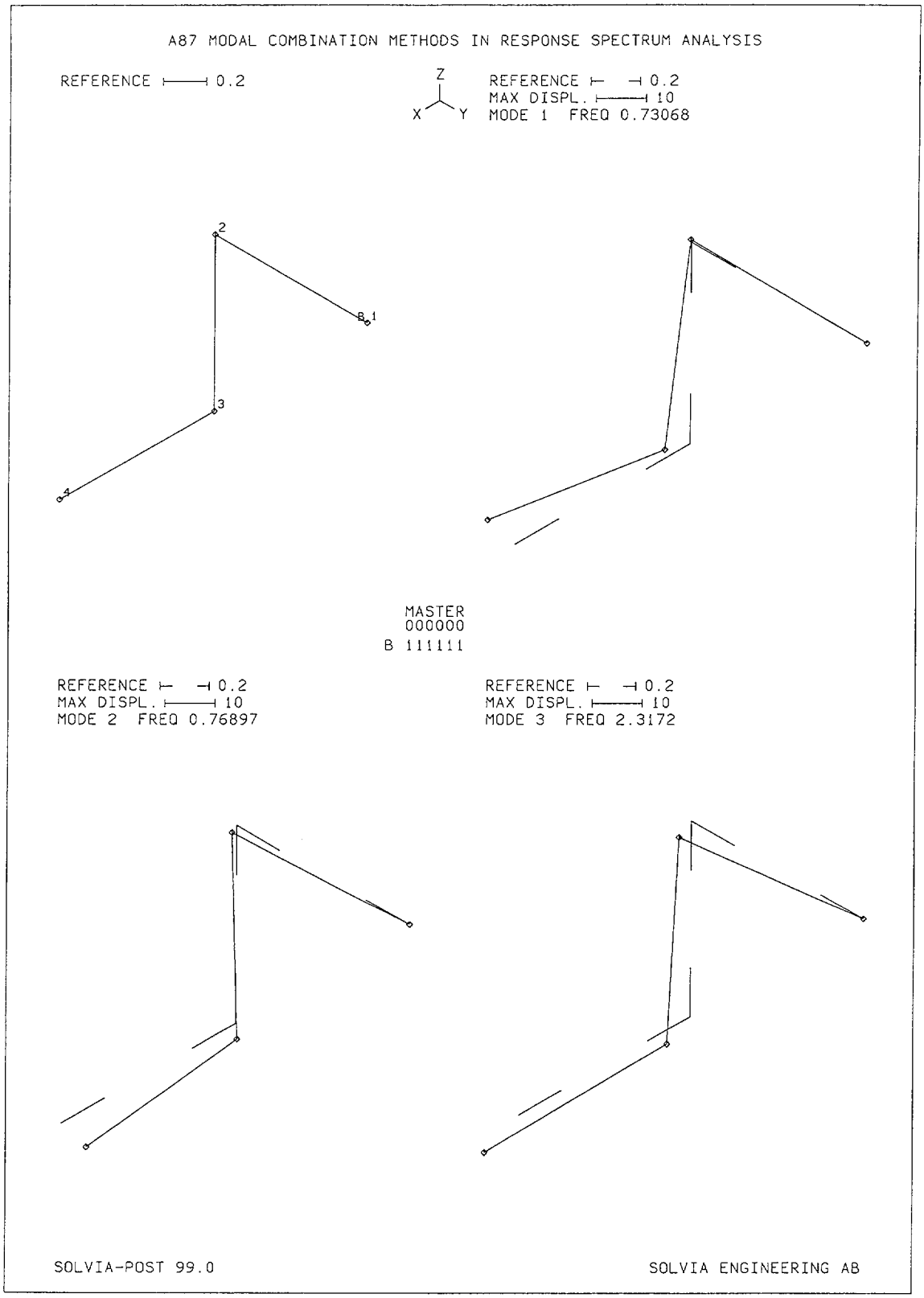
The results of the Absolute Sum method can be considered as an upper bound of the response.

A further discussion on the SRSS and CQC methods can be found in [1].

References

[1] Clough, R. W., and Penzien, J., Dynamics of Structures, Second Edition McGraw-Hill, Inc., 1993.

[2] SOLVIA-POST 99.0 Users Manual, SOLVIA Engineering AB, Report SE 99-2, 1999.


A87 MODAL COMBINATION METHODS IN RESPONSE SPECTRUM ANALYSIS
REFERENCE 0.2
Z
X
Y
REFERENCE 0.2
MAX DISPL. 10
MODE 1 FREQ 0.73068
2
1
3
4
MASTER
000000
B 111111
REFERENCE 0.2
MAX DISPL. 10
MODE 2 FREQ 0.76897
REFERENCE 0.2
MAX DISPL. 10
MODE 3 FREQ 2.3172
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING 'A87 MODAL COMBINATION METHODS IN RESPONSE SPECTRUM
ANALYSIS'
DATABASE CREATE
*
FREQUENCIES SUBSPACE-ITERATION NEIG=3
ANALYSIS DYNAMIC IMODS=2 NMODS=3
*
COORDINATES
 1  0  0  0
 2  0 -1  0
 3  0 -1 -1
 4  1 -1 -1
*
MASS
 4 0.01 0.01 0.01 0 0 0
*
MATERIAL 1 ELASTIC E=1. NU=0.
*
EGROUP 1 BEAM
SECTION  1 GENERAL RINERTIA=1.333333333 SINERTIA=1 ,
           TINERTIA=1  AREA=1E7
ENODES
 1 3 1 2
 2 1 2 3
 3 2 3 4
*
FIXBOUNDARIES / 1
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A87  MODAL COMBINATION METHODS IN RESPONSE SPECTRUM ANALYSIS
*
DATABASE CREATE
*
WRITE FILENAME='a87.lis'
FREQUENCIES
SET RESPONSETYPE=VIBRATION
NLIST N4 DIR=123 TSTART=1 TEND=3
*
SET PLOTORIENTATION=PORTRAIT HEIGHT=0.20
SET NSYMBOL=YES DMAX=1.
MESH ORIGINAL=YES DEFORMED=NO BCODE=ALL NSYMBOL=YES NNUMBER=YES
SUBFRAME=22
SET ORIGINAL=DASHED
SET AXES=NO
MESH TIME=1
MESH TIME=2
MESH TIME=3
*
RSPECTRUM 1 FACTOR=1
            0.      5.      10.
0.7        2.27    2.27    2.27
0.75       2.27    2.27    2.27
0.76       2.45    2.45    2.45
1.0        2.45    2.45    2.45
2.0        6.98    6.98    6.98
3.0        6.98    6.98    6.98
*
DAMPING 1
 1 5. TO 3 5.
*
MODAL-COMBINATION 1 RSPECTRUM=1 DIRECTION=X METHOD=SRSS
MODAL-COMBINATION 2 RSPECTRUM=1 DIRECTION=X METHOD=CQC
MODAL-COMBINATION 3 RSPECTRUM=1 DIRECTION=X METHOD=ABS
MODAL-COMBINATION 4 RSPECTRUM=1 DIRECTION=X METHOD=TENPERCENT
MODAL-COMBINATION 5 RSPECTRUM=1 DIRECTION=X METHOD=DOUBLESUM,
                    DURATION=100.
MODAL-COMBINATION 6 RSPECTRUM=1 DIRECTION=X METHOD=ALGEBRAIC
*
SET RESPONSETYPE=MODAL-COMBINATION
NLIST N4 DIR=123 TSTART=1 TEND=6
END
```

EXAMPLE A88

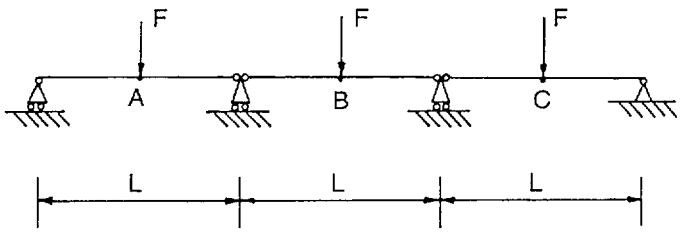
SIMPLY SUPPORTED BEAM CONDITIONS USING THE END RELEASE OPTION

Objective

To verify the BEAM element when using the end release option.

Physical Problem

Three beam spans as shown in the figure below are under central loads. The beam spans are simply supported without bending stiffness connection to the neighbouring span. The vertical displacements under the loads are to be calculated.



$L = 1$ m

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

$\nu = 0.3$

$F = 1$ N

Rectangular cross-section
10×10 mm

Finite Element Model

The model is shown on page A88.2 and consists of six BEAM elements where three of the beam elements use the end release option.

The bending stiffness about the local s-axis (global x-axis) is released at node 3 for both connecting elements and at node 5 for the left connecting element.

The selected end release options exemplify two possibilities to achieve simply supported end conditions for the beams connecting to nodes 3 and 5. Since the end release option is used twice at node 3 it is then necessary to also fix the x-rotation at node 3 since it is left without stiffness and would otherwise cause a zero diagonal element in the stiffness matrix.

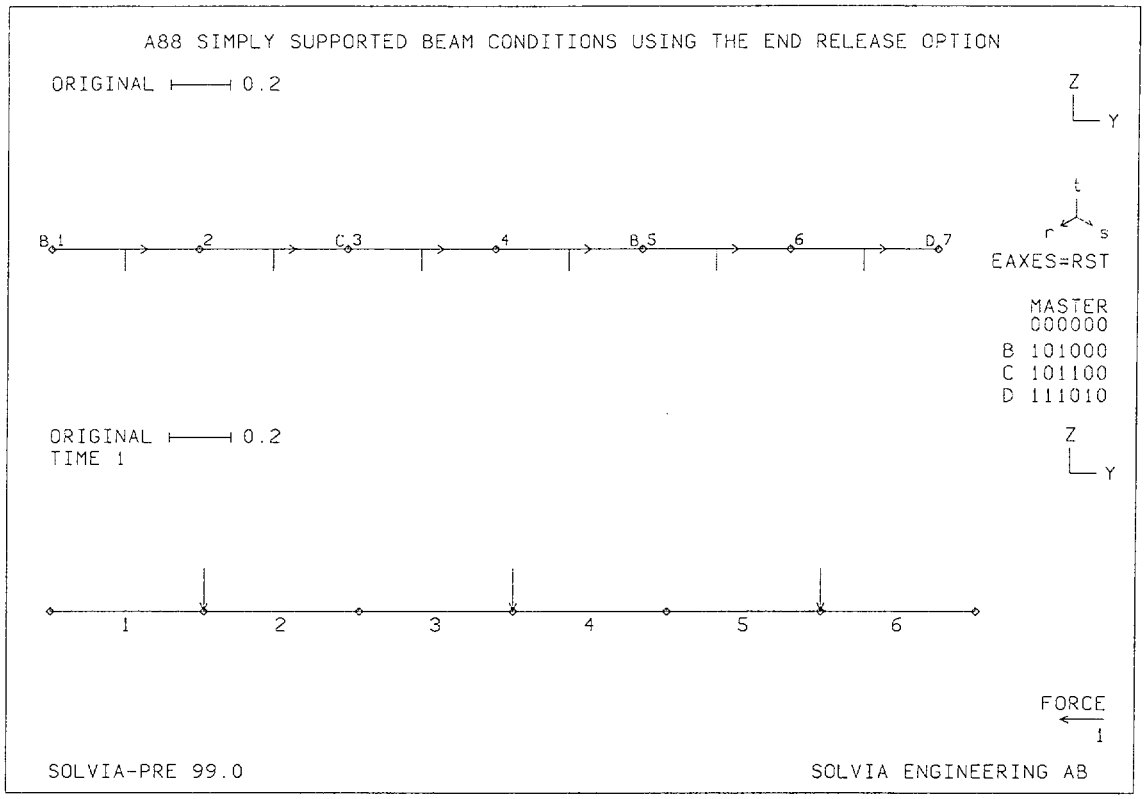
Solution Results

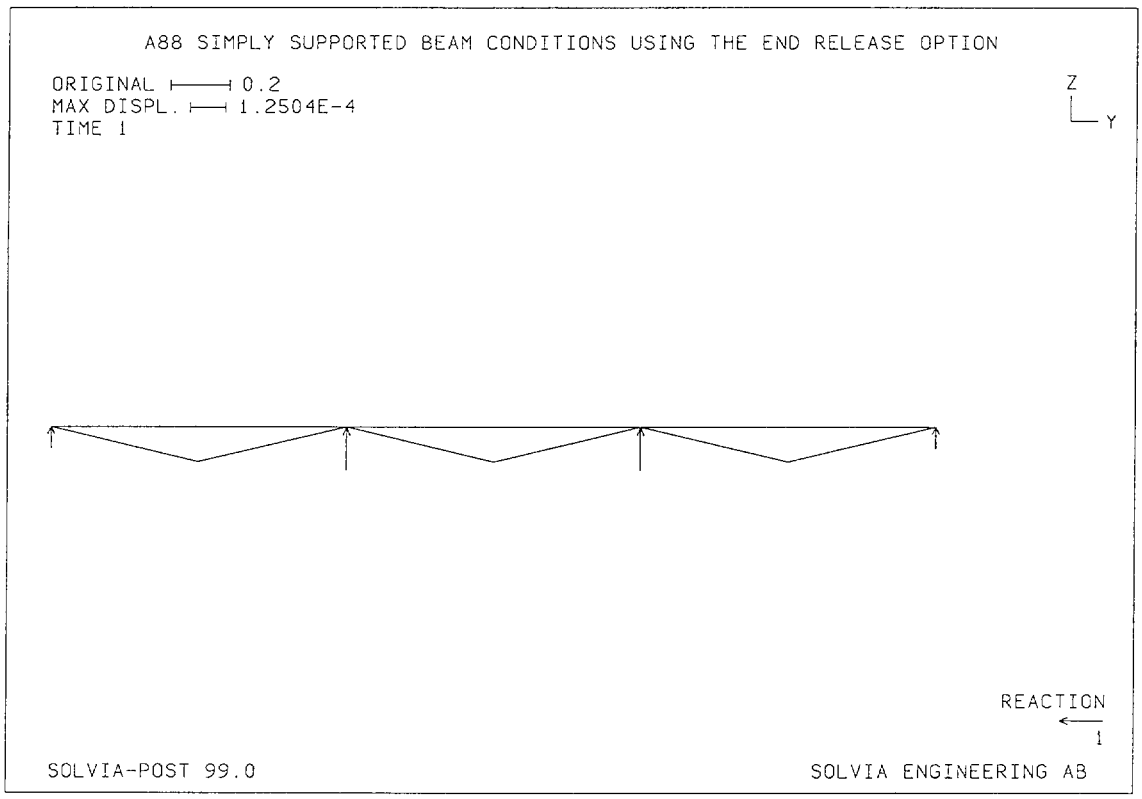
The theoretical displacement at mid-span for a simply supported beam is

$$u_z = \frac{-FL^3}{48\,EI}$$

The input data shown on page A88.3 gives the following vertical displacements in mm at the mid-span locations:

Theory	Point A	Point B	Point C
-0.1250	-0.1250	-0.1250	-0.1250


A88 SIMPLY SUPPORTED BEAM CONDITIONS USING THE END RELEASE OPTION
ORIGINAL 0.2
Z
Y
t
r
s
B 1
2
C 3
4
B 5
6
D 7
EAXES=RST
MASTER
000000
B 101000
C 101100
D 111010
ORIGINAL 0.2
TIME 1
Z
Y
1
2
3
4
5
6
FORCE
1
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A88 SIMPLY SUPPORTED BEAM CONDITIONS USING THE END RELEASE OPTION
ORIGINAL 0.2
MAX DISPL. 1.2504E-4
TIME 1
Z
Y
REACTION
1
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
DATABASE CREATE
HEADING 'A88 SIMPLY SUPPORTED BEAM CONDITIONS USING THE END RELEASE
OPTION'
*
COORDINATES
ENTRIES NODE Y
 1 0. TO 7 3.
*
MATERIAL 1 ELASTIC E=2.0E11 NU=0.3
*
EGROUP 1 BEAM
SECTION 1 RECTANGULAR WTOP=0.01 D=0.01
BEAMVECTORS
1 1. 0. 0.
ENODES
 1 -1 1 2 TO 6 -1 6 7
ENDRELEASE 1 REL1=11
ENDRELEASE 2 REL1=5
EDATA
ENTRIES EL SECTION ENDRELEASE
 1 1 0
 2 1 1
 3 1 2
 4 1 1
 5 1 0
 6 1 0
*
FIXBOUNDARIES 13 / 1 3 5 7
FIXBOUNDARIES 52 / 7
FIXBOUNDARIES  4 / 3
*
LOAD CONCENTRATED
 2 3 -1.
 4 3 -1.
 6 3 -1.
*
SET NSYMBOLS=YES VIEW=X
MESH BCODE=ALL NNUMBERS=YES EAXES=RST  SUBFRAME=12
MESH ENUMBERS=YES VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A88 SIMPLY SUPPORTED BEAM CONDITIONS USING THE END RELEASE OPTION
*
DATABASE CREATE
*
WRITE 'a88.lis'
NLIST
MESH ORIGINAL=YES VECTOR=REACTION VIEW=X
END
```

EXAMPLE A89

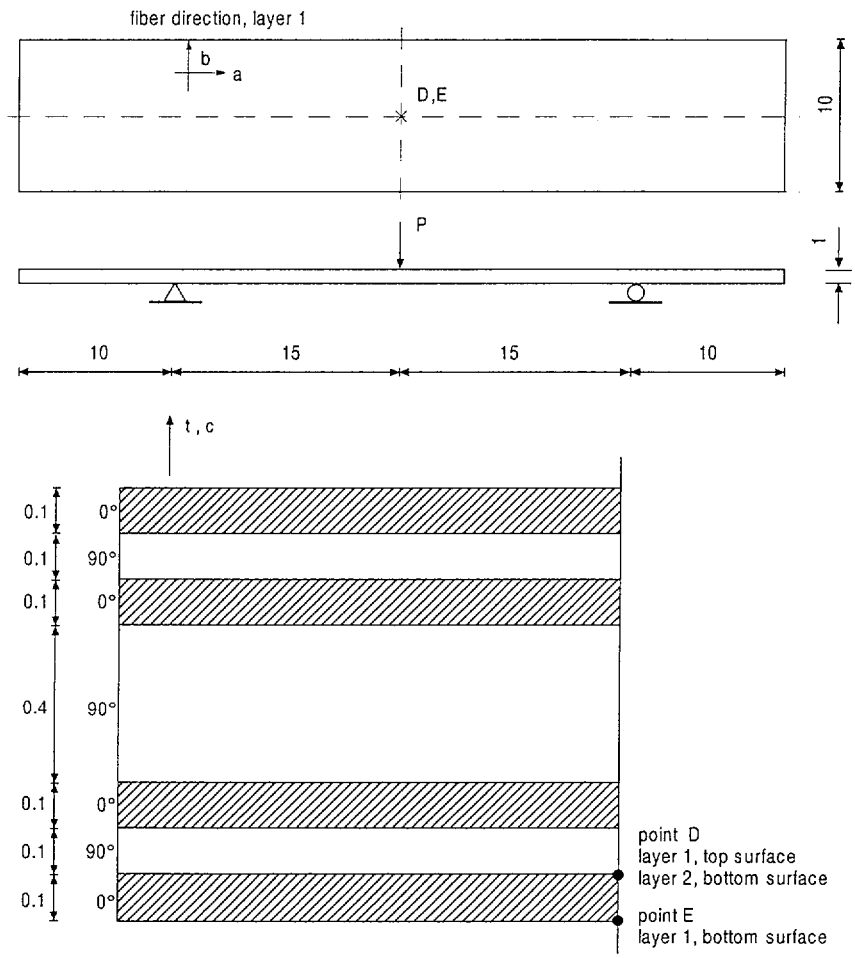
LAMINATED STRIP

Objective

To verify the laminated 16-node SHELL element.

Physical Problem

A simply supported laminated strip is subjected to a central line load. The laminate consists of seven layers as shown in the figure below. This problem is described in [1].



All dimensions in mm.

The line load, P = 10 N/mm at the center of the strip (X=25 mm)

Material properties:

$E_a = 1 \cdot 10^5$ MPa $\quad \nu_{ba} = 0.4 \quad G_{ab} = 3 \cdot 10^3$ MPa

$E_b = 5 \cdot 10^3$ MPa $\quad G_{ac} = 2 \cdot 10^3$ MPa

$G_{bc} = 2 \cdot 10^3$ MPa

Finite Element Model

One quarter of the strip is modeled using five 16-node SHELL elements. The finite element model with the symmetrical boundary condition is shown on page A89.2.

Solution Results

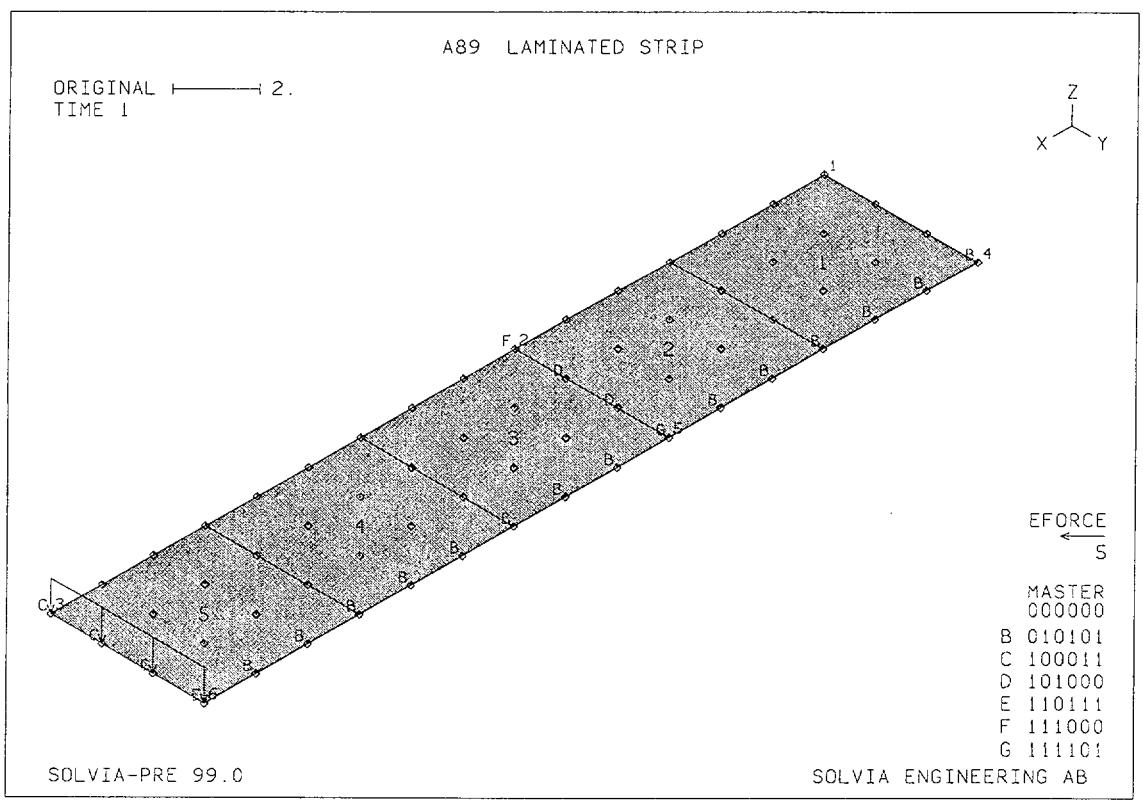
The input data shown on pages A89.3 and A89.4 is used in the SOLVIA analysis. Contour plots of σ_{xx} (bottom of layer 1) and σ_{xz} (top of layer 1) are shown on page A89.3.

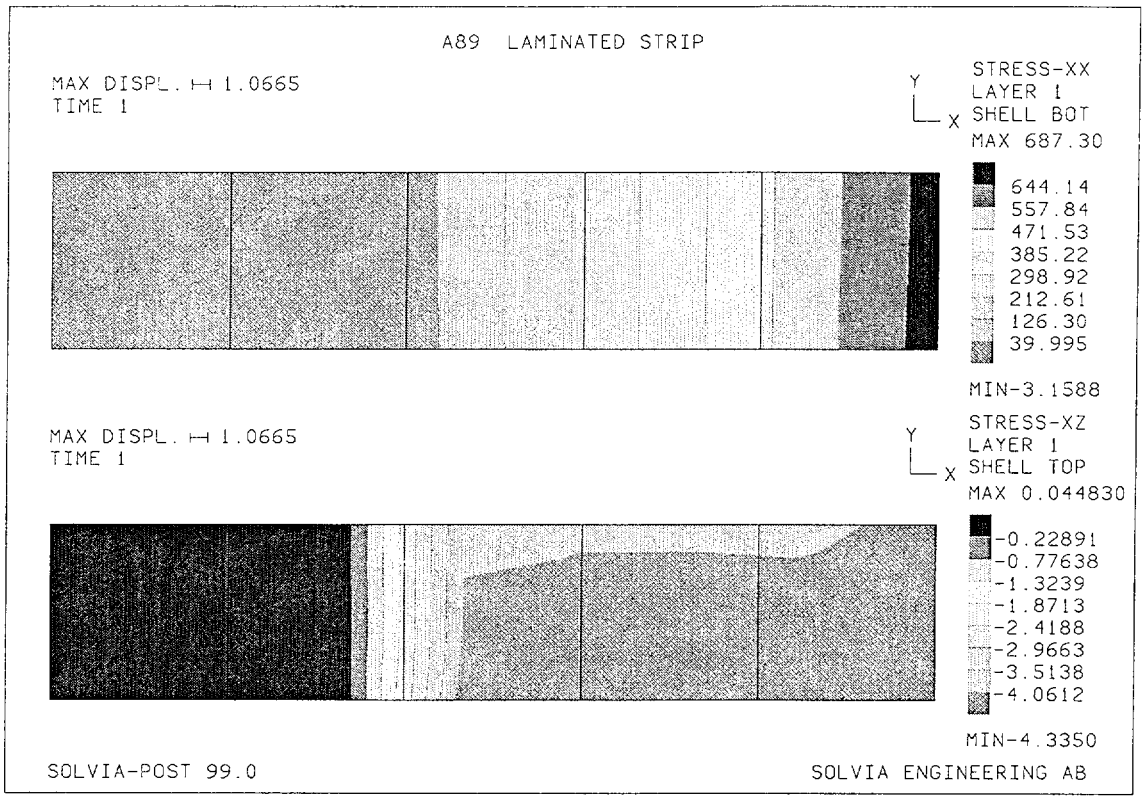
Results compared with NAFEMS target values.

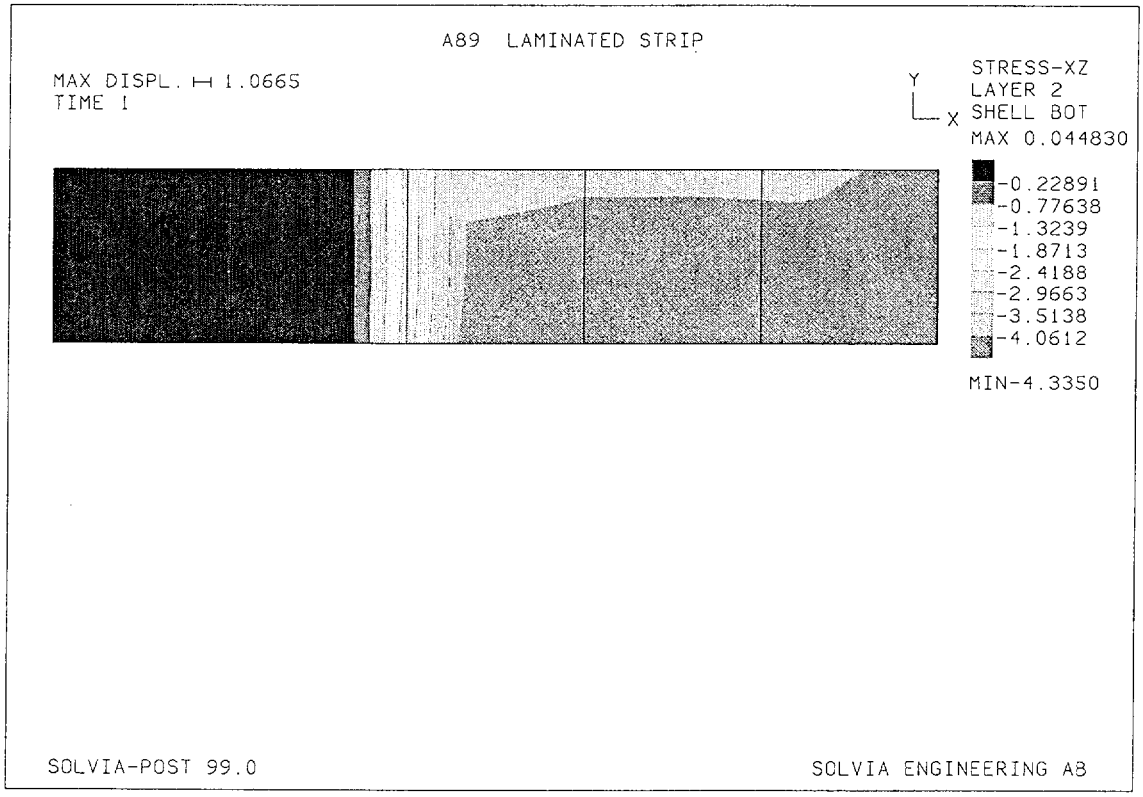
	SOLVIA	NAFEMS [1]
Central displacement [mm] (node 6, Z-direction)	-1.06	-1.06
Bending stress σ_{xx} point E [MPa] (element 5, layer 1, bottom, node 6)	682.6	683.9
Interlaminar shear stress σ_{xz} at point D [MPa] (element 5, layer 1, top, node 6)	-4.1	-4.1
(element 5, layer 2, bottom, node 6)	-4.1	-4.1

Reference

[1] Hardy, S.J, "Composite Benchmarks", NAFEMS Report R0031, 1994.




A89 LAMINATED STRIP
MAX DISPL. 1.0665
TIME 1
Y
X
STRESS-XX
LAYER 1
SHELL BOT
MAX 687.30
644.14
557.84
471.53
385.22
298.92
212.61
126.30
39.995
MIN-3.1588
MAX DISPL. 1.0665
TIME 1
Y
X
STRESS-XZ
LAYER 1
SHELL TOP
MAX 0.044830
-0.22891
-0.77638
-1.3239
-1.8713
-2.4188
-2.9663
-3.5138
-4.0612
MIN-4.3350
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A89 LAMINATED STRIP
MAX DISPL. 1.0665
TIME 1
Y
X
STRESS-XZ
LAYER 2
SHELL BOT
MAX 0.044830
-0.22891
-0.77638
-1.3239
-1.8713
-2.4188
-2.9663
-3.5138
-4.0612
MIN-4.3350
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
DATABASE CREATE
HEADING 'A89  LAMINATED STRIP'
*
COORDINATES
 ENTRIES  NODE   X   Y
           1     0   0
           2    10   0
           3    25   0
           4     0   5
           5    10   5
           6    25   5
*
MATERIAL 1 ORTHOTROPIC EA=1E5 EB=5E3 NUBA=0.4,
           GAB=3E3 GAC=2E3 GBC=2E3
*
MATERIAL 2 LAMINATE
 1 1 10  0
 2 1 10 90
 3 1 10  0
 4 1 40 90
 5 1 10  0
 6 1 10 90
 7 1 10  0
*
EGROUP 1 SHELL MATERIAL=2 RESULT=NSTRESS
SET NODES=16
GSURFACE  1 2 5 4 EL1=2
GSURFACE  2 3 6 5 EL1=3
THICKNESS 1 1.
LOAD ELEMENT TYPE=FORCE INPUT=LINES
 3 6 THICKNESS -5 -5
*
FIXBOUNDARIES  13 INPUT=LINES / 2 5
FIXBOUNDARIES  2 / 2
FIXBOUNDARIES 156 INPUT=LINES / 3 6
FIXBOUNDARIES 246 INPUT=LINES / 6 4
*
MESH NNUMBER=MYNODES NSYMBOL=YES VECTOR=LOAD,
     ENUMBER=YES BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A89 LAMINATED STRIP
*
DATABASE CREATE
*
WRITE FILENAME='a89.lis'
NLIST N6 DIR=3
SHELLSURFACE LAYER=1 PLOT=BOTTOM
ELIST EL5
MESH CONTOUR=SXX VIEW=Z SUBFRAME=12
SHELLSURFACE PLOT=TOP
MESH CONTOUR=SXZ
SHELLSURFACE LAYER=2 PLOT=BOTTOM
MESH CONTOUR=SXZ  SUBFRAME=12
ELIST EL5
END
```

EXAMPLE A90

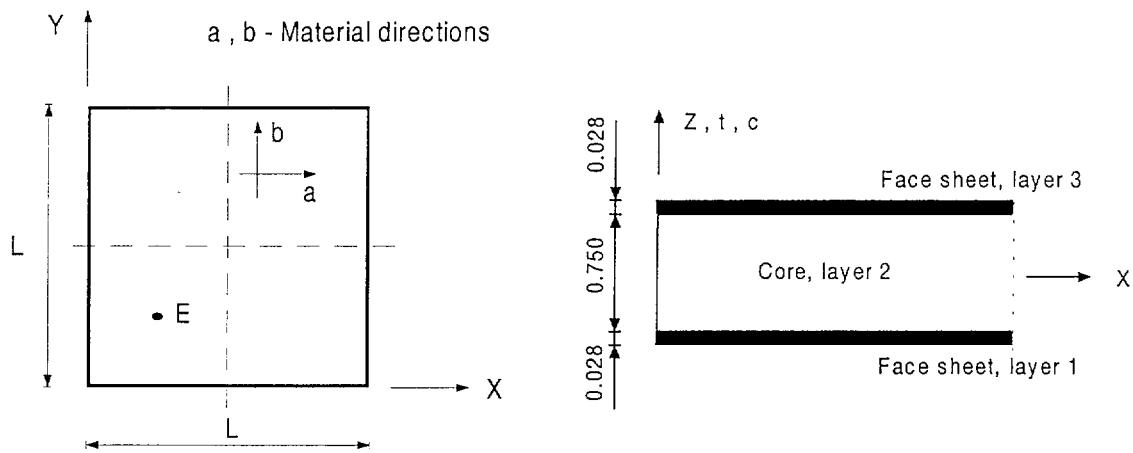
SANDWICH SHELL

Objective

To verify the laminated 9-node SHELL element in a sandwich plate case.

Physical Problem

A simply-supported square sandwich plate is subjected to a uniform normal pressure. The laminate consists of three layers with a thick central core as shown in the figure below. This problem is described in [1].



L = 10 inch Pressure 100 psi All dimensions in inches

Material properties

Face sheets:

$E_a = 10 \cdot 10^6$ psi $\nu_{ba} = 0.3$ $G_{ab} = 1.875 \cdot 10^6$ psi

$E_b = 4 \cdot 10^6$ psi $G_{ac} = 1 \cdot 10^5$ psi

$G_{bc} = 1 \cdot 10^5$ psi

Core:

$E_a = E_b = 1$ psi (≈ 0)

$G_{ac} = 3.0 \cdot 10^4$ psi

$G_{bc} = 1.2 \cdot 10^4$ psi

Note that assumed values of G_{ac} and G_{bc} for the sheets are used.

Finite Element Model

Due to symmetrical conditions only one quarter of the plate is considered. The model with sixteen 9-node SHELL elements and boundary conditions are shown on page A90.2.

Solution Results

The input data on pages A90.4 and A90.5 is used in the SOLVIA analysis.

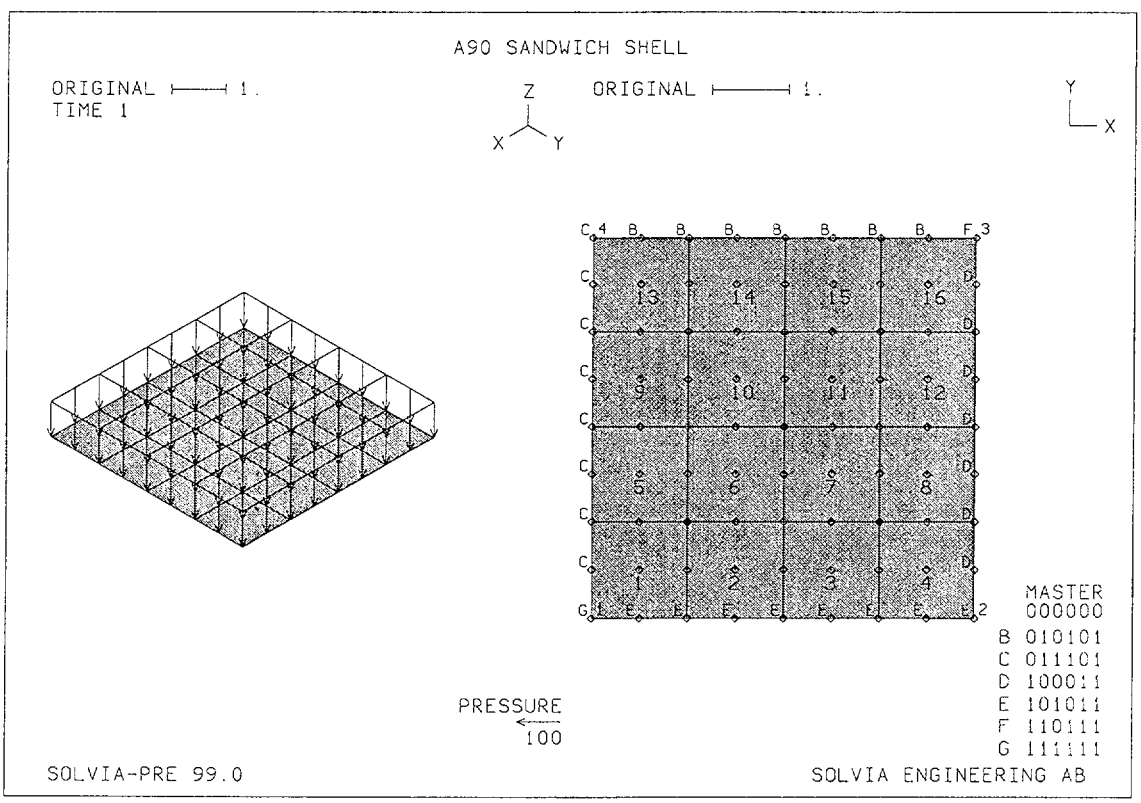
Contour plots of stresses in the principal material direction for layer number 1, midsurface location, are shown on page A90.3.

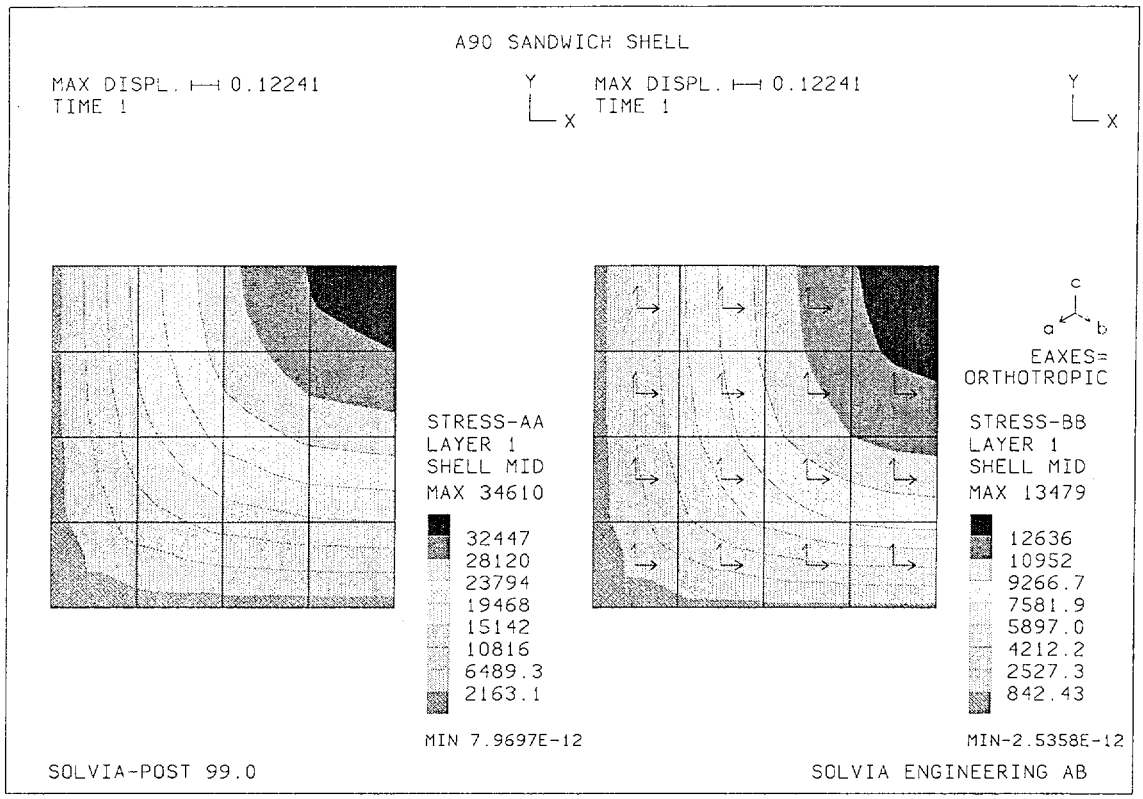
Results compared with NAFEMS' target values.

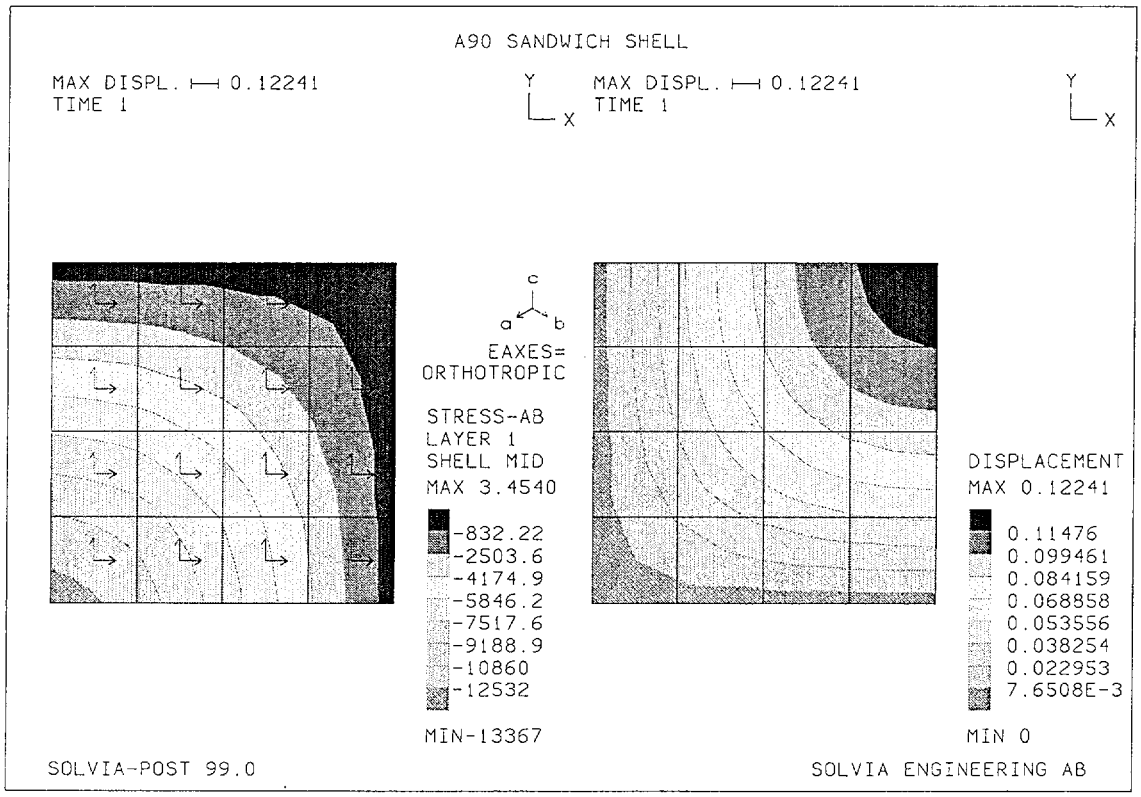
	SOLVIA	NAFEMS [1]
Central displacement [inch] (node 3, Z-direction)	-0.1224	-0.123
Stress in the material a-direction at central point [psi] (element 16, layer 1, midsurface, node 3)	34609	34449
Stress in the material b-direction at central point [psi] (element 16, layer 1, midsurface, node 3)	13479	13350
Quarter point, (E) in plane shear stress, σ_{ab} [psi]		-5067.5
(element 11, layer 1, midsurface, node 142)	-5106	
(element 10, layer 1, midsurface, node 142)	-5112	
(element 7, layer 1, midsurface, node 142)	-5117	
(element 6, layer 1, midsurface, node 142)	-5124	

Reference

[1] Hardy, S.J, "Composite Benchmarks", NAFEMS Report R0031, 1994.




A90 SANDWICH SHELL
MAX DISPL. 0.12241
TIME 1
Y
X
MAX DISPL. 0.12241
TIME 1
Y
X
STRESS-AA
LAYER 1
SHELL MID
MAX 34610
32447
28120
23794
19468
15142
10816
6489.3
2163.1
MIN 7.9697E-12
c
a
b
EAXES=
ORTHOTROPIC
STRESS-BB
LAYER 1
SHELL MID
MAX 13479
12636
10952
9266.7
7581.9
5897.0
4212.2
2527.3
842.43
MIN-2.5358E-12
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A90 SANDWICH SHELL
MAX DISPL. 0.12241
TIME 1
Y
X
MAX DISPL. 0.12241
TIME 1
Y
X
c
a
b
EAXES=
ORTHOTROPIC
STRESS-AB
LAYER 1
SHELL MID
MAX 3.4540
-832.22
-2503.6
-4174.9
-5846.2
-7517.6
-9188.9
-10860
-12532
MIN-13367
DISPLACEMENT
MAX 0.12241
0.11476
0.099461
0.084159
0.068858
0.053556
0.038254
0.022953
7.6508E-3
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
DATABASE CREATE
HEADING 'A90 SANDWICH SHELL'
*
COORDINATES
 ENTRIES  NODE   X   Y
           1     0   0
           2     5   0
           3     5   5
           4     0   5
*
MATERIAL 1 ORTHOTROPIC EA=10.E6 EB=4.E6 NUBA=0.3 ,
           GAB=1.875E6  GAC=1.E5 GBC=1.E5
MATERIAL 2 ORTHOTROPIC EA=1. EB=1. NUBA=0. ,
           GAB=1. GAC=3.0E4 GBC=1.2E4
*
MATERIAL 3 LAMINATE
 1 1  3.4739   0
 2 2 93.0522   0
 3 1  3.4739   0
*
EGROUP 1 SHELL MATERIAL=3 RESULT=NSTRESS STRESSREFERENCE=ORTHOTROPIC
SET NODES=9
GSURFACE  1 2 3 4 EL1=4 EL2=4
THICKNESS 1 0.806
LOAD ELEMENT TYPE=PRESSURE INPUT=SURFACE
 1 2 3 4 t 100. 100.
*
FIXBOUNDARIES  1356 INPUT=LINES / 1 2
FIXBOUNDARIES  2346 INPUT=LINES / 1 4
FIXBOUNDARIES  246  INPUT=LINES / 3 4
FIXBOUNDARIES  156  INPUT=LINES / 2 3
*
MESH VECTOR=LOAD SUBFRAME=21
MESH NNUMBER=MYNODES NSYMBOL=YES ENUMBER=YES BCODE=ALL VIEW=Z
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A90  SANDWICH SHELL
*
DATABASE CREATE
*
WRITE FILENAME='a90.lis'
*
NMAX DIR=3
SHELLSURFACE LAYER=1 PLOT=MID
ELIST EL16
EZONE E / 6 7 10 11
SHELLSURFACE  LIST=MID
ELIST E
*
MESH CONTOUR=SAA VIEW=Z SUBFRAME=21
MESH CONTOUR=SBB EAXES=ORTHO
MESH CONTOUR=SAB EAXES=ORTHO SUBFRAME=21
MESH CONTOUR=DISP
END
```

EXAMPLE A91

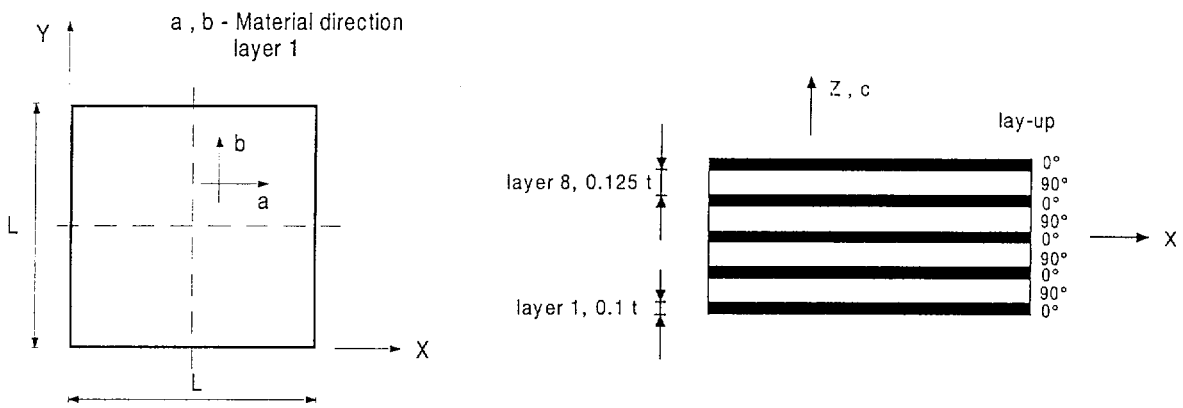
LAMINATED SQUARE PLATE UNDER NORMAL PRESSURE

Objective

To verify the laminated 4-node SHELL element.

Physical Problem

A simply supported square laminated plate is subjected to a uniform normal pressure. The laminate consists of 9 layers with lay-up 0°/90°/0°/90°/0°/90°/0°/90°/0° as shown in the figure below. This example is described in [1].



L = 10 inch
Thickness t = 0.1 inch
Pressure = 10 psi

Material properties

$E_a = 4.0 \cdot 10^7$ psi $\nu_{ba} = 0.25$

$E_b = 1.0 \cdot 10^6$ psi

$G_{ab} = 0.6 \cdot 10^6$ psi

$G_{ac} = 0.5 \cdot 10^6$ psi

$G_{bc} = 0.5 \cdot 10^6$ psi

Finite Element Model

Due to symmetry only one quarter of the plate is considered. The model with sixtyfour 4-node SHELL elements and the material directions for layers 1 and 2 are shown on pages A91.2 and A91.3.

Solution Results

The input data on pages A91.6 and A91.7 is used in the SOLVIA analysis.

Contour plots of the section moments for the laminate and principal stresses for layers 1 and 2 are shown on pages A91.3 and A91.4.

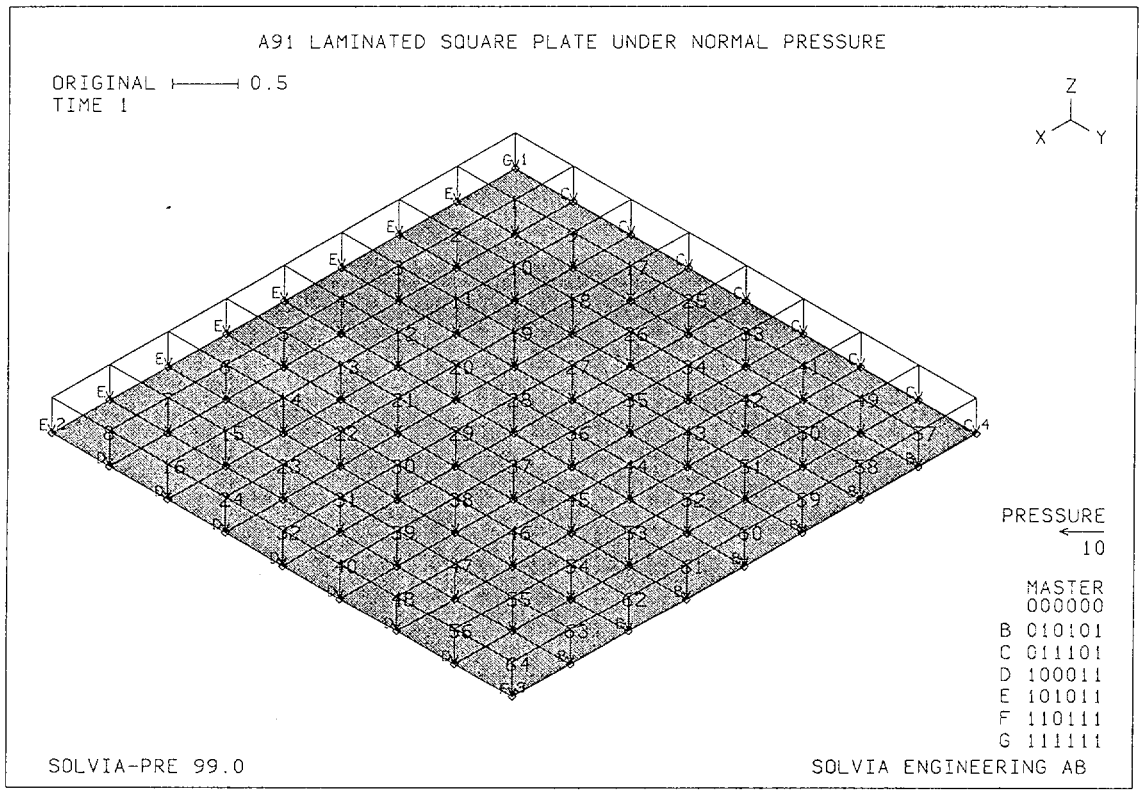
Stresses $\sigma_{xx}, \sigma_{yz}, \sigma_{yy}$ and σ_{xy} along a line in the thickness direction formed by the integration points nearest the centre are shown on pages A91.4 and A91.5.

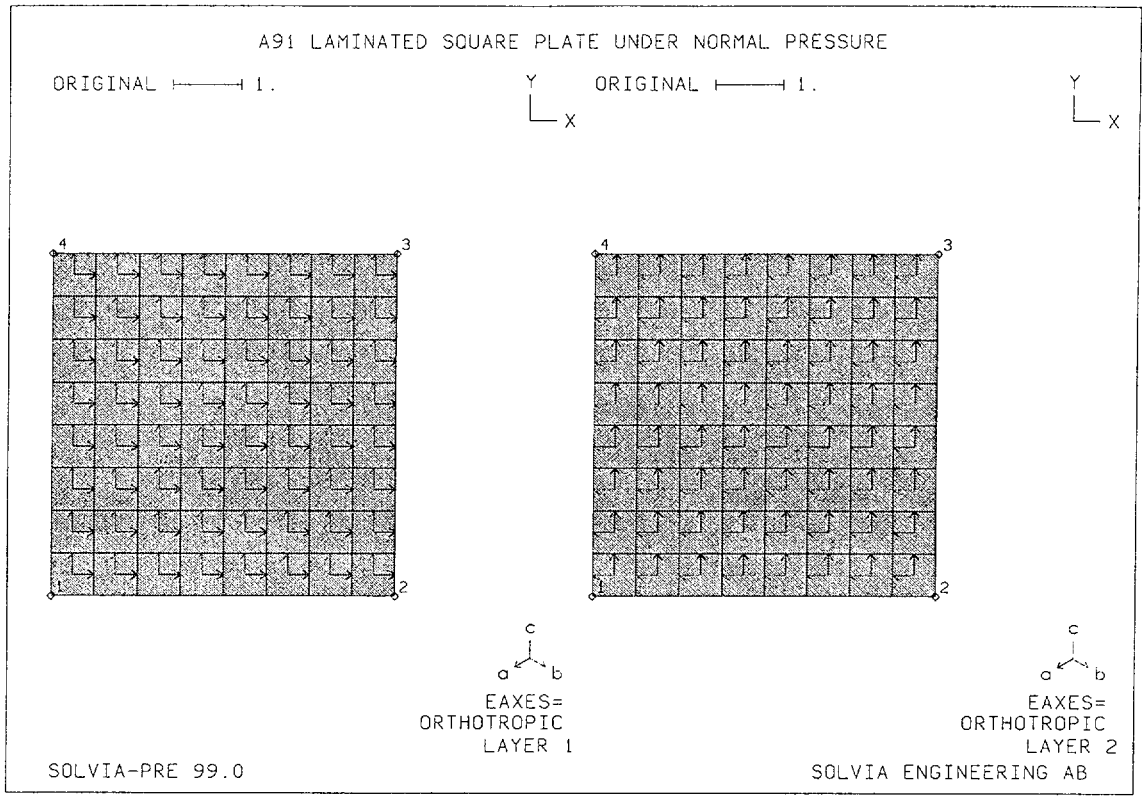
Results compared with NAFEMS' target values:

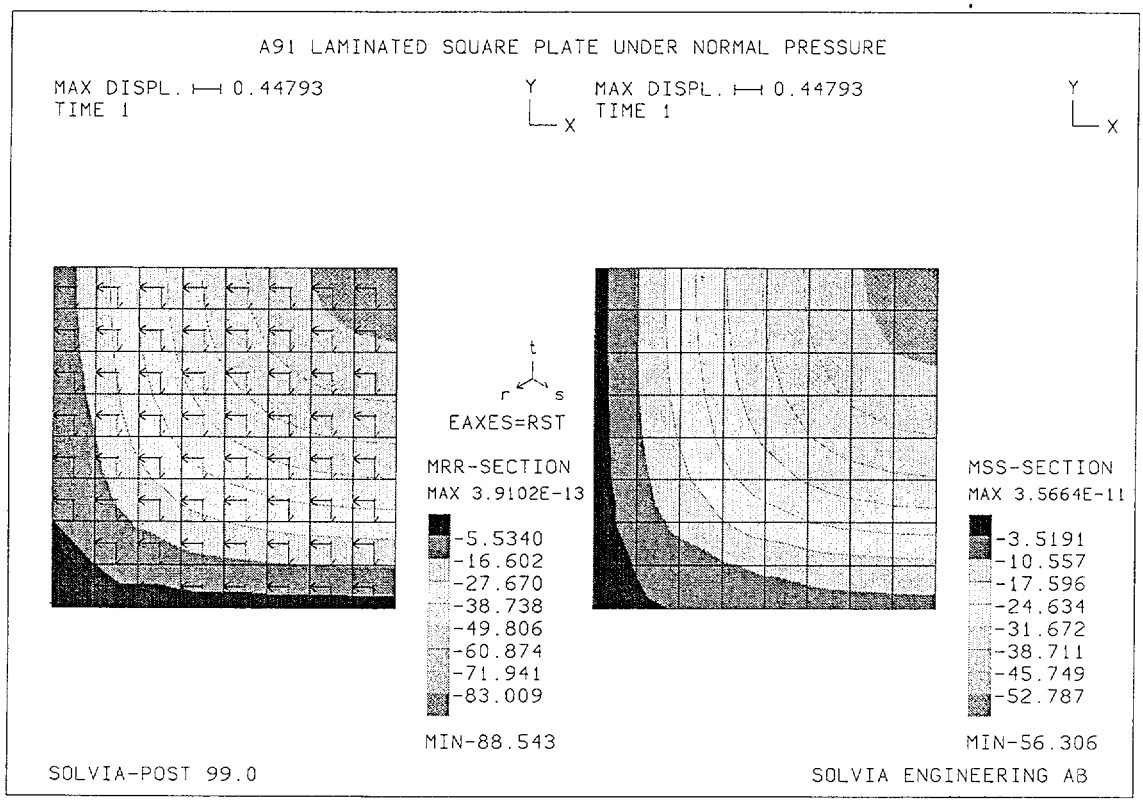
	SOLVIA	NAFEMS [1]
Central displacement [inch] (node 3, Z-direction)	-0.4479	-0.4486
Section moment, M_{rr} at centre, node 3 (SOLVIA value from contour plot)	-88.54	-88.79

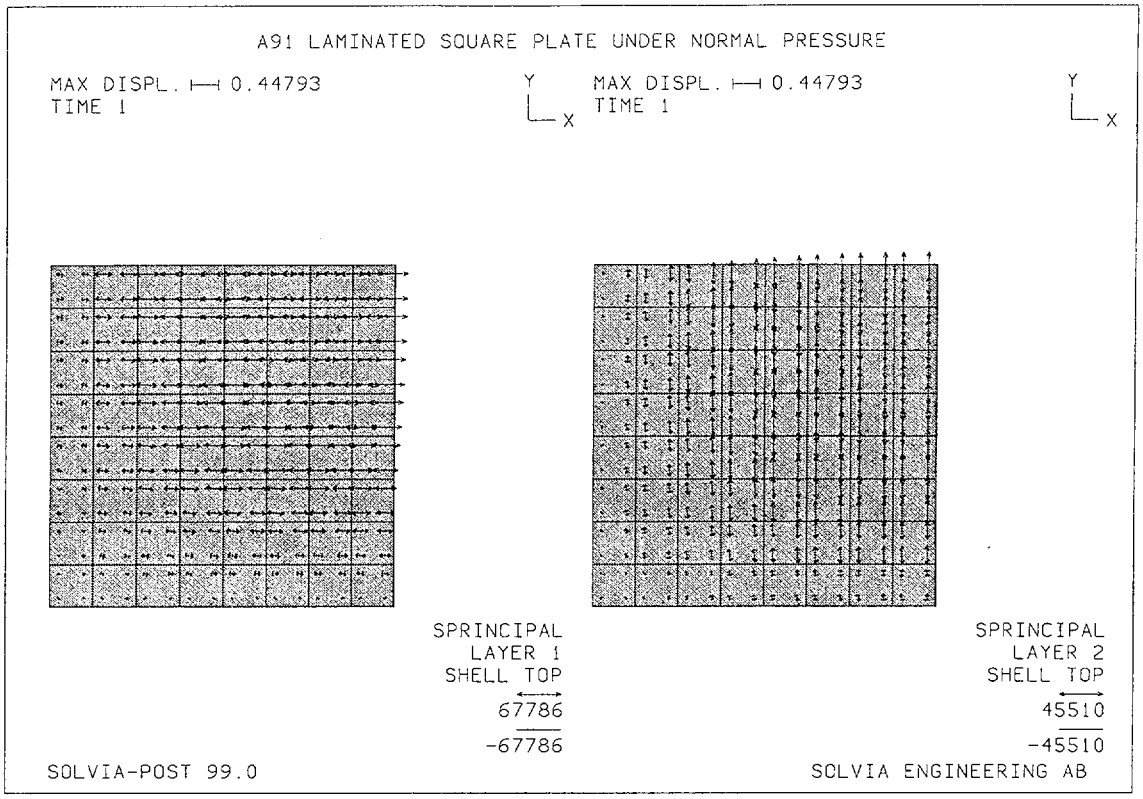
Reference

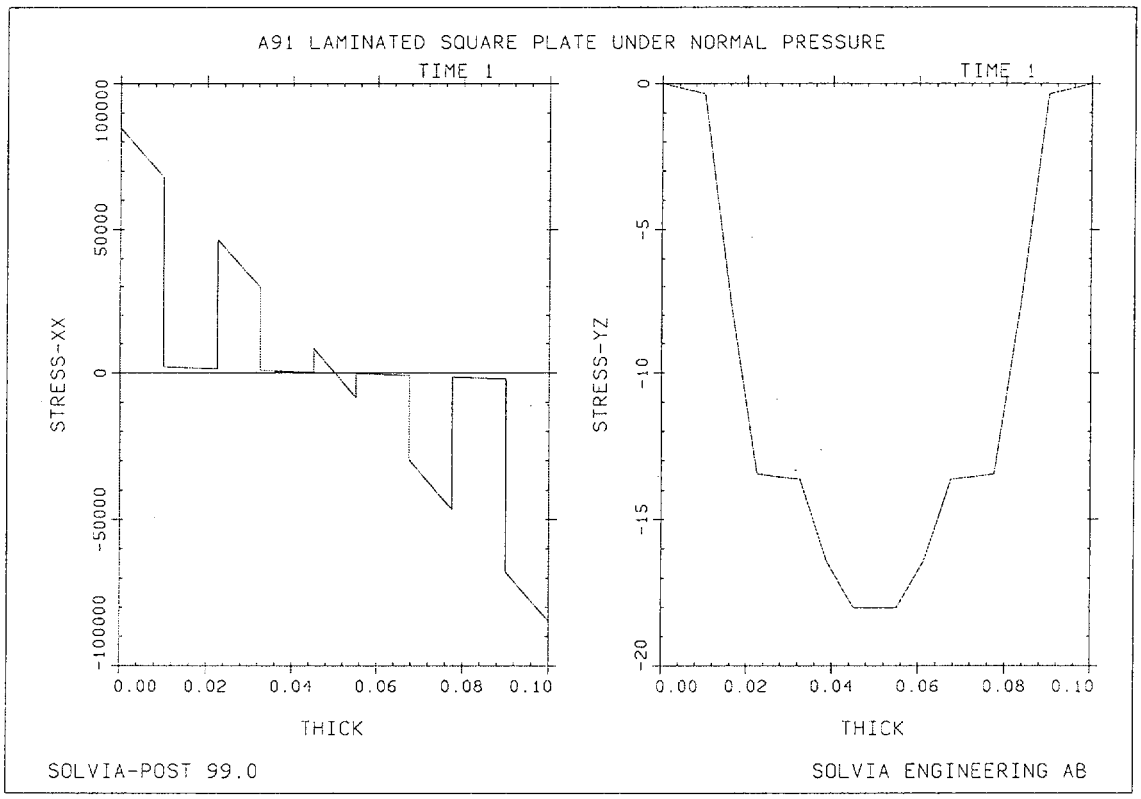
[1] Taig, I.C., "Finite Element Analysis of Composite Materials", NAFEMS Report R0003, 1994.

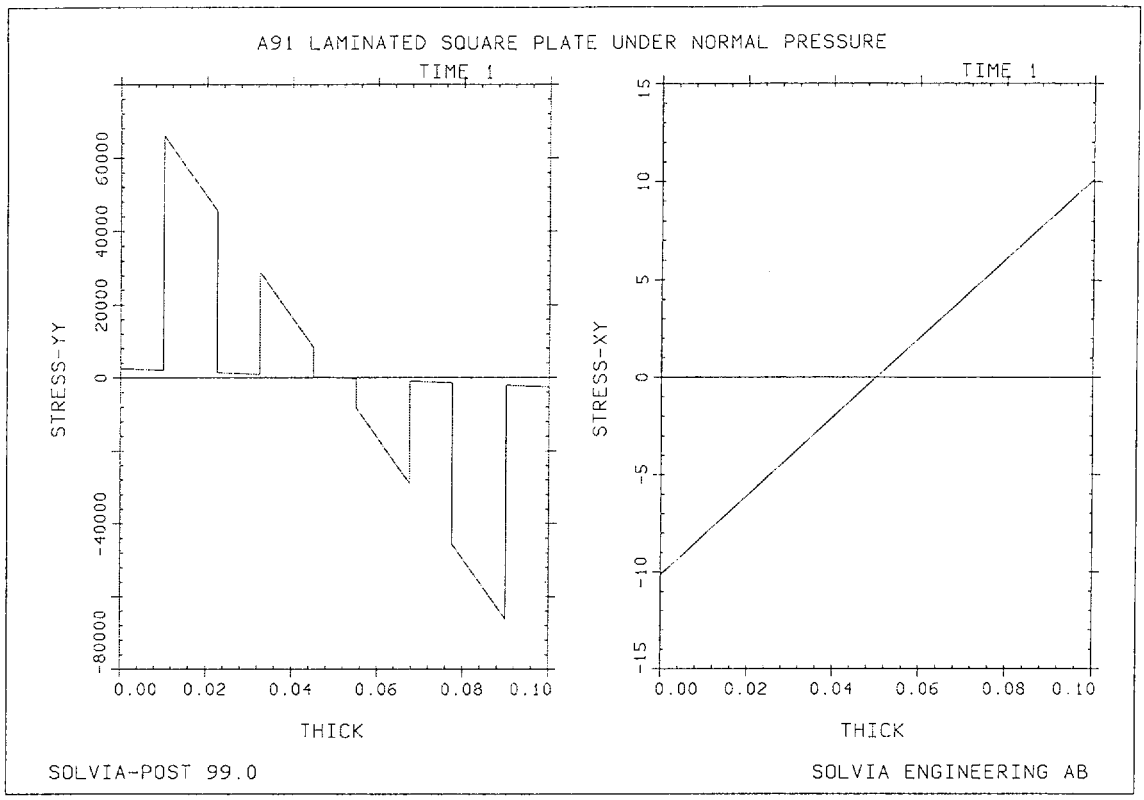



A91 LAMINATED SQUARE PLATE UNDER NORMAL PRESSURE
ORIGINAL 1.
Y
X
ORIGINAL 1.
Y
X
4
3
1
2
4
3
1
2
c
a
b
EAXES=
ORTHOTROPIC
LAYER 1
c
a
b
EAXES=
ORTHOTROPIC
LAYER 2
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A91 LAMINATED SQUARE PLATE UNDER NORMAL PRESSURE
MAX DISPL. 0.44793
TIME 1
Y
X
MAX DISPL. 0.44793
TIME 1
Y
X
t
r
s
EAXES=RST
MRR-SECTION
MAX 3.9102E-13
-5.5340
-16.602
-27.670
-38.738
-49.806
-60.874
-71.941
-83.009
MIN-88.543
MSS-SECTION
MAX 3.5664E-11
-3.5191
-10.557
-17.596
-24.634
-31.672
-38.711
-45.749
-52.787
MIN-56.306
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A91 LAMINATED SQUARE PLATE UNDER NORMAL PRESSURE
MAX DISPL. 0.44793
TIME 1
Y
X
MAX DISPL. 0.44793
TIME 1
Y
X
SPRINCIPAL
LAYER 1
SHELL TOP
67786
-67786
SPRINCIPAL
LAYER 2
SHELL TOP
45510
-45510
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A91 LAMINATED SQUARE PLATE UNDER NORMAL PRESSURE
TIME 1
STRESS-XX
100000
50000
0
-50000
-100000
0.00
0.02
0.04
0.06
0.08
0.10
THICK
TIME 1
STRESS-YZ
0
-5
-10
-15
-20
0.00
0.02
0.04
0.06
0.08
0.10
THICK
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A91 LAMINATED SQUARE PLATE UNDER NORMAL PRESSURE
TIME 1
STRESS-YY
60000
40000
20000
0
-40000
-80000
0.00 0.02 0.04 0.06 0.08 0.10
THICK
TIME 1
STRESS-XY
15
10
5
0
-5
-10
-15
0.00 0.02 0.04 0.06 0.08 0.10
THICK
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
DATABASE CREATE
HEADING 'A91 LAMINATED SQUARE PLATE UNDER NORMAL PRESSURE'
*
COORDINATES
 ENTRIES  NODE   X   Y
           1     0   0
           2     5   0
           3     5   5
           4     0   5
*
MATERIAL 1 ORTHOTROPIC EA=4E7 EB=1E6 NUBA=0.25 ,
           GAB=0.6E6  GAC=0.5E6 GBC=0.5E6
*
MATERIAL 2 LAMINATE
 1 1 10   0
 2 1 12.5 90
 3 1 10   0
 4 1 12.5 90
 5 1 10   0
 6 1 12.5 90
 7 1 10   0
 8 1 12.5 90
 9 1 10   0
*
EGROUP 1 SHELL MATERIAL=2 TINT=-3
SET NODES=4
GSURFACE  1 2 3 4 EL1=8 EL2=8
THICKNESS 1 1.0E-1
LOAD ELEMENT TYPE=PRESSURE INPUT=SURFACE
 1 2 3 4 t 10 10
*
FIXBOUNDARIES  1356 INPUT=LINES / 1 2
FIXBOUNDARIES  2346 INPUT=LINES / 1 4
FIXBOUNDARIES  246  INPUT=LINES / 3 4
FIXBOUNDARIES  156  INPUT=LINES / 2 3
*
MESH NNUMBER=MYNODES NSYMBOL=YES VECTOR=LOAD,
     ENUMBER=YES BCODE=ALL
SHELLSURFACE LAYER=1
SET VIEW=Z NNUMBER=MYNODES NSYMBOL=MYNODES
MESH EAXES=ORTHOTROPIC SUBFRAME=21
SHELLSURFACE LAYER=2
MESH EAXES=ORTHOTROPIC
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A91  LAMINATED SQUARE PLATE UNDER NORMAL PRESSURE
*
DATABASE CREATE
*
WRITE FILENAME='a91.lis'
NMAX DIR=3
MESH CONTOUR=MRR-SECTION EAXES=RST VIEW=Z SUBFRAME=21
MESH CONTOUR=MSS
*
SHELLSURFACE LAYER=1
MESH VECTOR=SPRINCIP SUBFRAME=21
SHELLSURFACE LAYER=2
MESH VECTOR=SPRINCIP
*
EPLINE THICK
64 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 ,
   21 22 23 24 25 26 27
ELINE THICK KIND=SXX SUBFRAME=21
ELINE THICK KIND=SYZ
ELINE THICK KIND=SYY SUBFRAME=21
ELINE THICK KIND=SXY
END
```

EXAMPLE A92

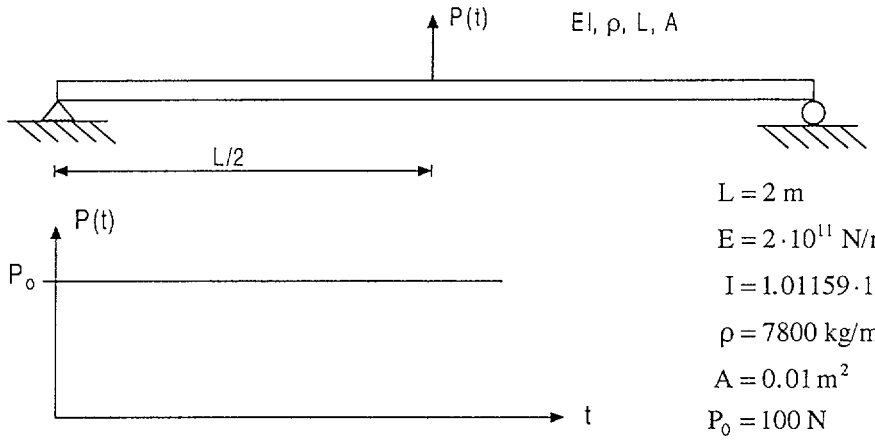
DYNAMIC ANALYSIS OF A BEAM

Objective

To verify the implicit direct time integration methods available in SOLVIA when used in a simply supported beam analysis.

Physical Problem

A simply supported uniform beam subjected to a central step-function loading as shown in the figure below. This problem is also analyzed in Example A93 using the mode superposition procedure.



$L = 2\ \text{m}$

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

$I = 1.01159 \cdot 10^{-6}\ \text{m}^4$

$\rho = 7800\ \text{kg/m}^3$

$A = 0.01\ \text{m}^2$

$P_0 = 100\ \text{N}$

Finite Element Model

The finite element model consists of 20 BEAM elements, see top figure on page A92.3. A consistent mass matrix is used.

The time integration methods used in the analyses are the Newmark method (trapezoidal rule), the Wilson theta-method and the Hilber-Hughes method. Four different time step sizes are used for each method, namely DT = T/50, DT = T/25, DT = T/10 and DT = T/5, where T is the time period for the lowest frequency mode of the beam structure.

Solution Results

The theoretical solution for this problem is presented in [1], pp. 401-403.

Displacement: $$u(x,t) = \frac{2P_0L^3}{\pi^4 EI} \sum_{n=1}^{\infty} \frac{\alpha_n}{n^4}(1-\cos\omega_n t)\sin\frac{n\pi x}{L}$$

where $$\alpha_n = \begin{cases} 1 & , \ n = 1,5,9,\ldots \\ -1 & , \ n = 3,7,11,\ldots \\ 0 & , \ n = \text{even numbers} \end{cases} \quad \text{and} \quad \omega_n^2 = \frac{n^4\pi^4}{L^4}\frac{EI}{\rho A}$$

Velocity: $$\dot{u}(x,t) = \frac{\partial u(x,t)}{\partial t} = \frac{2P_0L}{\pi^2\sqrt{EI\rho A}} \sum_{n=1}^{\infty} \frac{\alpha_n}{n^2}\sin\omega_n t \sin\frac{n\pi x}{L}$$

Acceleration: $$\ddot{u}(x,t) = \frac{\partial^2 u(x,t)}{\partial t^2} = \frac{2P_0}{L\rho A} \sum_{n=1}^{\infty} \alpha_n \cos\omega_n t \sin\frac{n\pi x}{L}$$

Bending moment: $$M(x,t) = EI \cdot \frac{\partial^2 u(x,t)}{\partial x^2} = -\frac{2P_0 L}{\pi^2} \sum_{n=1}^{\infty} \frac{\alpha_n}{n^2}(1-\cos\omega_n t)\sin\frac{n\pi x}{L}$$

Shear force: $$V(x,t) = EI \cdot \frac{\partial^3 u(x,t)}{\partial x^3} = -\frac{2P_0}{\pi} \sum_{n=1}^{\infty} \frac{\alpha_n}{n}(1-\cos\omega_n t)\cos\frac{n\pi x}{L}$$

Note that the acceleration series does not converge with increasing mode number n, which is due to the singular nature of the load. Therefore, the acceleration response is not treated further in this example. In practice, of course, the load is associated with a finite area and has a finite rise time.

The theoretical solution (using n = 12000) for the displacement, velocity and bending moment at the midspan of the beam and the shear force at the left support can be seen in the figures on page A92.3. Note the high frequency content in the shear force response compared to the bending moment and the velocity response, which in turn contain higher frequencies than the displacement response. The theoretical solution curves are also shown for comparison in the subsequent result figures.

The results from the Newmark time integration analysis can be seen on pages A92.4 and A92.5. The time integration parameters used are DELTA = 0.5 and ALPHA = 0.25. This method is also called the trapezoidal rule and the constant-average-acceleration method. Note that the trapezoidal rule has no algorithmic damping properties (non-dissipative) but it causes period elongation.

The results from the Wilson time integration analysis can be seen on pages A92.6 and A92.7 using time integration parameter THETA = 1.4. Note that the Wilson method is highly dissipative for larger time steps [2]. Note also that the Wilson method has larger period elongation as compared with the trapezoidal rule.

The results from the Hilber-Hughes time integration analysis can be seen on pages A92.8 and A92.9 using time integration parameter GAMMA = -0.1. Note that the Hilber-Hughes method equals the trapezoidal rule when GAMMA = 0 and that the algorithmic damping properties of the Hilber-Hughes method are a function of the GAMMA value [2]. The highest damping ratio is found when GAMMA = -1/3. The practical range of GAMMA is therefore $-1/3 \le \text{GAMMA} \le 0$. The solution curves for the Hilber-Hughes method and the trapezoidal rule are in close agreement, but we may note that the Hilber-Hughes method gives smoother curves for the velocity, the bending moment and the shear force when using smaller time steps, DT = T/50 and DT = T/25, due to the algorithmic damping of higher modes.

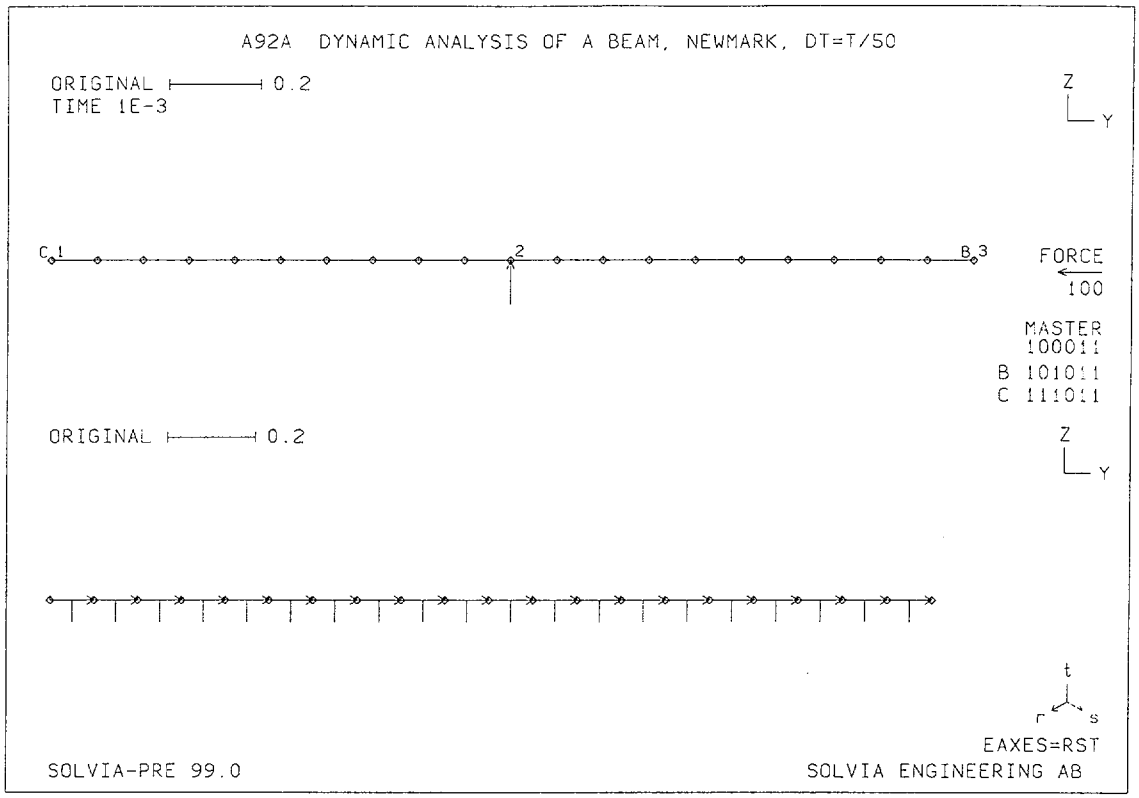
User Hints

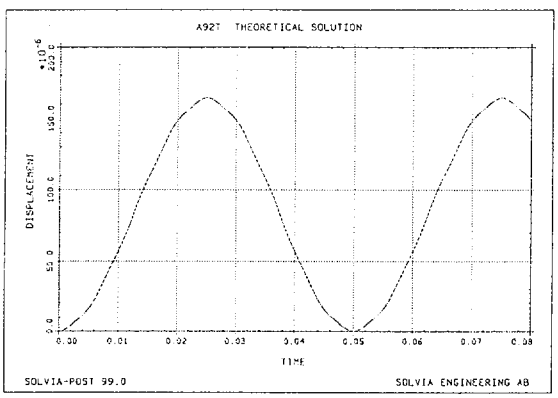
- A commonly used rule-of-thumb for non-dissipative algorithms requires at least 10 time steps per period to be taken for accuracy [2]. Hence, when DT = T/50 the response using non-dissipative algorithms contains frequencies up to 5 times the fundamental frequency with reasonable accuracy.
- Note also that an accurate solution of bending moments and shear forces requires small time steps as compared with the displacement solution in a dynamic analysis.
- The USERCURVE command can be used in SOLVIA-POST to easily include user-supplied curves.

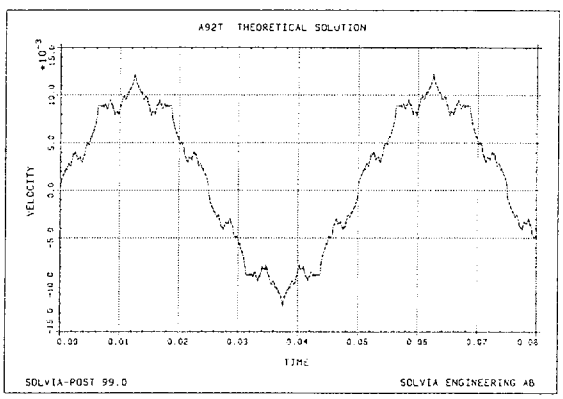
References

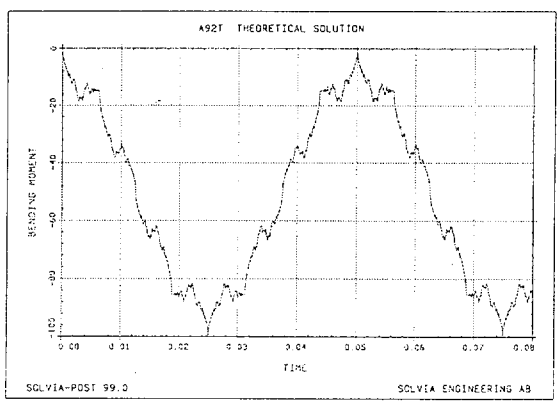
[1] Clough, R.W., and Penzien, J., Dynamics of Structures, Second Edition, McGraw-Hill, 1993.

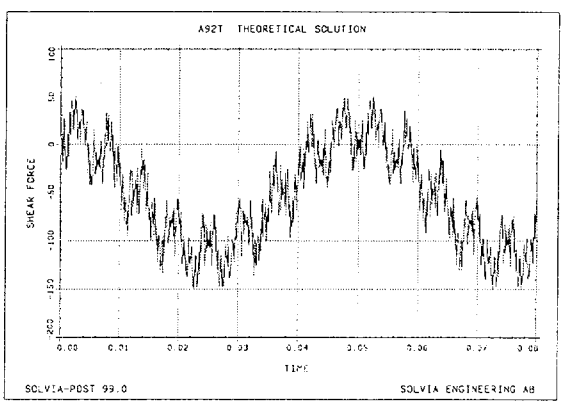
[2] Hilber, H.M., Hughes, T.J.R., and Taylor, R.L., "Improved Numerical Dissipation for Time Integration Algorithms in Structural Dynamics", Earthquake Engineering and Structural Dynamics, Vol. 5, pp. 283-292, 1977.

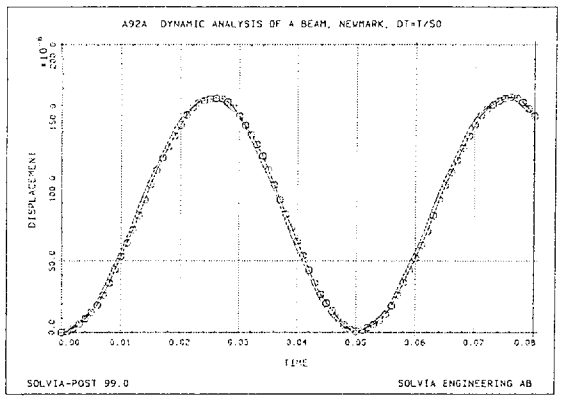

A92A DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/50
ORIGINAL 0.2
TIME 1E-3
Z
Y
C 1
2
B 3
FORCE
100
MASTER
100011
B 101011
C 111011
ORIGINAL 0.2
Z
Y
t
r
s
EAXES=RST
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

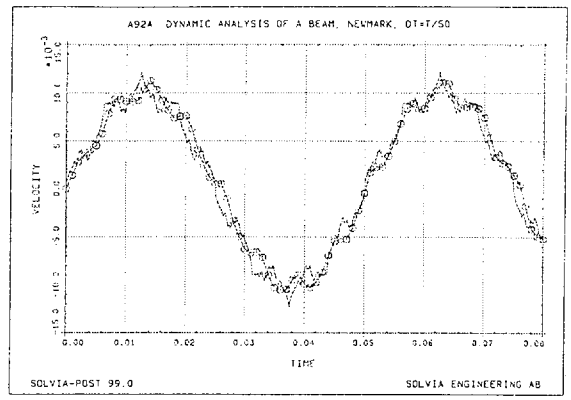

A92T THEORETICAL SOLUTION
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

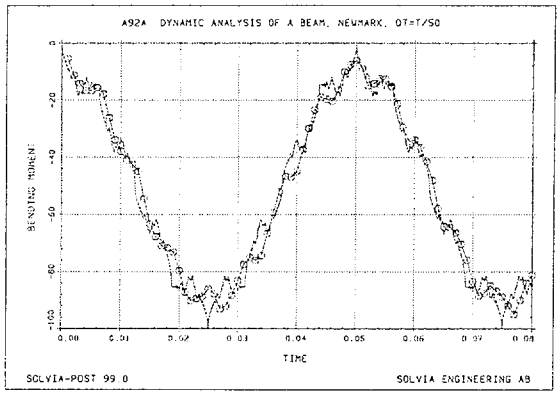

A92T THEORETICAL SOLUTION
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

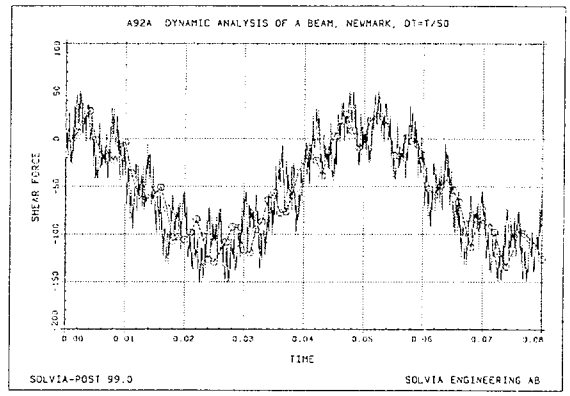

A92T THEORETICAL SOLUTION
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

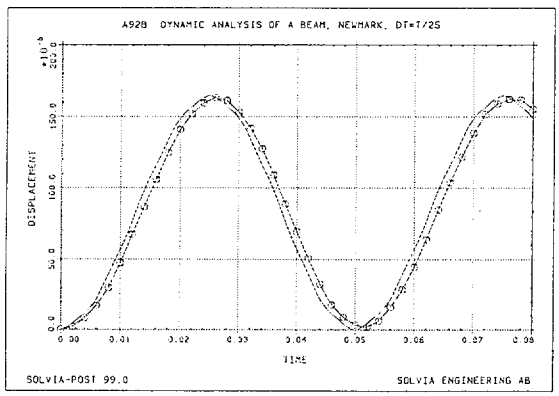

A92T THEORETICAL SOLUTION
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

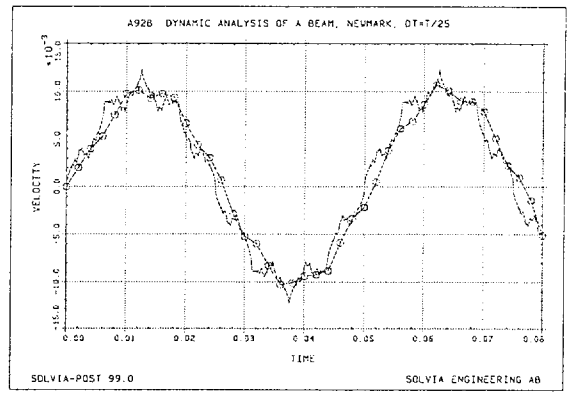

A92A DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/50
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

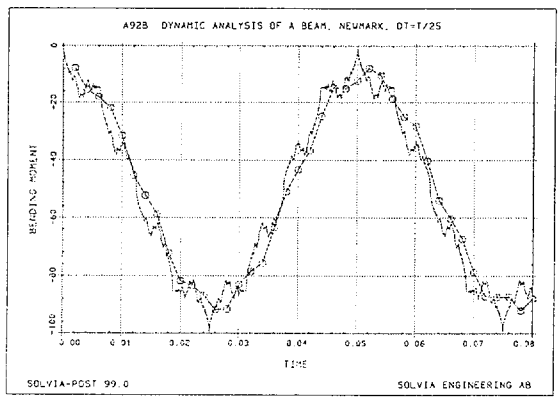

A92A DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/50
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

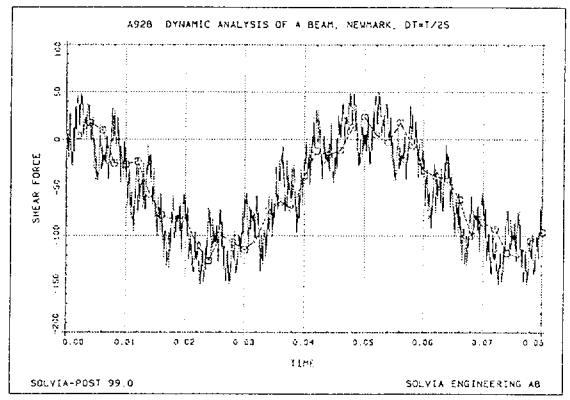

A92A DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/50
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

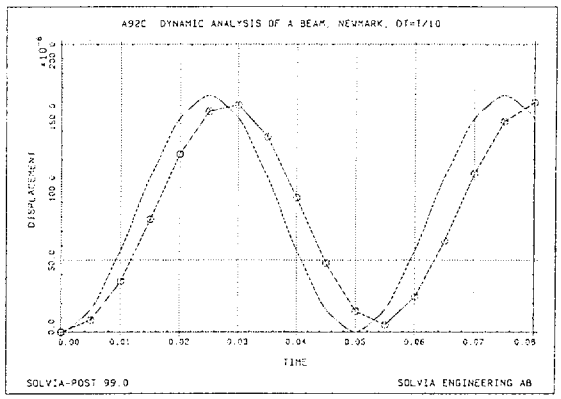

A92A DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/50
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

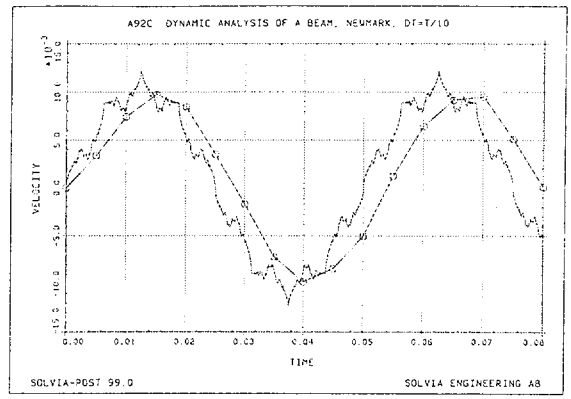

A92B DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/25
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

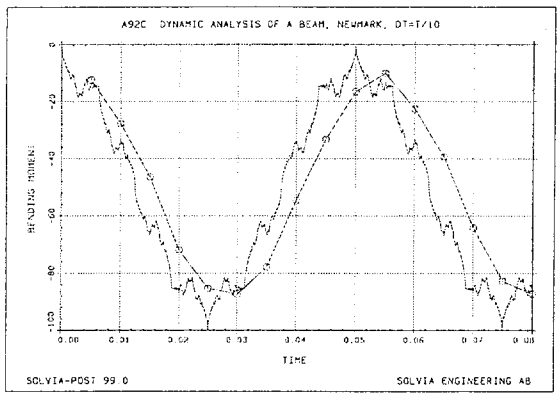

A92B DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/25
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

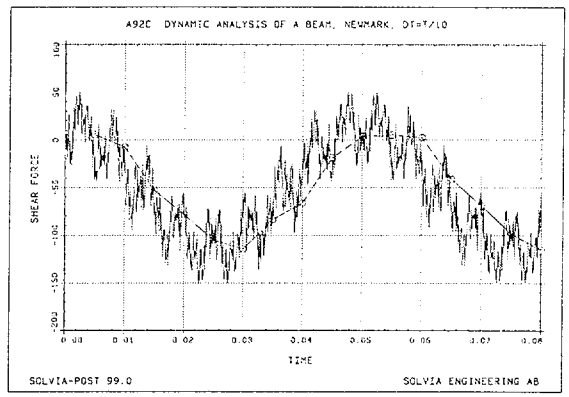

A92B DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/25
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

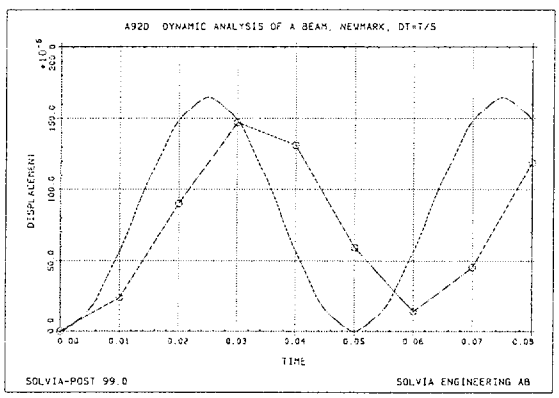

A92B DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/25
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

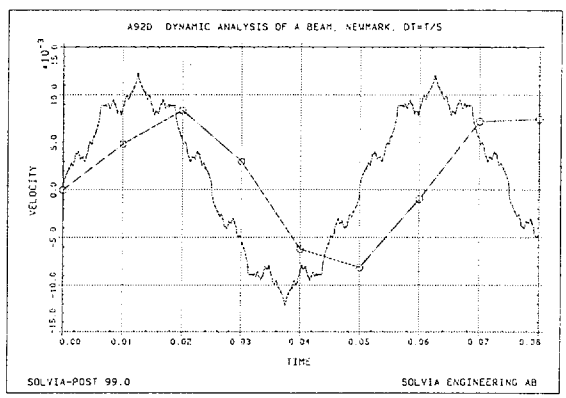

A92C DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/10
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

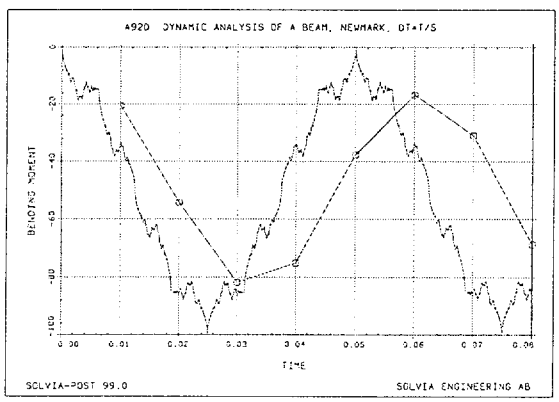

A92C DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/10
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

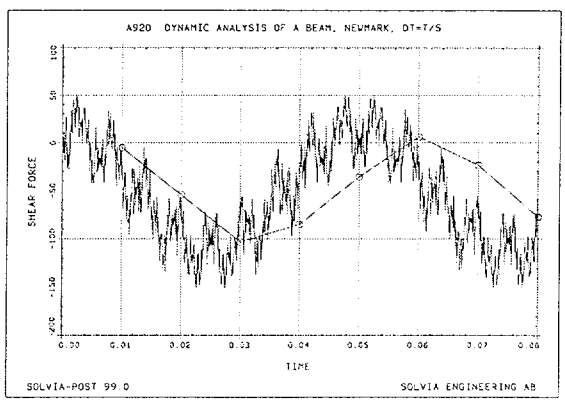

A92C DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/10
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

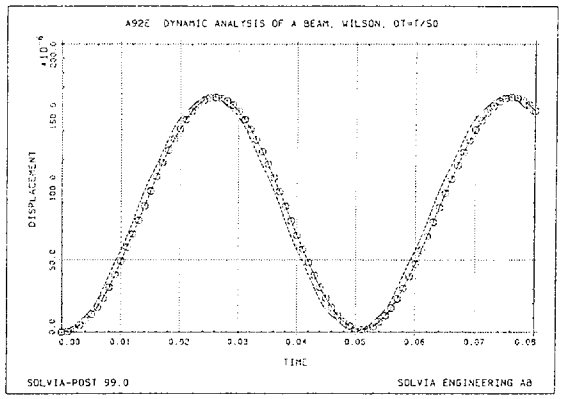

A92C DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/10
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

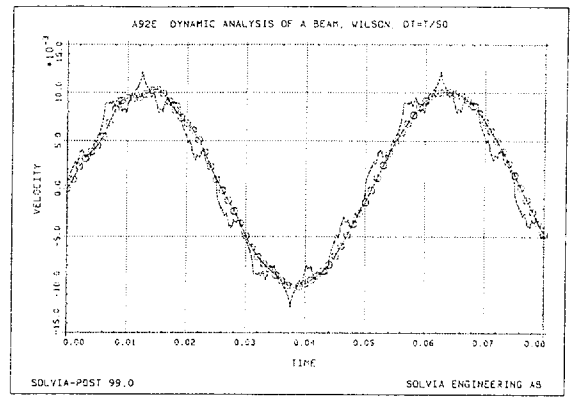

A92D DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/5
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

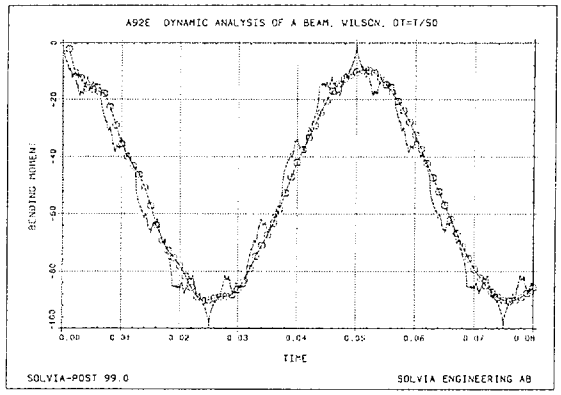

A92D DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/5
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

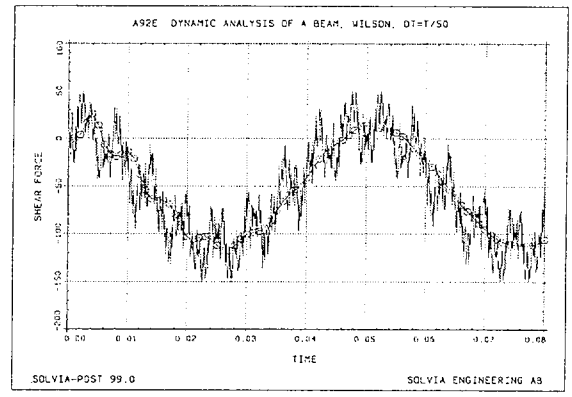

A92D DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/5
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

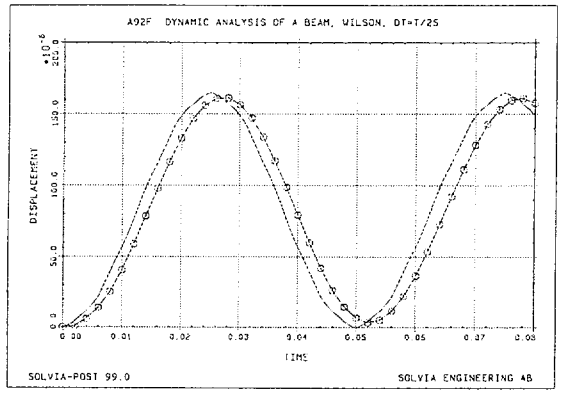

A92D DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/5
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

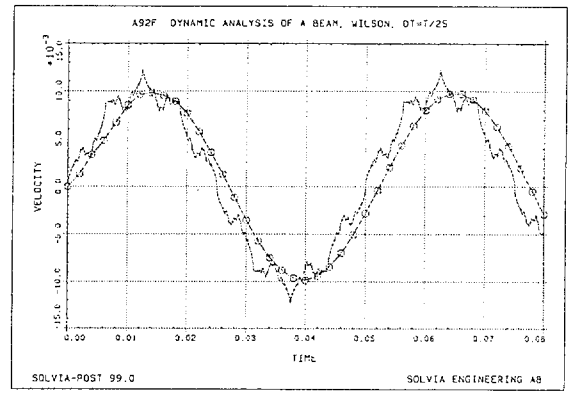

A92E DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/50
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

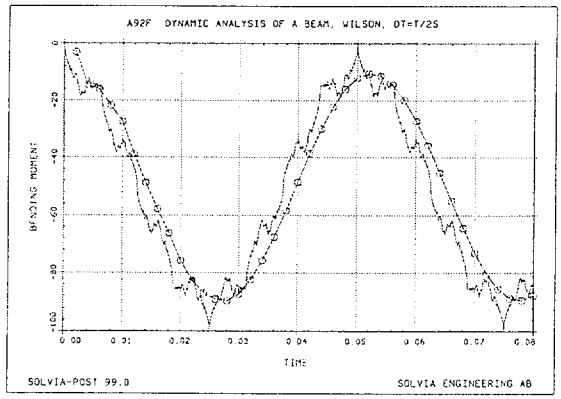

A92E DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/50
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

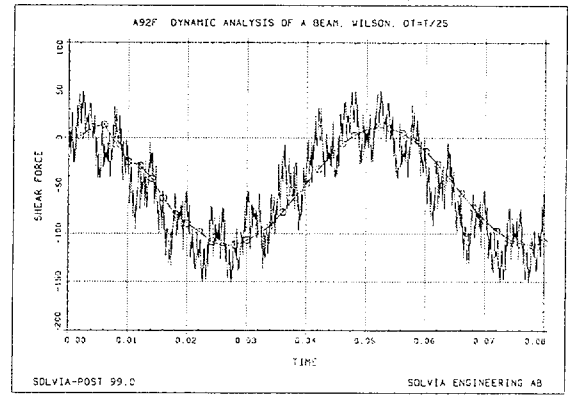

A92E DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/50
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

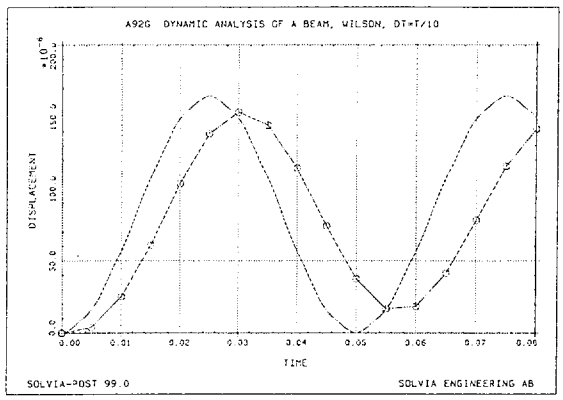

A92E DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/50
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

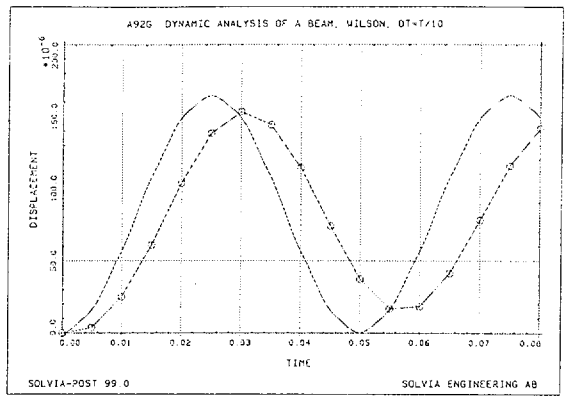

A92F DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/25
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

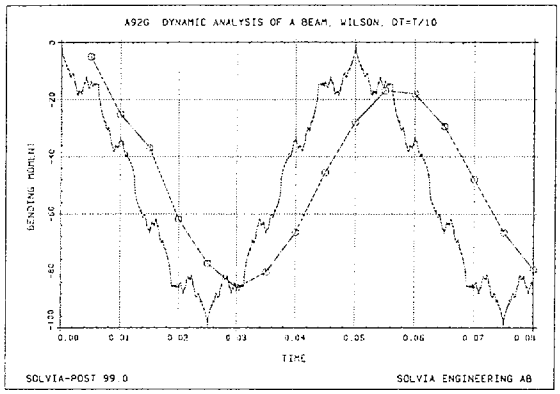

A92F DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/25
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

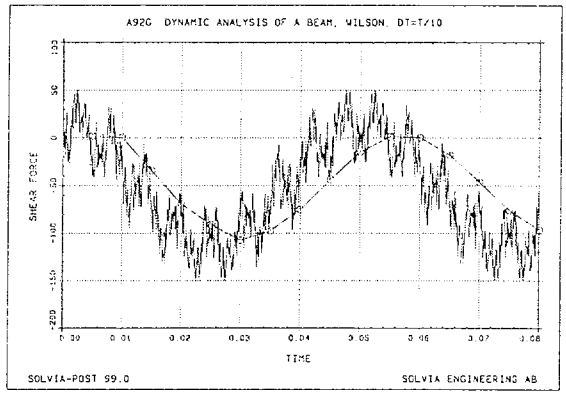

A92F DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/25
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

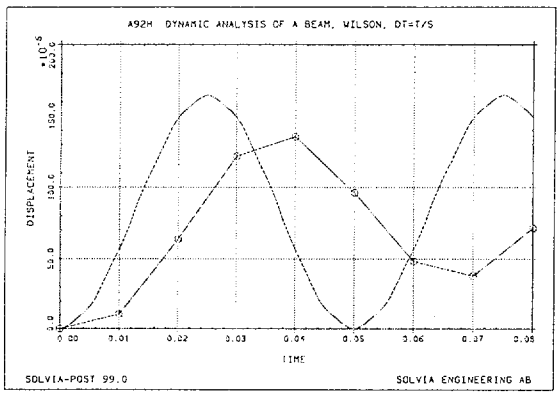

A92F DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/25
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

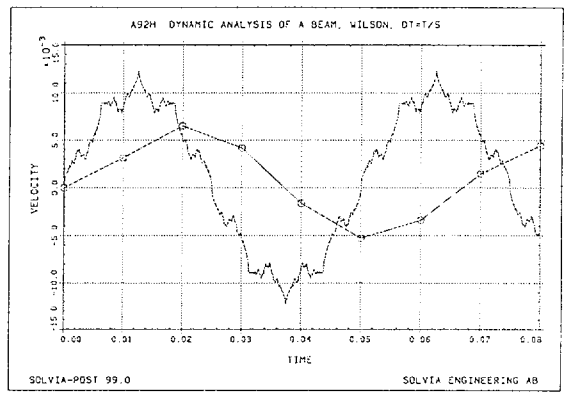

A92G DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/10
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

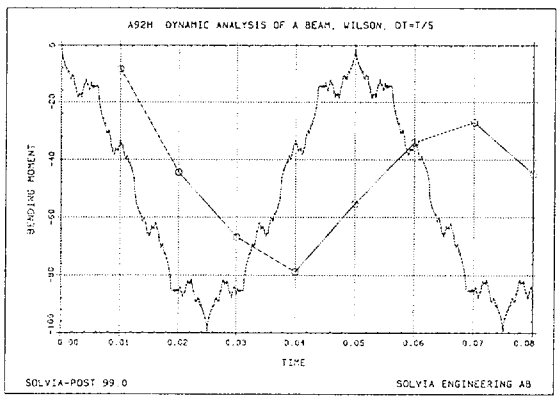

A92G DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/10
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

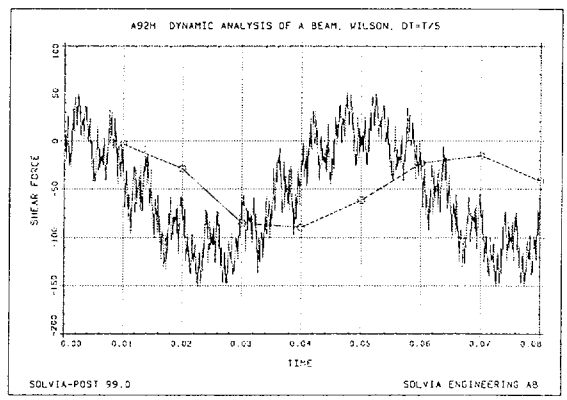

A92G DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/10
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

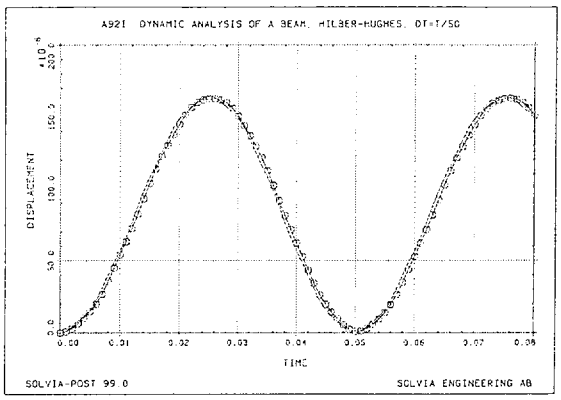

A92G DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/10
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

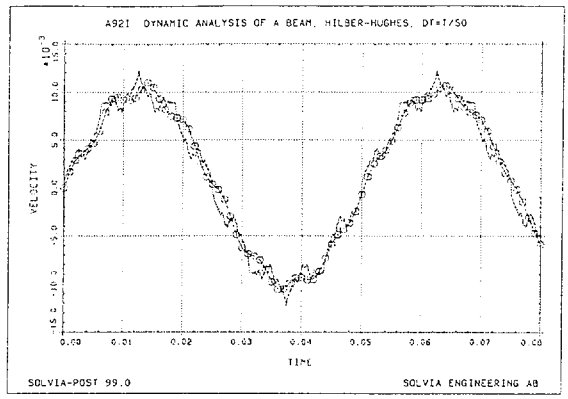

A92H DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/5
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

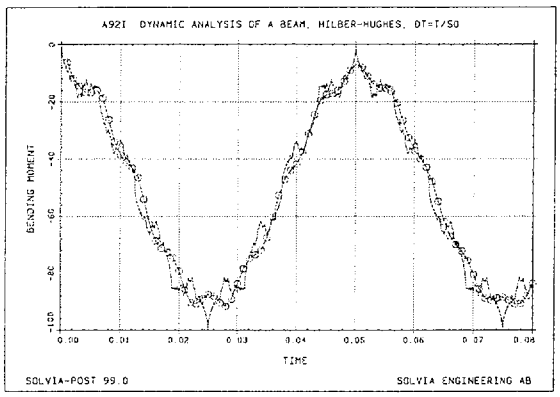

A92H DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/5
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

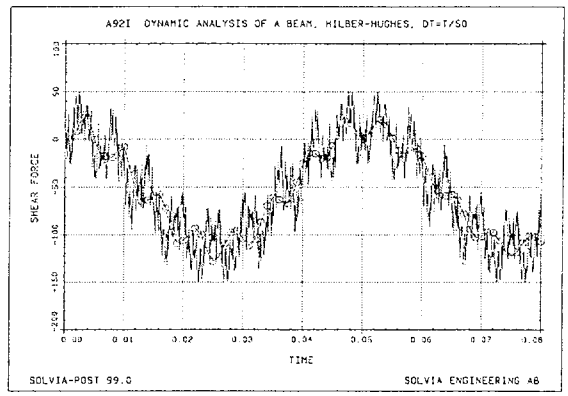

A92H DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/5
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

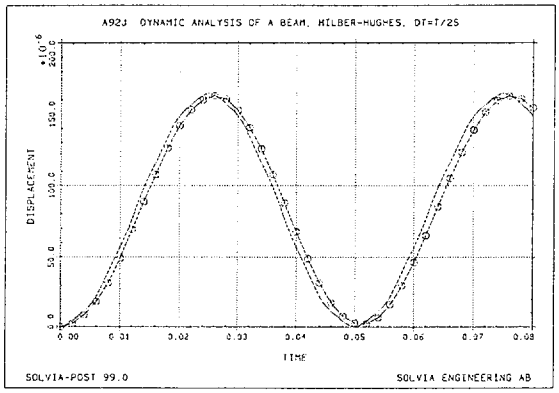

A92H DYNAMIC ANALYSIS OF A BEAM, WILSON, DT=T/5
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

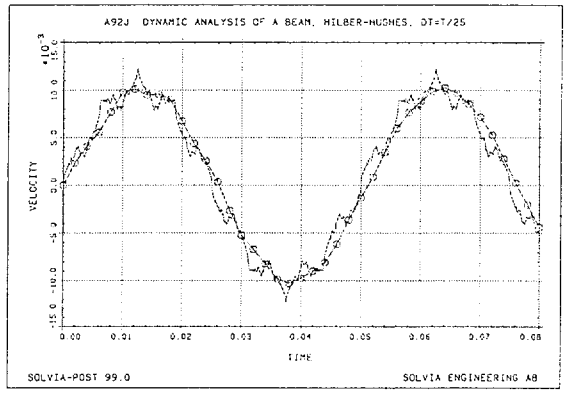

A92I DYNAMIC ANALYSIS OF A BEAM. HILBER-HUGHES. DT=T/50
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

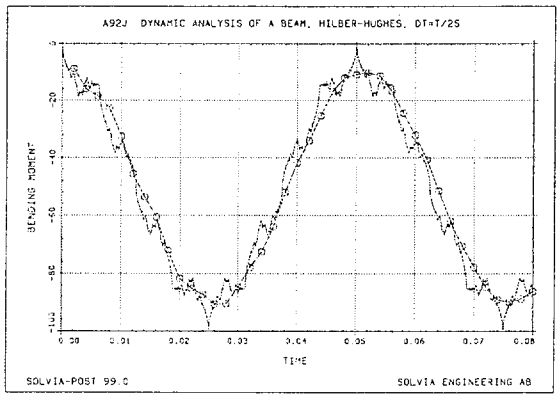

A92I DYNAMIC ANALYSIS OF A BEAM. HILBER-HUGHES. DT=T/50
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

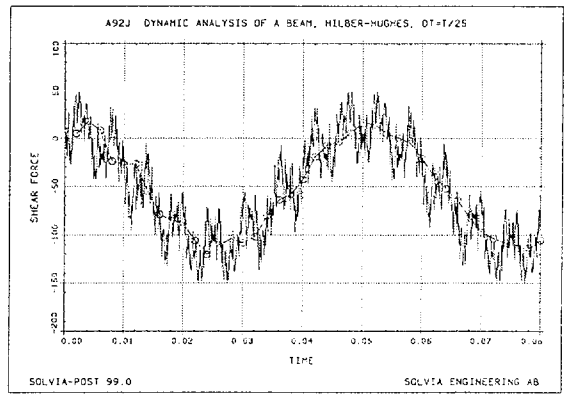

A92I DYNAMIC ANALYSIS OF A BEAM. HILBER-HUGHES. DT=T/50
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

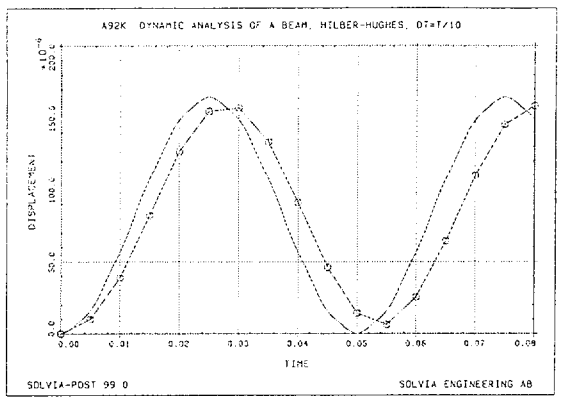

A92I DYNAMIC ANALYSIS OF A BEAM. HILBER-HUGHES. DT=T/50
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

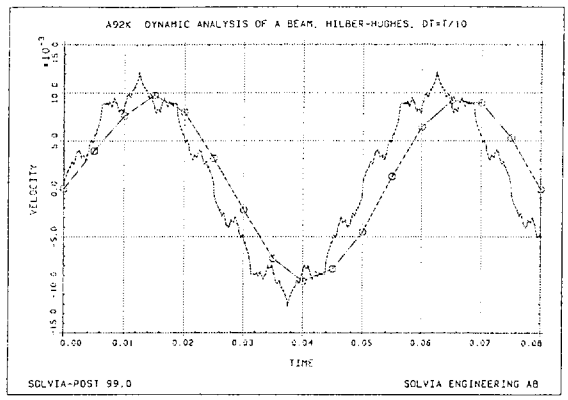

A92J DYNAMIC ANALYSIS OF A BEAM. HILBER-HUGHES. DT=T/25
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

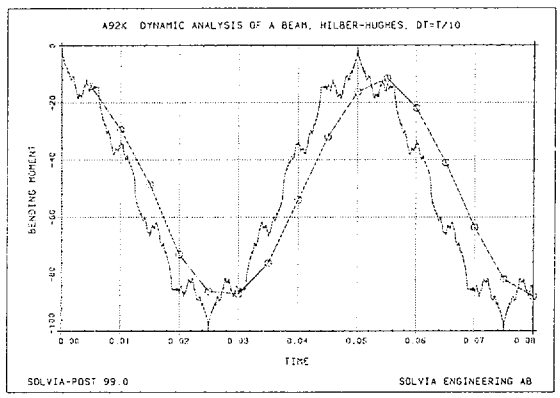

A92J DYNAMIC ANALYSIS OF A BEAM. HILBER-HUGHES. DT=T/25
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

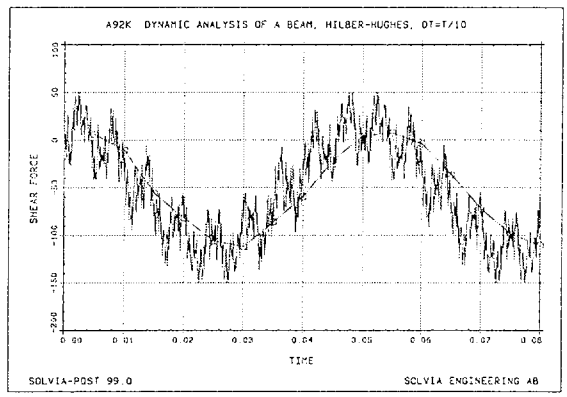

A92J DYNAMIC ANALYSIS OF A BEAM. HILBER-HUGHES. DT=T/25
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

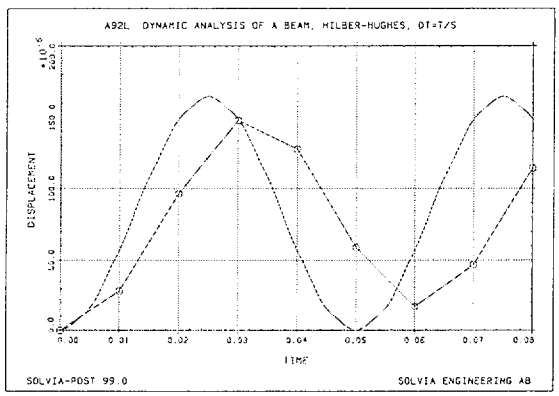

A92J DYNAMIC ANALYSIS OF A BEAM. HILBER-HUGHES. DT=T/25
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

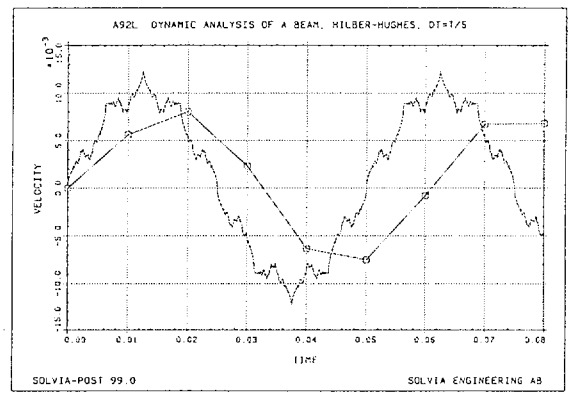

A92K DYNAMIC ANALYSIS OF A BEAM, HILBER-HUGHES, DT=T/10
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

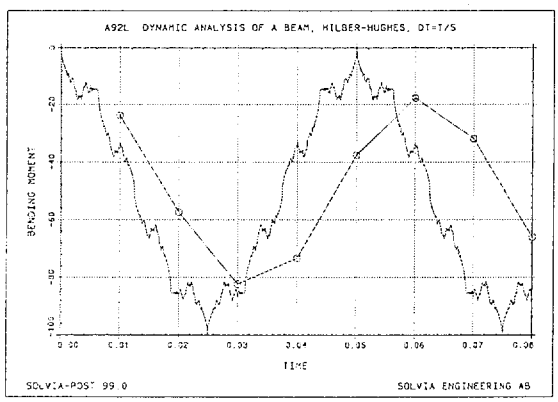

A92K DYNAMIC ANALYSIS OF A BEAM, HILBER-HUGHES, DT=T/10
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

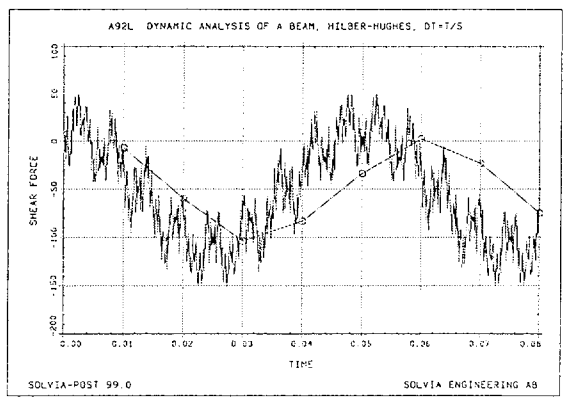

A92K DYNAMIC ANALYSIS OF A BEAM, HILBER-HUGHES, DT=T/10
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A92K DYNAMIC ANALYSIS OF A BEAM, HILBER-HUGHES, DT=T/10
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A92L DYNAMIC ANALYSIS OF A BEAM, HILBER-HUGHES, DT=T/5
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A92L DYNAMIC ANALYSIS OF A BEAM, HILBER-HUGHES, DT=T/5
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A92L DYNAMIC ANALYSIS OF A BEAM, HILBER-HUGHES, DT=T/5
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A92L DYNAMIC ANALYSIS OF A BEAM, HILBER-HUGHES, DT=T/5
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A92A  DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/50'
*
DATABASE  CREATE
MASTER  IDOF=100011 NSTEP=80 DT=0.001
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=CONSISTENT METHOD=NEWMARK,
          DELTA=0.5 ALPHA=0.25
*
TIMEFUNCTION  1
 0. 1.  /  1. 1.
*
COORDINATES
 1  /  2  0. 1.  /  3  0. 2.
*
MATERIAL  1  ELASTIC E=2.E11 NU=0. DENSITY=7800.
*
EGROUP  1  BEAM  RESULTS=FORCES
BEAMVECTOR
 1  1. 0. 0.
SECTION  1 GENERAL RINERTIA=1.E-6 SINERTIA=1.01159E-6,
                   TINERTIA=1.E-6 AREA=0.01
GLINE  1 3 -1  EL=20
*
FIXBOUNDARIES  23  /  1
FIXBOUNDARIES   3  /  3
LOADS  CONCENTRATED
 2 3 100.
*
SET  VIEW=X NSYMBOLS=YES
MESH  VECTOR=LOAD NNUMBERS=MYNODES BCODE=ALL SUBFRAME=12
MESH  EAXES=RST
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A92A  DYNAMIC ANALYSIS OF A BEAM, NEWMARK, DT=T/50
*
DATABASE  CREATE
*
AXIS  1 VMIN=0.      VMAX=0.08  LABELSTRING='TIME'
AXIS  2 VMIN=0.      VMAX=2.E-4 LABELSTRING='DISPLACEMENT'
AXIS  3 VMIN=-0.015 VMAX=0.015 LABELSTRING='VELOCITY'
AXIS  4 VMIN=-100.  VMAX=0.    LABELSTRING='BENDING MOMENT'
AXIS  5 VMIN=-200.  VMAX=100.  LABELSTRING='SHEAR FORCE'
*
USERCURVE  1
READ A92DISP.DAT
USERCURVE  2
READ A92VEL.DAT
USERCURVE  3
READ A92MOM.DAT
USERCURVE  4
READ A92FORCE.DAT
*
SET  DIAGRAM=GRID
NHISTORY  NODE=2 DIRECTION=3 XAXIS=1 YAXIS=2 SYMBOL=1
PLOT  USERCURVE  1 XAXIS=-1 YAXIS=-2 SUBFRAME=OLD
*
NHISTORY  NODE=2 DIRECTION=3 KIND=VELOCITY XAXIS=1 YAXIS=3 SYMBOL=1
PLOT  USERCURVE  2 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD
*
EHISTORY  ELEMENT=10 POINT=2 KIND=MS XAXIS=1 YAXIS=4 SYMBOL=1
PLOT  USERCURVE  3 XAXIS=-1 YAXIS=-4 SUBFRAME=OLD
*
EHISTORY  ELEMENT=1 POINT=1 KIND=FT XAXIS=1 YAXIS=5 SYMBOL=1
PLOT  USERCURVE  4 XAXIS=-1 YAXIS=-5 SUBFRAME=OLD
END
```

EXAMPLE A93

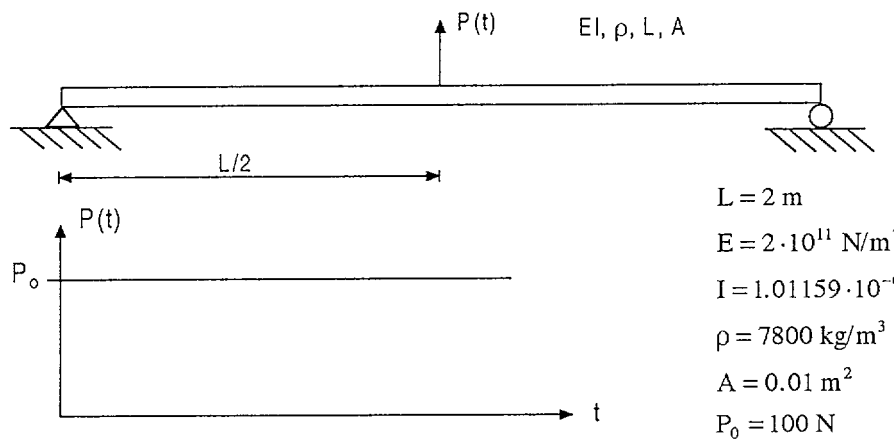
MODE SUPERPOSITION ANALYSIS OF A BEAM

Objective

To verify the mode superposition procedure in SOLVIA in a dynamic beam structure analysis.

Physical Problem

A simply supported uniform beam subjected to a central step-function loading as shown in the figure below will be evaluated. This problem has been analyzed using direct time integration methods as described in Example A92.



$L = 2$ m

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

$I = 1.01159 \cdot 10^{-6}$ m^4

$\rho = 7800$ kg/m^3

$A = 0.01$ m^2

$P_0 = 100$ N

Finite Element Model

Twenty BEAM elements are used to model the simply supported beam structure as shown in the top figure on page A93.3. A consistent mass discretization is applied. The modal time integration is performed with the Newmark method (trapezoidal rule) and the user has no control of it. Two different time step sizes are used, DT = T/25 and DT = T/10 where T is the time period for the lowest frequency mode of the beam structure. Ten modes are used in the mode superposition procedure. For comparison, a single mode analysis is also performed. The subspace iteration method is used in the evaluation of frequencies and mode shapes.

Solution Results

The theoretical solution for this problem is presented in [1], pp. 401-403.

$$\text{Displacement:}\quad u(x,t) = \frac{2P_0L^3}{\pi^4 \cdot EI} \sum_{n=1}^{\infty} \frac{\alpha_n}{n^4}(1-\cos\omega_n t)\sin\frac{n\pi x}{L}$$

$$\text{where}\quad \alpha_n = \begin{cases} 1 & , \; n = 1,5,9,\ldots \\ -1 & , \; n = 3,7,11,\ldots \\ 0 & , \; n = \text{even numbers} \end{cases} \qquad \text{and} \quad \omega_n^2 = \frac{n^4\pi^4}{L^4}\frac{EI}{\rho A}$$

$$\text{Velocity:}\quad \dot{u}(x,t) = \frac{\partial u(x,t)}{\partial t} = \frac{2P_0L}{\pi^2\sqrt{EI\rho A}} \sum_{n=1}^{\infty} \frac{\alpha_n}{n^2}\sin\omega_n t \sin\frac{n\pi x}{L}$$

Acceleration: $\ddot{u}(x,t) = \frac{\partial^2 u(x,t)}{\partial t^2} = \frac{2P_0}{L\rho A} \sum_{n=1}^{\infty} \alpha_n \cos\omega_n t \sin\frac{n\pi x}{L}$

Bending moment: $M(x,t) = EI \cdot \frac{\partial^2 u(x,t)}{\partial x^2} = -\frac{2P_0 L}{\pi^2} \sum_{n=1}^{\infty} \frac{\alpha_n}{n^2}(1-\cos\omega_n t)\sin\frac{n\pi x}{L}$

Shear force: $V(x,t) = EI \cdot \frac{\partial^3 u(x,t)}{\partial x^3} = -\frac{2P_0}{\pi} \sum_{n=1}^{\infty} \frac{\alpha_n}{n}(1-\cos\omega_n t)\cos\frac{n\pi x}{L}$

Note that the acceleration series does not converge with increasing mode number n, which is due to the singular nature of the load. Therefore, the acceleration response is not treated further in this example. In practice, of course, the load is associated with a finite area and has a finite rise time.

The theoretical solution (using n = 12000) for the displacement, velocity and bending moment at the midspan of the beam and the shear force at the left support can be seen in the bottom figures on page A93.3. Note the high frequency content in the shear force response compared to the bending moment and the velocity response, which in turn contain higher frequencies than the displacement response. The theoretical solution curves are also shown for comparison in the subsequent figures.

The frequency range for the ten modes is between 20.0 and 1608 Hz. The time steps DT = T/25 and DT = T/10 are only selected to allow a comparison with the response obtained by the corresponding direct time integration method shown in Example A92. The modes with higher frequency are in this case excited initially to some degree although the time step is large compared to their periods. Normally, about ten time steps per period are used for the highest frequency mode to participate in the modal superposition.

The input for the analysis using 10 modes and DT=T/25 in the mode superposition procedure is shown on pages A93.6 and A93.7. The results can be seen on page A93.4 together with the results for DT = T/10. We may note the period elongation of approximately 10 percent for the displacement response when DT = T/10. Using 10 modes we get almost identical results as for the corresponding analysis in Example A92 using the trapezoidal rule.

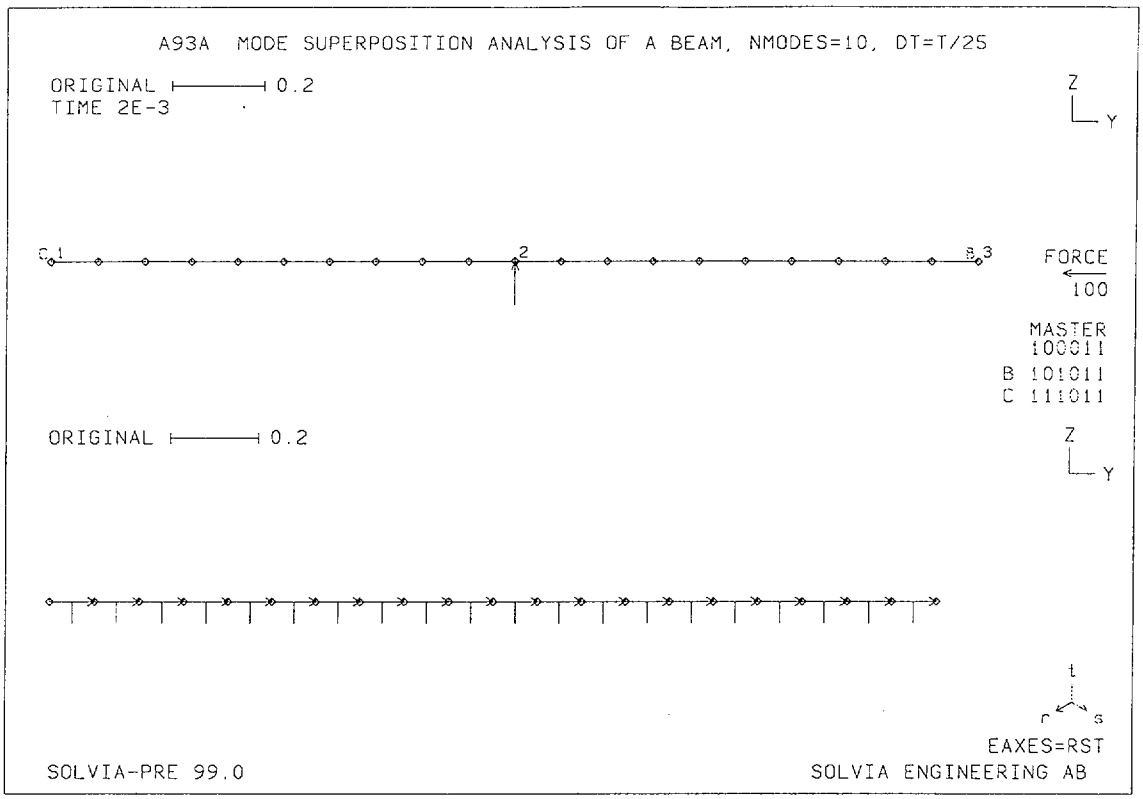
The solution results from the analysis using a single mode in the mode superposition procedure can be seen on page A93.5. The displacement response is almost the same as the 10 mode case but the velocity, the bending moment history and shear force history show somewhat different results since the higher frequency modes are not participating.

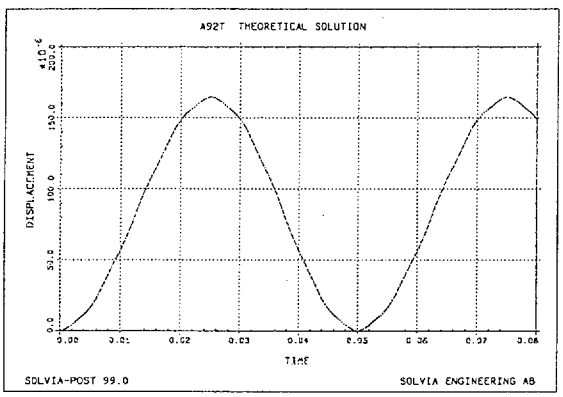
User Hints

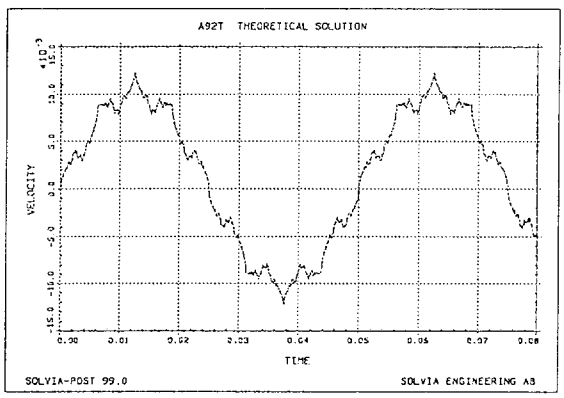
- Note that the higher modes contribute significantly to the bending moment response and even more significantly to the shear force response as compared with the displacement response. Therefore, when limiting the number of modes used in a mode superposition analysis one should consider the response quantity to be evaluated [1].
- A small enough time step is also important. In general, at least 10 time steps per period of the highest frequency mode to participate in the modal superposition are required for accuracy.

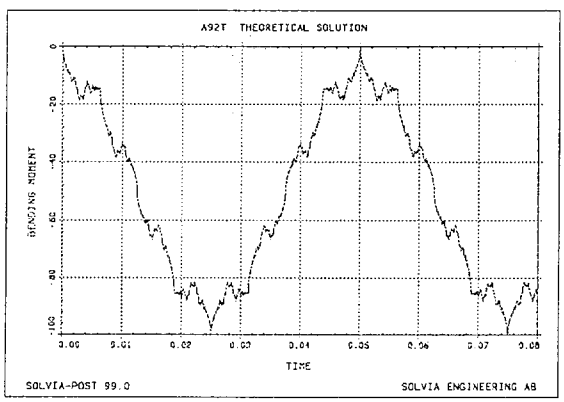
Reference

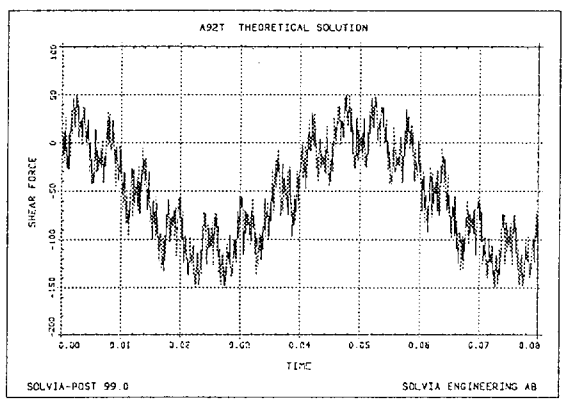
[1] Clough, R.W., and Penzien, J., Dynamics of Structures, Second Edition, McGraw-Hill, 1993.

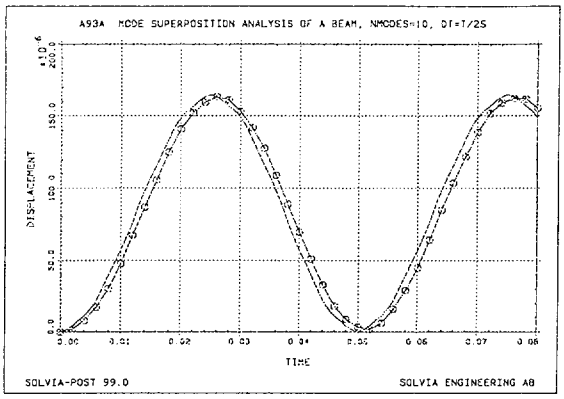

A93A MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=10, DT=T/25
ORIGINAL 0.2
TIME 2E-3
Z
Y
C 1
2
B 3
FORCE
100
MASTER
100011
B 101011
C 111011
ORIGINAL 0.2
Z
Y
t
r
s
EAXES=RST
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

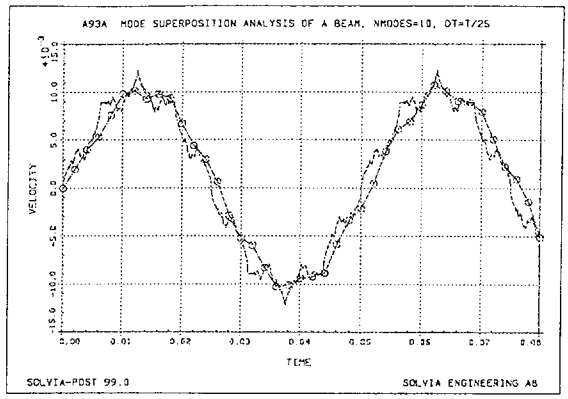

A92T THEORETICAL SOLUTION
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

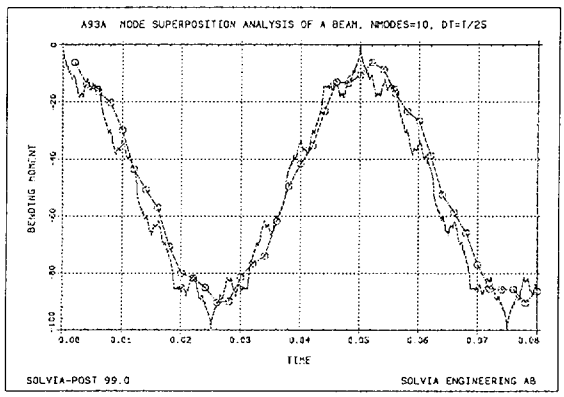

A92T THEORETICAL SOLUTION
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

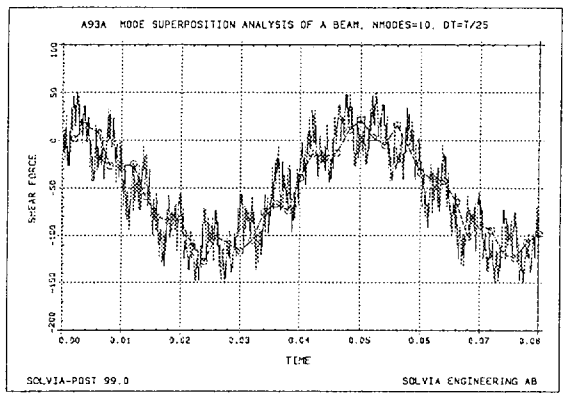

A92T THEORETICAL SOLUTION
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

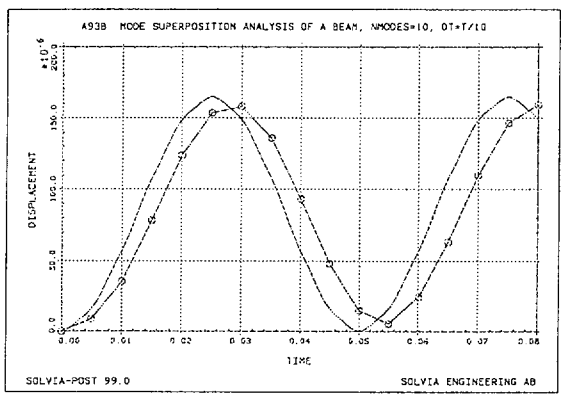

A92T THEORETICAL SOLUTION
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

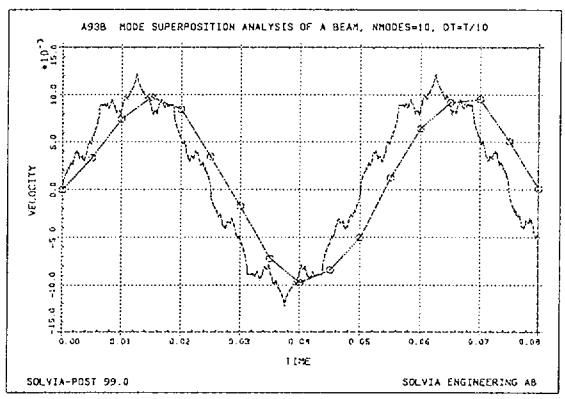

A93A MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=10, DT=T/25
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

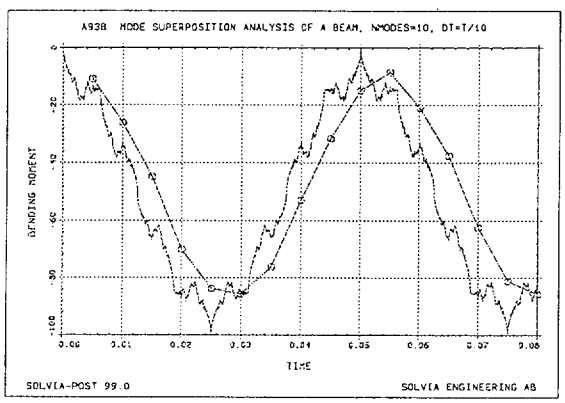

A93A MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=10, DT=T/25
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

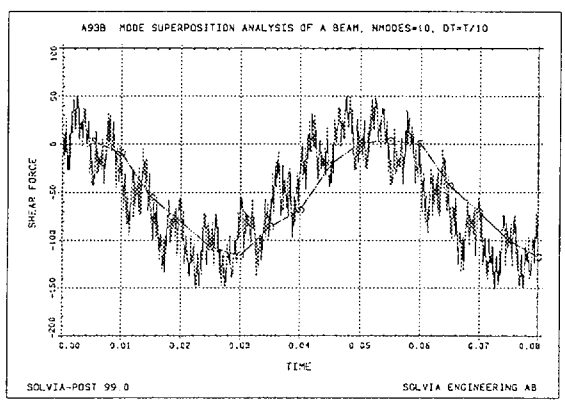

A93A MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=10, DT=T/25
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

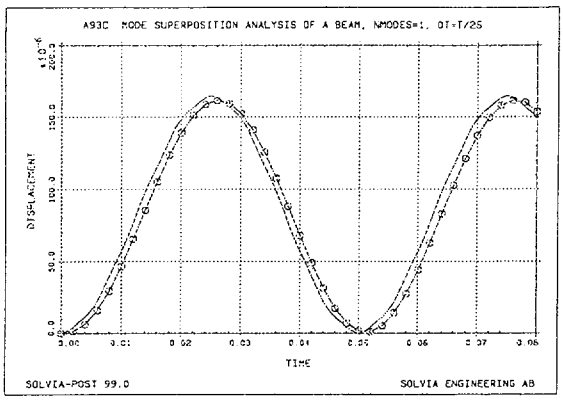

A93A MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=10, DT=T/25
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

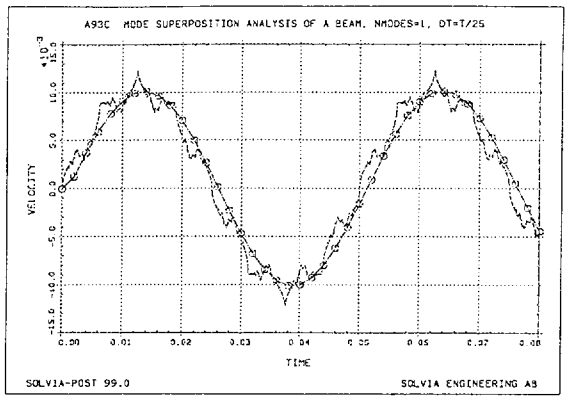

A93B MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=10, DT=T/10
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

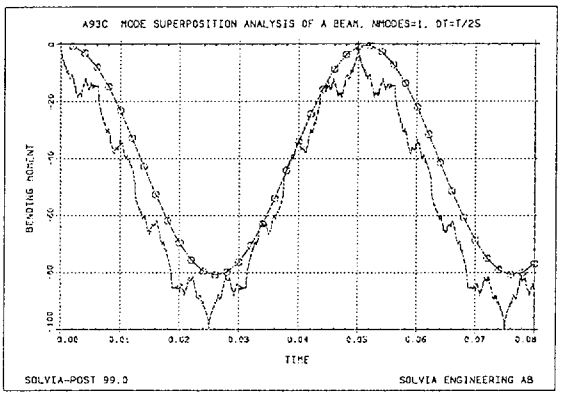

A93B MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=10, DT=T/10
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

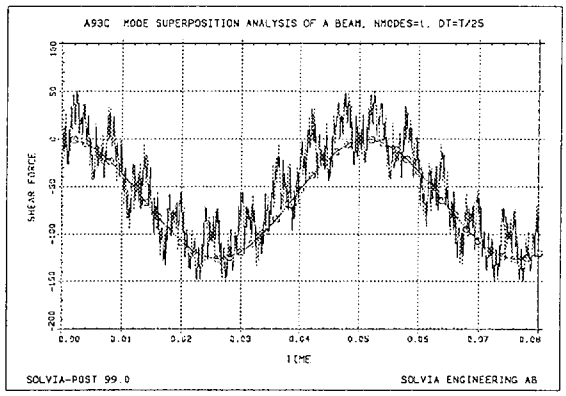

A93B MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=10, DT=T/10
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

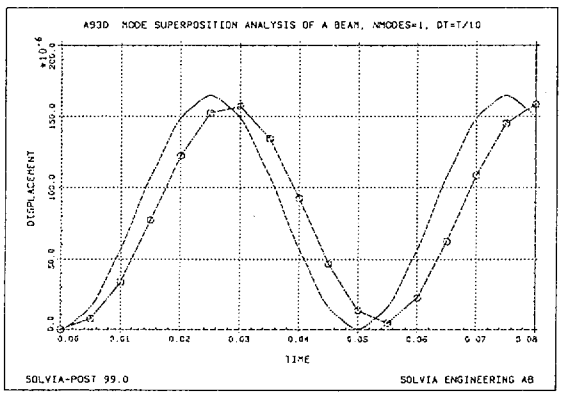

A93B MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=10, DT=T/10
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

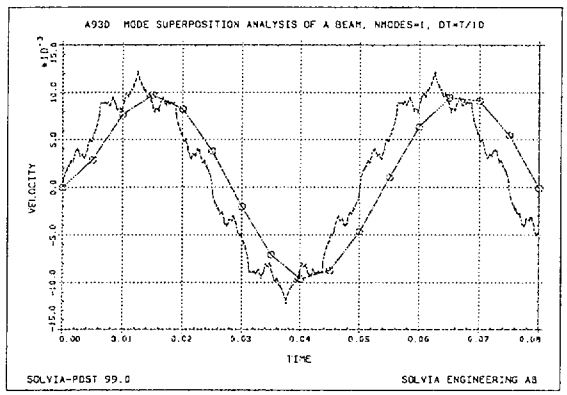

A93C MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=1, DT=T/25
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

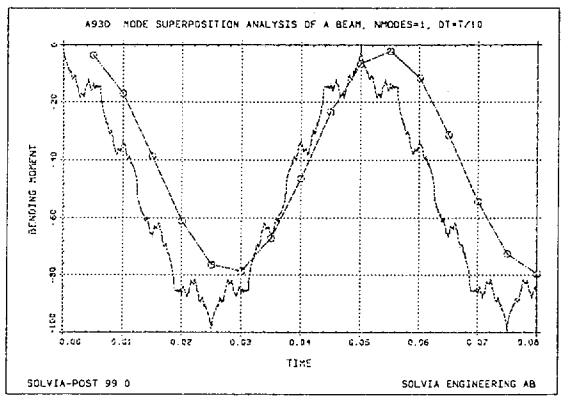

A93C MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=1, DT=T/25
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

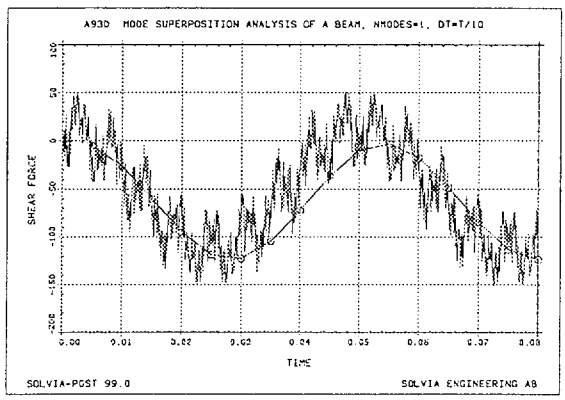

A93C MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=1, DT=T/25
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A93C MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=1, DT=T/25
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A93D MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=1, DT=T/10
DISPLACEMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A93D MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=1, DT=T/10
VELOCITY
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A93D MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=1, DT=T/10
BENDING MOMENT
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A93D MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=1, DT=T/10
SHEAR FORCE
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'A93A  MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=10,
DT=T/25'
*
DATABASE  CREATE
MASTER  IDOF=100011 NSTEP=40 DT=0.002
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=CONSISTENT IMODS=1 NMODES=10
FREQUENCIES  SUBSPACE-ITERATION  NEIG=10 SSTOL=1.E-10
*
TIMEFUNCTION  1
 0. 1.  /  1. 1.
*
COORDINATES
 1  /  2  0. 1.  /  3  0. 2.
*
MATERIAL  1  ELASTIC E=2.E11 NU=0. DENSITY=7800.
*
EGROUP  1  BEAM  RESULTS=FORCES
BEAMVECTOR
 1  1. 0. 0.
SECTION  1 GENERAL RINERTIA=1.E-6 SINERTIA=1.01159E-6,
                   TINERTIA=1.E-6 AREA=0.01
GLINE  1 3 -1  EL=20
*
FIXBOUNDARIES  23  /  1
FIXBOUNDARIES   3  /  3
LOADS  CONCENTRATED
 2 3 100.
*
SET  VIEW=X NSYMBOLS=YES
MESH  VECTOR=LOAD NNUMBERS=MYNODES BCODE=ALL SUBFRAME=12
MESH  EAXES=RST
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A93A  MODE SUPERPOSITION ANALYSIS OF A BEAM, NMODES=10, DT=T/25
*
DATABASE  CREATE
*
AXIS  1 VMIN=0.      VMAX=0.08  LABELSTRING='TIME'
AXIS  2 VMIN=0.      VMAX=2.E-4 LABELSTRING='DISPLACEMENT'
AXIS  3 VMIN=-0.015 VMAX=0.015 LABELSTRING='VELOCITY'
AXIS  4 VMIN=-100.  VMAX=0.    LABELSTRING='BENDING MOMENT'
AXIS  5 VMIN=-200.  VMAX=100.  LABELSTRING='SHEAR FORCE'
*
USERCURVE  1
READ A92DISP.DAT
USERCURVE  2
READ A92VEL.DAT
USERCURVE  3
READ A92MOM.DAT
USERCURVE  4
READ A92FORCE.DAT
*
SET  DIAGRAM=GRID
NHISTORY  NODE=2 DIRECTION=3 XAXIS=1 YAXIS=2 SYMBOL=1
PLOT  USERCURVE  1 XAXIS=-1 YAXIS=-2 SUBFRAME=OLD
*
NHISTORY  NODE=2 DIRECTION=3 KIND=VELOCITY XAXIS=1 YAXIS=3 SYMBOL=1
PLOT  USERCURVE  2 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD
*
EHISTORY  ELEMENT=10 POINT=2 KIND=MS XAXIS=1 YAXIS=4 SYMBOL=1
PLOT  USERCURVE  3 XAXIS=-1 YAXIS=-4 SUBFRAME=OLD
*
EHISTORY  ELEMENT=1 POINT=1 KIND=FT XAXIS=1 YAXIS=5 SYMBOL=1
PLOT  USERCURVE  4 XAXIS=-1 YAXIS=-5 SUBFRAME=OLD
END
```

EXAMPLE A94

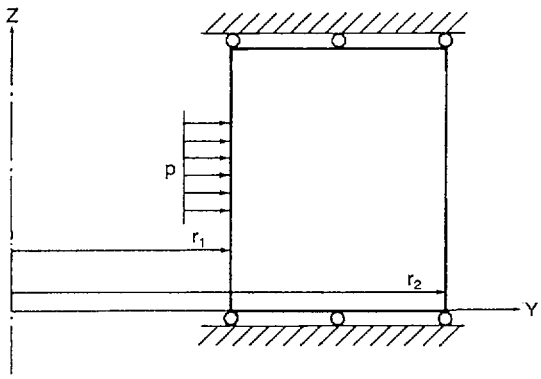
ORTHOTROPIC CYLINDER UNDER INTERNAL PRESSURE

Objective

To verify the PLANE AXISYMMETRIC element with pressure interpolation in a linear orthotropic analysis.

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. The material constant g is used to control the degree of orthotropy.



$E_a = 10^6 g \text{ N/mm}^2$ $\nu_{ab} = 0.25/g$

$E_b = 10^6 \text{ N/mm}^2$ $\nu_{ac} = 0.25/g$

$E_c = 10^6 \text{ N/mm}^2$ $\nu_{bc} = 0.25$

$G_{ab} = 4 \cdot 10^5 \text{ N/mm}^2$

$g = 0.16667$

$r_1 = 100 \text{ mm}$

$r_2 = 200 \text{ mm}$

$p = 10 \text{ N/mm}^2$

The orthotropic axes a, b and c correspond to the global axes Y (radial), Z (axial) and X (circumferential), respectively.

Finite Element Model

The cylinder is modeled using five 9-node PLANE AXISYMMETRIC elements with pressure interpolation included. The internal cylinder pressure is applied along the line between the nodes 2 and 3, see the figure on page A94.3.

Solution Results

The material constants are chosen to produce a semi-deformable material when g = 1/6.

When g = 1/6 the bulk modulus of the material becomes infinite and the requirement that the element pressure is finite means that the following sum of strains is zero:

$$\tfrac{1}{2} e_{aa} + e_{bb} + e_{cc} = 0$$

In our analysis we have used g = 0.16667 which gives $E_a = 166670 \text{ N/mm}^2$, $\nu_{ab} = \nu_{ac} = 1.49997$ and the bulk modulus $K = 2.31486 \cdot 10^{10} \text{ N/mm}^2$.

The analytical solution to this problem is [1]:

$$\sigma_{aa} = \frac{pc^{k+1}}{1-c^{2k}}\left(\frac{r}{r_2}\right)^{k-1} - \frac{pc^{k+1}}{1-c^{2k}}\left(\frac{r}{r_2}\right)^{-k-1}$$

$$\sigma_{cc} = \frac{pc^{k+1}}{1-c^{2k}}k\left(\frac{r}{r_2}\right)^{k-1} + \frac{pc^{k+1}}{1-c^{2k}}\left(\frac{r}{r_2}\right)^{-k-1}$$

$$\sigma_{bb} = \nu_{ab}\,\sigma_{aa} + \nu_{cb}\,\sigma_{cc}$$

$c = r_1 / r_2$, r is the radial coordinate and

$$k = \sqrt{\frac{(1-\nu_{ab}\cdot\nu_{ba})E_c}{(1-\nu_{cb}\cdot\nu_{bc})E_a}}$$

The column "Theory" in the tables below gives the analytical values for g = 1/6.

The input data used in the SOLVIA analysis can be seen on pages A94.5 and A94.6.

Stresses in N/mm^2 at nodes 2 and 1:

Y-coord. (mm)	σ_{aa}		σ_{bb}		σ_{cc}	
	SOLVIA	Theory	SOLVIA	Theory	SOLVIA	Theory
100	-9.7519	-10	-9.4300	-9.3333	22.5676	22.6667
200	0.014	0.	1.3283	1.3333	5.3281	5.3333

Using ten 4-node elements the following results are obtained:

Y-coord. (mm)	σ_{aa}		σ_{bb}		σ_{cc}	
	SOLVIA	Theory	SOLVIA	Theory	SOLVIA	Theory
100	-8.8169	-10	-9.7678	-9.3333	22.1573	22.6667
200	-0.1171	0.	1.3088	1.3333	5.290	5.3333

The element hydrostatic pressure values in N/mm^2 (negative mean stress) at nodes 2 and 1:

Y-coord. (mm)	9-node element SOLVIA	4-node element SOLVIA	Theory
100	-1.1286	-1.1909	-1.1111
200	-2.2235	-2.1613	-2.2222

The distribution of the orthotropic stresses as calculated by SOLVIA for the 9-node element analysis can be seen in the top figures on page A94.4 with symbols included. For comparison the theoretical solution curves are also included.

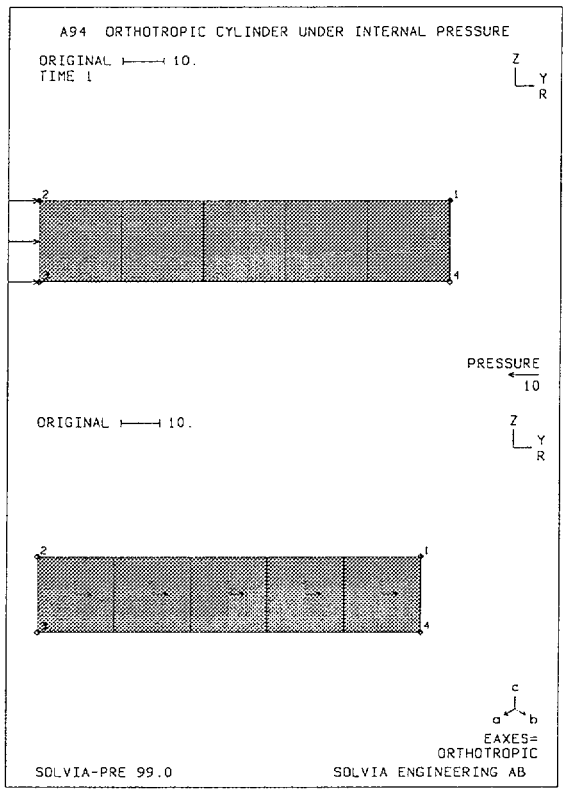
The solution results for the 4-node element analysis calculated by SOLVIA can be seen in the bottom figures on page A94.4.

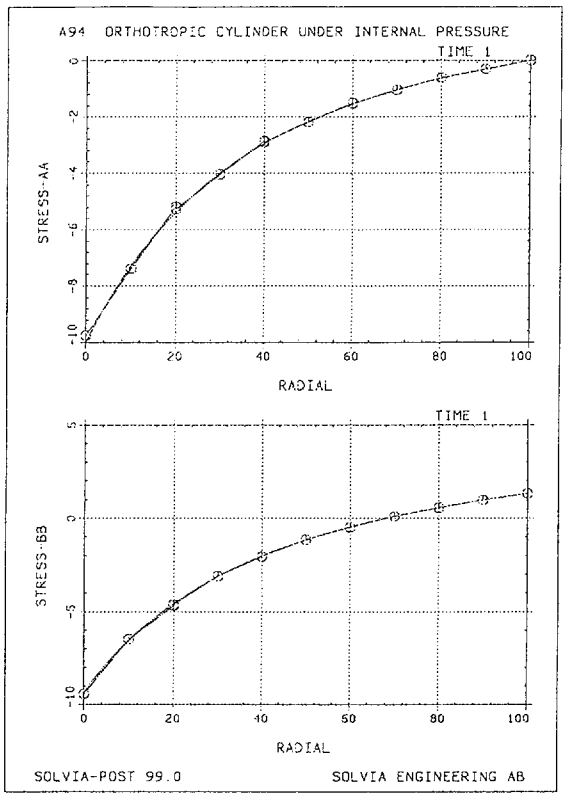
User Hints

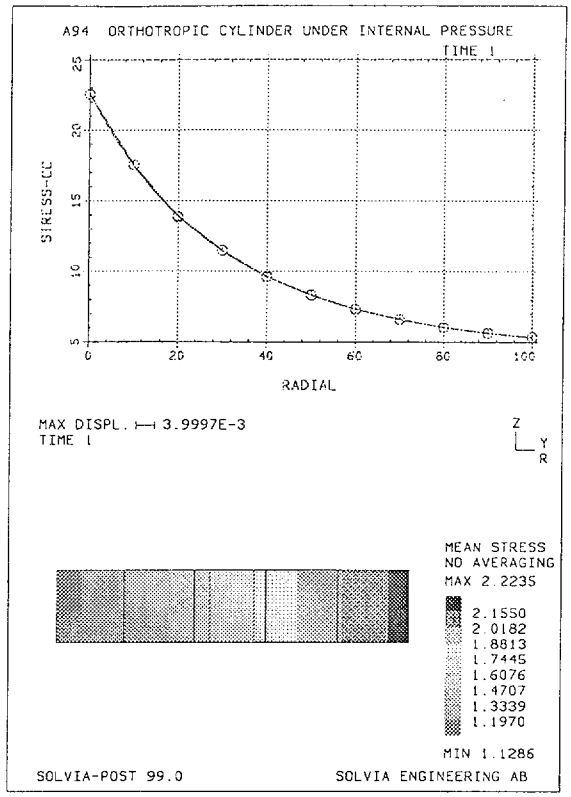
- Note that the 9-node PLANE element has a linear pressure interpolation assumption. The 4-node PLANE element has a constant pressure assumption.

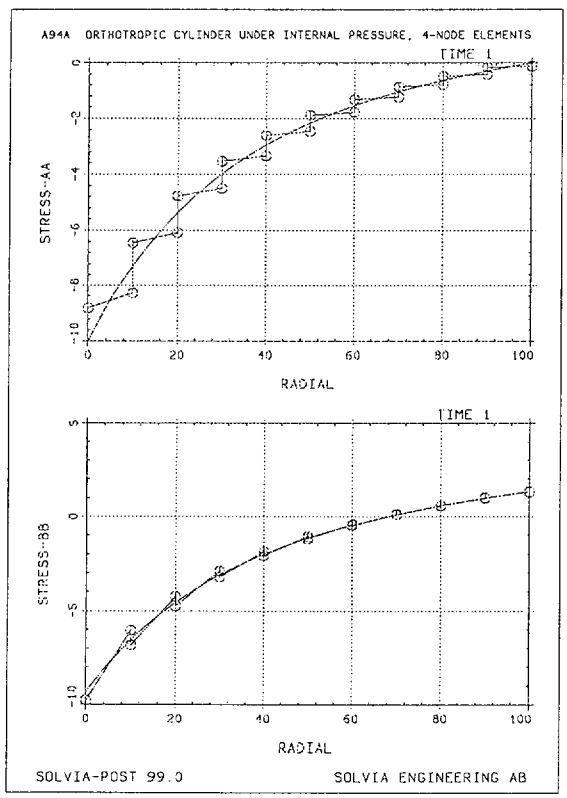
Reference

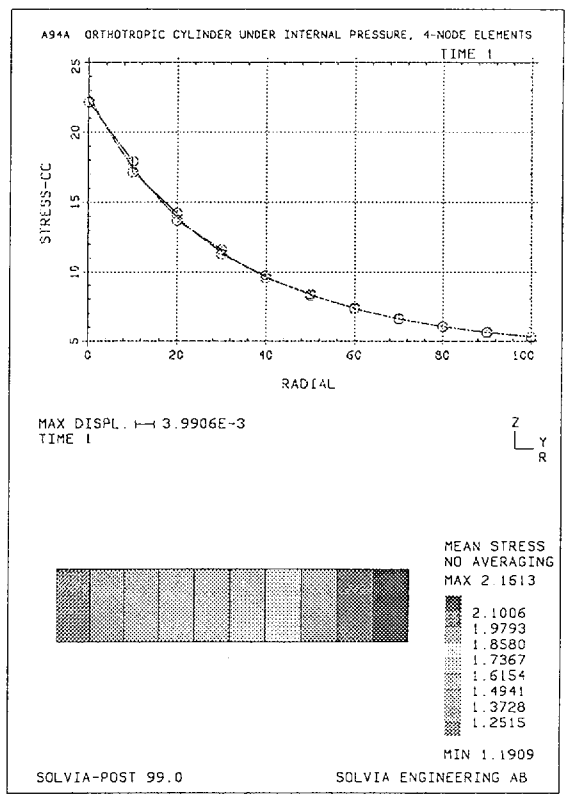
[1] Lekhnitski, S.G., "Theory of Elasticity of an Anisotropic Body", Mir, Moscow, 1977.




A94 ORTHOTROPIC CYLINDER UNDER INTERNAL PRESSURE
TIME 1
STRESS-AA
RADIAL
TIME 1
STRESS-BB
RADIAL
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A94 ORTHOTROPIC CYLINDER UNDER INTERNAL PRESSURE
TIME 1
STRESS-CC
RADIAL
MAX DISPL. 3.9997E-3
TIME 1
Z
Y
R
MEAN STRESS
NO AVERAGING
MAX 2.2235
2.1550
2.0182
1.8813
1.7445
1.6076
1.4707
1.3339
1.1970
MIN 1.1286
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A94A ORTHOTROPIC CYLINDER UNDER INTERNAL PRESSURE, 4-NODE ELEMENTS
TIME 1
STRESS-AA
RADIAL
TIME 1
STRESS-BB
RADIAL
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A94A ORTHOTROPIC CYLINDER UNDER INTERNAL PRESSURE, 4-NODE ELEMENTS
TIME 1
STRESS-CC
RADIAL
MAX DISPL. 3.9906E-3
TIME 1
Z
Y
R
MEAN STRESS
NO AVERAGING
MAX 2.1613
2.1006
1.9793
1.8580
1.7367
1.6154
1.4941
1.3728
1.2515
MIN 1.1909
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A94  ORTHOTROPIC CYLINDER UNDER INTERNAL PRESSURE'
*
DATABASE  CREATE
*
MASTER  IDOF=100111
*
COORDINATES  /  ENTRIES  NODE  Y  Z
 1  200. 20.  /  2  100.  20.  /  3  100.  /  4  200.
*
MATERIAL  1  ORTHOTROPIC  EA=166670. EB=1.E6 EC=1.E6 NUAB=1.49997,
                          NUAC=1.49997 NUBC=0.25 GAB=4.E5
*
EGROUP  1  PLANE AXISYMMETRIC  RESULTS=NSTRESSES,
                 PRESSURE-INTERPOLATION=YES
GSURFACE  3 4 1 2  EL1=5 EL2=1 NODES=9
LOADS  ELEMENT  INPUT=LINE
 2 3 10. 10.
*
FIXBOUNDARIES  3 INPUT=LINE  /  1 2  /  3 4
*
SET  NSYMBOLS=MYNODES NNUMBERS=MYNODES PLOTORIENTATION=PORTRAIT
*
MESH  VECTOR=LOAD SUBFRAME=12
MESH  EAXES=ORTHOTROPIC
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A94  ORTHOTROPIC CYLINDER UNDER INTERNAL PRESSURE
*
DATABASE  CREATE  STRESSREFERENCE=ORTHOTROPIC
WRITE  FILENAME='a94.lis'
*
AXIS  1  VMIN=0.   VMAX=100. LABEL='RADIAL'
AXIS  2  VMIN=-10. VMAX=0.   LABEL='STRESS-AA'
AXIS  3  VMIN=-10. VMAX=5.   LABEL='STRESS-BB'
AXIS  4  VMIN=5.   VMAX=25.  LABEL='STRESS-CC'
*
EPLINE  NAME=RADIAL  /  1 4 7 3  TO  5 4 7 3
*
USERCURVE  1
READ  A94SAA.DAT
USERCURVE  2
READ  A94SBB.DAT
USERCURVE  3
READ  A94SCC.DAT
*
SET  PLOTORIENTATION=PORTRAIT DIAGRAM=GRID
SUBFRAME  12
ELINE  LINENAME=RADIAL KIND=SAA XAXIS=1 YAXIS=2 SYMBOL=1 OUTPUT=ALL
PLOT  USERCURVE  1 XAXIS=-1 YAXIS=-2 SUBFRAME=OLD
*
ELINE  LINENAME=RADIAL KIND=SBB XAXIS=1 YAXIS=3 SYMBOL=1 OUTPUT=ALL
PLOT  USERCURVE  2 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD
*
SUBFRAME  12
ELINE  LINENAME=RADIAL KIND=SCC XAXIS=1 YAXIS=4 SYMBOL=1 OUTPUT=ALL
PLOT  USERCURVE  3 XAXIS=-1 YAXIS=-4 SUBFRAME=OLD
*
CONTOUR  AVERAGE=NO
MESH  CONTOUR=SMEAN
*
ELIST  SELECT=S-EFFECTIVE
END
```

EXAMPLE A95

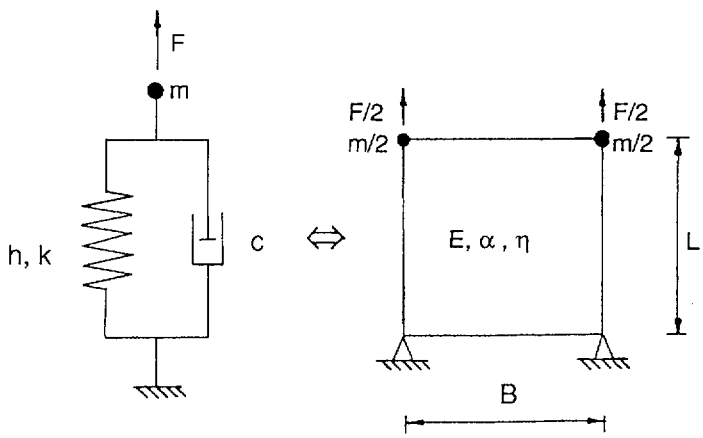
MATERIAL DAMPING IN COMPLEX-HARMONIC ANALYSIS

Objective

To verify the PLANE, SOLID and SHELL elements with viscous and hysteretic material damping in a COMPLEX-HARMONIC analysis.

Physical Problem

A sinusoidal force is applied to a damped one degree-of-freedom system.



Thickness t = 1
L = 1 m
B = 1 m
m = 1 kg
F = 1 N

Stiffness

k = 100 N/m

Elastic material

$$E = \frac{Lk}{Bt} = 100 \ \text{N/m}^2$$

$\nu = 0.$

Viscous damping

c = 5 Ns/m

Viscous damping coefficient

$$\alpha = \frac{L}{EBt} \cdot c = 0.05 \ \text{s}$$

Hysteretic damping

h = 30 N/m

Hysteretic damping coefficient

$$\eta = \frac{L}{EBt} \cdot h = 0.3$$

Finite Element Model

One element of each type is used and shown on page A95.3.

Solution Results

The theoretical displacement amplitude in complex form at the mass point of the spring/damper system is

$$\underline{u} = \frac{\underline{R}}{\left(-\omega^2 m + i\omega C + k\right)}$$

where $\underline{R}$ load amplitude in complex form

ω excitation circular frequency

$$C = c + \frac{h}{\omega}$$

The velocity amplitude in complex form at the mass point is

$$\dot{\underline{u}} = i\omega\,\underline{u}_z$$

The displacement and velocity at the midpoint of the spring/damper system are

$$\underline{u}_{mid} = \underline{u}/2$$
$$\dot{\underline{u}}_{mid} = \dot{\underline{u}}/2$$

The forces in the spring and the damper are

$$\underline{F}_{spring} = k\underline{u}$$
$$\underline{F}_{damp} = C\dot{\underline{u}}$$

The time average of the power flow in the spring/damper system at mid point is

$$\underline{S}_r = -\frac{1}{2}\left(\underline{F}^*_{spring} + \underline{F}^*_{damp}\right)\dot{\underline{u}}_{mid}$$

where $\underline{F}^*_{spring}$ and $\underline{F}^*_{damp}$ are complex conjugate of the corresponding element forces.

The time average of the power flow into the spring/damper system is

$$\underline{S}_{in} = \frac{1}{2}\,\underline{F}^*\underline{u}$$

where $\underline{F} = \underline{F}^* = F$ since a sinusoidal force is applied with zero phase angle.

The time average of the strain energy in the spring/damper system is

$$U = \frac{1}{4}\underline{u}^* k\,\underline{u}$$

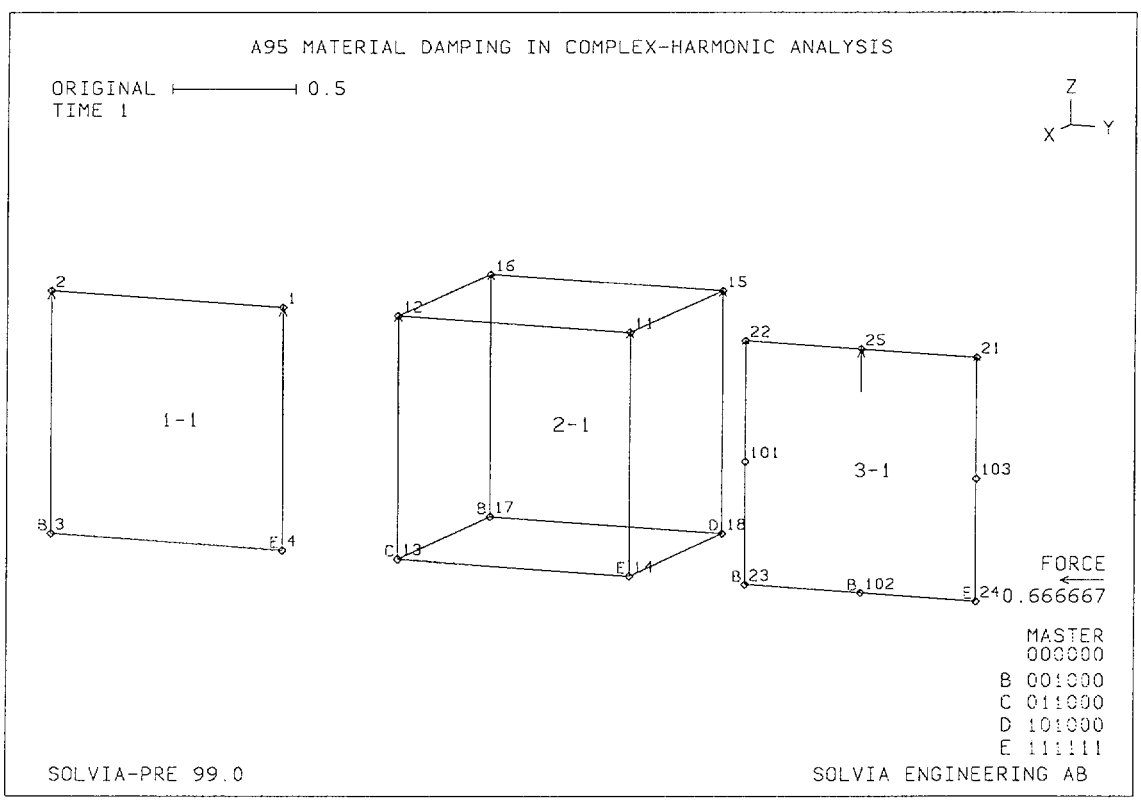
The time average of the dissipated energy in the spring/damper system is

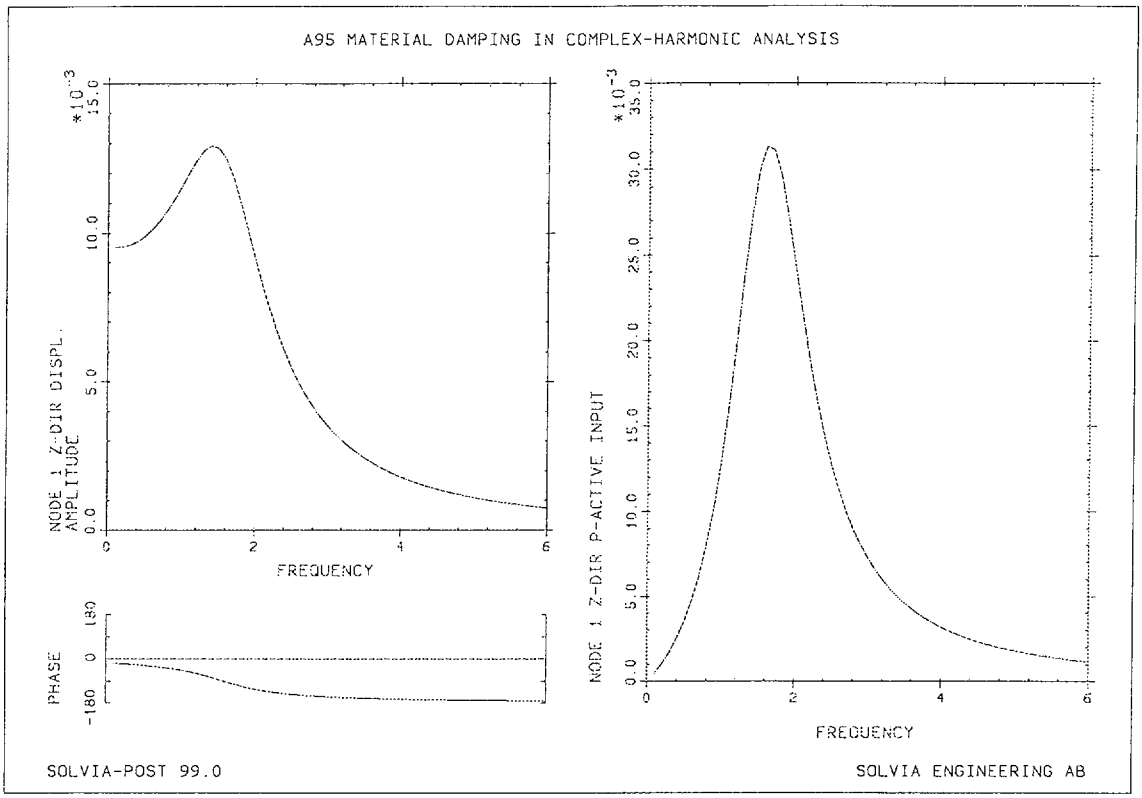
$$\Pi = \frac{1}{2}\dot{\underline{u}}^* C\dot{\underline{u}}$$

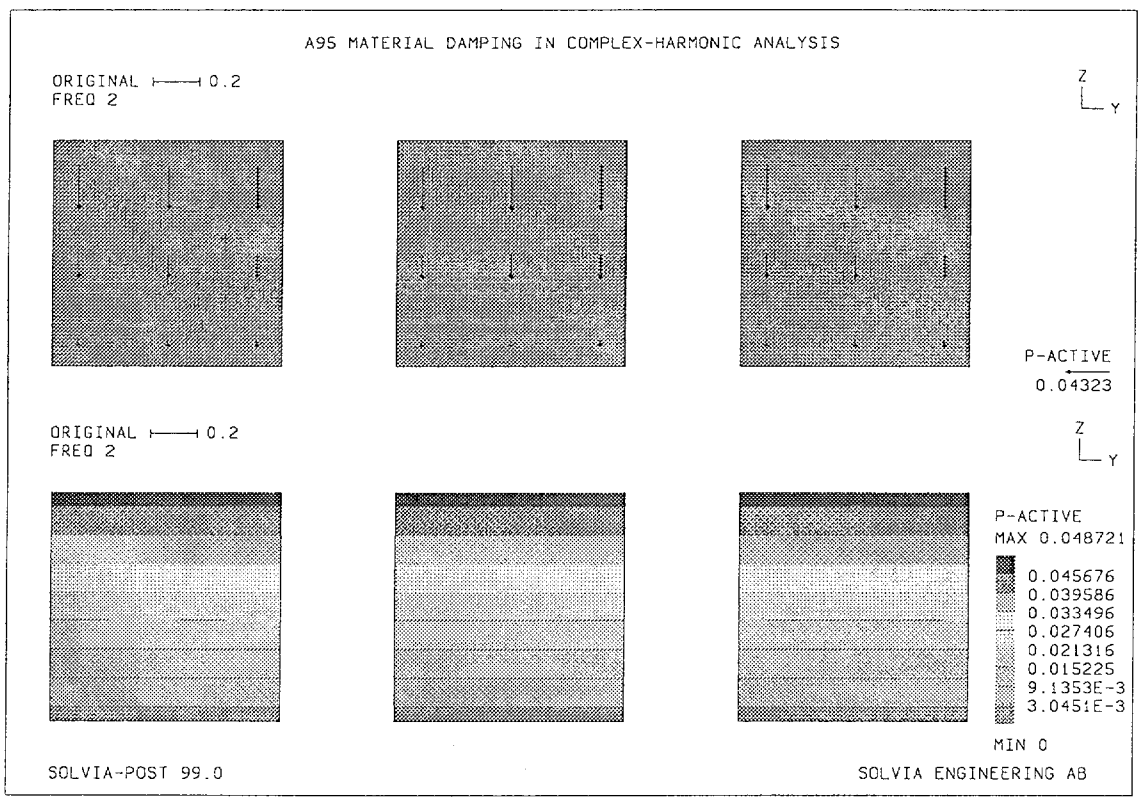
The numerical solution of time averages obtained at the excitation frequency 2 Hz using the input data on pages A95.4 and A95.5 gives the following results:

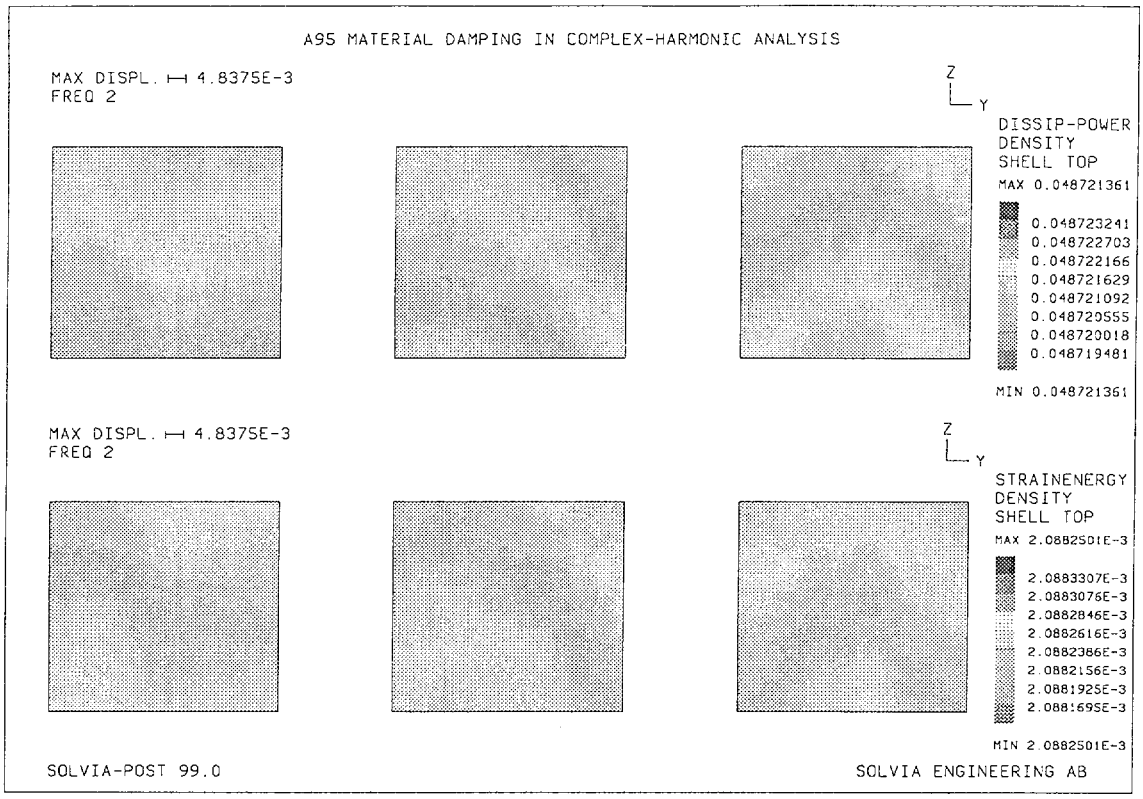
	Theory	PLANE element	SOLID element	SHELL element
Active power flow into the structure, $\mathrm{Re}(\underline{S}_{in})$, in Nm/s Contribution from all loaded nodes of the element	$4.872 \cdot 10^{-2}$	$4.872 \cdot 10^{-2}$	$4.872 \cdot 10^{-2}$	$4.872 \cdot 10^{-2}$
Active power flow in the structure, $\mathrm{Re}(\underline{S}_r)$, in Nm/(m^2,s), mid point	$2.436 \cdot 10^{-2}$	$2.436 \cdot 10^{-2}$	$2.436 \cdot 10^{-2}$	$2.436 \cdot 10^{-2}$
Total dissipated power, Π, in Nm/s	$4.872 \cdot 10^{-2}$	$4.872 \cdot 10^{-2}$	$4.872 \cdot 10^{-2}$	$4.872 \cdot 10^{-2}$
Total strain energy, U, in Nm	$2.088 \cdot 10^{-3}$	$2.088 \cdot 10^{-3}$	$2.088 \cdot 10^{-3}$	$2.088 \cdot 10^{-3}$

Note that the input active power into the structure is equal to the dissipated power in the material damping. The active power flow at 2 Hz is shown as vector and contour plots on page A95.4. The dissipated power density and strain energy density are constant over the elements, see contour plots on page A95.5.




A95 MATERIAL DAMPING IN COMPLEX-HARMONIC ANALYSIS
NODE 1 Z-DIR DISPL. AMPLITUDE
*10^-3
15.0
10.0
5.0
0.0
0
2
4
6
FREQUENCY
PHASE
180
0
-180
NODE 1 Z-DIR P-ACTIVE INPUT
*10^-3
35.0
30.0
25.0
20.0
15.0
10.0
5.0
0.0
FREQUENCY
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A95 MATERIAL DAMPING IN COMPLEX-HARMONIC ANALYSIS
ORIGINAL 0.2
FREQ 2
Z
Y
P-ACTIVE
0.04323
ORIGINAL 0.2
FREQ 2
Z
Y
P-ACTIVE
MAX 0.048721
0.045676
0.039586
0.033496
0.027406
0.021316
0.015225
9.1353E-3
3.0451E-3
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB



SOLVIA-PRE input

```
DATABASE CREATE
*
HEAD 'A95 MATERIAL DAMPING IN COMPLEX-HARMONIC ANALYSIS'
COMPLEX-HARMONIC
0.1 59 6.0
TIMEFUNCTION 1
0 1 / 500 1
*
SYSTEM 1 Y=1.5
SYSTEM 2 Y=3.0
COORDINTES SYSTEM=0
ENTRIES NODE Y Z
 1  1. 1. / 2  .0 1. / 3  .0 .0 / 4  1. .0
COORDINTES SYSTEM=1
11  .0 1. 1. /12  .0 .0 1. /13  .0 .0 .0 /14  .0 1. .0
15 -1. 1. 1. /16 -1.  0 1. /17 -1. .0 .0 /18 -1. 1. .0
COORDINTES SYSTEM=2
ENTRIES NODE Y Z
21  1. 1. /22   0  1 / 23  .0 .0 /24  1. 0. / 25 .5 1.
*
MASSES
 1 0.50 /   2 0.50
11 0.25 /  12 0.25 /  15 0.25 /  16 0.25
21 0.166666666 /  22 0.166666666 /  25 0.666666666
*
MATERIAL 1 ELASTIC E=100 DENSITY=0 ,
  VISCOUS-DAMPING=0.05 HYSTERETIC-DAMPING=0.3
*
EGROUP 1 PLANE STRESS2 INT=3
```

SOLVIA-PRE input (cont.)

```
GSURFACE  2 3 4 1 EL1=1 EL2=1 NODES=4
EDATA / 1 1.
*
EGROUP 2 SOLID TINT=3 RSINT=3
GVOLUME 11 12 13 14 15 16 17 18 EL1=1 EL2=1 NODES=8
*
EGROUP 3 SHELL RINT=3 SINT=3
GSURFACE  22 23 24 21 EL1=1 EL2=1 NODES=8
THICK 1 1.
*
FIXBOUNDARIES 3 INPUT=LINE/ 3 4 / 23 24
FIXBOUNDARIES 3 INPUT=SURFACE/ 13 14 18 17
FIXBOUNDARIES  / 4 14 24
FIXBOUNDARIES 1 / 18
FIXBOUNDARIES 2 / 13
*
LOAD CONC
 1 3 0.50 /  2 3 0.50
11 3 0.25 / 12 3 0.25 / 15 3 0.25 / 16 3 0.25
21 3 0.166666666 / 22 3 0.166666666 / 25 3 0.666666666
*
SOLVIA
COLOR LINE
VIEW ID=C XVIEW=5 YVIEW=2 ZVIEW=1
SET HIDDEN=N0 NSYMBOL=YES NNUMBER=YES ENUMBER=GROUP
MESH VIEW=C VECTOR=LOAD BCODE=ALL
END
```

SOLVIA-POST input

```
* A95 MATERIAL DAMPING IN COMPLEX-HARMONIC ANALYSIS
*
DATABASE CREATE
WRITE 'a95.lis'
*
SET RESPONSETYPE=REAL HEIGHT=0.25 VIEW=X
NCOMPLEX-HARMONIC NODE=1 DIRECTION=3 KIND=DISPLACEMENT SUBFRAME=21
NCOMPLEX-HARMONIC NODE=1 DIRECTION=3 KIND=P-ACTIVE OUTPUT=ALL
*
SET TIME=2
MESH VECTOR=P-ACTIVE SUBFRAME=12
MESH CONTOUR=P-ACTIVE
MESH CONTOUR=DISSIPATED-POWER SUBFRAME=12
MESH CONTOUR=ENERGY
*
ELIST SELECT=POWER  TSTART=2
SUMMATION KIND=ENERGY DETAILS=YES TSTART=2
NCOMPLEX-HARMONIC NODE=2  DIRE=3 KIND=P-ACTIVE OUTPUT=LIST FEND=2
NCOMPLEX-HARMONIC NODE=11 DIRE=3 KIND=P-ACTIVE OUTPUT=LIST FEND=2
NCOMPLEX-HARMONIC NODE=12 DIRE=3 KIND=P-ACTIVE OUTPUT=LIST FEND=2
NCOMPLEX-HARMONIC NODE=15 DIRE=3 KIND=P-ACTIVE OUTPUT=LIST FEND=2
NCOMPLEX-HARMONIC NODE=16 DIRE=3 KIND=P-ACTIVE OUTPUT=LIST FEND=2
NCOMPLEX-HARMONIC NODE=21 DIRE=3 KIND=P-ACTIVE OUTPUT=LIST FEND=2
NCOMPLEX-HARMONIC NODE=22 DIRE=3 KIND=P-ACTIVE OUTPUT=LIST FEND=2
NCOMPLEX-HARMONIC NODE=25 DIRE=3 KIND=P-ACTIVE OUTPUT=LIST FEND=2
END
```

EXAMPLE A96

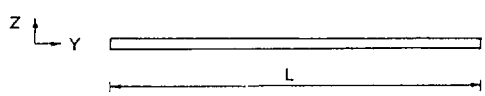
FREQUENCIES OF A BEAM WITH U CROSS-SECTION

Objective

To verify the dynamic behaviour of the BEAM element when the shear center and the center of gravity are not coinciding.

Physical Problem

The natural frequencies and mode shapes of a free-free beam with a U cross-section as shown in the figure below are to be calculated.



$L = 5$ m

$WTOP = 0.1$ m

$D = 0.1$ m

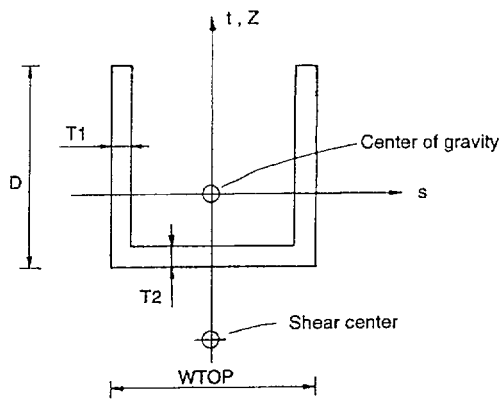
$T1 = 0.01$ m

$T2 = 0.01$ mm

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

$\nu = 0.3$

$\rho = 7800$ kg/m^3



Finite Element Model

The finite element model consists of twenty BEAM elements, see figure on page A96.3. A consistent mass matrix is used. Rigid links are attached to the beam in the transverse direction to make it easier to show the torsion modes. A SHELL element model of the beam is used as a reference solution.

Solution Results

The input data for the BEAM element model is shown on page A96.6 and the input data for the SHELL element model is shown on page A96.7.

Calculated natural frequencies, Hz:

Mode	BEAM model	SHELL model
7	23.5	23.4
8	25.2	25.4
9	35.5	32.7
10	53.2	55.8
11	64.4	64.2
12	71.2	75.9
13	90.6	92.9
14	97.8	123.4

The first six modes are rigid body modes with zero frequency since the beam has no support.

Mode 7 and 11 are pure bending modes in the plane X = 0. The pure bending modes are possible since both the shear center and the center of gravity have zero X-coordinates. The BEAM and the SHELL element models give almost the same frequencies from the pure bending modes.

The remaining of the listed and displayed modes are coupled modes with both bending and torsion. A pure torsion mode is not possible since the shear center and the center of gravity do not coincide.

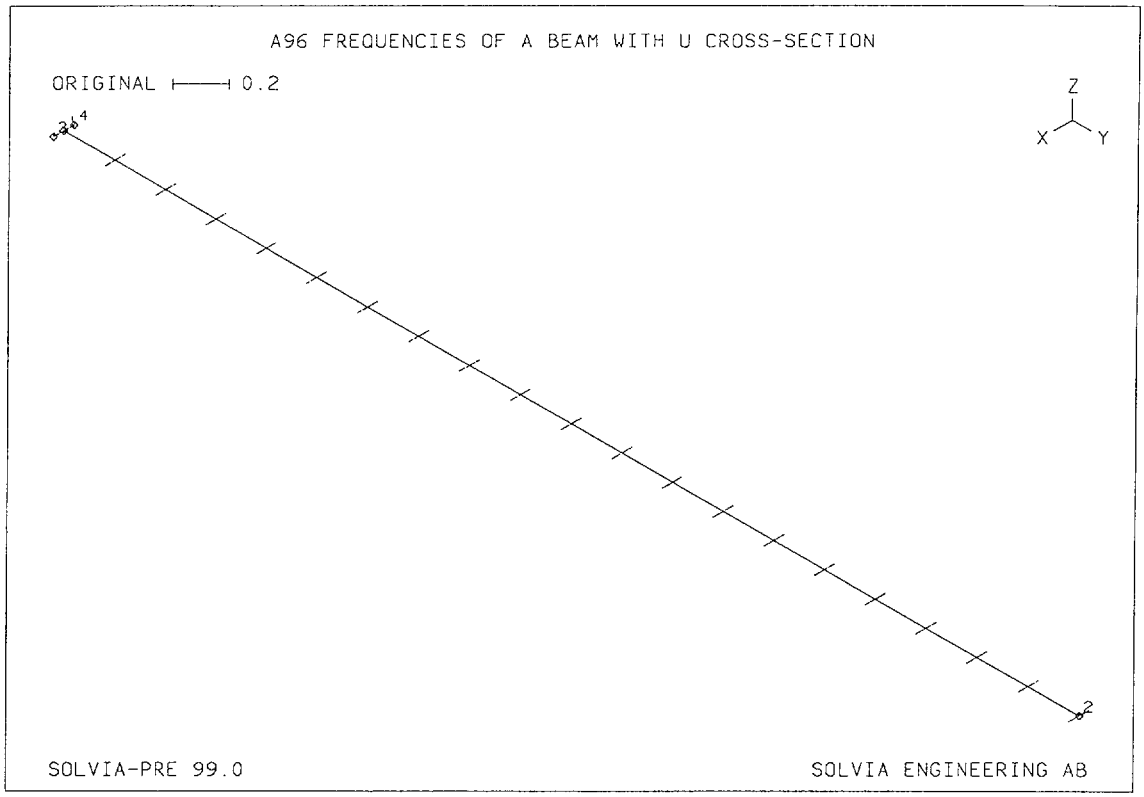
The frequencies of the BEAM and the SHELL element models agree reasonably well up to mode 13. The torsional half wave length in mode 14 is, however, only about 1 meter. The warping of the cross-section becomes then significantly constrained which increases the stiffness. This effect is modelled by the SHELL element but not by the BEAM element.

The torsional stiffness of a BEAM element with U cross-section is calculated based on Saint Venant's theory of torsion for thin rectangular areas. Any warping of the cross-section is unrestrained.

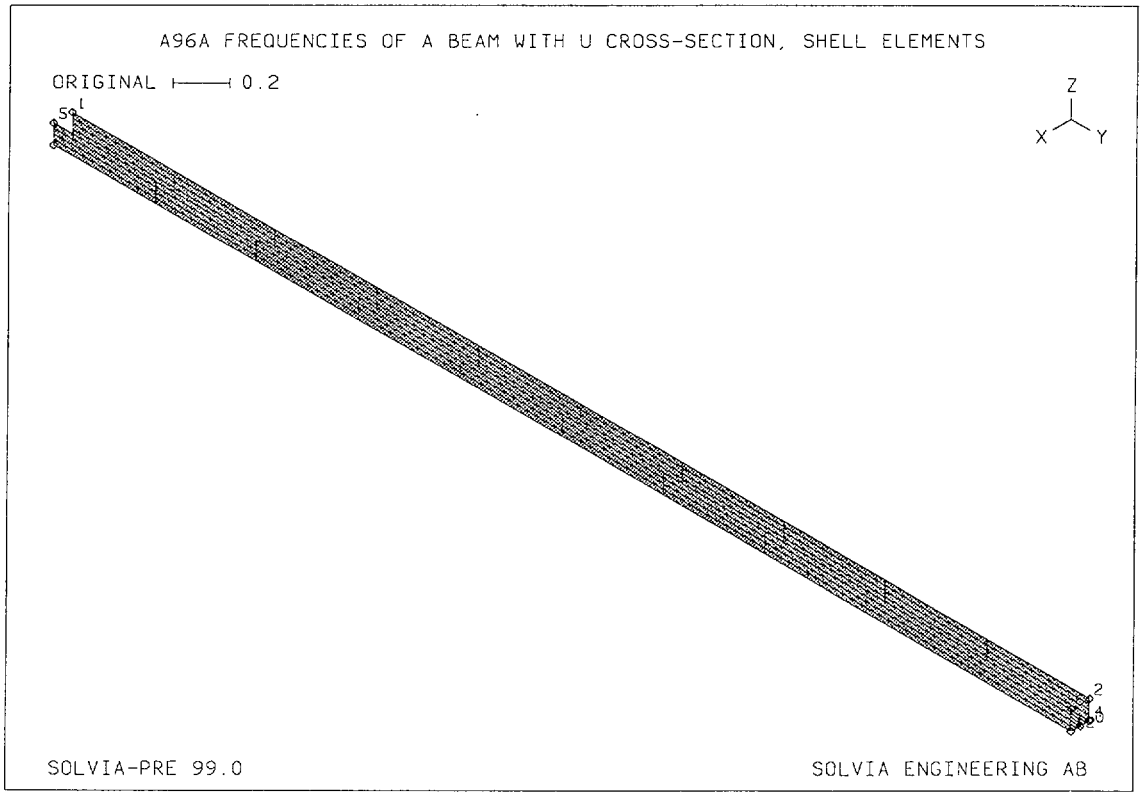
User Hints

- In the command FREQUENCIES SUBSPACE-ITERATION the parameter RIGIDMODES = YES is necessary when rigid body modes are present and parameter FSHIFT = $-f_1$ improves the convergence rate where f_1 is an approximation of the first non-zero natural frequency in Hz.

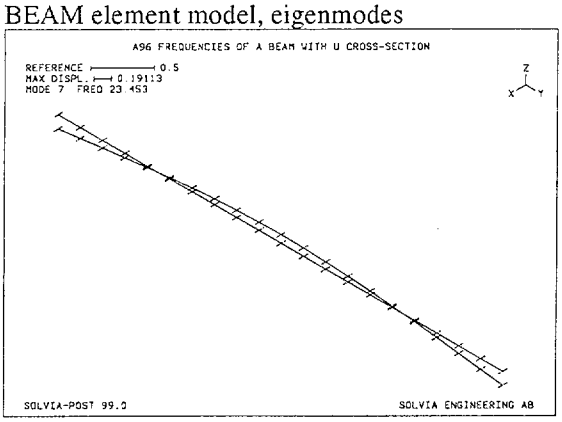
BEAM element model

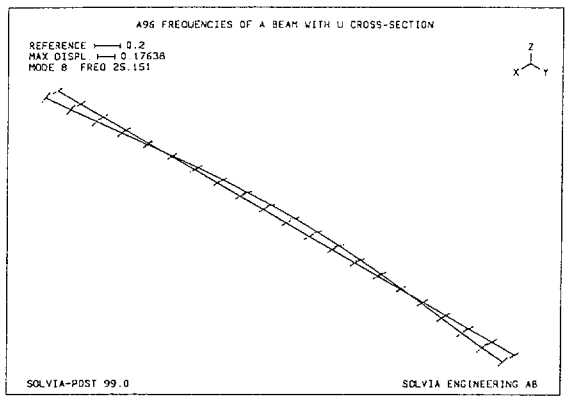


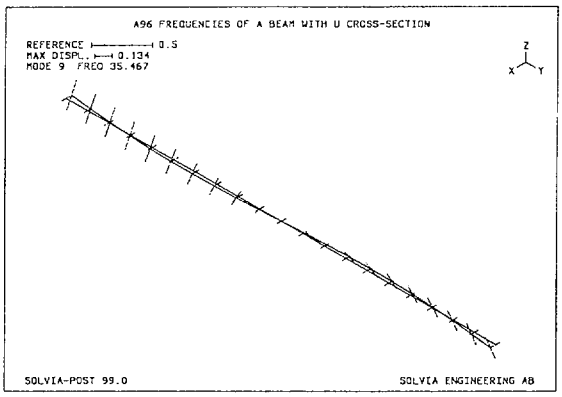
SHELL element model

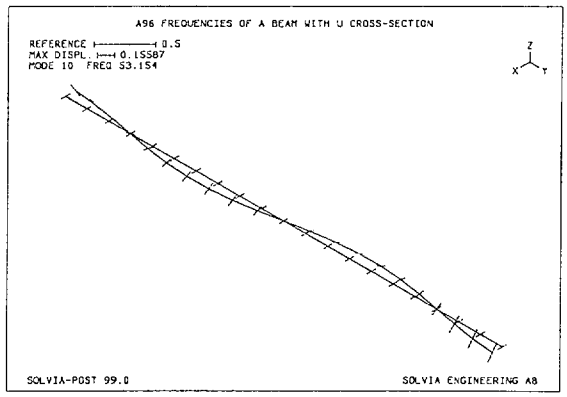


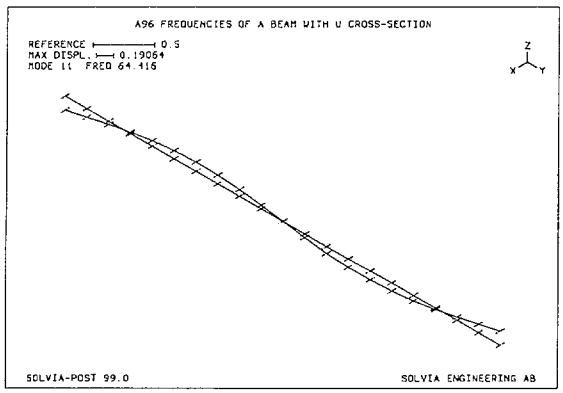
BEAM element model, eigenmodes

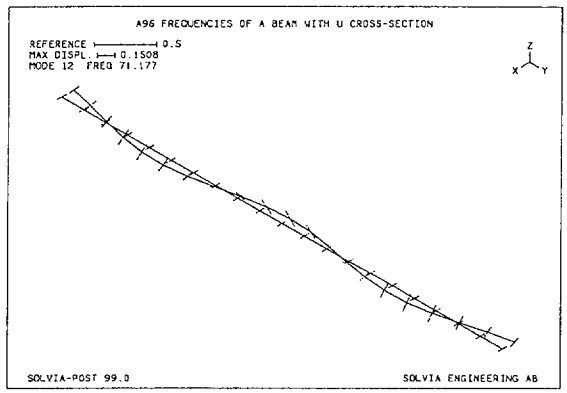


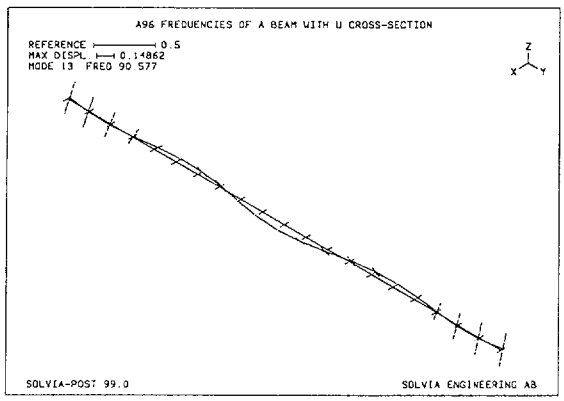


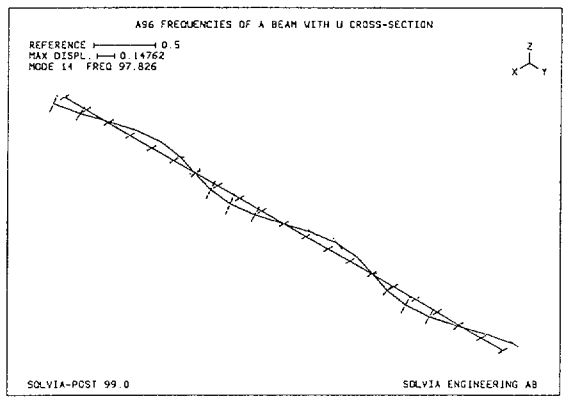




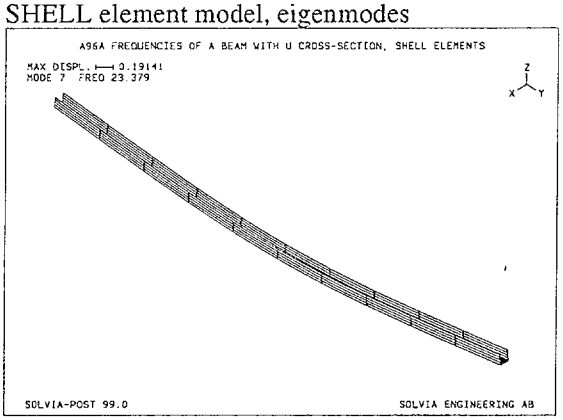


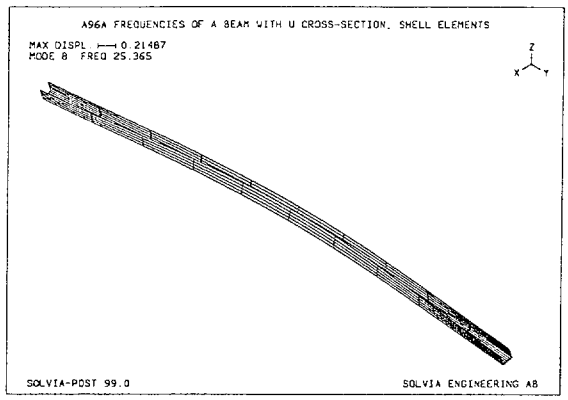


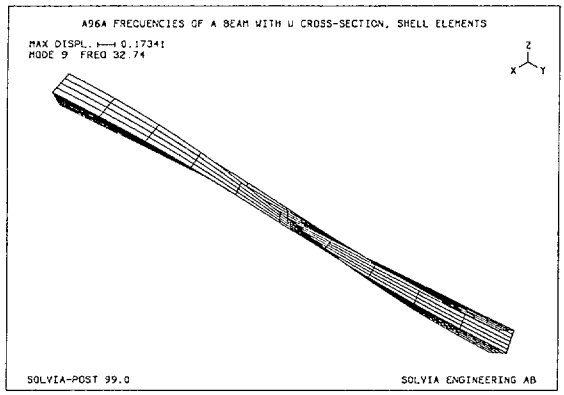


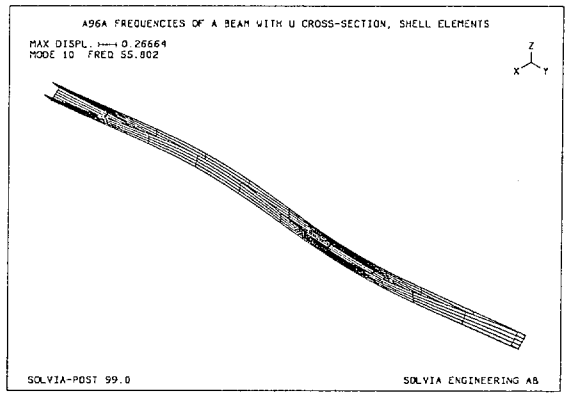


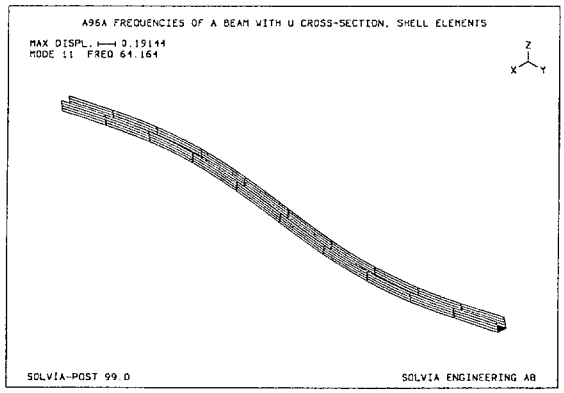
SHELL element model, eigenmodes

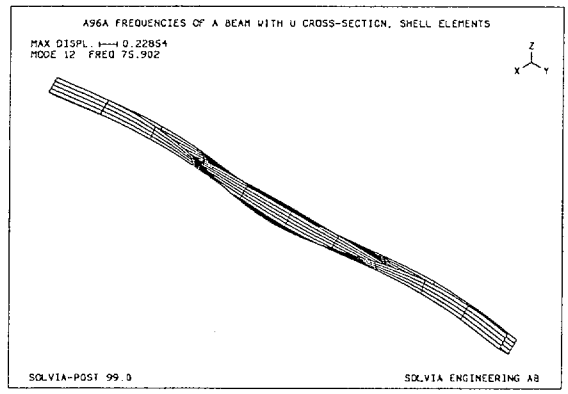


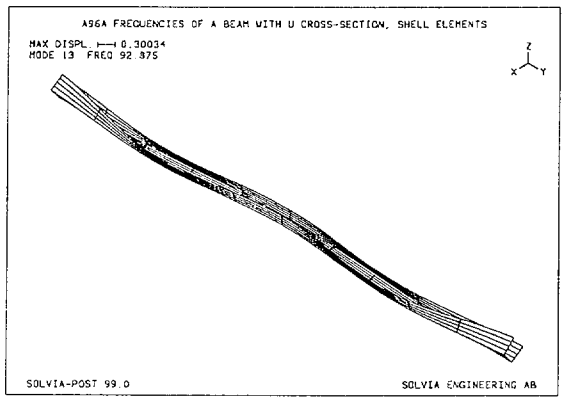


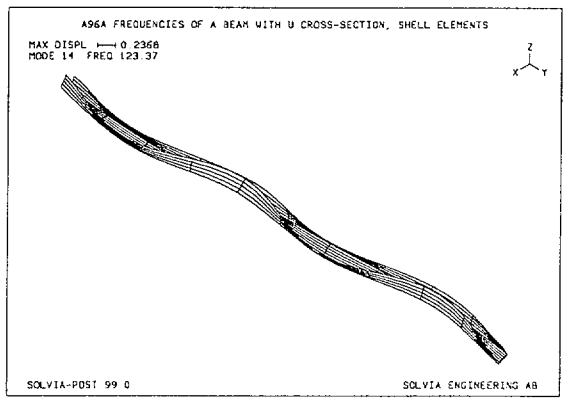












SOLVIA-PRE input, BEAM element model

```
HEADING  'A96 FREQUENCIES OF A BEAM WITH U CROSS-SECTION'
*
DATABASE  CREATE
*
FREQUENCIES SUBSPACE-ITERATION NEIG=15 SSTOL=1E-10 ,
 FSHIFT=-20 NQ=40 RIGIDMODES=YES
*
COORDINATES
 1
 2    0.   5.
 3   .05   0.
 4  -.05   0.
*
MATERIAL  1  ELASTIC  E=2.1E11 NU=0.3 DENSITY=7800.
*
EGROUP  1  BEAM
SECTION  1  U  WTOP=0.1 D=0.1 T1=0.01 T2=0.01
BEAMVECTOR
 1  -1. 0. 0.
GLINE  1 2  AUX=-1 EL=20
*
RIGIDLINK ADDZONE=AUX
 3 1 / 4 1
TRANSLATE ZONE=AUX COPIES=20 YSTEP=0.25
*
MESH NNUMBER=MYNODES NSYMBOL=MYNODES
SOLVIA
END
```

SOLVIA-POST input, BEAM element model

```
*  A96 FREQUENCIES OF A BEAM WITH U CROSS-SECTION
*
DATABASE  CREATE
*
WRITE 'a96.lis'
FREQUENCIES
MASS-PROPERTIES
*
SET  RESPONSE-TYPE=VIBRATIONMODE ORIGINAL=YES
MESH TIME=7
MESH TIME=8
MESH TIME=9
MESH TIME=10
MESH TIME=11
MESH TIME=12
MESH TIME=13
MESH TIME=14
*
END
```

SOLVIA-PRE input, SHELL element model

```
HEADING  'A96A FREQUENCIES OF A BEAM WITH U CROSS-SECTION,
 SHELL ELEMENTS'
*
DATABASE  CREATE
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=CONSISTENT
FREQUENCIES  SUBSPACE-ITERATION  NEIG=15 RIGIDMODES=YES ,
  SSTOL=1E-10 FSHIFT=-20
MEMORY SOLVIA=24
*
COORDINATES
  1  -0.045 0. 0.1    /  2  -0.045 5. 0.1
  3  -0.045 0. 0.005  /  4  -0.045 5. 0.005
  5   0.045 0. 0.1    /  6   0.045 5. 0.1
  9   0     0  0.005  / 10   0     5. 0.005
 11   0.045 0. 0.005  / 12   0.045 5. 0.005
*
MATERIAL  1  ELASTIC  E=2.1E11 NU=0.3 DENSITY=7800.
*
EGROUP  1  SHELL
THICKNESS  1  0.01
GSURFACE  6  5 11 12  EL1=10 EL2=4 NODES=16
GSURFACE 12 11  3  4  EL1=10 EL2=6 NODES=16
GSURFACE  4  3  1  2  EL1=10 EL2=4 NODES=16
*
MESH NNUMBER=MYNODES NSYMBOL=MYNODES
SOLVIA  MODEL-SUMMARY=DETAIL
END
```

SOLVIA-POST input, SHELL element model

```
*  A96A FREQUENCIES OF A BEAM WITH U CROSS-SECTION, SHELL ELEMENTS
*
DATABASE  CREATE
*
WRITE 'a96a.lis'
FREQUENCIES
MASS-PROPERTIES
*
SET  RESPONSE-TYPE=VIBRATIONMODE
COLOR LINE
MESH TIME=7
MESH TIME=8
MESH TIME=9
MESH TIME=10
MESH TIME=11
MESH TIME=12
MESH TIME=13
MESH TIME=14
*
END
```

EXAMPLE A97

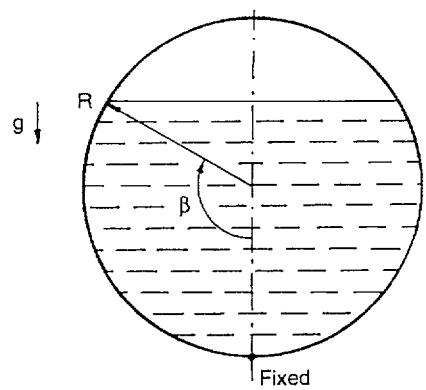
SEGMENT OF A PIPE PARTLY FILLED WITH WATER

Objective

To verify the BEAM element load with interpolation of intensity along a global coordinate axis.

Physical Problem

The figure below shows a segment of a pipe partly filled with water. The segment length is 1 m and plane stress conditions are assumed.



$R = 1$ m

Thickness $= 0.001$ m

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

$\nu = 0.3$

$\rho_{pipe} = 0$ kg/m^3

$\rho_{water} = 1000$ kg/m^3

$g = 9.81$ m/s^2

$\beta = 120°$

Finite Element Model

The finite element model is shown on page A97.2 and consists of 144 BEAM elements.

Solution Results

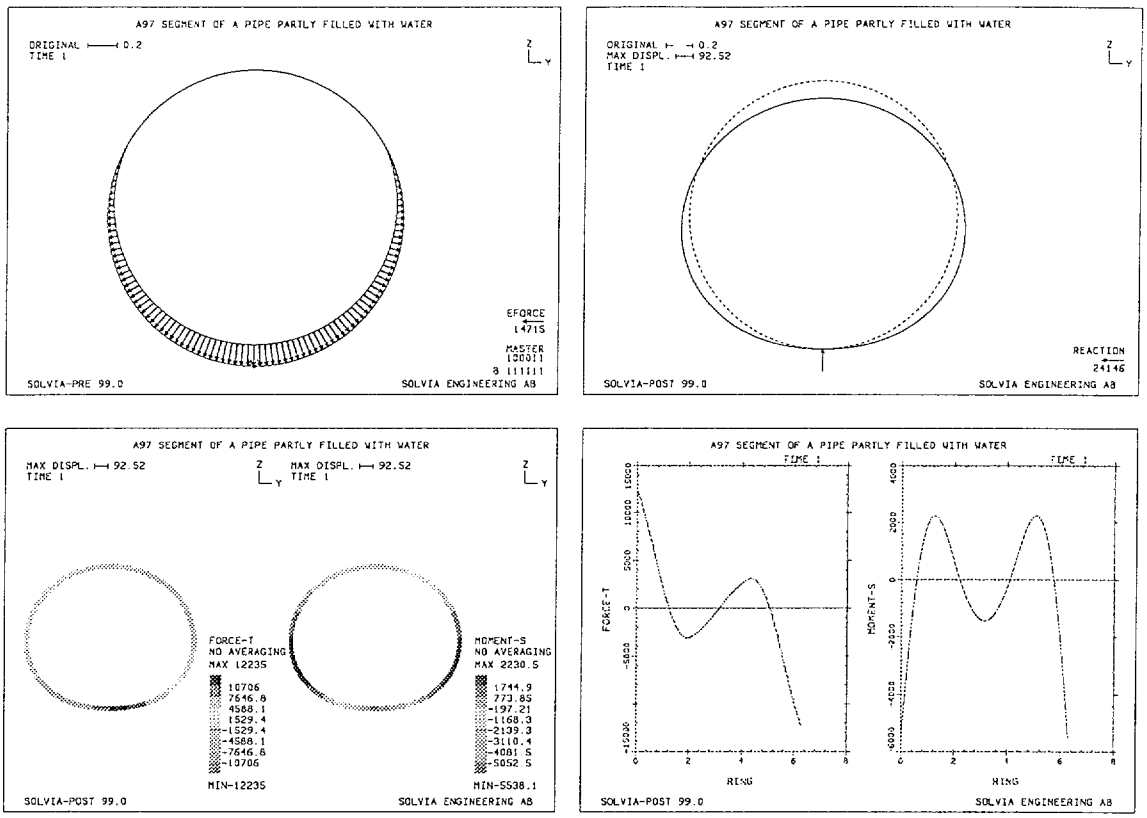
The theoretical solution for this example can be found in reference [1]. Input data used in the SOLVIA analysis is found on pages A97.2 and A97.3.

Element results at the fixed point:

	Theory	SOLVIA element 1, node 1
Axial force, N (r-direction)	$7.57 \cdot 10^3$	$7.30 \cdot 10^3$
Shear force, N	$1.24 \cdot 10^4$	$1.22 \cdot 10^4$
Bending moment, Nm	$-5.54 \cdot 10^3$	$-5.54 \cdot 10^3$

Reference

[1] Roark, R. J., Formulas for Stress and Strain, Fourth Ed., McGraw-Hill Book Company, 1965.



SOLVIA-PRE input

```
DATABASE CREATE
HEADING 'A97 SEGMENT OF A PIPE PARTLY FILLED WITH WATER'
*
MASTER IDOF=100011
SYSTEM 1 CYLINDRIC
COORDINATES
ENTRIES NODE R THETA
         1   1. -90.
*
MATERIAL 1 ELASTIC E=2.0E11 NU=0.3
*
EGROUP 1 BEAM RESULT=FORCE
BEAMVECTOR / 1  1. 0. 0.
GLINE 1 1 AUX=-1 EL=144 SYSTEM=1
SECTION 1 RECTANGULAR D=1.E-3 WTOP=1.
LOADS ELEMENT INPUT=LINES ,
 INTERPOLATION=Z COORD1=0.5 V1=0. COORD2=-1. V2=14715. VMIN=0.
 1 1 t
*
FIXBOUNDARIES / 1
*
SET VECTOR=LOAD BCODE=ALL NNUMBER=MYNODES NSYMBOL=MYNODES
MESH VIEW=X
SOLVIA
END
```

SOLVIA-POST input

```
*  A97 SEGMENT OF A PIPE PARTLY FILLED WITH WATER
*
DATABASE CREATE
WRITE 'a97.lis'
*
CONTOUR AVERAGING=NO
MESH ORIGINAL=DASHED VECTOR=REACTION VIEW=X
MESH CONTOUR=FT SUBFRAME=21
MESH CONTOUR=MS
*
EPLINE RING
1 1 2 TO 144 1 2
ELINE RING KIND=FT SUBFRAME=21
ELINE RING KIND=MS
ELIST EL1
END
```

EXAMPLE A98

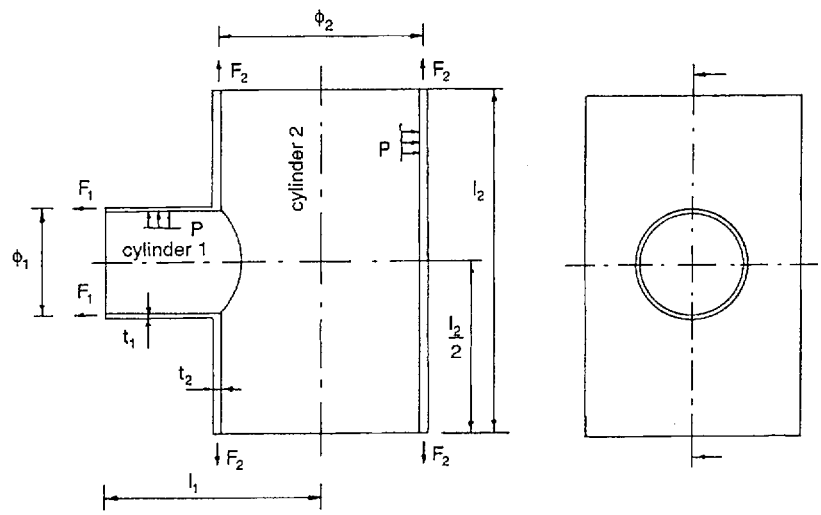
PIPE JOINT

Objective

To demonstrate the model generation of a pipe joint using SHELL elements.

Physical Problem

A pipe joint under internal pressure as shown in the figure below is considered.



$\phi_1 = 250$ mm

$l_1 = 1000$ mm

$t_1 = 5$ mm

$F_1 = \frac{p \cdot \phi_1}{4} = 125 \cdot 10^3$ N/m

$\phi_2 = 500$ mm

$l_2 = 1600$ mm

$t_2 = 10$ mm

$F_2 = \frac{p \cdot \phi_2}{4} = 250 \cdot 10^3$ N/m

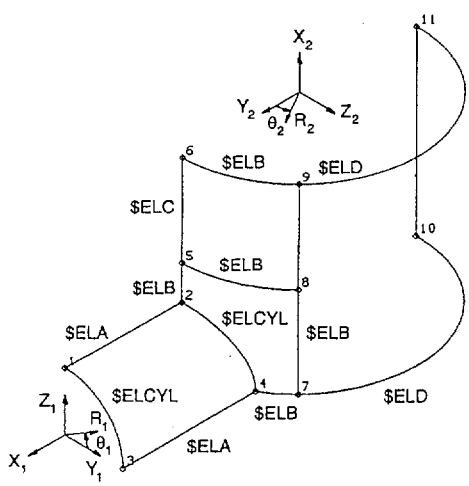
$E = 2.1 \cdot 10^{11}$ N/m^2

$\nu = 0.3$

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

Finite Element Model

One quarter of the pipe joint is considered due to symmetrical conditions. The SHELL element mesh is generated in two local systems. The location of the local systems, keynodes and the parameters used to define the mesh density are shown in the figure on the next page.



Two mathematical surfaces of type cylinder are used to find the intersection line between node 2 and node 4. Using the command GSURFACE, 16-node SHELL elements are generated and the mesh is shown on top of page A98.3.

Two symmetrical planes and a fixed X-displacement at node 11 are used as boundary conditions.

Solution Results

A deformation plot of the model is shown in the left bottom figure on page A98.3. Since the cylindrical shells are rather thin there are considerable ovalizations in the cylinders, which have not dampened out at the free ends. The influence on the stresses at the intersection from conditions at the free ends is, however, small.

The von Mises effective stress at the intersection is shown in a contour plot in the right bottom figure on page A98.3. Note that averaging of nodal stresses should not be used since there is a natural stress discontinuity due to the different thicknesses of the cylinders.

The fineness of the mesh can be checked by stress deviation plots, see the top figures on page A98.4. It can be seen that the stress deviation (from the nodal mean values) is of the order one percent of the von Mises effective stress.

The degree of accuracy is also illustrated by the smoothness in stress variations along the intersecting line, see the bottom figures on page A98.4. The stress values in the large cylinder are marked with triangle symbols.

The radial displacement, hoop stress and meridional stress along the line node 2 - 1 (cylinder 1) and along the line node 2 - 6 (cylinder 2) are shown on page A98.5. The ovalizations of the cylinders can be seen to cause a significant portion of the radial displacements. The stresses at the intersection dominate, however, the stress response.

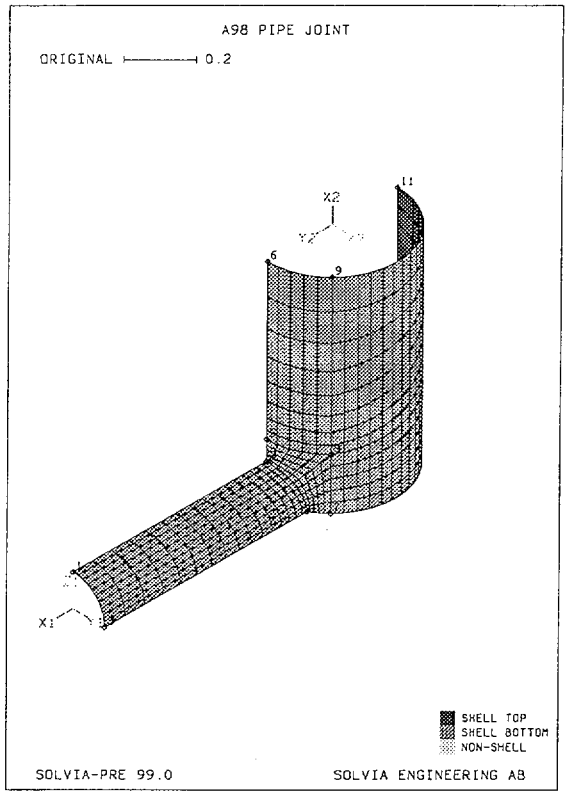
The membrane stresses σ_m (meridional) and σ_h (hoop) for a thin vessel without any disturbance are:

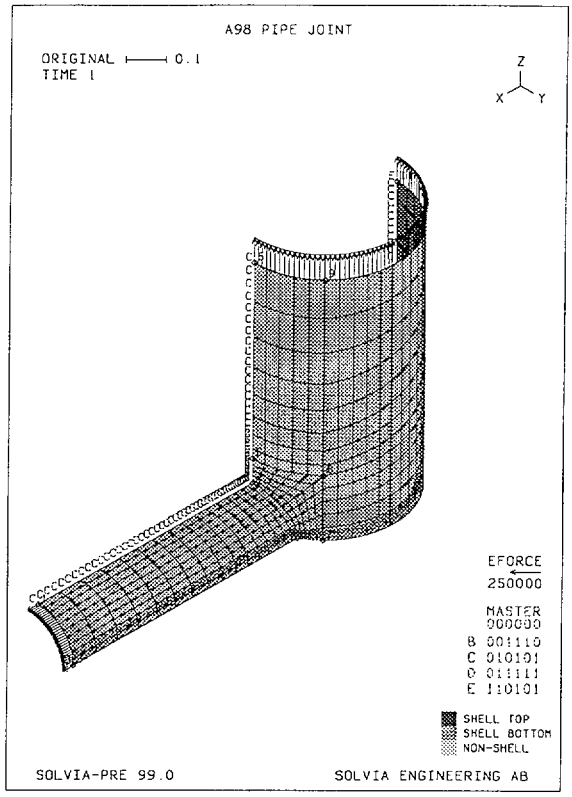
$$\sigma_m = \frac{p \cdot R}{2t} \qquad \sigma_h = \frac{p \cdot R}{t}$$

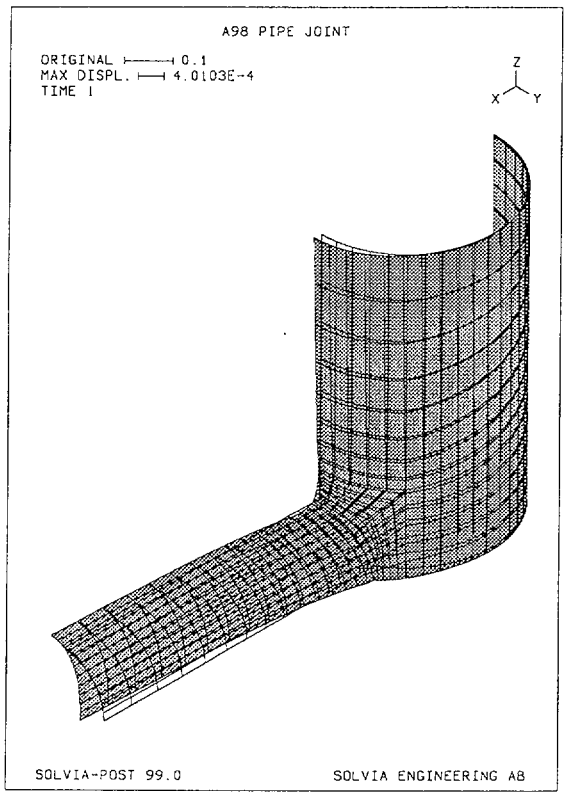
The radial displacement due to this membrane stress state is:

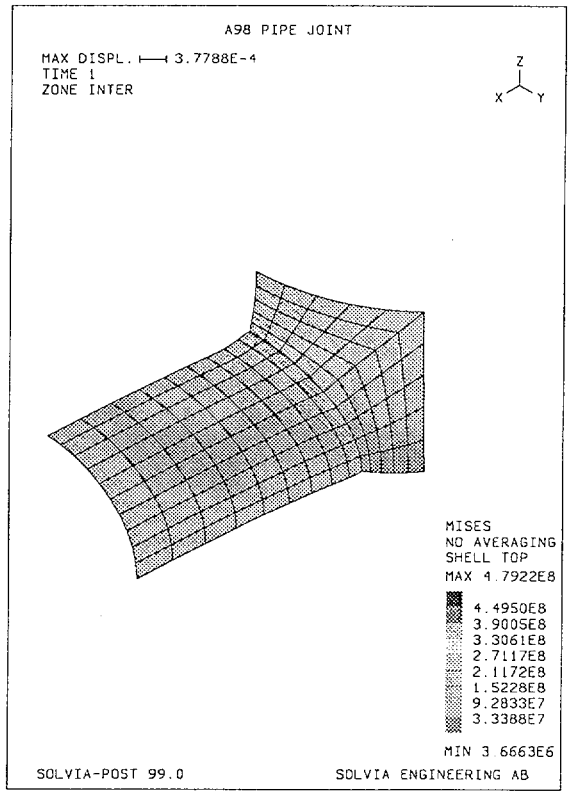
$$\delta_R = \frac{R}{E}(\sigma_h - \nu\,\sigma_m)$$

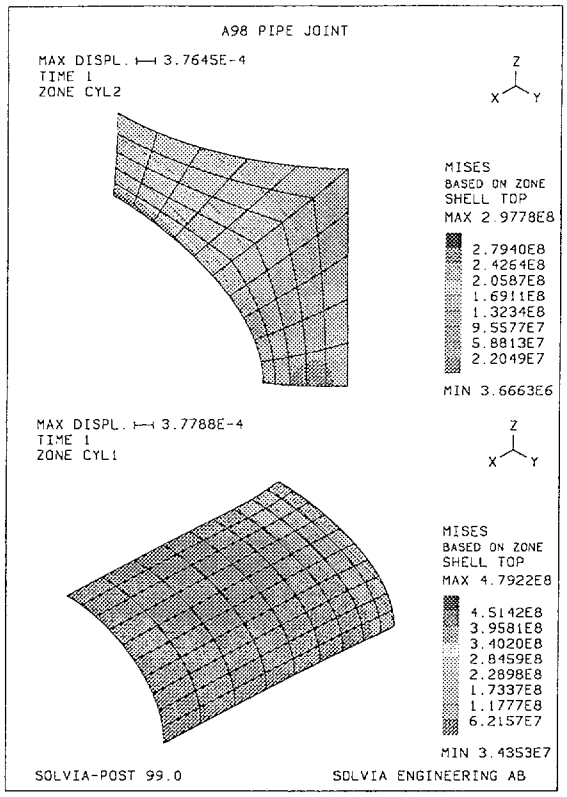
	σ_m N/m^2	σ_h N/m^2	δ_R m
Cylinder 1, $\phi = 250$ mm	$25 \cdot 10^6$	$50 \cdot 10^6$	$25.30 \cdot 10^{-6}$
Cylinder 2, $\phi = 500$ mm	$25 \cdot 10^6$	$50 \cdot 10^6$	$50.60 \cdot 10^{-6}$

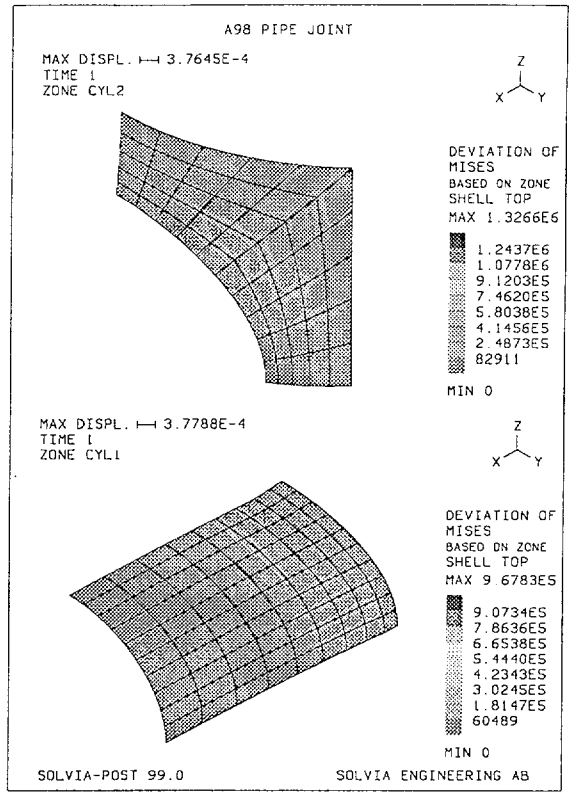

A98 PIPE JOINT
ORIGINAL 0.2
SHELL TOP
SHELL BOTTOM
NON-SHELL
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

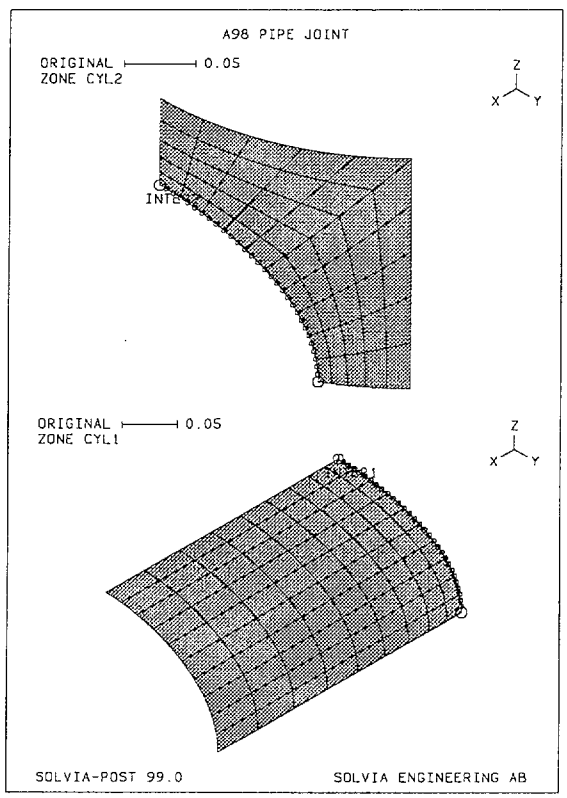

A98 PIPE JOINT
ORIGINAL 0.1
TIME 1
EFORCE
250000
MASTER
000000
B 001110
C 010101
D 011111
E 110101
SHELL TOP
SHELL BOTTOM
NON-SHELL
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

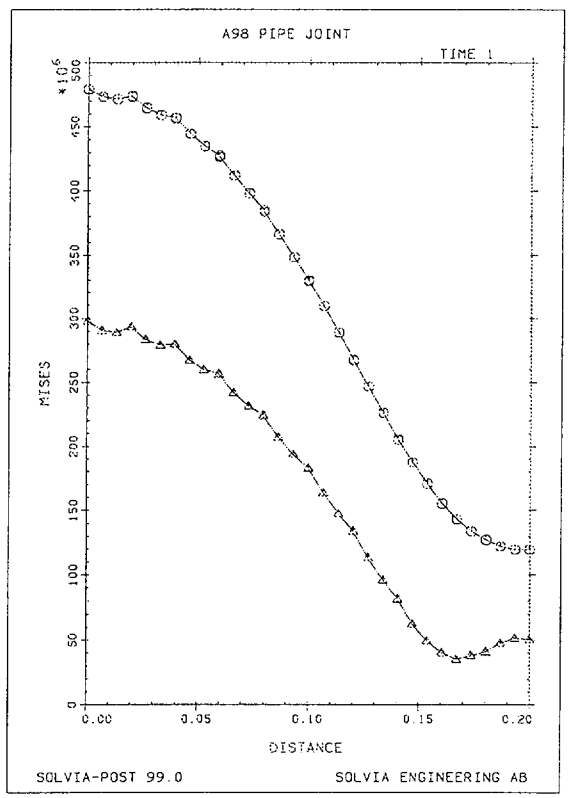

A98 PIPE JOINT
ORIGINAL 0.1
MAX DISPL. 4.0103E-4
TIME 1
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

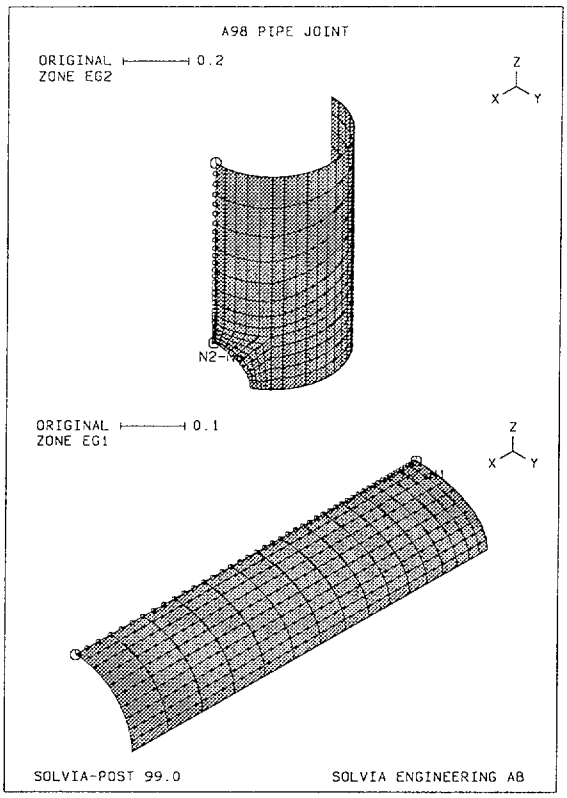

A98 PIPE JOINT
MAX DISPL. 3.7788E-4
TIME 1
ZONE INTER
MISES
NO AVERAGING
SHELL TOP
MAX 4.7922E8
4.4950E8
3.9005E8
3.3061E8
2.7117E8
2.1172E8
1.5228E8
9.2833E7
3.3388E7
MIN 3.6663E6
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

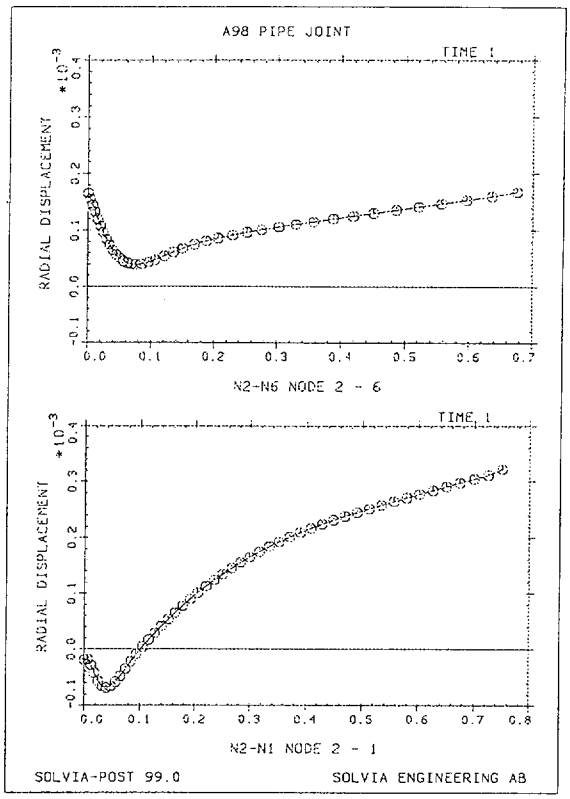

A98 PIPE JOINT
MAX DISPL. 3.7645E-4
TIME 1
ZONE CYL2
MISES
BASED ON ZONE
SHELL TOP
MAX 2.9778E8
2.7940E8
2.4264E8
2.0587E8
1.6911E8
1.3234E8
9.5577E7
5.8813E7
2.2049E7
MIN 3.6663E6
MAX DISPL. 3.7788E-4
TIME 1
ZONE CYL1
MISES
BASED ON ZONE
SHELL TOP
MAX 4.7922E8
4.5142E8
3.9581E8
3.4020E8
2.8459E8
2.2898E8
1.7337E8
1.1777E8
6.2157E7
MIN 3.4353E7
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

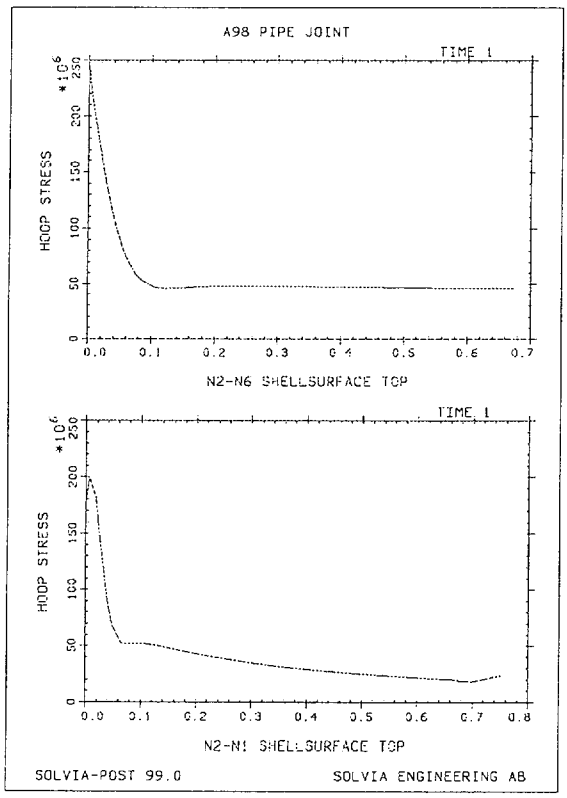

A98 PIPE JOINT
MAX DISPL. 3.7645E-4
TIME 1
ZONE CYL2
DEVIATION OF
MISES
BASED ON ZONE
SHELL TOP
MAX 1.3266E6
1.2437E6
1.0778E6
9.1203E5
7.4620E5
5.8038E5
4.1456E5
2.4873E5
82911
MIN 0
MAX DISPL. 3.7788E-4
TIME 1
ZONE CYL1
DEVIATION OF
MISES
BASED ON ZONE
SHELL TOP
MAX 9.6783E5
9.0734E5
7.8636E5
6.6538E5
5.4440E5
4.2343E5
3.0245E5
1.8147E5
60489
MIN 0
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

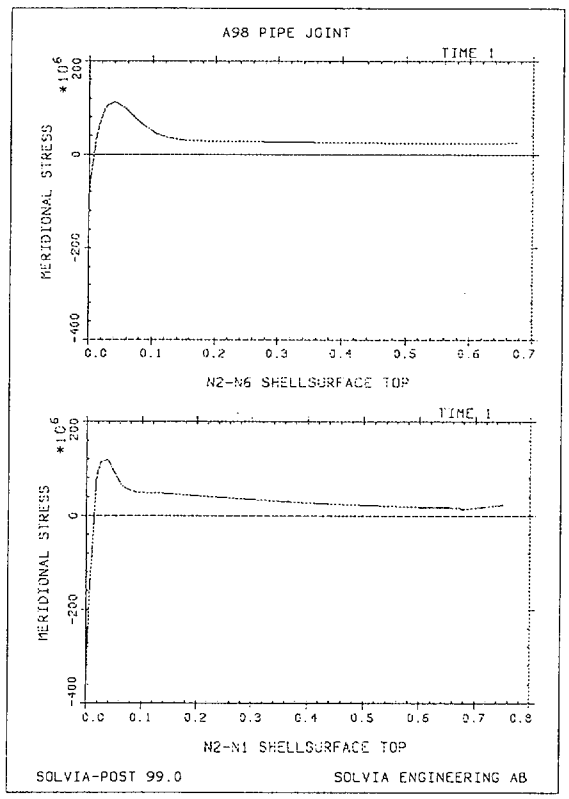

A98 PIPE JOINT
ORIGINAL 0.05
ZONE CYL2
ORIGINAL 0.05
ZONE CYL1
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A98 PIPE JOINT
TIME 1
MISES
DISTANCE
0.00
0.05
0.10
0.15
0.20
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A98 PIPE JOINT
ORIGINAL
ZONE EG2
0.2
Z
X
Y
N2-N6
ORIGINAL
ZONE EG1
0.1
Z
X
Y
N1
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A98 PIPE JOINT
TIME 1
RADIAL DISPLACEMENT
*10^-3
N2-N6 NODE 2 - 6
TIME 1
RADIAL DISPLACEMENT
*10^-3
N2-N1 NODE 2 - 1
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A98 PIPE JOINT
TIME 1
HOOP STRESS
*10^6
N2-N6 SHELLSURFACE TOP
TIME 1
HOOP STRESS
*10^6
N2-N1 SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A98 PIPE JOINT
TIME 1
MERIDIONAL STRESS
*10^6
N2-N6 SHELLSURFACE TOP
TIME 1
MERIDIONAL STRESS
*10^6
N2-N1 SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
DATABASE CREATE
HEAD 'A98 PIPE JOINT'
MEMORY SOLVIA=20
PARAMETER $ELA=16 $ELB=5 $ELCYL=10 $ELC=8 $ELD=16
SET NODES=16 MIDNODES=2
SET NNUMBERS=MYNODES NSYMBOL=MYNODES PLOTORIENTATION=PORTRAIT
COLOR EFILL=SHELL
*
SYSTEM 1 CYLINDRICAL X=1.0
SYSTEM 2 CYLINDRICAL Z=0.8 AX=0 AZ=1 BX=1 BY=0 BZ=0
MATERIAL 1 ELASTIC E=2.1E11 NU=0.3
*
COORDINATES SYSTEM=1
ENTRIES  NODE  R      THETA  XL
          1    0.125  90.    0.
          2    0.125  90.   -0.75
          3    0.125  0.     0.
          4    0.125  0.    -0.75
COORDINATES SYSTEM=2
ENTRIES  NODE  R      THETA  XL
          5    0.250  0.    -0.6
          6    0.250  0.     0.
          7    0.250  45.   -0.8
          8    0.250  45.   -0.6
          9    0.250  45.    0.
         10    0.250  180.  -0.8
         11    0.250  180.   0.
*
SET SYSTEM=1
LINE CYLINDRIC 2 4 EL=$ELCYL
MSURFACE 1 CYLINDER R=0.125 SYSTEM=1
MSURFACE 2 CYLINDER R=0.250 SYSTEM=2
LINE INTERSECTION 2 4 MSURFACE=1 TARGETSURFACE=2
LINE CYLINDRIC 2 1 EL=$ELA RATIO=5
LINE CYLINDRIC 4 3 EL=$ELA RATIO=5
*
EGROUP 1 SHELL MATERIAL=1 RESULT=NSTRESS
GSURFACE 1 2 4 3 SYSTEM=1
THICKNESS 1 5E-3
LOAD ELEMENTS INPUT=SURFACE /  1 2 4 3 t 2.E6
LOAD ELEMENTS TYPE=FORCE INPUT=LINE /  1 3 out 125.E3
*
SET SYSTEM=2
LINE CYLINDRIC 5 8 EL=$ELB
LINE CYLINDRIC 8 7 EL=$ELB
LINE COMBINED 5 7 8
LINE CYLINDRIC 2 5 EL=$ELB RATIO=2
LINE CYLINDRIC 4 7 EL=$ELB RATIO=2
LINE STRAIGHT  5 6 EL=$ELC RATIO=5
LINE CYLINDRIC 9 11 EL=$ELD RATIO=1
*
EGROUP 2 SHELL MATERIAL=1 RESULT=NSTRESS
GSURFACE 2 5 7 4  ADDZONE=CYL2
GSURFACE 5 6 9 8
LINE COMBINES 7 9 8
```

SOLVIA-PRE input (cont.)

```
GSURFACE 7 9 11 10
THICKNESS 1 10E-3
LOAD ELEMENTS INPUT=SURFACE
 2 5 7 4 t 2.E6 / 5 6 9 8 t 2.E6 / 7 9 11 10 t 2.E6
*
LOAD ELEMENTS TYPE=FORCE INPUT=LINE
 6 9  out 250.E3 /  9 11 out 250.E3
*
ZONE NAME=BOUND-Z GLOBAL-LIMITS ZMAX=1.E-6
FIXBOUNDARIES 345 INPUT=ZONE ZONE1=BOUND-Z
ZONE NAME=BOUND-SYM GLOBAL-LIMITS YMAX=1.E-6
FIXBOUNDARIES 246 INPUT=ZONE ZONE1=BOUND-SYM
FIXBOUNDARIES 1 / 11
*
MESH  AXES=LOCAL
MESH  BCODE=ALL VECTOR=EFORCE
SOLVIA
END
```

SOLVIA-POST input

```
*  A98 PIPE JOINT
*
DATABASE CREATE
WRITE 'a98.lis'
*
ZONE NAME=CYL1 INPUT=EGROUP / 1
ZONE NAME=CLOSE GLOBAL-LIMITS XMAX=0.57
ZONE NAME=CYL1 INPUT=ZONE OPERATION=INTERSECT ZONE1=CLOSE
ZONE NAME=INTER INPUT=ZONE OPERATION=ADD ZONE1=CYL1 CYL2
*
EGROUP 1
EPLINE INTER1 / 16 2 6 10 3 STEP 16 TO 160 2 6 10 3
EPLINE N2-N1  / 16 2 9 1 TO 1 2 9 1
NPLINE N2-N1  / 2 130 TO 176 1
EGROUP 2
EPLINE INTER2 / 1 1 12 8 4 STEP 5  TO 46 1 12 8 4
EPLINE N2-N6  / 1 1 5 2 TO 5 1 5 2 / 51 1 5 2 TO 58 1 5 2
NPLINE N2-N6  / 2 1644 TO 1657 5 1672 TO 1694 6
*
SET PLOTORIENTATION=PORTRAIT
MESH ORGINAL=YES
CONTOUR AVERAGING=NO
MESH INTER CONTOUR=MISES
*
CONTOUR AVERAGING=ZONE
MESH CYL2 CONTOUR=MISES SUBFRAME=12
MESH CYL1 CONTOUR=MISES
 *
MESH CYL2 CONTOUR=SDEVIATION SUBFRAME=12
MESH CYL1 CONTOUR=SDEVIATION
*
```

SOLVIA-POST input (cont.)

```
MESH CYL2 PLINES=INTER2 SUBFRAME=12
MESH CYL1 PLINES=INTER1
*
AXIS 1 VMIN=0. VMAX=0.20 LABEL='DISTANCE'
AXIS 2 VMIN=0. VMAX=500.E6 LABEL='MISES'
ELINE INTER2 KIND=MISES SYMBOL=2 XAXIS=1  YAXIS=2
ELINE INTER1 KIND=MISES SYMBOL=1 XAXIS=-1 YAXIS=-2 SUBFRAME=OLD
*
MESH EG2 PLINES=N2-N6 SUBFRAME=12
MESH EG1 PLINES=N2-N1
*
AXIS 3 VMIN=-100E-6 VMAX=400E-6 LABEL='RADIAL DISPLACEMENT'
NLINE N2-N6 DIRECTION=1 SYMBOL=1 YAXIS=3 SUBFRAME=12
NLINE N2-N1 DIRECTION=3 SYMBOL=1 YAXIS=3
*
AXIS 4 VMIN=0 VMAX=250E6 LABEL='HOOP STRESS'
ELINE N2-N6 KIND=SYY YAXIS=4 SUBFRAME=12
ELINE N2-N1 KIND=SYY YAXIS=4
*
AXIS 5 VMIN=-400E6 VMAX=200E6 LABEL='MERIDIONAL STRESS'
ELINE N2-N6 KIND=SZZ YAXIS=5 SUBFRAME=12
ELINE N2-N1 KIND=SXX YAXIS=5
*
SUMMATION KIND=LOAD
END
```

EXAMPLE A99

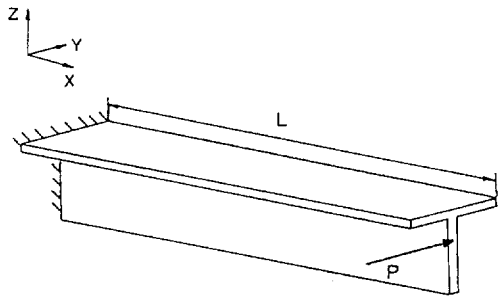
CANTILEVER BEAM, STANDARD T CROSS-SECTION

Objective

To verify the BEAM element stress calculations when using a standard T cross-section.

Physical Problem

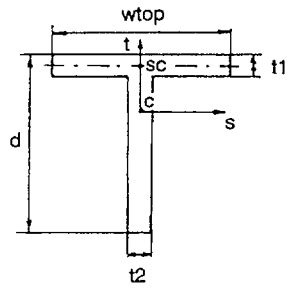
A cantilever beam as shown in the figure. One end is fixed and the other end is subjected to a transverse force in the Y-direction acting at the center of gravity.



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

$\nu = 0.3$

$P = 2 \cdot 10^3$ N



$L = 1$ m

$wtop = 0.1$ m

$d = 0.1$ m

$t1 = 0.01$ m

$t2 = 0.01$ m

$c =$ center of gravity

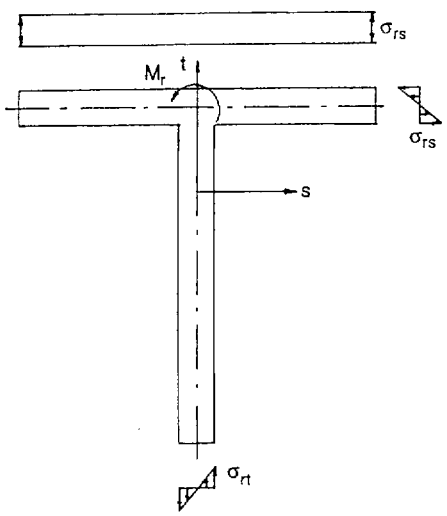
$sc =$ shear center

Finite Element Model

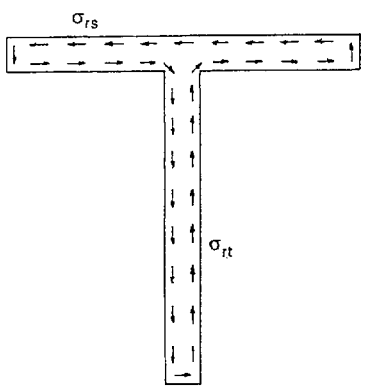
The cantilever beam is modeled using one BEAM element with standard T cross-section. A reference model using 256 cubic SHELL elements, example A99A, has also been analyzed. The reference model can be seen in the figure on page A99.5.

Solution Results

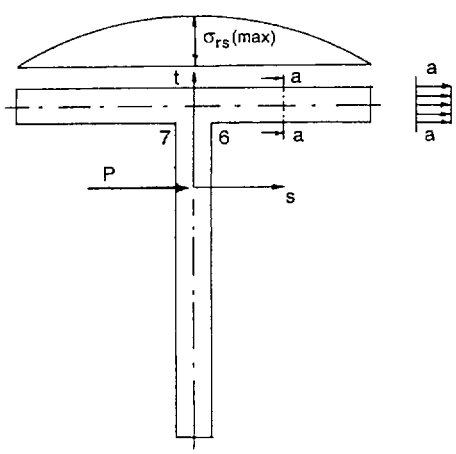
If no BEAM section offset is defined the BEAM element nodes are located at the center of gravity. The concentrated force is defined to act at the center of gravity and can then be directly applied to the BEAM element end node. For a standard T cross-section the shear center does not coincide with the center of gravity and the transverse load will produce a twisting of the beam element. With the orientation of the local element r-s-t system as shown in the figure above the twisting moment is $M_r = P \cdot \Delta$, where Δ is the distance between the shear center and the center of gravity.



The torsional rigidity of the T cross-section is assumed to be equal to the sum of two narrow rectangular sections. The shear stress distribution in the T cross-section due to a twisting moment as calculated in SOLVIA can be seen in the figure to the left. The shear stress σ_{rs} is assumed constant along the flange in the s-direction and the shear stress σ_{rt} is assumed constant along the web in the t-direction.



The detailed distribution of the shear stress due to Saint-Venant torsion is, therefore, neglected at the ends and at the re-entrant corners. The figure to the left shows schematically the detailed stress distribution. Example A65 indicates one method to calculate detailed torsional stresses.



The shear stress distribution in the T cross-section due to a shear force P in the s-direction can be seen in the figure to the left. A parabolic variation in the s-direction of the flange is assumed with a maximum value of

$$\sigma_{rs}(\max) = \frac{1.5 \cdot P}{\text{wtop} \cdot \text{t1}}$$

and zero at the corners $s = \pm \text{wtop} / 2$.

The shear stresses at stress points 6 and 7 due to a shear force P in s-direction are calculated as

$$\sigma_{rs} = \sigma_{rs}(\max) \cdot \left(1 - \left(\frac{\text{t2}}{\text{wtop}} \right)^2 \right)$$

Transverse shear effects are included in the standard BEAM sections in SOLVIA. The shear areas in the s- and t-direction for the T cross-section are $A_s = \text{wtop} \cdot \text{t1}$ and $A_t = \text{d} \cdot \text{t2}$, respectively.

The axial stress distribution, σ_{rr}, in a standard BEAM section is calculated as

$$\sigma_{rr} = \frac{F_r}{\text{Area}} + \frac{M_s \cdot t_i}{I_s} - \frac{M_t \cdot s_i}{I_t}$$

where s_i , t_i are the coordinates for the considered stress point.

Using the input data shown on page A99.10 in the SOLVIA analysis the following displacements at the tip node are obtained:

$$u_y = 4.02 \cdot 10^{-3} \text{ m}$$

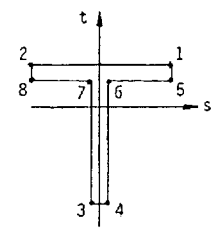
$$\varphi_x = 9.29 \cdot 10^{-3}$$

The stresses from the beam model at the section X = L/2 are shown in the table below at selected stress points. This section is away from local disturbances at the concentrated load and the boundary conditions.

Stresses [MPa]

Point	σ_{rr}	σ_{rs}	σ_{rt}	von Mises
1	-59.46	-7.50	0	60.87
4	-5.95	0	7.50	14.29
7	5.95	10.47	-7.50	19.09

Stress point at local node N1



Note that for the standard BEAM section the von Mises effective stress is calculated as

$$\sigma_e = \sqrt{\sigma_{rr}^2 + 3\tau_{max}^2} \text{ , where}$$

$$\tau_{max} = \text{Max}\left(|\sigma_{rs}|, |\sigma_{rt}|\right)$$

The nominal values are used without any stress concentration.

Note that the shear stress due to the twisting moment is ± 7.50 MPa in the web and in the flange. The shear stress due to the shear force is 2.97 MPa in the flange at stress points 6 and 7.

A 16-node SHELL model with 16 × 8 elements for both the flange and the web was analyzed using the input data on pages A99.11 and A99.12. The following results were obtained:

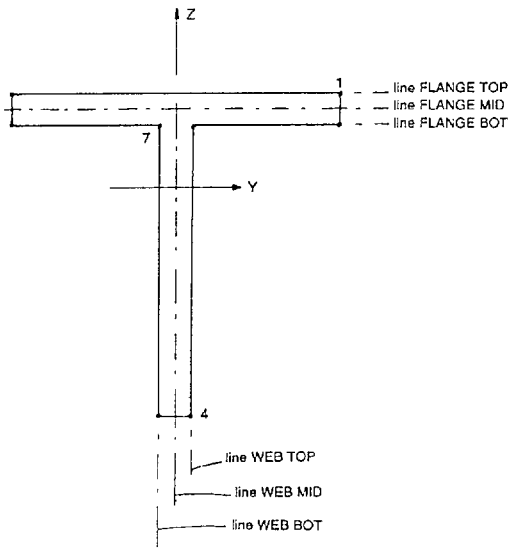
The displacements at center of gravity (node 9) are

$$u_y = 4.03 \cdot 10^{-3} \text{ m}$$

$$\varphi_x = 9.66 \cdot 10^{-3}$$

The distribution of the Y- and Z-displacements in the beam can be seen as contours on page A99.6.

Element point lines called FLANGE and WEB are defined at the middle section of the beam, see the figure of page A99.6.



The axial stress σ_{xx} and the two shear stresses σ_{xy} and σ_{xz} are shown along these lines for the location TOP, MID and BOTTOM of the shell on pages A99.7 to A99.9.

The axial stress can be seen to vary practically linearly over the cross-section. This is in accordance with the beam theory where the normals to the beam axis are assumed to remain straight during deformation. The calculated axial stresses in MPa at the stress point locations 1, 4 and 7 are -59.44, -5.94 and 5.94, respectively, which are in excellent agreement with the results for the BEAM element.

The variation of the shear stress σ_{xy} along the line FLANGE at the MID surface is only due to the transverse shear force and has no torsional component. The maximum value is 2.97 MPa, which agrees with the BEAM element results. The values along the TOP and BOTTOM surface includes both the parabolic distribution due to the shear force and the portion due to Saint-Venant torsion. The value at the location for stress point 7 is 10.54 MPa which agrees very well with the BEAM results of 10.47 MPa.

Neglecting the variations near the end of the flange the value of σ_{xy} due to Saint-Venant torsion is -7.60 MPa on the TOP surface, which compares well with the value of -7.50 MPa for the BEAM element.

The variation of the shear stress σ_{xz} along the line WEB is practically only due to Saint-Venant torsion and can be seen to be ± 7.73 MPa with some small variation along the web. This value also compares well with the BEAM result of ± 7.50 MPa.

The shear stress σ_{xz} in the thickness direction of the flange is generally zero except at the two ends where Saint-Venant shear stress develop also in the Z-direction. A physical picture is provided by the membrane (soap film) analogy, [1] page 268. A uniformly tensioned membrane shaped as the cross-section and supported at the boundaries is loaded by pressure. The slope of the membrane is then proportional to the Saint-Venant torsional shear stress.

Another analogy is provided in example A65 where uniform heat generation over the cross-section is used. The heat flux is then proportional to the shear stress.

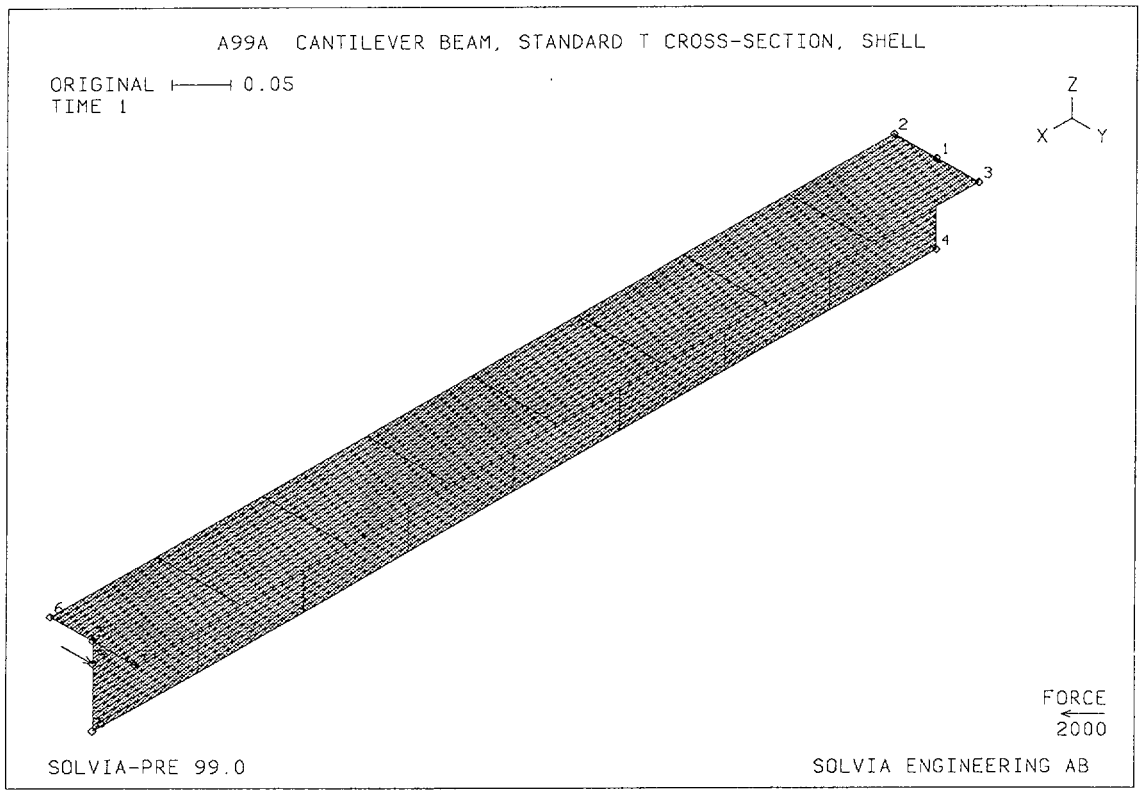
Note that the shear stress in the thickness direction of a SHELL element is always constant. The reason is that the normal to the midsurface of the SHELL is remaining straight during deformation. All points along the rotated but straight normal have, therefore, the same shear strain. The actual shear stress distribution in the thickness direction would be parabolic with zero values at the surfaces and 1.5 times the average shear stress value at the midsurface.

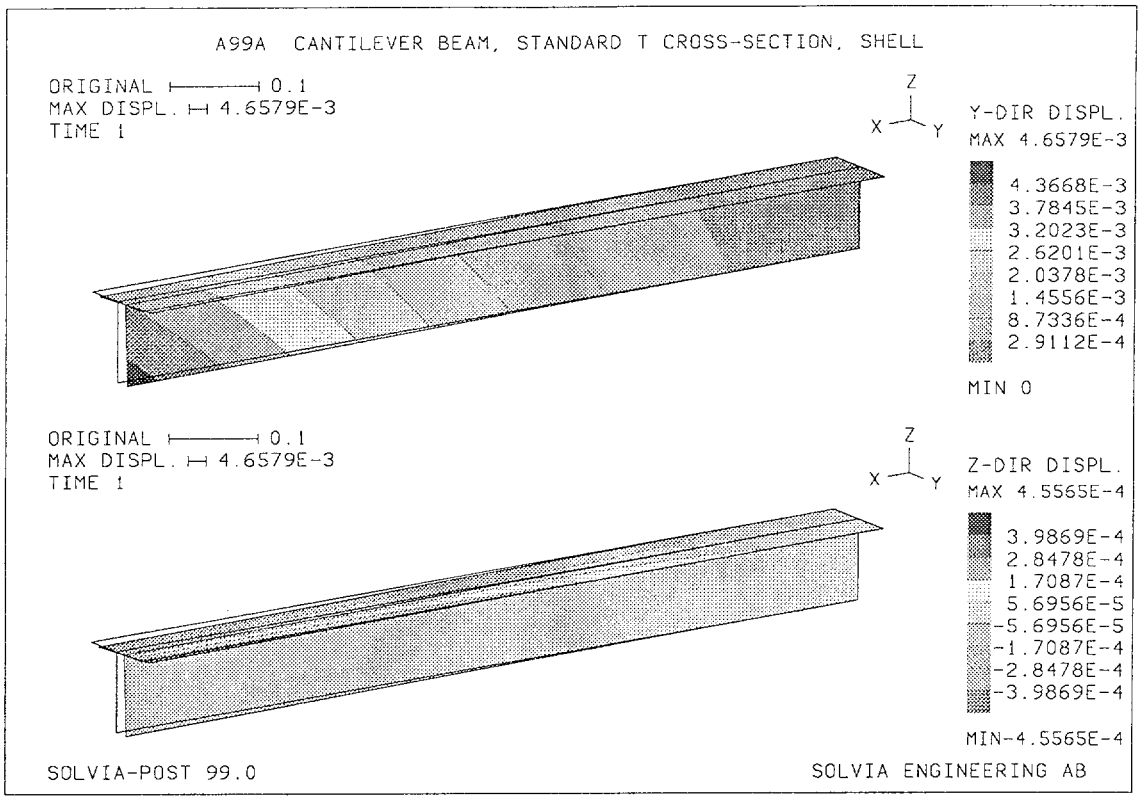
User Hints

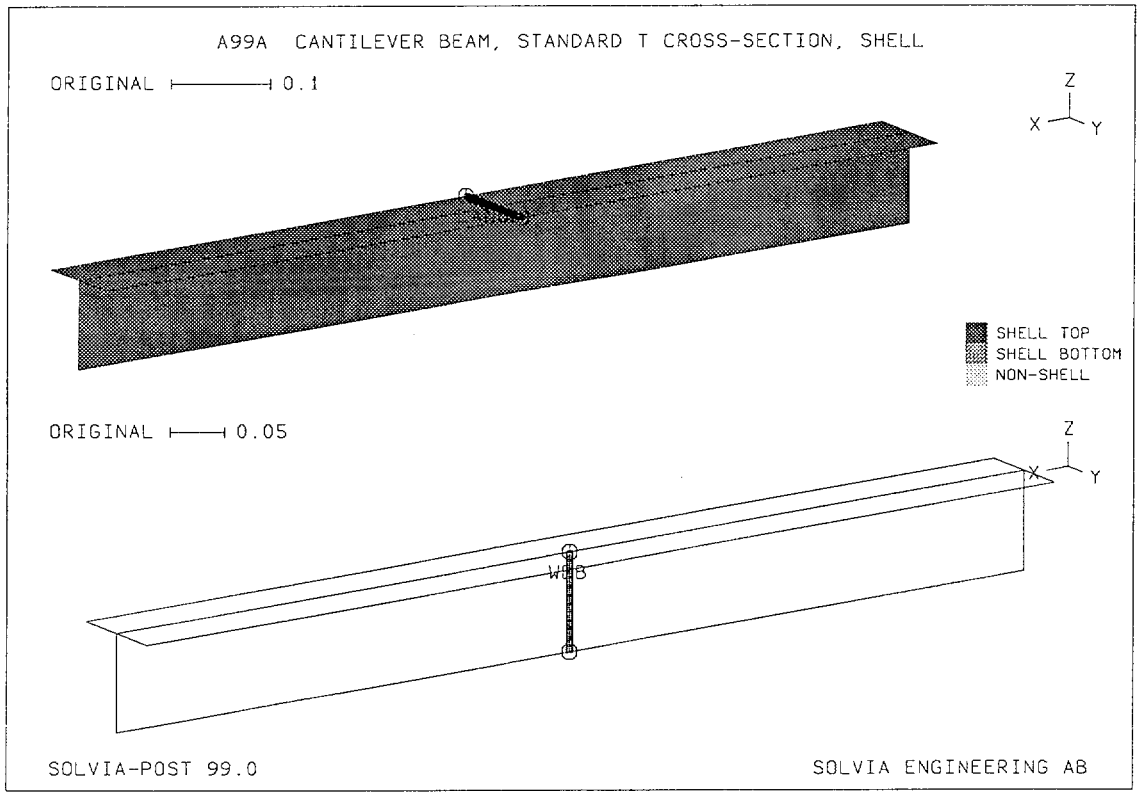
- Note that a fine SHELL element model must be used to calculate the shear stress distribution correctly (within the shell theory).
- Note that the reaction forces and moments are evaluated at the nodal points. If no BEAM section offset is defined the reactions are defined at the center of gravity for the BEAM standard section.

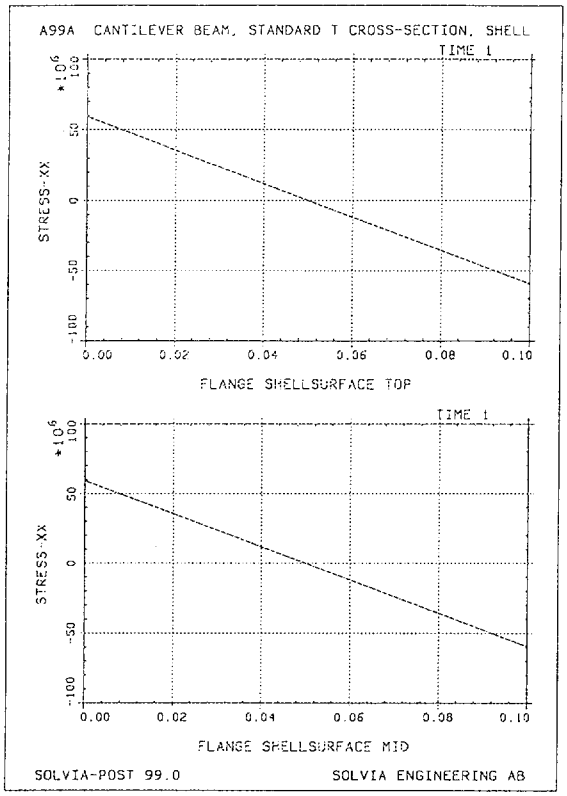
Reference

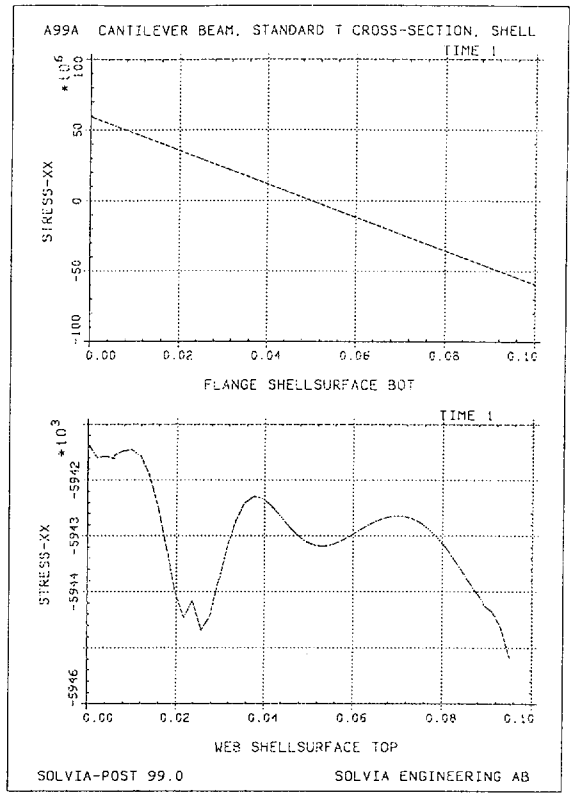
[1] Timoshenko, S., and Goodier, J.N., Theory of Elasticity, 2nd edition, McGraw Hill, 1951.

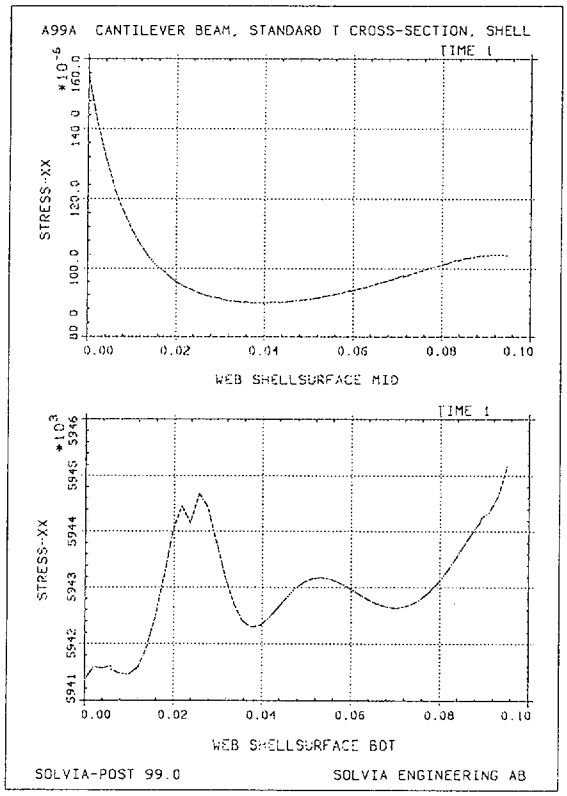


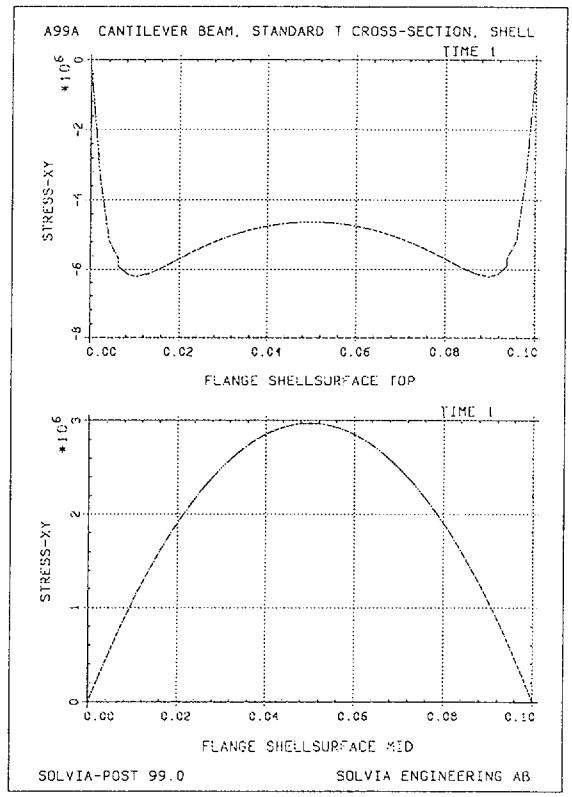

A99A CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL
ORIGINAL 0.1
MAX DISPL. 4.6579E-3
TIME 1
Z
X Y
Y-DIR DISPL.
MAX 4.6579E-3
4.3668E-3
3.7845E-3
3.2023E-3
2.6201E-3
2.0378E-3
1.4556E-3
8.7336E-4
2.9112E-4
MIN 0
ORIGINAL 0.1
MAX DISPL. 4.6579E-3
TIME 1
Z
X Y
Z-DIR DISPL.
MAX 4.5565E-4
3.9869E-4
2.8478E-4
1.7087E-4
5.6956E-5
-5.6956E-5
-1.7087E-4
-2.8478E-4
-3.9869E-4
MIN-4.5565E-4
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

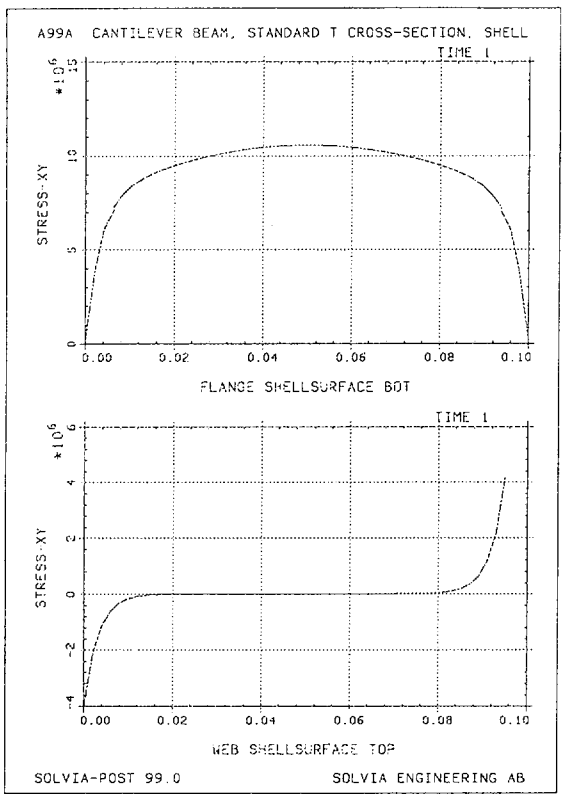

A99A CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL
ORIGINAL 0.1
Z
X Y
SHELL TOP
SHELL BOTTOM
NON-SHELL
ORIGINAL 0.05
Z
X Y
WEB
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

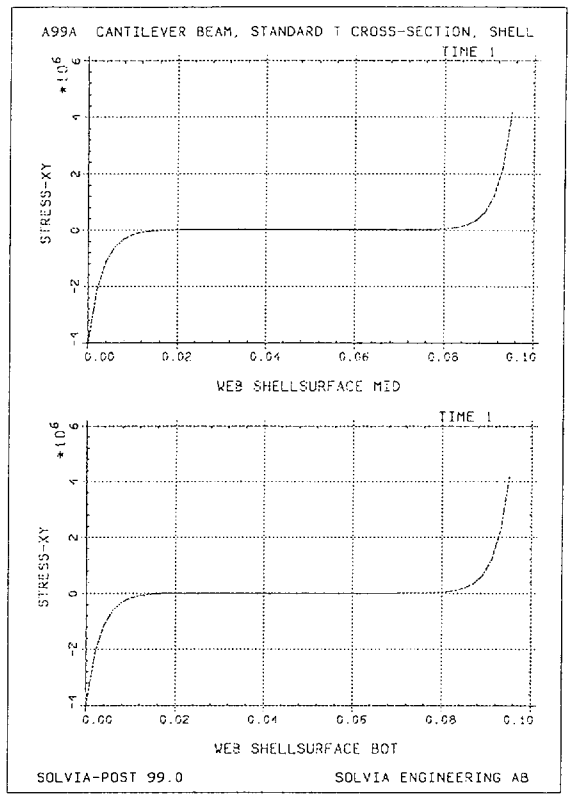

A99A CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL
TIME 1
STRESS-XX
FLANGE SHELLSURFACE TOP
FLANGE SHELLSURFACE MID
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

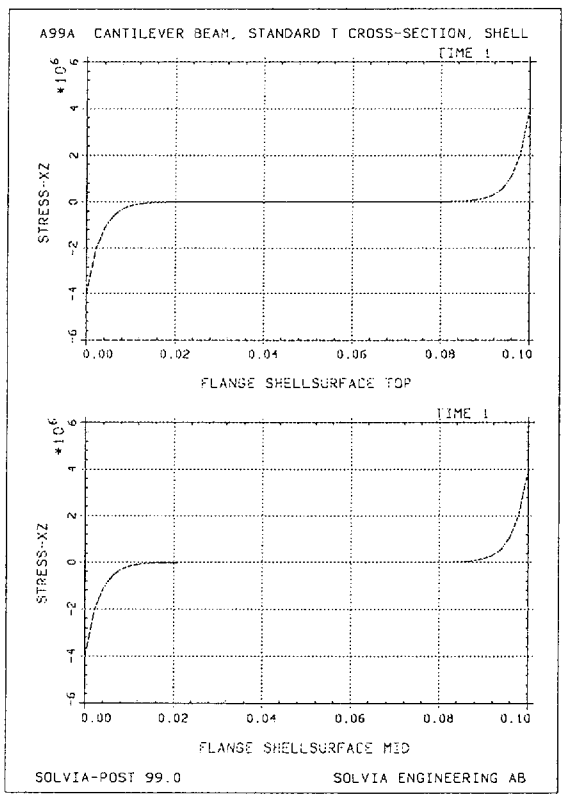

A99A CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL
TIME 1
STRESS-XX
FLANGE SHELLSURFACE BOT
WEB SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

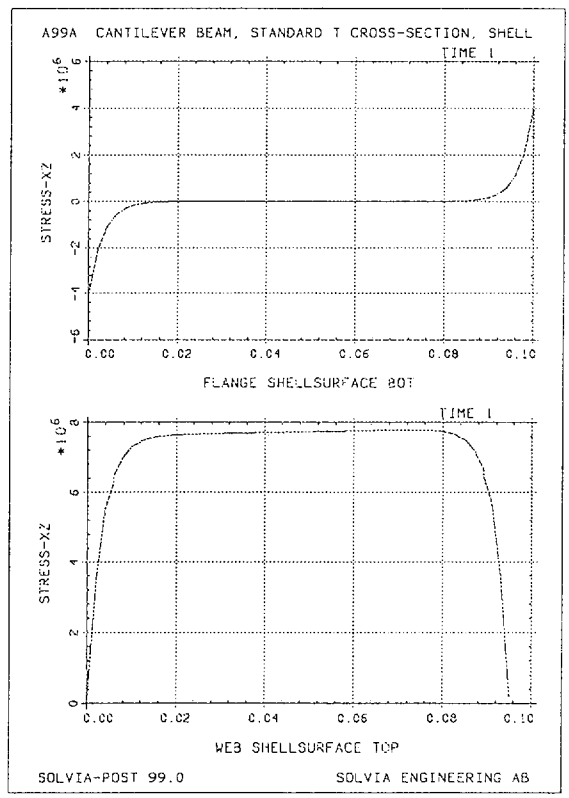

A99A CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL
TIME 1
STRESS-XX
WEB SHELLSURFACE MID
WEB SHELLSURFACE BOT
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

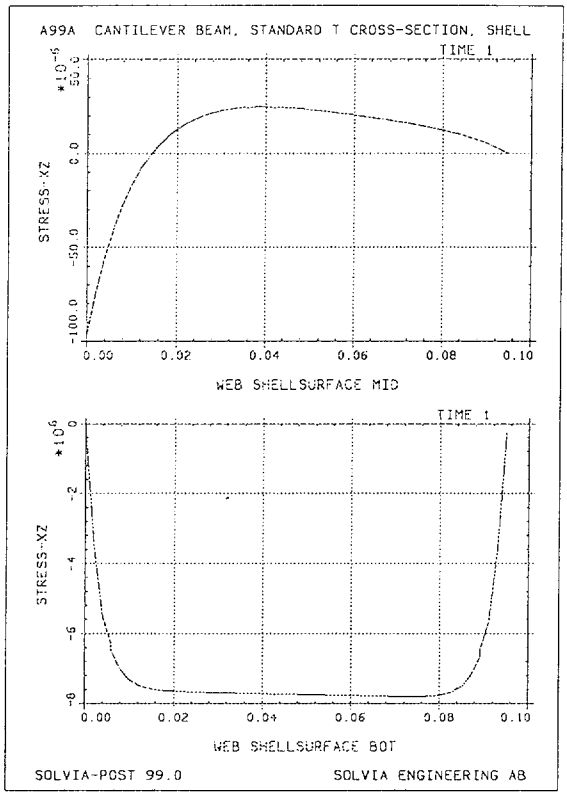

A99A CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL
TIME 1
STRESS-XY
FLANGE SHELLSURFACE TOP
FLANGE SHELLSURFACE MID
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

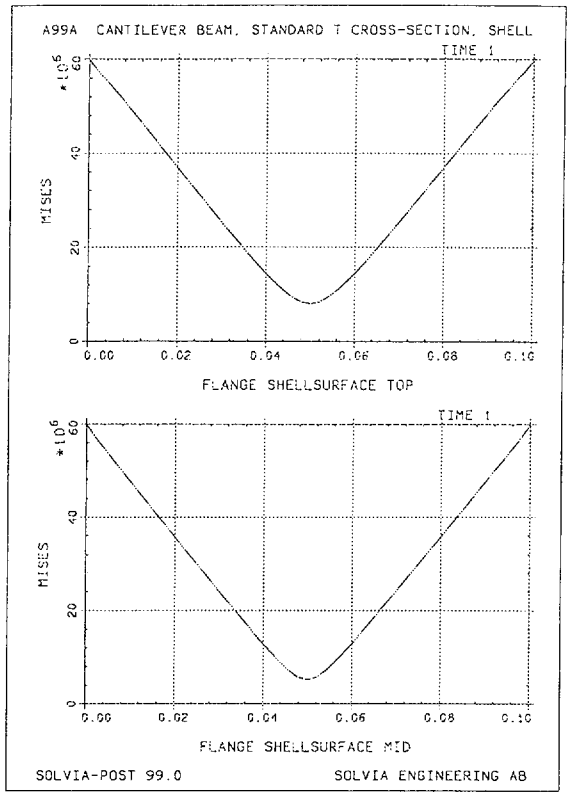

A99A CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL
TIME 1
STRESS-XY
FLANGE SHELLSURFACE BOT
TIME 1
STRESS-XY
WEB SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

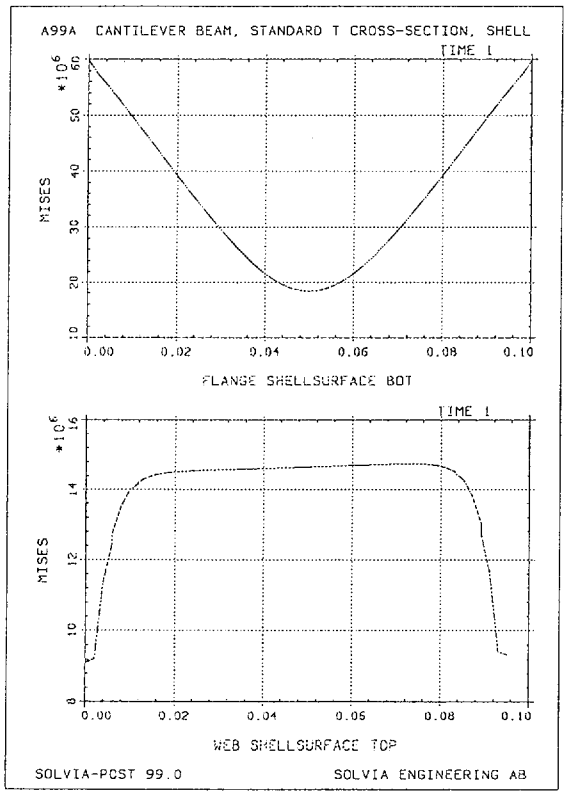

A99A CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL
TIME 1
STRESS-XY
WEB SHELLSURFACE MID
TIME 1
STRESS-XY
WEB SHELLSURFACE BOT
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

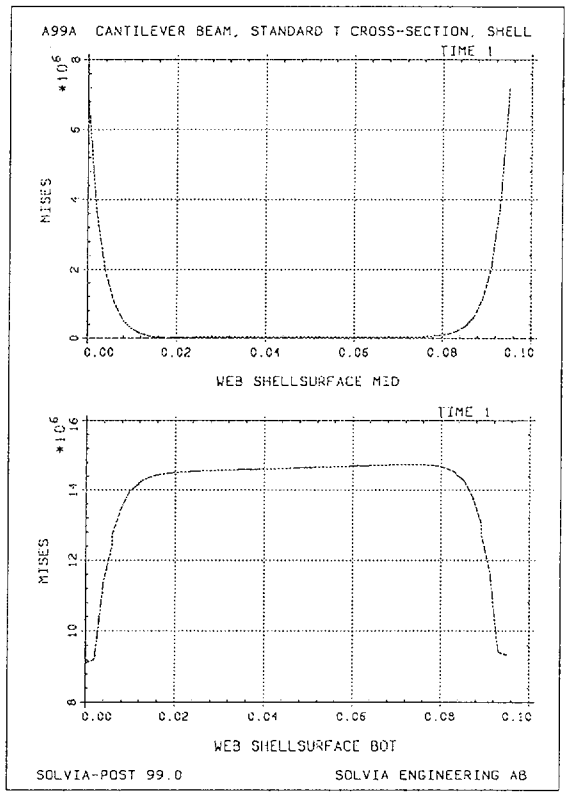

A99A CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL
TIME 1
STRESS-XZ
FLANGE SHELLSURFACE TOP
TIME 1
STRESS-XZ
FLANGE SHELLSURFACE MID
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A99A CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL
TIME 1
STRESS-XZ
FLANGE SHELLSURFACE BOT
TIME 1
STRESS-XZ
WEB SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A99A CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL
TIME 1
STRESS-XZ
WEB SHELLSURFACE MID
TIME 1
STRESS-XZ
WEB SHELLSURFACE BOT
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A99A CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL
TIME 1
MISES
FLANGE SHELLSURFACE TOP
TIME 1
MISES
FLANGE SHELLSURFACE MID
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A99A CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL
TIME 1
MISES
FLANGE SHELLSURFACE BOT
TIME 1
MISES
WEB SHELLSURFACE TOP
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A99A CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL
TIME 1
MISES
WEB SHELLSURFACE MID
TIME 1
MISES
WEB SHELLSURFACE BOT
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A99  CANTILEVER BEAM, STANDARD T CROSS-SECTION'
*
DATABASE  CREATE
*
COORDINATES
 1  /  2  0.5 / 3 1.0
*
MATERIAL  1  ELASTIC  E=2.1E11 NU=0.3
*
EGROUP  1  BEAM  RESULTS=STRESSES
SECTION  1  T  WTOP=0.1 D=0.1 T1=0.01 T2=0.01
LIST  SECTION
BEAMVECTOR
 1  0 1 0
ENODES
 1  -1 1 2
 2  -1 2 3
*
FIXBOUNDARIES  /  1
LOADS  CONCENTRATED
 3 2 2.E3
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A99  CANTILVER BEAM, STANDARD T CROSS-SECTION
*
DATABASE  CREATE
*
WRITE  FILENAME='a99.lis'
*
NLIST  DIRECTION=246
NLIST  DIRECTION=246 KIND=REACTIONS
*
ELIST  SELECT=BASICS
ELIST  SELECT=S-EFFECTIVE
END
```

SOLVIA-PRE input

```
HEADING  'A99A  CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL'
*
MEMORY SOLVIA=21
DATABASE  CREATE
*
COORDINATES
  1   0.   0.    0.095  /  2   0.  -0.05  0.095
  3   0.   0.05  0.095  /  4   0.   0.    0.
  5   1.   0.    0.095  /  6   1.  -0.05  0.095
  7   1.   0.05  0.095  /  8   1.   0.    0.
  9   1.   0.    0.07125
*
MATERIAL  1  ELASTIC  E=2.1E11 NU=0.3
*
EGROUP  1  SHELL RESULTS=NSTRESSES STRESSREFERENCE=GLOBAL
THICKNESS  1  0.01
GSURFACE  6 5 1 2  EL1=8 EL2=8 NODES=16
GSURFACE  5 7 3 1  EL1=8 EL2=8 NODES=16
GSURFACE  5 8 4 1  EL1=16 EL2=8 NODES=16
*
FIXBOUNDARIES  INPUT=LINE  /  1 4  /  2 1  /  1 3
LOADS  CONCENTRATED  /  9 2 2.E3
*
MESH  VECTOR=LOAD NSYMBOLS=MYNODES NNUMBERS=MYNODES
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A99A  CANTILEVER BEAM, STANDARD T CROSS-SECTION, SHELL
*
DATABASE  CREATE
WRITE  FILENAME='a99a.lis'
*
EPLINE  NAME=FLANGE
   25  4 11 7 3  TO  32  4 11 7 3
   89  4 11 7 3  TO  96  4 11 7 3
EPLINE  NAME=WEB
  177  4 11 7 3  TO  192 4 11 7 3
*
NLIST  N9
*
VIEW ABC 2 3 1
SET  DIAGRAM=GRID OUTLINE=SHELL
MESH  CONTOUR=DY ORGINAL=YES VIEW=ABC SUBFRAME=12
MESH  CONTOUR=DZ ORGINAL=YES
COLOR EFILL=SHELL
MESH  PLINES=FLANGE SUBFRAME=12 OUTLINE=SHELL
COLOR LINE
MESH  PLINES=WEB OUTLINE=SHELL HIDDEN=NO
*
SHELLSURFACE  PLOTRESULTS=TOP
SET  PLOTORIENTATION=PORTRAIT
ELINE  LINENAME=FLANGE KIND=SXX OUTPUT=ALL SUBFRAME=12
SHELLSURFACE  PLOTRESULTS=MID
ELINE  LINENAME=FLANGE KIND=SXX
SHELLSURFACE  PLOTRESULTS=BOTTOM
ELINE  LINENAME=FLANGE KIND=SXX OUTPUT=ALL SUBFRAME=12
*
```

SOLVIA-POST input (cont.)

```
SHELLSURFACE  PLOTRESULTS=TOP
ELINE  LINENAME=WEB KIND=SXX OUTPUT=ALL
SHELLSURFACE  PLOTRESULTS=MID
ELINE  LINENAME=WEB KIND=SXX SUBFRAME=12
SHELLSURFACE  PLOTRESULTS=BOTTOM
ELINE  LINENAME=WEB KIND=SXX OUTPUT=ALL
*
SHELLSURFACE  PLOTRESULTS=TOP
SET  PLOTORIENTATION=PORTRAIT
ELINE  LINENAME=FLANGE KIND=SXY OUTPUT=ALL SUBFRAME=12
SHELLSURFACE  PLOTRESULTS=MID
ELINE  LINENAME=FLANGE KIND=SXY
SHELLSURFACE  PLOTRESULTS=BOTTOM
ELINE  LINENAME=FLANGE KIND=SXY OUTPUT=ALL SUBFRAME=12
*
SHELLSURFACE  PLOTRESULTS=TOP
ELINE  LINENAME=WEB KIND=SXY OUTPUT=ALL
SHELLSURFACE  PLOTRESULTS=MID
ELINE  LINENAME=WEB KIND=SXY OUTPUT=ALL SUBFRAME=12
SHELLSURFACE  PLOTRESULTS=BOTTOM
ELINE  LINENAME=WEB KIND=SXY OUTPUT=ALL
*
SHELLSURFACE  PLOTRESULTS=TOP
ELINE  LINENAME=FLANGE KIND=SXZ OUTPUT=ALL SUBFRAME=12
SHELLSURFACE  PLOTRESULTS=MID
ELINE  LINENAME=FLANGE KIND=SXZ
SHELLSURFACE  PLOTRESULTS=BOTTOM
ELINE  LINENAME=FLANGE KIND=SXZ OUTPUT=ALL SUBFRAME=12
*
SHELLSURFACE  PLOTRESULTS=TOP
ELINE  LINENAME=WEB KIND=SXZ OUTPUT=ALL
SHELLSURFACE  PLOTRESULTS=MID
ELINE  LINENAME=WEB KIND=SXZ SUBFRAME=12
SHELLSURFACE  PLOTRESULTS=BOTTOM
ELINE  LINENAME=WEB KIND=SXZ OUTPUT=ALL
*
SHELLSURFACE  PLOTRESULTS=TOP
ELINE  LINENAME=FLANGE KIND=MISES OUTPUT=ALL SUBFRAME=12
SHELLSURFACE  PLOTRESULTS=MID
ELINE  LINENAME=FLANGE KIND=MISES
SHELLSURFACE  PLOTRESULTS=BOTTOM
ELINE  LINENAME=FLANGE KIND=MISES OUTPUT=ALL SUBFRAME=12
*
SHELLSURFACE  PLOTRESULTS=TOP
ELINE  LINENAME=WEB KIND=MISES OUTPUT=ALL
SHELLSURFACE  PLOTRESULTS=MID
ELINE  LINENAME=WEB KIND=MISES SUBFRAME=12
SHELLSURFACE  PLOTRESULTS=BOTTOM
ELINE  LINENAME=WEB KIND=MISES OUTPUT=ALL
*
END
```

EXAMPLE A100

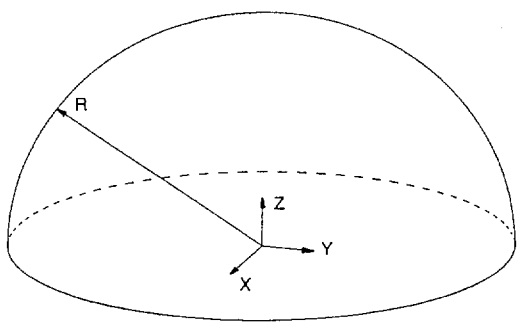
FREQUENCY ANALYSIS OF A SPHERICAL DOME, PLANE STRESS3

Objective

To verify the PLANE STRESS3 (membrane) element to model a spherical surface in a frequency analysis.

Physical Problem

A hemisphere is considered as shown in the figure below. The boundary conditions at Z = 0 are symmetrical.



$R = 4.5$ m

$h = 0.08$ m

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

$\nu = 0.3$

$\rho = 3000$ kg/m^3

Finite Element Model

A sector of one radian of the spherical surface is modeled using 160 nine node PLANE STRESS3 elements, see left top figure on page A100.3. Skew system directions are used to define the boundary conditions in the non-global circumferential directions. A consistent mass matrix assumption is used in the analysis and the subspace iteration method is used to calculate the lowest membrane frequency of the hemisphere.

Solution Results

The analytical solution to this problem of natural frequencies of a closed spherical membrane shell can be found in [1], page 338. The lower branch solution with n = 2 gives

$$2\left(1-\nu^2\right)\Omega^2 = 7 + 3\nu - \sqrt{\left(7+3\nu\right)^2 - 16\left(1-\nu^2\right)} \quad \text{where} \quad \Omega^2 = \frac{\rho \cdot \omega^2 \cdot R^2}{E}$$

Insertion of numerical values gives

$$f = \frac{\omega}{2\pi} = 67.1 \text{ Hz}$$

The input data of the SOLVIA analysis can be found on page A100.4.

SOLVIA calculates the fundamental frequency $f = 67.1$ Hz for the PLANE STRESS3 element. The mode shape is shown in the right top figure on page A100.3.

Using a PLANE AXISYMMETRIC finite element model the two lowest frequencies are calculated as

$f_1 = 67.1$ Hz

$f_2 = 84.7$ Hz

The second frequency corresponds to the lower branch solution with n = 4 which gives the analytical solution f = 84.3 Hz. The mode shapes are shown in the bottom figures on page A100.3.

User Hints

- The upper branch solution for n = 0 gives the natural frequency solution $\Omega^2 = \frac{2}{(1-\nu)}$

  This mode is normally referred to as the fundamental mode for a closed spherical membrane shell. To calculate this mode the boundary conditions along the line $\varphi = 90$ need to be antisymmetric. Further discussion can be found in [1].

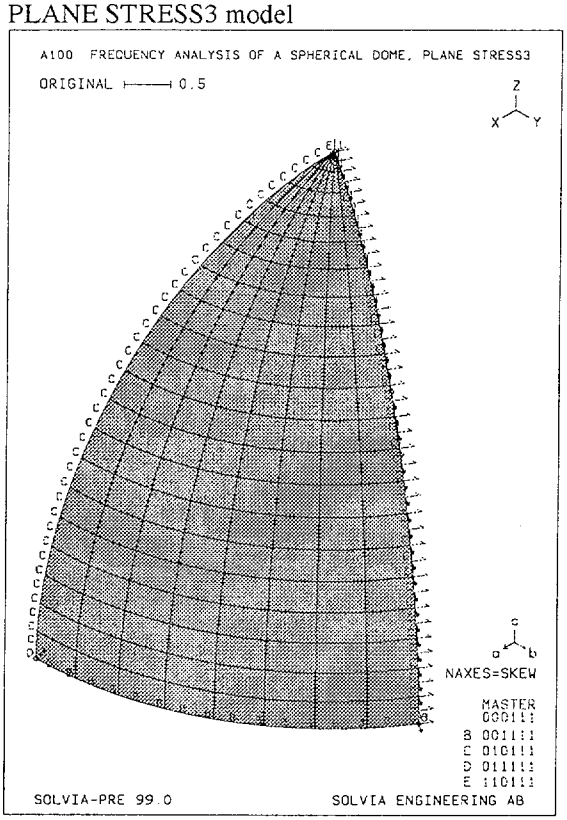
- A sector of one radian is considered in the PLANE STRESS3 model and in the AXISYMMETRIC model. The translational mass is $4.860 \cdot 10^3$ kg in the models and can be listed with the command MASSPROPERTIES. The AXISYMMETRIC model has only translational mass in the radial (Y) and the axial (Z) directions.

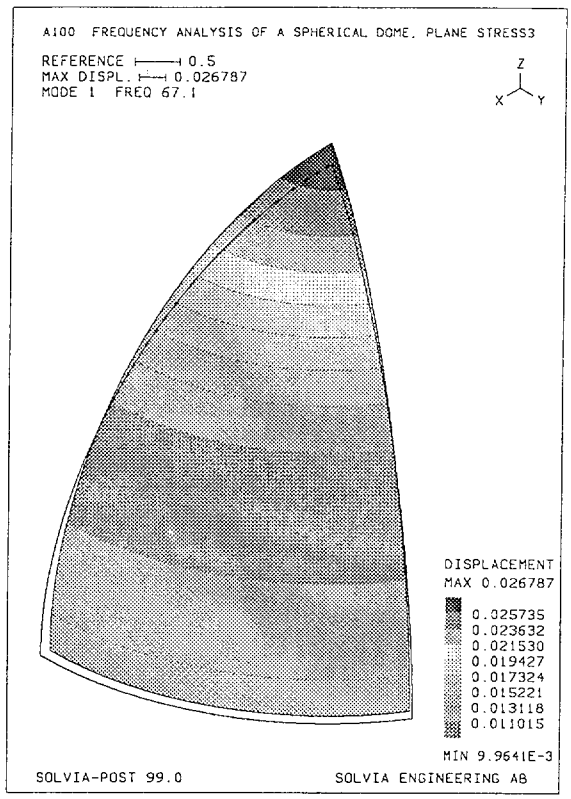
- The bending stiffness is not important in this example for the considered modes. Hence, the model with PLANE STRESS3 elements and the model with PLANE AXISYMMETRIC elements give practically the same results.

Reference

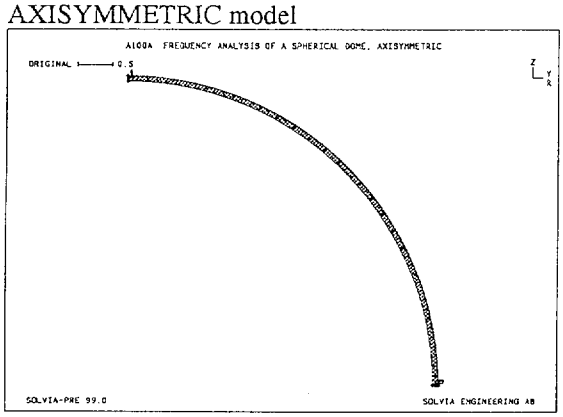
[1] Kraus, H., Thin Elastic Shells, John Wiley & Sons, 1967, pp. 336-339.

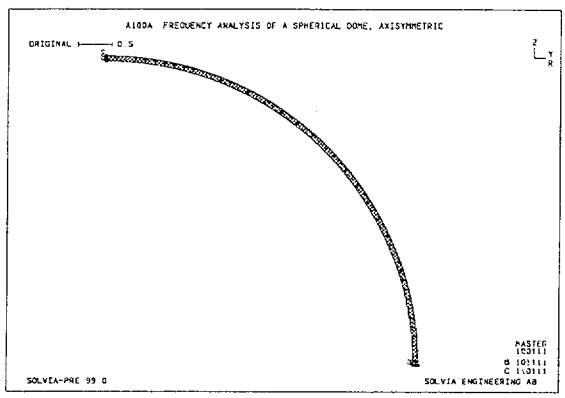
PLANE STRESS3 model

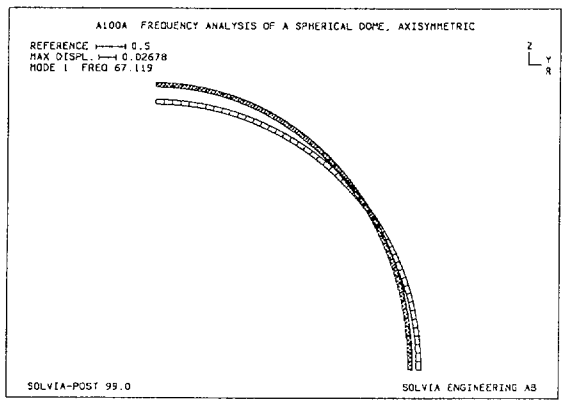


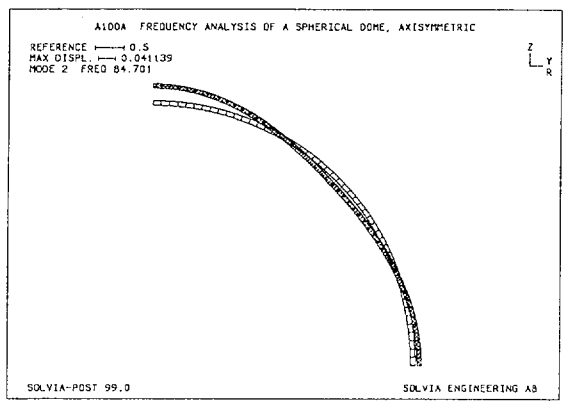


AXISYMMETRIC model









SOLVIA-PRE input, PLANE STRESS3 model

```
HEADING  'A100  FREQUENCY ANALYSIS OF A SPHERICAL DOME, PLANE
STRESS3'
*
DATABASE  CREATE
MASTER  IDOF=000111
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=CONSISTENT
FREQUENCIES  SUBSPACE-ITERATION  NEIG=1 SSTOL=1.E-10 IACCN=1
*
SYSTEM  1 SPHERICAL
COORDINATES
ENTRIES NODE  R    THETA       PHI
         1   4.5     0.         0.
         2   4.5     0.        90.
         3   4.5    57.2957795 90.
*
SKEWSYSTEMS  VECTORS
 1  0.5403023 0.841471 0. -0.841471 0.5403023 0.
*
MATERIAL  1  ELASTIC  E=2.E10 NU=0.3 DENSITY=3000.
*
EGROUP  1  PLANE STRESS3
EDATA  /  1  0.08
GSURFACE  1 2 3 1  EL1=20 EL2=8 NODES=9 SYSTEM=1 BLENDING=ANGLES
*
NSKEWS  INPUT=LINE  /  1 3  1
FIXBOUNDARIES  2  INPUT=LINES  /  1 2  /  1 3
FIXBOUNDARIES  3  INPUT=LINES  /  2 3
FIXBOUNDARIES  1  INPUT=NODES  /  1
*
SET  PLOTORIENTATION=PORTRAIT
MESH  BCODE=ALL NNUMBERS=MYNODES NAXES=SKEW NSYMBOLS=MYNODES
SOLVIA
END
```

SOLVIA-POST input

```
*  A100  FREQUENCY ANALYSIS OF A SPHERICAL DOME, PLANE STRESS3
*
DATABASE  CREATE
*
WRITE 'a100.lis'
MASS-PROPERITES
FREQUENCIES
*
SET  PLOTORIENTATION=PORTRAIT
SET  RESPONSETYPE=VIBRATIONMODE ORIGINAL=YES OUTLINE=YES
MESH  CONTOUR=DISPLACEMENTS
END
```

SOLVIA-PRE input, AXISYMMETRIC model

```
HEADING  'A100A  FREQUENCY ANALYSIS OF A SPHERICAL DOME,
AXISYMMETRIC'
*
DATABASE  CREATE
MASTER  IDOF=100111
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=CONSISTENT
FREQUENCIES  SUBSPACE-ITERATIONS  NEIG=2 SSTOL=1.E-10
*
SYSTEM  1 CYLINDRICAL
COORDINATES
ENTRIES NODE  R    THETA   XL
         1   4.46    0.  TO
         3   4.54    0.
         4   4.46   90.  TO
         6   4.54   90.
*
MATERIAL  1  ELASTIC  E=2.E10 NU=0.3 DENSITY=3000.
*
EGROUP  1  PLANE AXISYMMETRIC
GSURFACE  3 6 4 1  EL1=50 EL2=1 SYSTEM=1 NODES=8
*
FIXBOUNDARIES  2  /  4 5 6
FIXBOUNDARIES  3  /  1 2 3
*
SET HEIGHT=0.25
MESH  SMOOTHNESS=YES NSYMBOLS=MYNODES NNUMBER=MYNODES
MESH  SMOOTHNESS=YES NSYMBOLS=MYNODES BCODE=ALL
SOLVIA
END
```

SOLVIA-POST input

```
*  A100A  FREQUENCY ANALYSIS OF A SPHERICAL DOME, AXISYMMETRIC
*
DATABASE  CREATE
*
WRITE 'a100a.lis'
MASS-PROPERTIES
FREQUENCIES
*
SET  RESPONSETYPE=VIBRATIONMODE
SET  SMOOTHNESS=YES ORIGINAL=YES
MESH  TIME=1
MESH  TIME=2
END
```

EXAMPLE A101

DYNAMIC EXCITATION OF A BEAM STRUCTURE

Objective

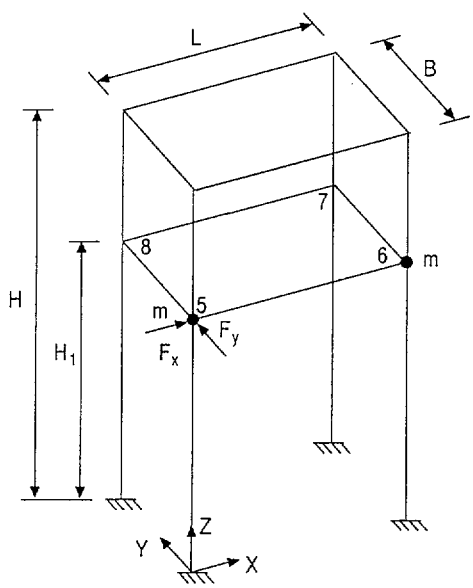
To verify the use of NHIST-PORTHOLE = COMBINATION in the SOLVIA-POST command DATABASE when different boundary conditions of a model are used in a dynamic analysis.

Physical Problem

A beam structure is subjected to a dynamic excitation by two concentrated loads acting at a corner node of the structure in the horizontal directions X and Y, see figure. The time dependence of the loads is described by the time functions shown in figures on page A101.4. The time functions are the same as used in Example A79.

The geometry is also the same as used in Example A79 except that an extra point mass has been added at node 5 to create a convenient geometric symmetry plane at X=0.75.

The response at the structure nodes 5 and 7 are to be calculated.



Geometry

$H = 3.0$ m $\quad L = 1.5$ m

$H_1 = 2.0$ m $\quad B = 1.0$ m

Circular section BEAM elements

Diameter $D = 0.020$ m

Material

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

$\nu = 0.3$

$\rho = 7850$ kg/m^3

Rayleigh damping $\alpha = 0$, $\beta = 5 \cdot 10^{-3}$

Point mass m = 10 kg

Loads

$F_x = 10 \cdot$ (time function 1)

$F_y = 10 \cdot$ (time function 2)

Finite Element Model

Due to the geometric symmetry a half model of the structure can be used. The load is nonsymmetric so symmetric and antisymmetric boundary conditions at nodes in the plane X=0.75 are used in the runs A101A and A101B, respectively. The corresponding models with boundary conditions are shown on page A101.4 and the input data is shown on pages A101.8 - A101.10. Direct integration is used with a time step of 0.005 sec.

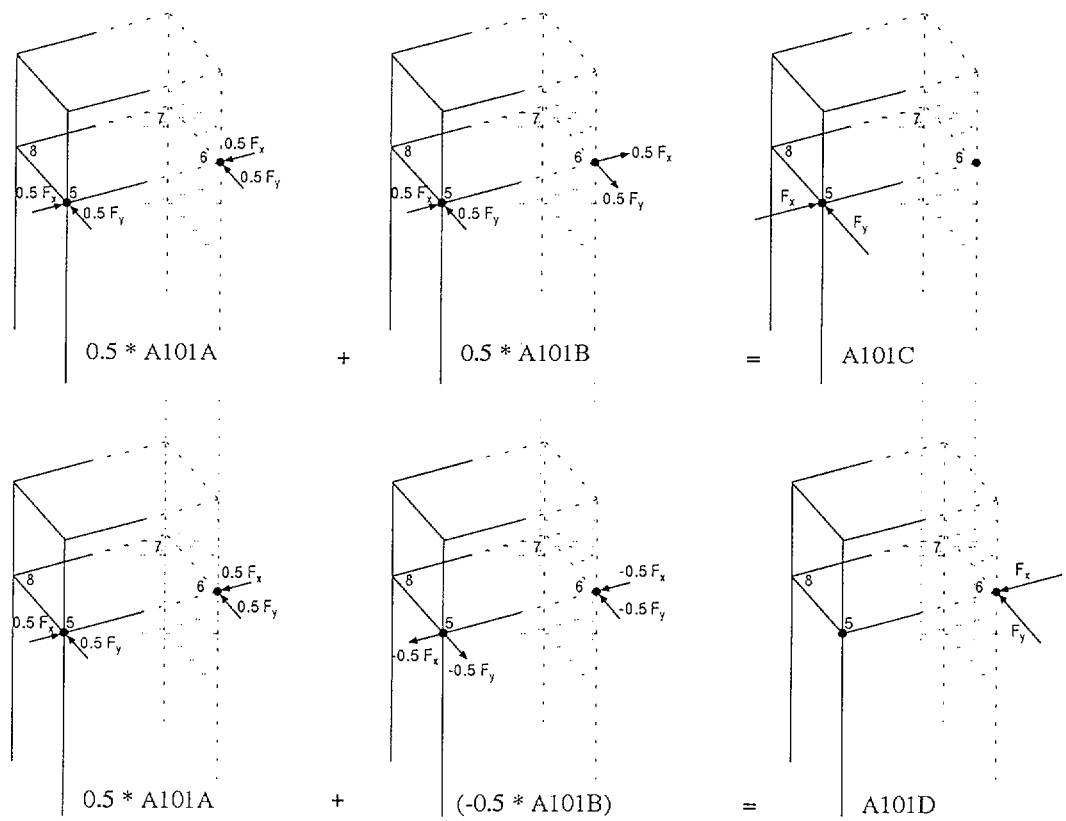
Solution results

Displacements and accelerations in the horizontal directions X and Y are saved on the nhist-porthole file for the nodes 5 and 8. For all other nodes results are only saved at every 550:th step, thus at 4 solution steps since the total number of steps is 2200. The use of the nhist-porthole file for selective saving of nodal results reduces dramatically the amount of result data.

The nhist-porthole standard file SOLVIA63.DAT is renamed to A101A.DAT and A101B.DAT, respectively, for the runs A101A and A101B using the DISKFILE command. The nhist-porthole files are added in the SOLVIA-POST run A101C using the factor 0.5 for both files, see figure below. Thus total results for node 5 (and 8) are obtained and the displacements, accelerations and response spectra in the X and Y directions at node 5 are shown on pages A101.5 - A101.6.

In run A101D the model response is the mirror image of the unmodelled portion of the structure since the factor 0.5 is used for the nhist-porthole file A101A.DAT and the factor -0.5 for the file A101B.DAT, see figure below. The results at the model node 8 are then the mirror image of the results at the structure node 7. Hence, the Y-direction results of the model node 8 are the same as for the structure node 7, but the X-direction results of the model node 8 are the negative of those for structure node 7. The model node 8 results are shown on pages A101.6.

For comparison, the complete structure has been modeled in run A101E, see figure on page A101.4, and results at the structure nodes 5 and 7 are shown on pages A101.7 - A101.8 as a reference solution. A summary of the analysis runs and the maximum and minimum displacements/accelerations are given in the tables on the next page. The combined results using the symmetric model are identical with the reference solution of the complete structure model when the proper signs of the response are considered.



Input data

Run	Name	Purpose	SOLVIA-PRE input on pages	SOLVIA-POST input on pages
1	A101A	Symmetric boundaries	A101.8 - A101.9	A101.9
2	A101B	Anti-symmetric boundaries	A101.9	A101.10
3	A101C	NHIST COMBINATION results at node 5	____	A101.10
4	A101D	NHIST COMBINATION results at node 7	____	A101.10 - A101.11
5	A101E	Reference solution full model	A101.11 - A101.12	A101.12

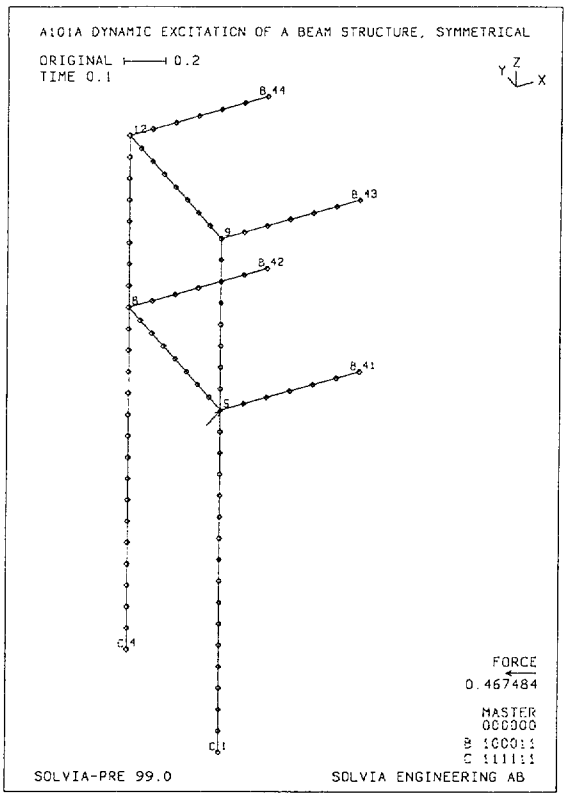
Extreme values of the response

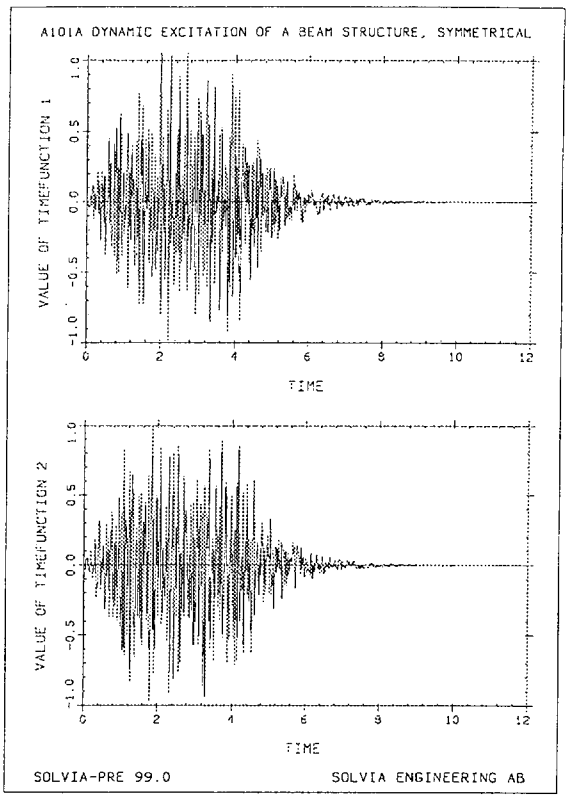
	Reference solution		Combined results using half model	
	Node 5	**Node 7**	**A101C: node 5**	**A101D: node 8**
Min. displacement X-direction	$-5.13882 \cdot 10^{-4}$	$-5.36335 \cdot 10^{-4}$	$-5.13882 \cdot 10^{-4}$	$-4.96034 \cdot 10^{-4}$ (note 1)
Max. displacement X-direction	$+5.88536 \cdot 10^{-4}$	$4.96034 \cdot 10^{-4}$	$+5.88536 \cdot 10^{-4}$	$5.36335 \cdot 10^{-4}$ (note 1)
Min. acceleration Y-direction	$-4.85002 \cdot 10^{-4}$	$-1.37974 \cdot 10^{-1}$	$-4.85002 \cdot 10^{-1}$	$-1.37974 \cdot 10^{-1}$
Max. acceleration Y-direction	$+5.10907 \cdot 10^{-1}$	$+1.68328 \cdot 10^{-1}$	$+5.10907 \cdot 10^{-1}$	$1.68328 \cdot 10^{-1}$

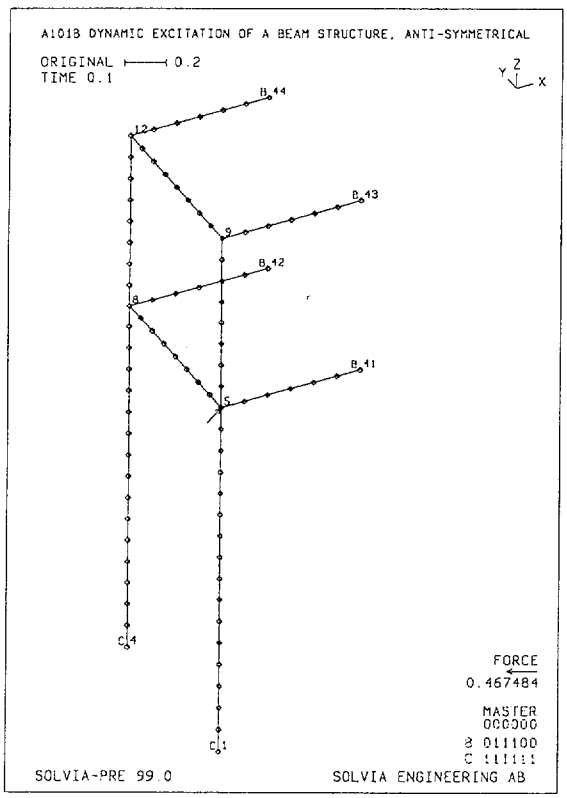
Note 1: Reverse the sign to obtain the extreme X-direction results for node 7 since the results for node 8 of the half model in run A101D are the mirror images of the results for structure node 7.

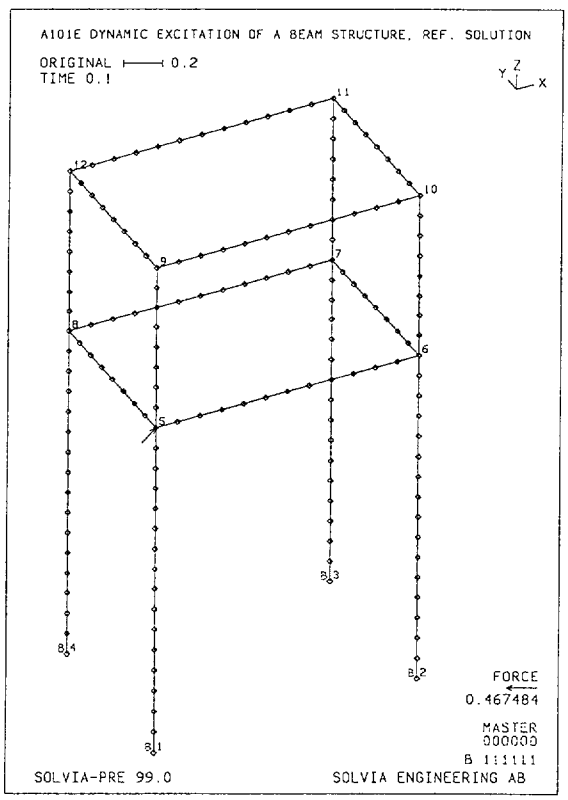
User Hints

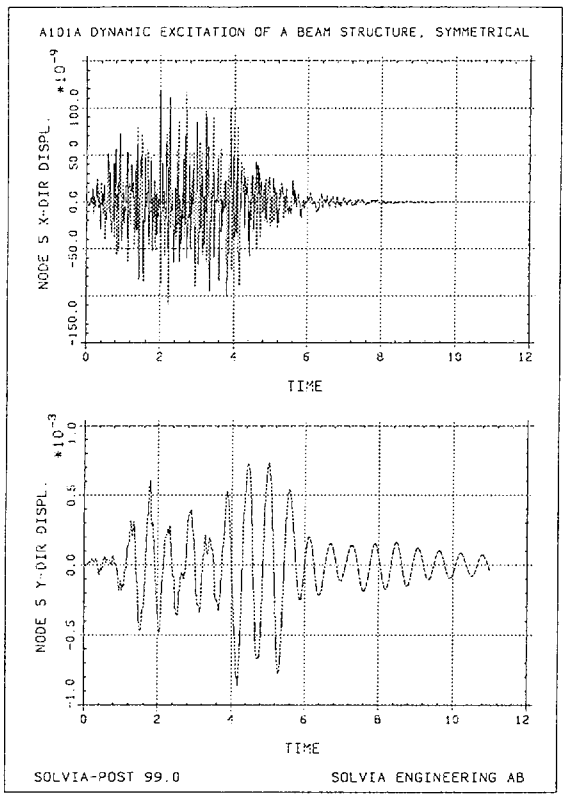
- The combination of nhist-porthole file results may be of interest in the dynamic analysis of a very large model which has symmetry properties. Only a portion of the full model need then be considered but using analysis cases with different boundary conditions. The results from these cases may then be superimposed to produce the full solution. This procedure may speed up the solution considerably when the symmetry portion can be solved in-core but the full model yields an out-of-core solution.

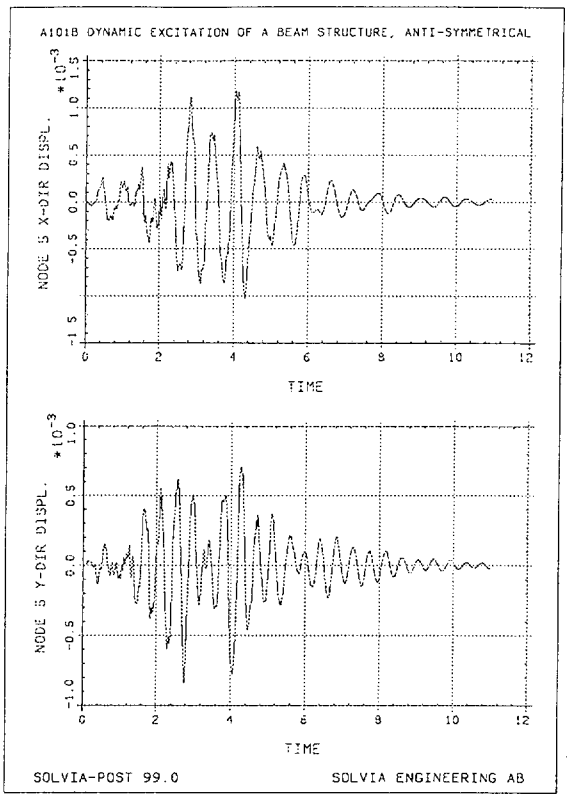

A101A DYNAMIC EXCITATION OF A BEAM STRUCTURE, SYMMETRICAL
ORIGINAL 0.2
TIME 0.1
FORCE
0.467484
MASTER
000000
B 100011
C 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

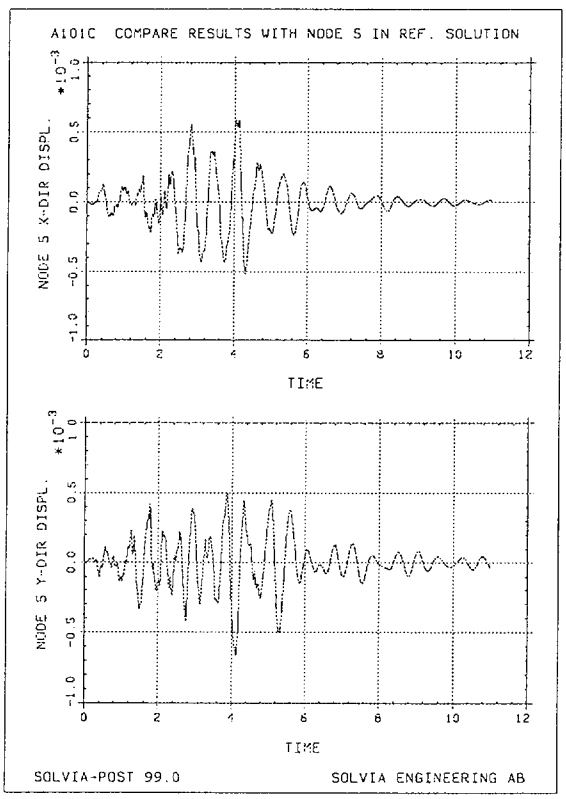

A101A DYNAMIC EXCITATION OF A BEAM STRUCTURE, SYMMETRICAL
VALUE OF TIMEFUNCTION 1
TIME
VALUE OF TIMEFUNCTION 2
TIME
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

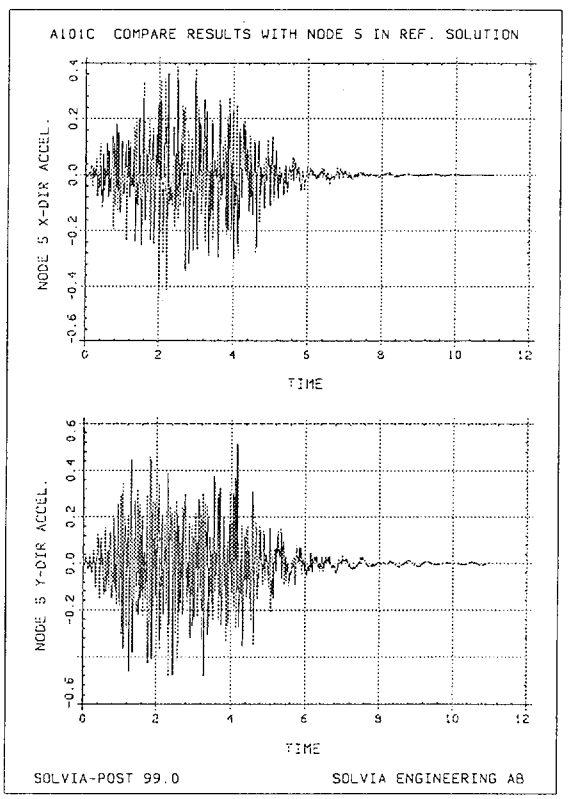

A101B DYNAMIC EXCITATION OF A BEAM STRUCTURE, ANTI-SYMMETRICAL
ORIGINAL 0.2
TIME 0.1
FORCE
0.467484
MASTER
000000
B 011100
C 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

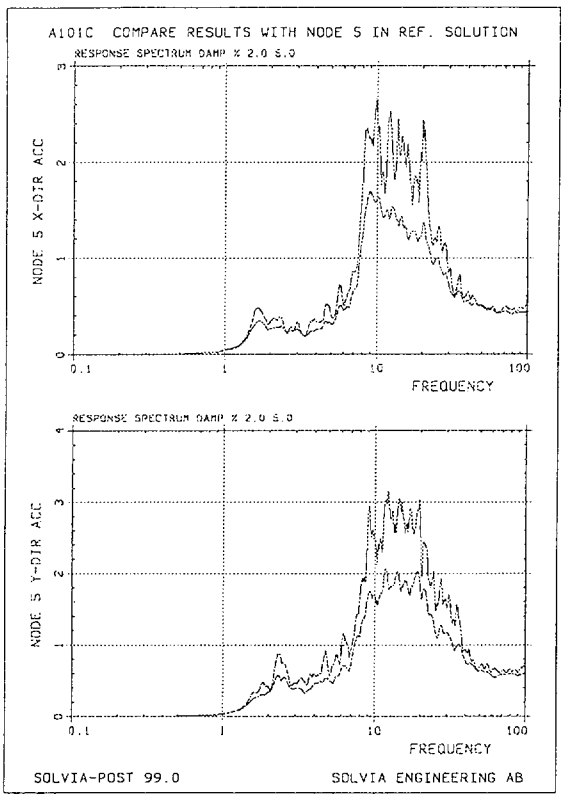

A101E DYNAMIC EXCITATION OF A BEAM STRUCTURE, REF. SOLUTION
ORIGINAL 0.2
TIME 0.1
FORCE
0.467484
MASTER
000000
B 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

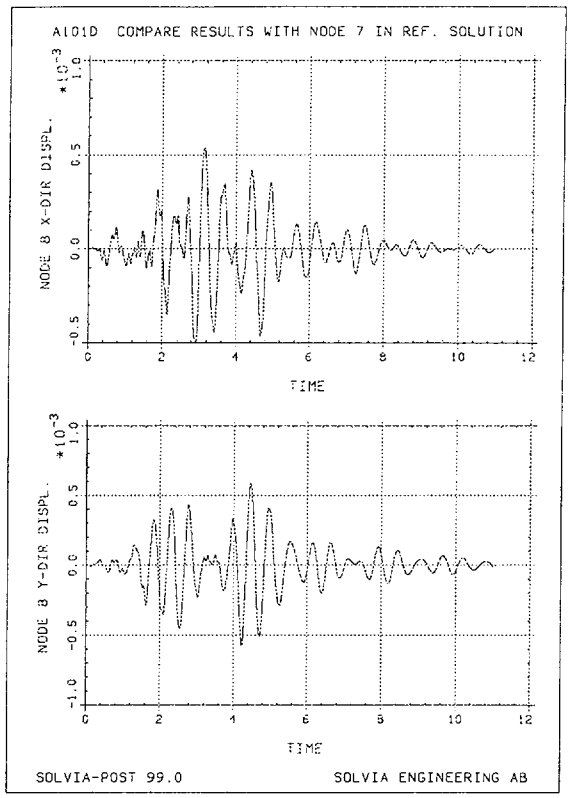

A101A DYNAMIC EXCITATION OF A BEAM STRUCTURE, SYMMETRICAL
NODE 5 X-DIR DISPL.
*10^-4
TIME
NODE 5 Y-DIR DISPL.
*10^-3
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

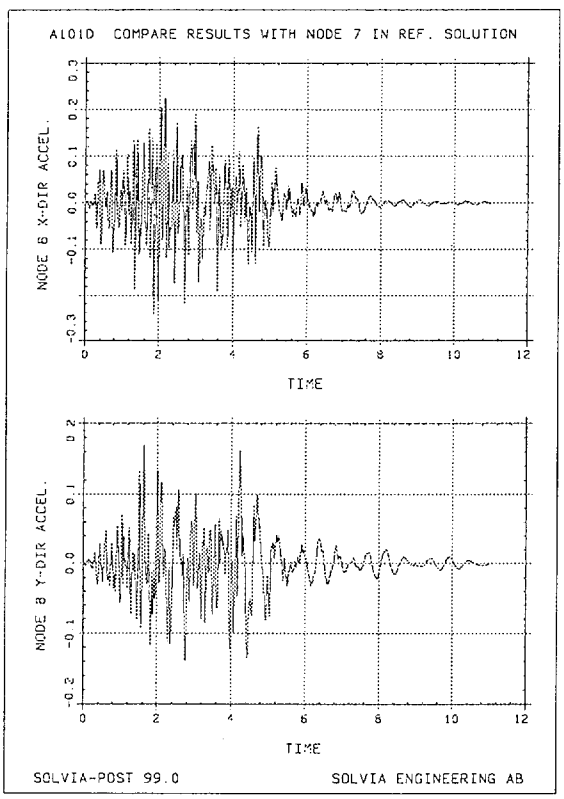

A101B DYNAMIC EXCITATION OF A BEAM STRUCTURE, ANTI-SYMMETRICAL
NODE 5 X-DIR DISPL.
*10^-3
TIME
NODE 5 Y-DIR DISPL.
*10^-3
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

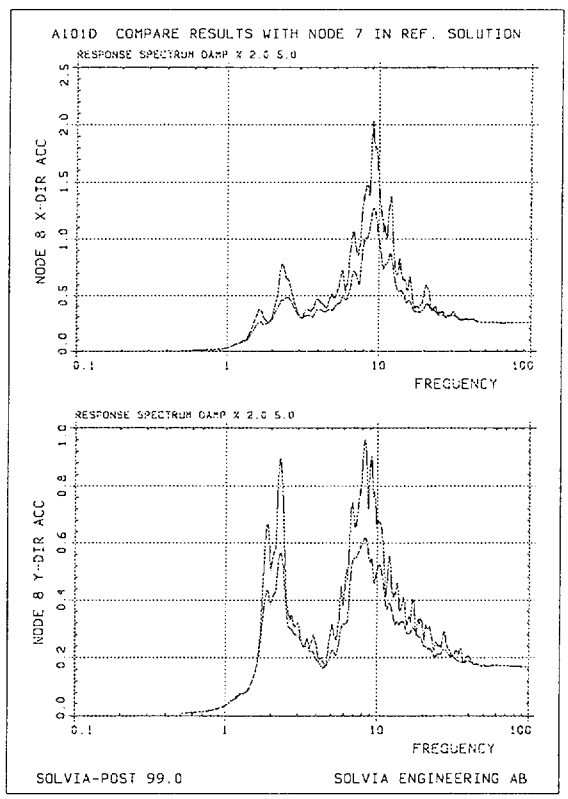

A101C COMPARE RESULTS WITH NODE 5 IN REF. SOLUTION
NODE 5 X-DIR DISPL.
*10^-3
TIME
NODE 5 Y-DIR DISPL.
*10^-3
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

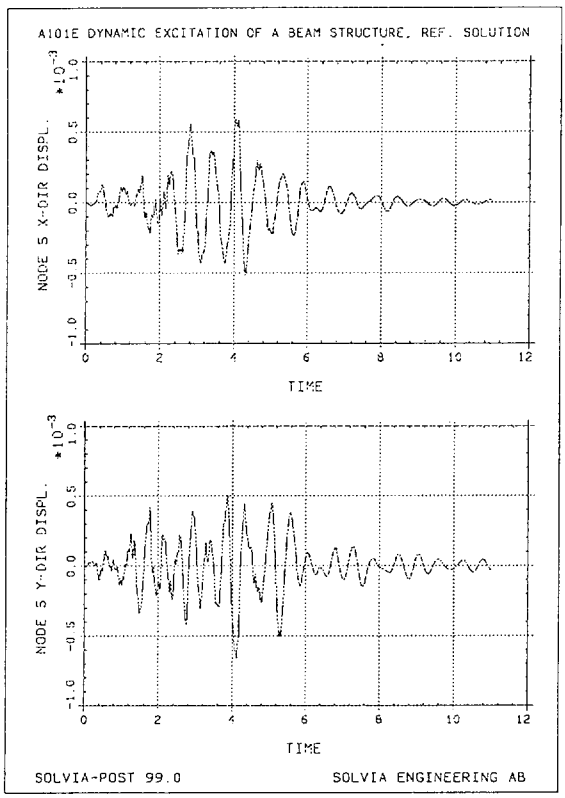

A101C COMPARE RESULTS WITH NODE 5 IN REF. SOLUTION
NODE 5 X-DIR ACCEL.
TIME
NODE 5 Y-DIR ACCEL.
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

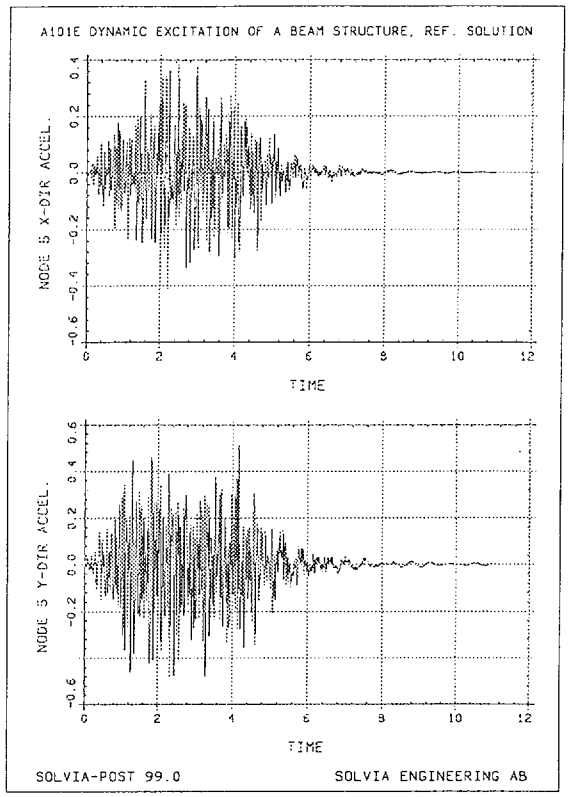

A101C COMPARE RESULTS WITH NODE 5 IN REF. SOLUTION
RESPONSE SPECTRUM DAMP % 2.0 5.0
NODE 5 X-DIR ACC
FREQUENCY
RESPONSE SPECTRUM DAMP % 2.0 5.0
NODE 5 Y-DIR ACC
FREQUENCY
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

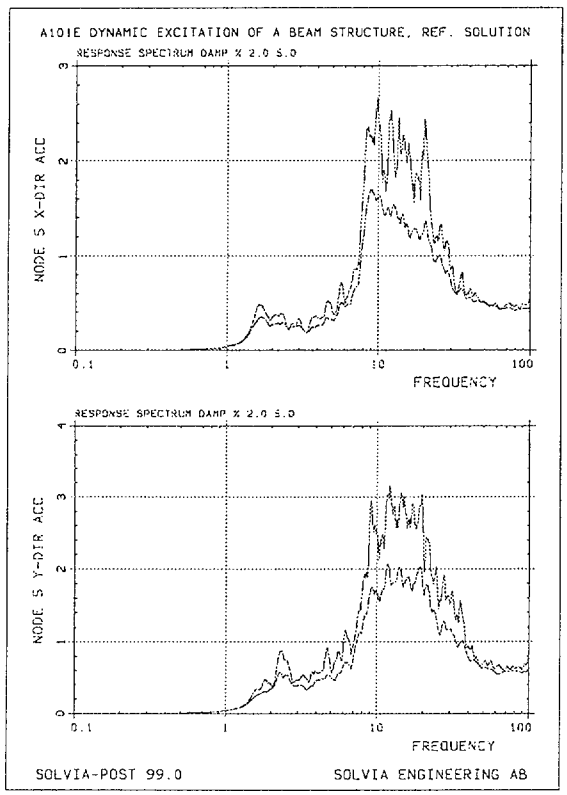

A101D COMPARE RESULTS WITH NODE 7 IN REF. SOLUTION
NODE 8 X-DIR DISPL.
TIME
NODE 8 Y-DIR DISPL.
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

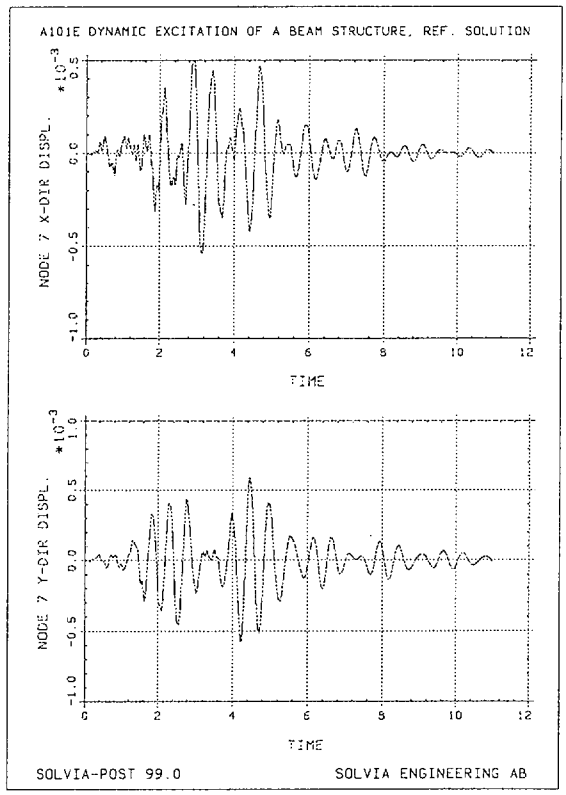

A101D COMPARE RESULTS WITH NODE 7 IN REF. SOLUTION
NODE 8 X-DIR ACCEL.
TIME
NODE 8 Y-DIR ACCEL.
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

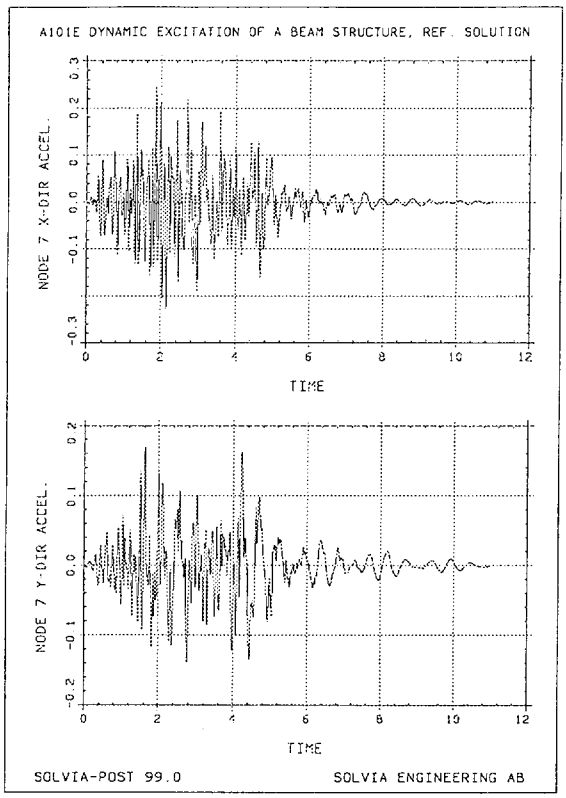

A101D COMPARE RESULTS WITH NODE 7 IN REF. SOLUTION
RESPONSE SPECTRUM DAMP % 2.0 5.0
NODE 8 X-DIR ACC
FREQUENCY
RESPONSE SPECTRUM DAMP % 2.0 5.0
NODE 8 Y-DIR ACC
FREQUENCY
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

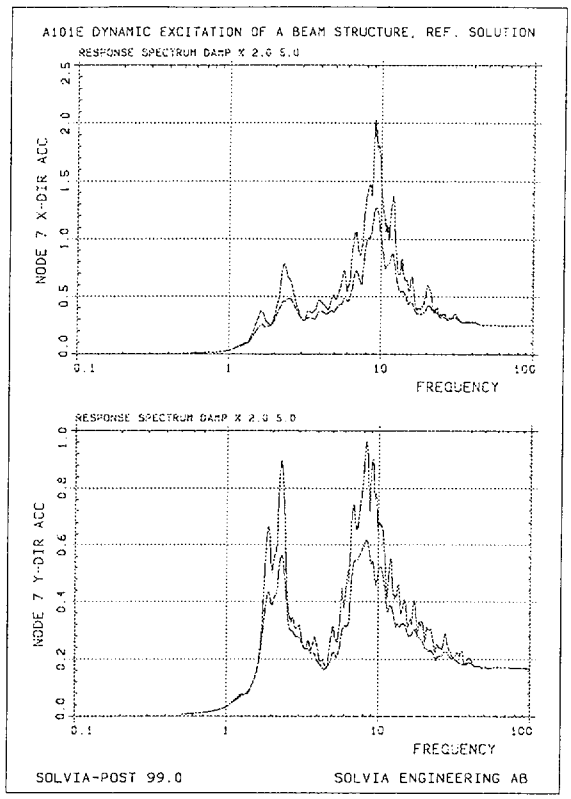

A101E DYNAMIC EXCITATION OF A BEAM STRUCTURE, REF. SOLUTION
NODE 5 X-DIR DISPL.
*10^-3
TIME
NODE 5 Y-DIR DISPL.
*10^-3
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A101E DYNAMIC EXCITATION OF A BEAM STRUCTURE, REF. SOLUTION
NODE 5 X-DIR ACCEL.
TIME
NODE 5 Y-DIR ACCEL.
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A101E DYNAMIC EXCITATION OF A BEAM STRUCTURE, REF. SOLUTION
RESPONSE SPECTRUM DAMP % 2.0 5.0
NODE 5 X-DIR ACC
FREQUENCY
RESPONSE SPECTRUM DAMP % 2.0 5.0
NODE 5 Y-DIR ACC
FREQUENCY
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A101E DYNAMIC EXCITATION OF A BEAM STRUCTURE, REF. SOLUTION
NODE 7 X-DIR DISPL.
*10^-3
TIME
NODE 7 Y-DIR DISPL.
*10^-3
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB





SOLVIA-PRE input

```
DATABASE CREATE
*
HEAD 'A101A DYNAMIC EXCITATION OF A BEAM STRUCTURE, SYMMETRICAL'
MASTER  NSTEP=2200 DT=0.005
ANALYSIS  DYNAMIC SPARSE=YES
RAYLEIGH-DAMPING ALPHA=0 BETA=5.E-3
PORTHOLE SAVEDEFAULT=NO
NHIST-PORTHOLE
 5 1 DISP / 8 1 DISP
 5 2 DISP / 8 2 DISP
 5 1 ACC / 8 1 ACC
 5 2 ACC / 8 2 ACC
NSAVESTEPS FIRST1=550 LAST1=2200 INCR1=550
ESAVESTEPS FIRST1=550 LAST1=2200 INCR1=550
*
MATERIAL 1 ELASTIC E=2.0E11 NU=0.3 DENSITY=7850
*
COORDINATES
 1 0 0 0  /  4 0 1 0 / 5  0 0 2
 8 0  1 2 /  9 0 0 3 / 12 0 1 3
41 0.75 0 2 / 42 0.75 1 2 / 43 0.75 0 3 / 44 0.75 1 3
*
EGROUP   1  BEAM MATERIAL=1
GLINE  N1=1  N2=5  AUX=41 EL=16 / GLINE  N1=4  N2=8  AUX=42 EL=16
GLINE  N1=5  N2=9  AUX=41 EL=8  / GLINE  N1=8  N2=12 AUX=42 EL=8
GLINE  N1=5  N2=41 AUX=9  EL=6  / GLINE  N1=42 N2=8  AUX=44 EL=6
GLINE  N1=8  N2=5  AUX=12 EL=8  / GLINE  N1=9  N2=43 AUX=5  EL=6
GLINE  N1=44 N2=12 AUX=42 EL=6  / GLINE  N1=12 N2=9  AUX=41 EL=8
```

SOLVIA-PRE input (cont.)

```
SECTION 1 CIRCULARSOLID D=20E-3
*
BOUNDARIES 111111 NODES / 1 2 3 4
BOUNDARIES 100011 NODES / 41 42 43 44
MASS / 5 10.
*
TIMEFUNCTION  1 / READ 'A79XDIR.DAT'
TIMEFUNCTION  2 / READ 'A79YDIR.DAT'
*
LOADS  CONCENTRATED
 * node dir fac timef
    5    1  10.  1
    5    2  10.  2
*
DISKFILE SOLVIA63.DAT 'A101A.DAT'
*
VIEW V1 -1 -2 1.5
SET PLOTORIENTATION=PORTRAIT VIEW=V1 NSYMBOL=Y NNNUMBERS=MYNODES
MESH BCODE=ALL VECTOR=LOAD TIME=0.100
AXIS 1 VMIN=-1. VMAX=1.
PLOT TIMEFUNCTION 1 YAXIS=1  SUBFRAME=12
PLOT TIMEFUNCTION 2 YAXIS=1
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A101A DYNAMIC EXCITATION OF A BEAM STRUCTURE, SYMMETRICAL
*
DISKFILE SOLVIA63.DAT 'A101A.DAT'
DATABASE  CREATE NHIST-PORTHOLE=YES
*
SET  PLOTORIENTATION=PORTRAIT
SET  DIAGRAM=GRID
NHISTORY  NODE=5 DIRECTION=1 KIND=DISP SUBFRAME=12
NHISTORY  NODE=5 DIRECTION=2 KIND=DISP
*
END
```

SOLVIA-PRE input

```
DATABASE OPEN
*
HEAD 'A101B DYNAMIC EXCITATION OF A BEAM STRUCTURE, ANTI-
SYMMETRICAL'
*
BOUNDARIES 011100 NODES / 41 42 43 44
*
DISKFILE SOLVIA63.DAT 'A101B.DAT'
*
SET PLOTORIENTATION=PORTRAIT VIEW=V1 NSYMBOL=Y NNNUMBERS=MYNODES
MESH BCODE=ALL VECTOR=LOAD TIME=0.100
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A101B DYNAMIC EXCITATION OF A BEAM STUCTURE
*
DISKFILE SOLVIA63.DAT 'A101B.DAT'
DATABASE  CREATE NHIST-PORTHOLE=YES
*
SET DIAGRAM=GRID PLOTORIENTATION=PORTRAIT
NHISTORY  NODE=5 DIRECTION=1 KIND=DISP SUBFRAME=12
NHISTORY  NODE=5 DIRECTION=2 KIND=DISP
*
END
```

SOLVIA-POST input

```
*  A101C DYNAMIC EXCITATION OF A BEAM STRUCTURE
*
DATABASE  CREATE NHIST-PORTHOLE=COMBINATION
 0.5 'A101A.DAT'
 0.5 'A101B.DAT'
*
HEAD 'A101C  COMPARE RESULTS WITH NODE 5 IN REF. SOLUTION'
WRITE  FILENAME='a101c.lis'
*
SET PLOTORIENTATION=PORTRAIT NEWPAGE=NO DIAGRAM=GRID
*
NHISTORY  NODE=5 DIRECTION=1 KIND=DISP SUBFRAME=12
NHISTORY  NODE=5 DIRECTION=2 KIND=DISP
*
NHISTORY  NODE=5 DIRECTION=1 KIND=ACC SUBFRAME=12
NHISTORY  NODE=5 DIRECTION=2 KIND=ACC
*
RESPONSE-SPECTRUM  KIND=ACCELERATION  NODE=5 DIRECTION=1,
          FSTART=0.5 FEND=100 FINC=1  AXES=4  SUBFRAME=12
          2. 5.
RESPONSE-SPECTRUM  KIND=ACCELERATION  NODE=5 DIRECTION=2,
          FSTART=0.5 FEND=100 FINC=1  AXES=4
          2. 5.
*
END
```

SOLVIA-POST input

```
*  A101D DYNAMIC EXCITATION OF A BEAM STRUCTURE
*
*
DATABASE  CREATE NHIST-PORTHOLE=COMBINATION
  0.5 'A101A.DAT'
 -0.5 'A101B.DAT'
*
HEAD 'A101D  COMPARE RESULTS WITH NODE 7 IN REF. SOLUTION'
WRITE  FILENAME='a101d.lis'
*
SET PLOTORIENTATION=PORTRAIT NEWPAGE=NO
SET RESPONSETYPE=TIMERESULT DIAGRAM=GRID
*
```

SOLVIA-POST input (cont.)

```
NHISTORY  NODE=8 DIRECTION=1 KIND=DISP SUBFRAME=12
NHISTORY  NODE=8 DIRECTION=2 KIND=DISP
*
NHISTORY  NODE=8 DIRECTION=1 KIND=ACC SUBFRAME=12
NHISTORY  NODE=8 DIRECTION=2 KIND=ACC
*
RESPONSE-SPECTRUM  KIND=ACCELERATION  NODE=8 DIRECTION=1,
          FSTART=0.5 FEND=100 FINC=1  AXES=4  SUBFRAME=12
          2. 5.
RESPONSE-SPECTRUM  KIND=ACCELERATION  NODE=8 DIRECTION=2,
          FSTART=0.5 FEND=100 FINC=1 AXES=4
          2. 5.
*
END
```

SOLVIA-PRE input

```
DATABASE CREATE
*
HEAD 'A101E DYNAMIC EXCITATION OF A BEAM STRUCTURE, REF. SOLUTION'
MASTER  NSTEP=2200 DT=0.005
ANALYSIS  DYNAMIC SPARSE=YES
RAYLEIGH-DAMPING ALPHA=0 BETA=5.E-3
PORTHOLE SAVEDEFAULT=NO
NHIST-PORTHOLE DISPSCAN=ABSMAX VELSCAN=ABSMAX ACCSCAN=ABSMAX
 5 1 DISP TO 8 1 DISP
 5 2 DISP TO 8 2 DISP
 5 1 ACC TO 8 1 ACC
 5 2 ACC TO 8 2 ACC
NSAVESTEPS FIRST1=550 LAST1=2200 INCR1=550
ESAVESTEPS FIRST1=550 LAST1=2200 INCR1=550
*
MATERIAL 1 ELASTIC E=2.0E11 NU=0.3 DENSITY=7850
*
COORDINATES
 1 0 0 0  /  2 1.5 0 0 /  3 1.5 1 0 /  4 0   1 0
 5 0 0 2  /  6 1.5 0 2 /  7 1.5 1 2 /  8 0   1 2
 9 0 0 3  / 10 1.5 0 3 / 11 1.5 1 3 / 12 0   1 3
*
EGROUP   1  BEAM MATERIAL=1
GLINE  N1=1  N2=5  AUX=2  EL=16 / GLINE  N1=2  N2=6  AUX=1  EL=16
GLINE  N1=3  N2=7  AUX=4  EL=16 / GLINE  N1=4  N2=8  AUX=3  EL=16
GLINE  N1=5  N2=9  AUX=2  EL=8  / GLINE  N1=6  N2=10 AUX=1  EL=8
GLINE  N1=7  N2=11 AUX=4  EL=8  / GLINE  N1=8  N2=12 AUX=3  EL=8
GLINE  N1=5  N2=6  AUX=9  EL=12 / GLINE  N1=6  N2=7  AUX=10 EL=8
GLINE  N1=7  N2=8  AUX=11 EL=12 / GLINE  N1=8  N2=5  AUX=12 EL=8
GLINE  N1=9  N2=10 AUX=5  EL=12 / GLINE  N1=10 N2=11 AUX=6  EL=8
GLINE  N1=11 N2=12 AUX=7  EL=12 / GLINE  N1=12 N2=9  AUX=8  EL=8
SECTION 1 CIRCULARSOLID D=20E-3
*
BOUNDARIES 111111 NODES / 1 2 3 4
MASSES
  5 10.
  6 10.
*
```

SOLVIA-PRE input (cont.)

```
TIMEFUNCTION  1 / READ 'A79XDIR.DAT'
TIMEFUNCTION  2 / READ 'A79YDIR.DAT'
*
LOADS  CONCENTRATED
 * node dir fac timef
    5    1  10.  1
    5    2  10.  2
*
VIEW V1 -1 -2 1.5
SET PLOTORIENTATION=PORTRAIT VIEW=V1 NSYMBOL=Y NNNUMBERS=MYNODES
MESH BCODE=ALL VECTOR=LOAD TIME=0.100
AXIS 1 VMIN=-1. VMAX=1.
PLOT TIMEFUNCTION 1 YAXIS=1  SUBFRAME=12
PLOT TIMEFUNCTION 2 YAXIS=1
*
SOLVIA
END
```

SOLVIA-POST input

```
*  A101E DYNAMIC EXCITATION OF A BEAM STRUCTURE, REF SOLUTION
*
*
DATABASE  CREATE NHIST-PORTHOLE=YES
WRITE  FILENAME='a101e.lis'
*
SET DIAGRAM=GRID
SET PLOTORIENTATION=PORTRAIT NEWPAGE=NO
*
NHISTORY  NODE=5 DIRECTION=1 KIND=DISP SUBFRAME=12
NHISTORY  NODE=5 DIRECTION=2 KIND=DISP
*
NHISTORY  NODE=5 DIRECTION=1 KIND=ACC SUBFRAME=12
NHISTORY  NODE=5 DIRECTION=2 KIND=ACC
*
RESPONSE-SPECTRUM  KIND=ACCELERATION  NODE=5 DIRECTION=1,
          FSTART=0.5 FEND=100 FINC=1  AXES=4  SUBFRAME=12
          2. 5.
RESPONSE-SPECTRUM  KIND=ACCELERATION  NODE=5 DIRECTION=2,
          FSTART=0.5 FEND=100 FINC=1  AXES=4
          2. 5.
*
NHISTORY  NODE=7 DIRECTION=1 KIND=DISP SUBFRAME=12
NHISTORY  NODE=7 DIRECTION=2 KIND=DISP
*
NHISTORY  NODE=7 DIRECTION=1 KIND=ACC SUBFRAME=12
NHISTORY  NODE=7 DIRECTION=2 KIND=ACC
*
RESPONSE-SPECTRUM  KIND=ACCELERATION  NODE=7 DIRECTION=1,
          FSTART=0.5 FEND=100 FINC=1  AXES=4  SUBFRAME=12
          2. 5.
RESPONSE-SPECTRUM  KIND=ACCELERATION  NODE=7 DIRECTION=2,
          FSTART=0.5 FEND=100 FINC=1  AXES=4
          2. 5.
*
END
```

EXAMPLE A102

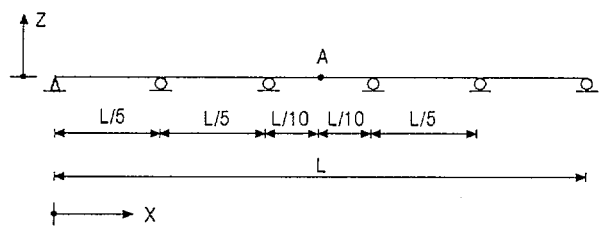
CONDITIONAL CASE COMBINATIONS FOR A MULTISPAN BEAM

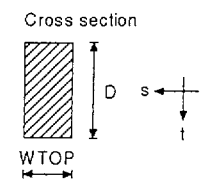
Objective

To demonstrate the use of conditional case combinations in SOLVIA-POST.

Physical Problem

A multispan beam structure is subjected to five independent load cases, see figures below. The objective is to determine the combinations of the five load cases that produce the maximum and the minimum moments at point A and to show the distributions of moments, shear forces , displacements and loads that correspond to these combinations.



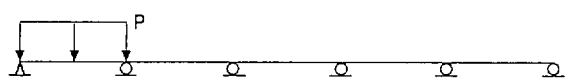


$L = 20$ m $\quad E = 30 \cdot 10^9$ N/m^2

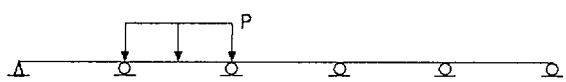
WTOP $= 0.1$ m $\quad \nu = 0.2$

$D = 0.2$ m $\quad P = 10000$ N/m

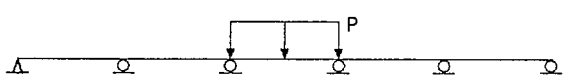
Load Case 1



Load Case 2

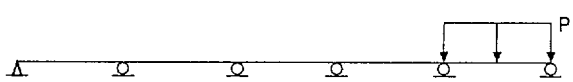


Load Case 3



Load Case 4

Load Case 5



Finite Element Model

The input data on pages A102.5 and A102.6 is used in the analysis. The model consists of 20 BEAM elements per span. Node 7 is positioned at the point where the maximum and minimum moments are to be calculated. The model including the boundary conditions and the five load cases is shown on page A102.3.

Solution results

Element 50, point 2 is located at node 7, see figure on page A102.3. Load cases with positive values of the moment MS at element 50, point 2, are added to case combination 1 using the SOLVIA-POST command CASE-COMBINATION N EPK-CONDITIONAL FUNCTION=ADDIFPOSITIVE, see the SOLVIA-POST input data. Load cases with negative moment MS are similarly added to case combination 2. The following moment values for the load cases and the case combinations were obtained:

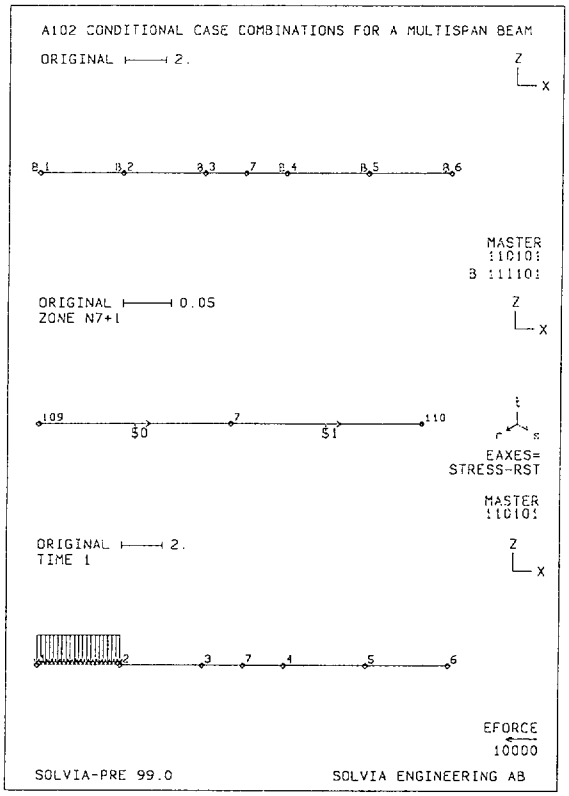
Load Case	Moment MS (Nm) at node 7
1	1045.7
2	-3159.7
3	11622.6
4	-3159.7
5	1045.7

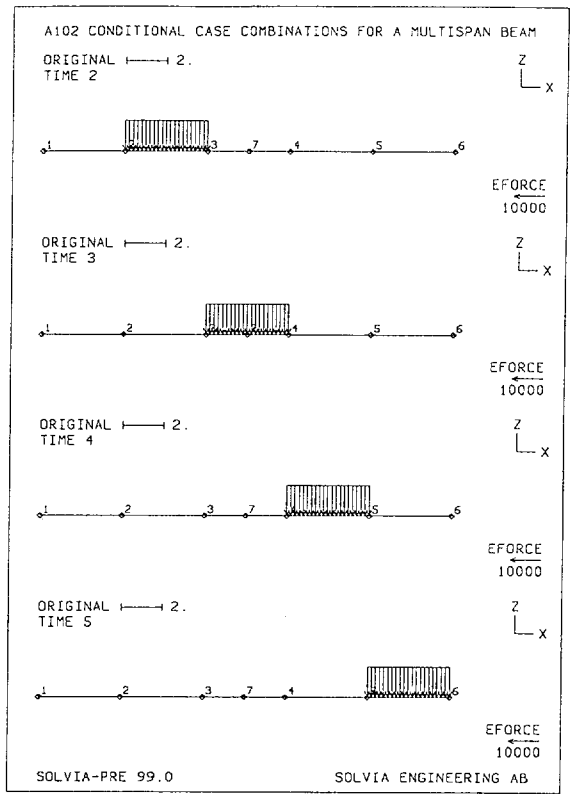
Case Combination	Moment MS (Nm) at node 7
1	13714.0
2	-6319.4

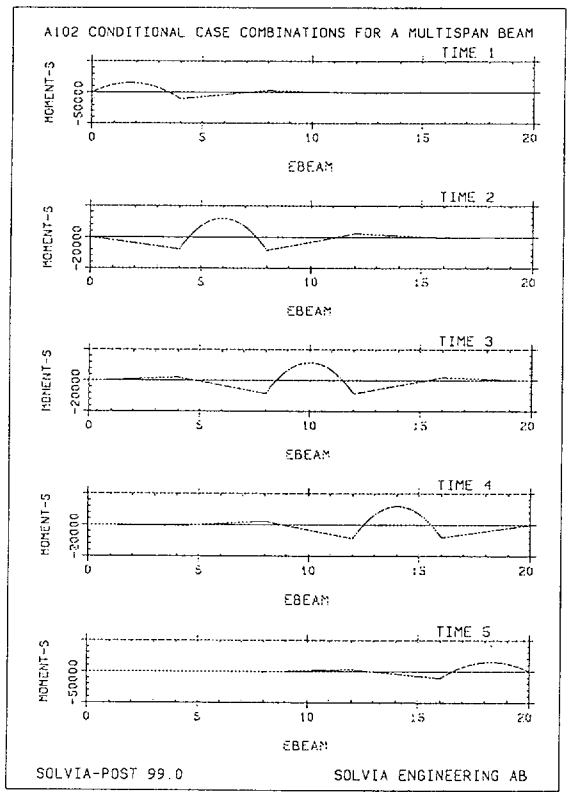
The distribution of moments and shear forces for the load cases and the case combinations are shown on pages A102.3 and A102.4. The applied load, the reactions and the deformations for the two case combinations are shown on page A102.4.

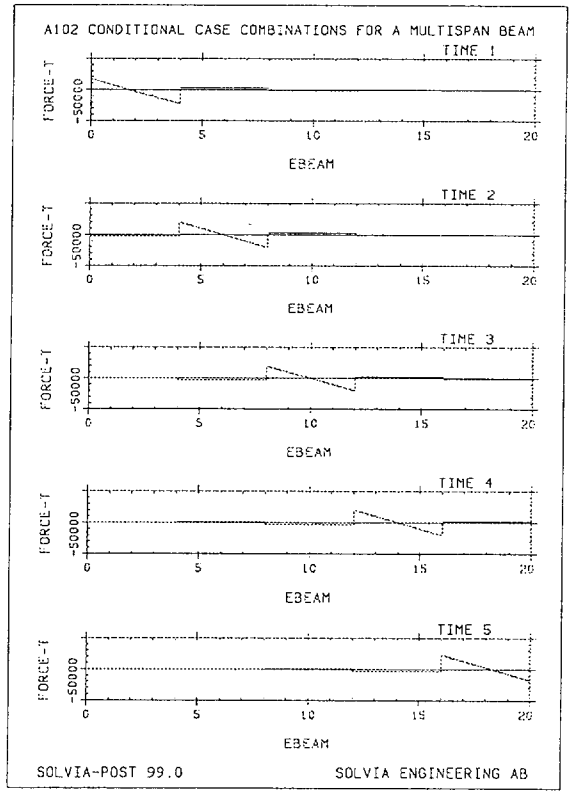
User Hints

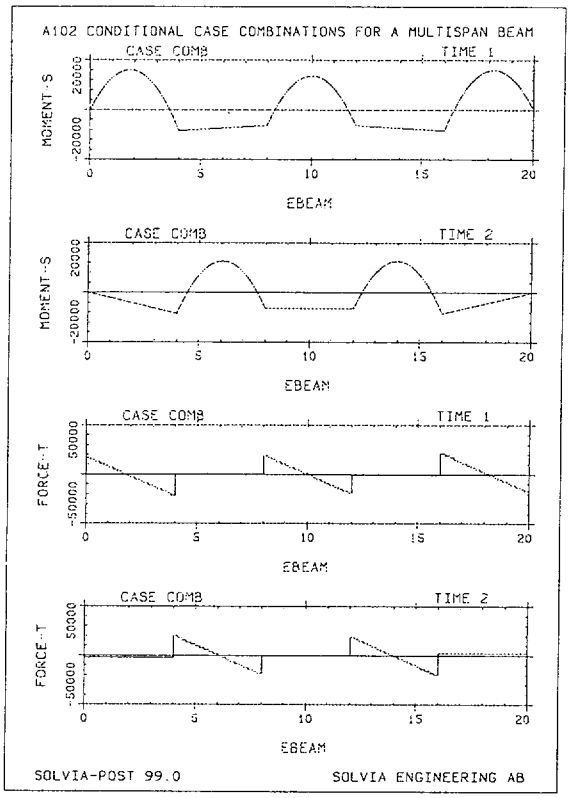
- For good accuracy in the distribution of moments, shear forces and reactions due to distributed loading it is important to use a sufficiently large number of elements in each span. The distributed loading of BEAM elements are applied using reversed fixed end forces/moments, which are statically equivalent to a linearly varying distributed loading under zero end displacements/-rotations of each element. If such a fixed end force/moment acts in a displacement/rotation degree-of-freedom that is fixed to zero, it is ignored when the load vector is assembled. Hence, such a fixed end force/moment is also not included in the reactions. In the present example with 20 elements per span the accuracy of the calculated reaction forces is about 5 percent. The accuracy of the calculated positive moment at node 7 is much better, namely about 0.3 percent.
- The current example consists of a small structure for ease of demonstration. However, when applied to a larger structure with hundreds of load cases, which shall be combined in some "worst" manner, the SOLVIA-POST command CASE-COMBINATION can be very powerful.

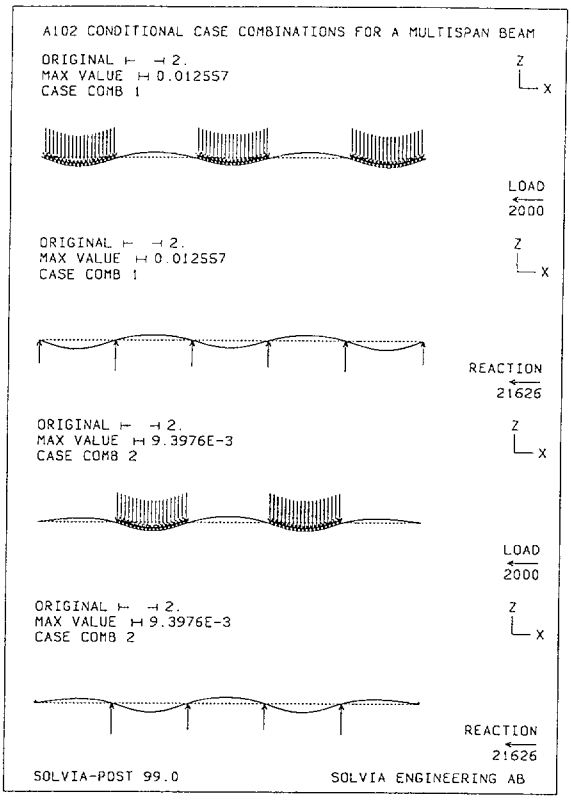

A102 CONDITIONAL CASE COMBINATIONS FOR A MULTISPAN BEAM
ORIGINAL 2.
MASTER
ORIGINAL 0.05
ZONE N7+1
EAXES=
STRESS-RST
MASTER
ORIGINAL 2.
TIME 1
EFORCE
10000
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A102 CONDITIONAL CASE COMBINATIONS FOR A MULTISPAN BEAM
ORIGINAL 2.
TIME 2
EFORCE
10000
ORIGINAL 2.
TIME 3
EFORCE
10000
ORIGINAL 2.
TIME 4
EFORCE
10000
ORIGINAL 2.
TIME 5
EFORCE
10000
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


A102 CONDITIONAL CASE COMBINATIONS FOR A MULTISPAN BEAM
TIME 1
TIME 2
TIME 3
TIME 4
TIME 5
MOMENT-S
EBEAM
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A102 CONDITIONAL CASE COMBINATIONS FOR A MULTISPAN BEAM
TIME 1
TIME 2
TIME 3
TIME 4
TIME 5
FORCE-T
EBEAM
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A102 CONDITIONAL CASE COMBINATIONS FOR A MULTISPAN BEAM
CASE COMB
TIME 1
MOMENT-S
25000
-20000
0
5
10
15
20
EBEAM
CASE COMB
TIME 2
MOMENT-S
20000
-20000
EBEAM
CASE COMB
TIME 1
FORCE-T
50000
-50000
EBEAM
CASE COMB
TIME 2
FORCE-T
50000
-50000
EBEAM
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


A102 CONDITIONAL CASE COMBINATIONS FOR A MULTISPAN BEAM
ORIGINAL ⊢ ⊣ 2.
MAX VALUE ⊢ 0.012557
CASE COMB 1
Z
X
LOAD
2000
ORIGINAL ⊢ ⊣ 2.
MAX VALUE ⊢ 0.012557
CASE COMB 1
Z
X
REACTION
21626
ORIGINAL ⊢ ⊣ 2.
MAX VALUE ⊢ 9.3976E-3
CASE COMB 2
Z
X
LOAD
2000
ORIGINAL ⊢ ⊣ 2.
MAX VALUE ⊢ 9.3976E-3
CASE COMB 2
Z
X
REACTION
21626
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'A102 CONDITIONAL CASE COMBINATIONS FOR A MULTISPAN BEAM'
*
DATABASE  CREATE
MASTER IDOF=110101
PARAMETER $E=20 $E1=10
*
COORDINATES
ENTRIES NODE X
         1   0. TO
         6  20.
         7  10.
*
MATERIAL 1 ELASTIC E=30E9 NU=0.2
*
EGROUP 1 BEAM RESULT=FORCE
SECTION 1 RECTANGULAR WTOP=0.1 D=0.2
BEAMVECTOR / 1  0. -1. 0.
*
LINE STRAIGHT 3 7 EL=$E1
LINE STRAIGHT 7 4 EL=$E1
LINE COMBINE  3 4 7
GLINE 1 2 -1 EL=$E
GLINE 2 3 -1 EL=$E
GLINE 3 4 -1 EL=$E
GLINE 4 5 -1 EL=$E
GLINE 5 6 -1 EL=$E
*
FIXBOUNDARIES  3 / 1 2 3 4 5 6
*
LCASE 1
LOADS  ELEMENT INPUT=LINES /  1 2 T 10.E3
LCASE 2
LOADS  ELEMENT INPUT=LINES /  2 3 T 10.E3
LCASE 3
LOADS  ELEMENT INPUT=LINES /  3 4 T 10.E3
LCASE 4
LOADS  ELEMENT INPUT=LINES /  4 5 T 10.E3
LCASE 5
LOADS  ELEMENT INPUT=LINES /  5 6 T 10.E3
*
SET  NSYMBOLS=MY NNUMBERS=MY VIEW=-Y PLOTORIENTATION=PORTRAIT
MESH  BCODE=ALL SUBFRAME=13
MESH  N7+1 ENUM=YES NNUM=YES NSYM=YES BCODE=ALL EAXES=STRESS
MESH  VECTOR=LOAD TIME=1
MESH  VECTOR=LOAD TIME=2 SUBF=14
MESH  VECTOR=LOAD TIME=3
MESH  VECTOR=LOAD TIME=4
MESH  VECTOR=LOAD TIME=5
*
SOLVIA
END
```

SOLVIA-POST input:

```
*  A102 CONDITIONAL CASECOMBINATIONS FOR A MULTISPAN BEAM
*
DATABASE  CREATE
WRITE  FILENAME='a102.lis'
SET PLOTORIENTATION=PORTRAIT
*
CASE-COMBINATION 1 EPK-CONDITIONAL FUNCTION=ADDIFPOSITIVE ,
    ELEMENT=50 POINT=2 KIND=MS
    1 1. TO 5 1.
    END DATA
CASE-COMBINATION 2 EPK-CONDITIONAL FUNCTION=ADDIFNEGATIVE ,
    ELEMENT=50 POINT=2 KIND=MS
    1 1. TO 5 1.
    END DATA
LIST CASE-COMBINATION
*
EPLINE  NAME=EBEAM  /  1  1 2  TO  100  1 2
SUBFRAME 15
ELINE EBEAM KIND=MS TIME=1
ELINE EBEAM KIND=MS TIME=2
ELINE EBEAM KIND=MS TIME=3
ELINE EBEAM KIND=MS TIME=4
ELINE EBEAM KIND=MS TIME=5
*
SUBFRAME 15
ELINE EBEAM KIND=FT TIME=1
ELINE EBEAM KIND=FT TIME=2
ELINE EBEAM KIND=FT TIME=3
ELINE EBEAM KIND=FT TIME=4
ELINE EBEAM KIND=FT TIME=5
*
SUBFRAME 14
SET RESPONSETYPE=CASECOMBINATION
ELINE EBEAM KIND=MS TIME=1
ELINE EBEAM KIND=MS TIME=2
ELINE EBEAM KIND=FT TIME=1
ELINE EBEAM KIND=FT TIME=2
*
SUBFRAME 14
SET ORIGINAL=DASHED VIEW=-Y
MESH VECTOR=LOAD TIME=1
MESH VECTOR=REAC TIME=1
MESH VECTOR=LOAD TIME=2
MESH VECTOR=REAC TIME=2
*
NLIST N7  TSTART=1 TEND=5 RESPONSETYPE=TIMERESULTS
NLIST N7  TSTART=1 TEND=2 RESPONSETYPE=CASECOMBINATION
ELIST E50 TSTART=1 TEND=5 RESPONSETYPE=TIMERESULTS
ELIST E50 TSTART=1 TEND=2 RESPONSETYPE=CASECOMBINATION
SUMMATION KIND=REACTIONS TSTART=1 TEND=2 RESPONSETYPE=CASECOMBINATION
END
```

INDEX

Acoustic cavity A56
Axisymmetric shell A2
Base motion A54
Beam
  curved A11, A12, A57
  on elastic foundation A41
Cantilever
  end loads A7, A8
  ground motion A54
  stiffened A23, A82
  tapered A22
  truss A13
  T-section A99
  Z-section A35
Cantilever under distributed load
  BEAM element A17
  ISOBEAM element A18
  PLANE STRAIN element A15
  PLANE STRESS element A14
  PLATE element A19
  SHELL element A20
  SOLID element A16
Cantilever frequencies
  BEAM elements A47
  ISOBEAM elements A48
  off-center masses A37
  PLANE STRAIN elements A45
  PLANE STRESS elements A44
  PLATE elements A49
  SHELL 4 node A66
  SHELL elements A50
  SOLID elements A46
Case combinations
  conditional A102
  NHIST-PORTHOLE A101
  dynamic A101
Centrifugal loading A69
Combining results from different boundary conditions
    static load cases A80
    dynamic A101
    harmonic response A81
Complex harmonic response A85, A86, A95
Cylinder
  ring A5, A6
  edge bending A2
  fluid-filled A34
  hemispherical end A36
  intersections A98
  orthotropic A94
  pinched A32, A33
  Scordelis-Lo roof A30, A31, A67
  thick A1
  thin A4
Damping
  weighted A26
  viscous A95
  hysteretic A95
Dome, spherical A24, A100
Dynamic analysis
  direct integration A52, A54, A55, A92, A101
  modal superposition A79, A93
Earthquake analysis
  response spectrum method A76, A87
  time history by modal superposition A79
Electric potential A63
End release of BEAM elements A88
Floor response spectra A79
FLUID elements A34, A53
Fracture mechanics A43
Ground motion A54
Harmonic response A77, A81
Heat convection A84
Heat conduction, transient
  PLANE element A71, A72
  TRUSS element A60, A73
Heat conduction, steady state
  AXISYMMETRIC element A62
  PLANE element A61
Heat flux loading A73
Heat generation A74, A75
Hemisphere
  gravity load A24
  point load A40
Intersection of cylindrical surfaces A98
Laminated shell A89, A90, A91
Membrane shell A24, A100
Offset of BEAM element axis A82
Pipe
  elbow A68
  T connection A98

Plate
circular A3
clamped A25
frequencies A51
laminated A91
orthotropic A39, A89, A90
sandwich A90
simply supported A27, A51
skew A38
stiffened A23
triangular A29
twisting A27, A28, A29
Plate frequencies
simply supported A51
clamped rhombic A59
Power flow A85
Response spectrum
load pulses A78
modal combination methods A87
Rigid body modes A96
Rigid links A23, A37
Ring, pinched
BEAM elements A10
ISOBEAM elements A9
PLATE elements A6
SHELL elements A5
Ring under water pressure A97
Roof
spherical A24
Scordelis-Lo A30, A31, A67
Sandwich plate A90
Shear center A99
Spherical dome frequencies A100
Stiffened shell A23, A82
Strain energy density A85
Stress concentration
around circular hole A42
Stress deviation quality measure A27, A42
Thermal eigenvalues A64
Thickness variation using SHELL
elements A22
Toroidal shell A68
Torsional stresses A65, A96, A99
Torsional frequencies
off-center masses A37
spring system A83
free-free U beam A96
Transition SHELL element A22
Travelling load A58
Truss structure A13, A21
Twisting of plate A27, A28, A29
Wave propagation
using FLUID3 elements A53
using GENERAL elements A70
using TRUSS elements A52
Water pressure load A97