Stevenson & Associates

A structural-mechanical consulting engineering firm

10 State Street, Woburn, Massachusetts 01801
Tel 617 932 9580 Fax 617 933 4428
BOSTON•CLEVELAND

July 2, 2002

American Electric Power
Nuclear Generation Group
500 Circle Drive
Buchanan, MI 49107

Subject: License for SOLVIA Computer Code

Let this letter serve as a license for American Electric Power Company to use SOLVIA copyrighted verification documentation[1] for the purposes of regulatory compliance in answering of questions as posed by the USNRC. This grants American Electric Power the right have a copy of this documentation and to send a copy of the same to the USNRC for their use in the review of the SOLVIA code.

The SOLVIA verification documents transmitted herewith each bear a SOLVIA copyright notice. The NRC is permitted to make the number of copies of the information contained in these reports which are necessary for its internal use in connection with generic and plant-specific reviews and approvals as well as the issuance, denial, amendment, transfer, renewal, modification, suspension, revocation or violation of a license, permit, order, or regulation, copyright protection notwithstanding. The NRC is permitted to make the number of copies beyond those necessary for its internal use which are necessary in order to have one copy available for public viewing in the appropriate docket files in the public document room in Washington, DC and in local public document rooms as may be required by NRC regulations if the number of copies submitted is insufficient for this purpose. Copies made by the NRC must include the copyright notice in all instances.

Stevenson & Associates hereby grants this license as US distributor of the SOLVIA code.

Very truly yours,

Walter Djordjevic
President

---

1 a) SOLVIA Finite Element Program, Version 99.0, SOLVIA Verification Manual Linear Examples, Report SE 99-4, SOLVIA Engineering AB, 2000.
b) SOLVIA Finite Element Program, Version 99.0, SOLVIA Verification Manual Nonlinear Examples, Report SE 99-5, SOLVIA Engineering AB, 2000.

ATTACHMENT 6 TO AEP:NRC:2520

SOLVIA ENGINEERING REPORT SE 99-5,
"SOLVIA VERIFICATION MANUAL NON-LINEAR EXAMPLES"

SOLVIA® Finite Element System
Version 99.0

SOLVIA Verification Manual
Nonlinear Examples

Report SE 99-5

SOLVIA Engineering AB

SOLVIA Verification Manual, Nonlinear Examples.

SOLVIA Engineering AB
Trefasgatan 3
SE-721 30 Västerås
Sweden

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

INTRODUCTORY REMARKS

The objective with this report is to present example solutions obtained with the SOLVIA-PRE, SOLVIA and SOLVIA-POST computer programs (the SOLVIA System) that verify and demonstrate their usage. Solutions to nonlinear analyses are presented in this report. Linear example solutions are presented in the companion report SE 99-4.

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

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

CONTENTS

EXAMPLE B1

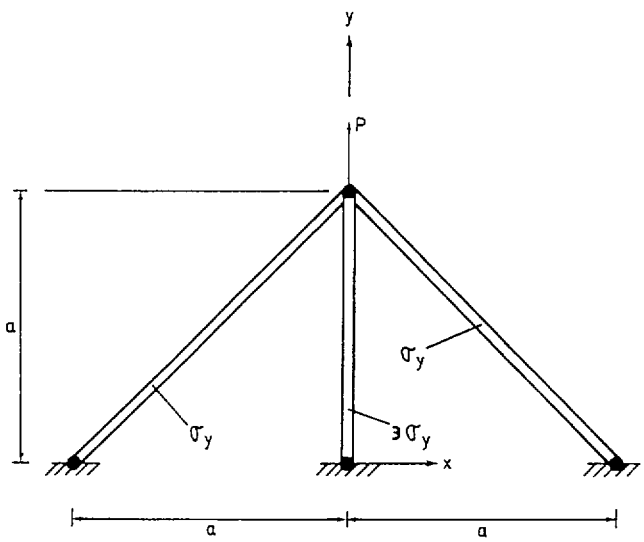
ELASTIC-PLASTIC ANALYSIS OF TRUSS STRUCTURE

Objective

To verify the geometric and material nonlinear behaviour of the TRUSS element.

Physical Problem

A three-member truss structure is subjected to a concentrated load as shown in the figure below. The material of the structure is assumed to be elastic-perfectly-plastic. The variation of the applied load with time is also shown in the figure below. P_y is the limit load considering small displacements only $\left(P_y = 100\sqrt{2} + 300 = 441.42 \text{ kips}\right)$.



Bar Areas $= 1.0 \text{ ft}^2$

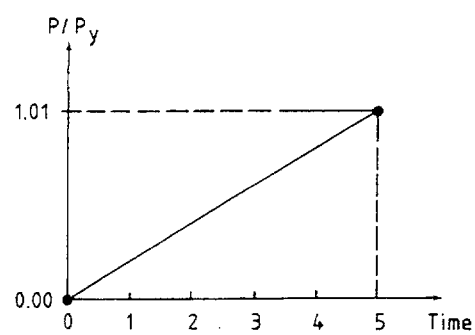
$E = 2.0 \cdot 10^5 \text{ kips/ft}^2$

$E_T = 0.0$

$a = 5.0 \text{ ft}$

$\sigma_y = 100.0 \text{ kips/ft}^2$

$P_y = 441.42 \text{ kips}$



Finite Element Model

In the finite element model considered, each member is represented by a 2-node elastic-plastic TRUSS element as shown in the top figure on page B1.3. Large displacements are prescribed with stiffness reformation and BFGS equilibrium iterations at every step.

Solution Results

The input data on page B1.4 gives the results as shown in the table below. Due to symmetry, the force in element 1 in element group 1 is the same as in element 2.

Time	Element group 1 Truss 1 Force [kips]	Element group 2 Truss 1 Force [kips]
1	26.12	52.23
2	52.23	104.45
3	78.35	156.66
4	100.00*	215.17
5	100.00*	300.00*

* Plastic

A check of equilibrium at time 5 gives, since the displacement in the Y-direction for node 4 is calculated to be 0.327614 inch:

Applied force : $1.01 \cdot 441.42 = 445.8342$ kips

Internal truss forces in the Y-direction:

$$2 \cdot 100 \cdot \frac{5.327614}{\sqrt{5^2 + 5.327614^2}} + 300 = 445.8342 \text{ kips}$$

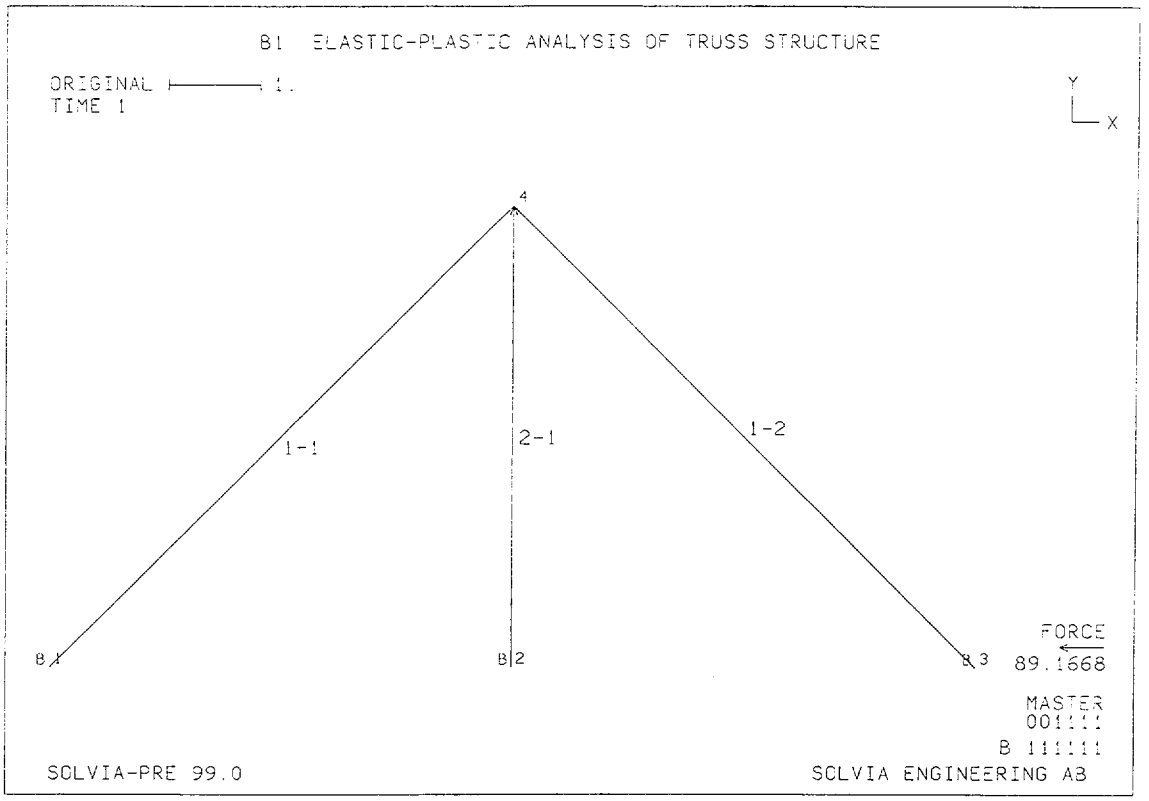
The stress-strain solution points for the left and center elements as predicted in the finite element analysis are shown in the right bottom figure on page B1.3. Note that the curves in the figures are drawn as straight lines between the solution points, which may not be the true stress-strain relation if, for example, there is a change from elastic to plastic condition. Deformed mesh and Y-displacement history for node 4 are shown in the left bottom figure on page B1.3.

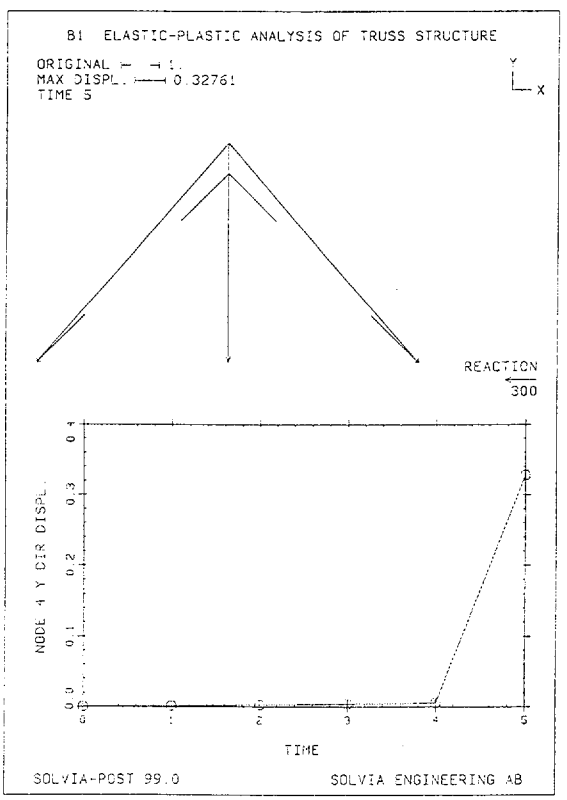
User Hints

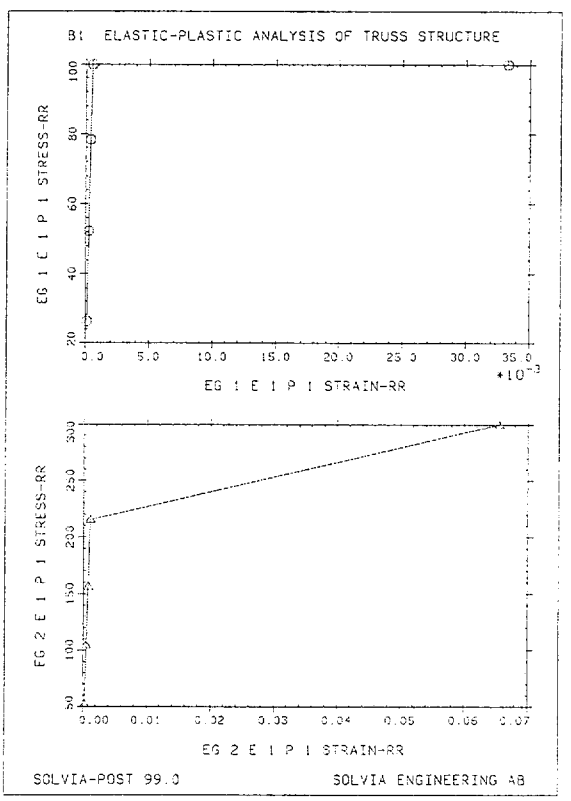
- The option of large displacements must be used for a physically realistic modeling of the structure. If a material-nonlinear-only analysis is employed, the structure is predicted to become unstable once the left and right elements become plastic since at that state there is no stiffness in the X-translational degree-of-freedom at node 4. When large displacements are considered, the analysis proceeds beyond this load point, because the forces in all the members provide a geometric stiffness to the structural model.

- The TRUSS element is formulated for small strain conditions only. Therefore, the area of the TRUSS element is assumed to remain constant during the analysis and Poisson's ratio is not included in the input to the material models. Also, the strain calculated by SOLVIA for the TRUSS element in large displacement analysis is engineering strain and not Green-Lagrange strain. For example, the axial strain in element 2 at time 5 is

$$e = \frac{u}{L} = \frac{0.327614}{5.000000} = 0.06552$$


B1 ELASTIC-PLASTIC ANALYSIS OF TRUSS STRUCTURE
ORIGINAL 1.
TIME 1
Y
X
4
1-1
2-1
1-2
B1
B2
B3
FORCE
89.1668
MASTER
001111
B 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


B1 ELASTIC-PLASTIC ANALYSIS OF TRUSS STRUCTURE
ORIGINAL 1.
MAX DISPL. 0.32761
TIME 5
Y
X
REACTION
300
NODE 4 Y DIR DISPL.
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B1 ELASTIC-PLASTIC ANALYSIS OF TRUSS STRUCTURE
EG 1 E 1 P 1 STRESS-RR
EG 1 E 1 P 1 STRAIN-RR
EG 2 E 1 P 1 STRESS-RR
EG 2 E 1 P 1 STRAIN-RR
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B1  ELASTIC-PLASTIC ANALYSIS OF TRUSS STRUCTURE'
*
DATABASE  CREATE
MASTER  IDOF=001111 NSTEP=5
KINEMATICS  DISPLACEMENT=LARGE
*
TIMEFUNCTION  1
 0. 0.  /   5. 1.01
*
COORDINATES
 1  -5.  TO  3  5.  /  4  0. 5.
*
MATERIAL  1  PLASTIC  E=2.E5 YIELD=100. ET=0.
MATERIAL  2  PLASTIC  E=2.E5 YIELD=300. ET=0.
*
EGROUP  1  TRUSS  MATERIAL=1
ENODES  /  1  1 4  /  2  3 4
EDATA  /  1  1.
*
EGROUP  2  TRUSS  MATERIAL=2
ENODES  /  1  2 4
EDATA  /  1  1.
*
FIXBOUNDARIES  /  1  TO  3
LOADS  CONCENTRATED
 4 2 441.42
*
MESH  VIEW=Z NNUMBERS=YES ENUMBERS=GROUP BCODE=ALL VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B1  ELASTIC-PLASTIC ANALYSIS OF TRUSS STRUCTURE
*
DATABASE  CREATE
WRITE  FILENAME='b1.lis'
*
SET  PLOTORIENTATION=PORTRAIT
MESH  VIEW=Z ORIGINAL=DASHED DMAX=1 VECTOR=REACTION SUBFRAME=12
NHISTORY  NODE=4 DIRECTION=2 OUTPUT=ALL SYMBOL=1
*
EXYPLOT  ELEMENT=1 POINT=1 XKIND=ERR YKIND=SRR SYMBOL=1 SUBFRAME=12
EGROUP  2
EXYPLOT  ELEMENT=1 POINT=1 XKIND=ERR YKIND=SRR SYMBOL=2
*
ELIST  TSTART=1 TEND=5
END
```

EXAMPLE B2

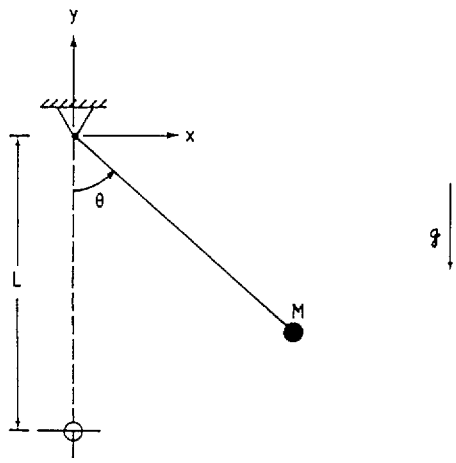
LARGE AMPLITUDE OSCILLATION OF A PENDULUM

Objective

To verify the dynamical behaviour of the TRUSS element in large displacement analysis.

Physical Problem

The simple pendulum shown in the figure below is released from a horizontal position at time t=0. The response for one period of oscillation is to be determined.



$L = 3.0443$ m

$EA = 1.0 \cdot 10^5$ N

$M = 10.0$ kg

$g = 9.8$ m/s^2

Finite Element Model

The pendulum is idealized as a TRUSS element capable of large displacements with a concentrated mass at its free end. The trapezoidal rule is employed to obtain the step-by-step response with a time step size of 0.1 second. Full-Newton iterations are performed at each step of the solution. The parameters used for measuring equilibrium iteration force convergence are RTOL=0.0001 and RNORM=98.0 (pendulum weight) where RTOL is the force tolerance and RNORM is the reference load.

Solution Results

The input data on pages B2.4 and B2.5 is used in the finite element analysis. The bottom figure on page B2.3 shows the time history of the angle θ, as defined in figure above, and the angular velocity $\dot{\theta}$, respectively, as calculated and displayed by SOLVIA-POST. The top figure on page B2.3 shows the time history of the axial force in the truss and the deformed mesh for time 1.0.

The maximum absolute value of the angular acceleration is predicted to be 3.219 rad/s^2 which can be compared to the theoretical value

$$g / L = 9.8 / 3.0443 = 3.219 \text{ rad/s}^2$$

The maximum absolute value of the angular velocity is predicted to be 2.50 rad/s (at time 3.1 sec) which can be compared to the theoretical value

$$\sqrt{2g/L} = \sqrt{2 \cdot 9.8 / 3.0443} = 2.54 \text{ rad/s}$$

The maximum axial force in the truss is predicted by SOLVIA to be 292.26 N which can be compared to the theoretical value

$$3\,mg = 3 \cdot 10 \cdot 9.8 = 294.00 \text{ N}$$

User Hints

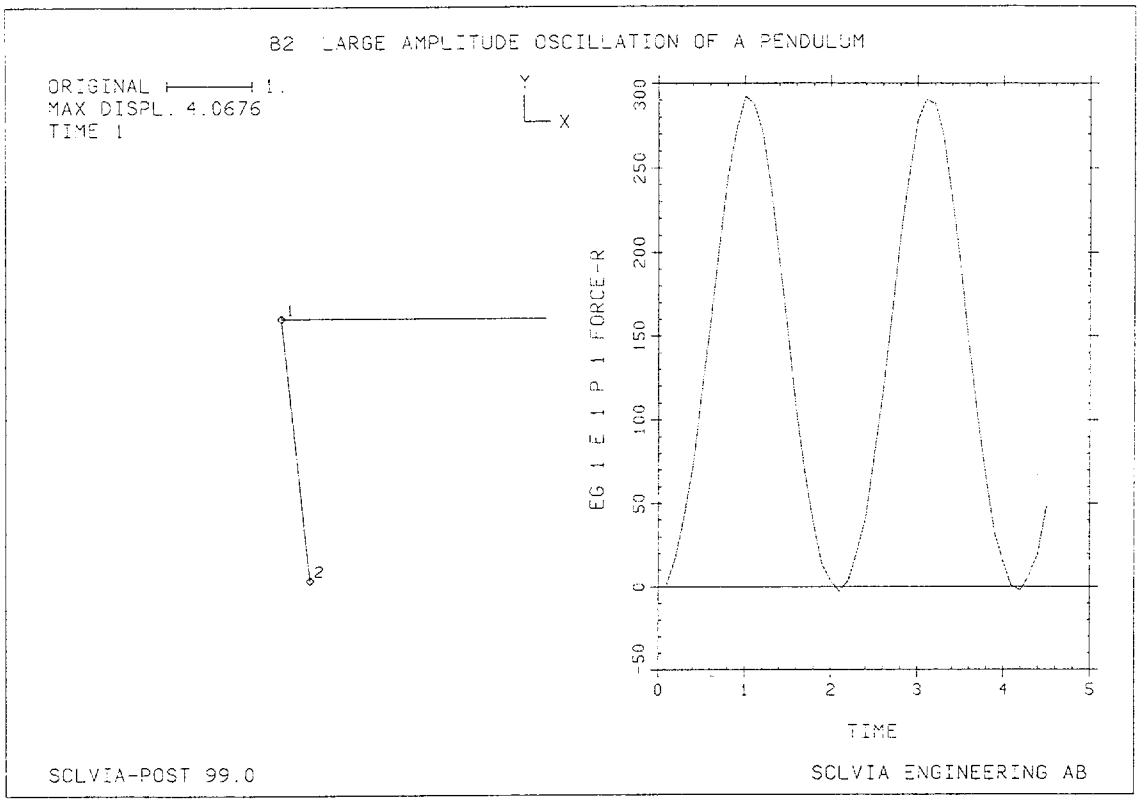
- To avoid spurious oscillations in the element force, it is important to select a small enough value of the Young's modulus times area for the truss material. In this example, the natural period for vibrations of the mass in the direction of the element is

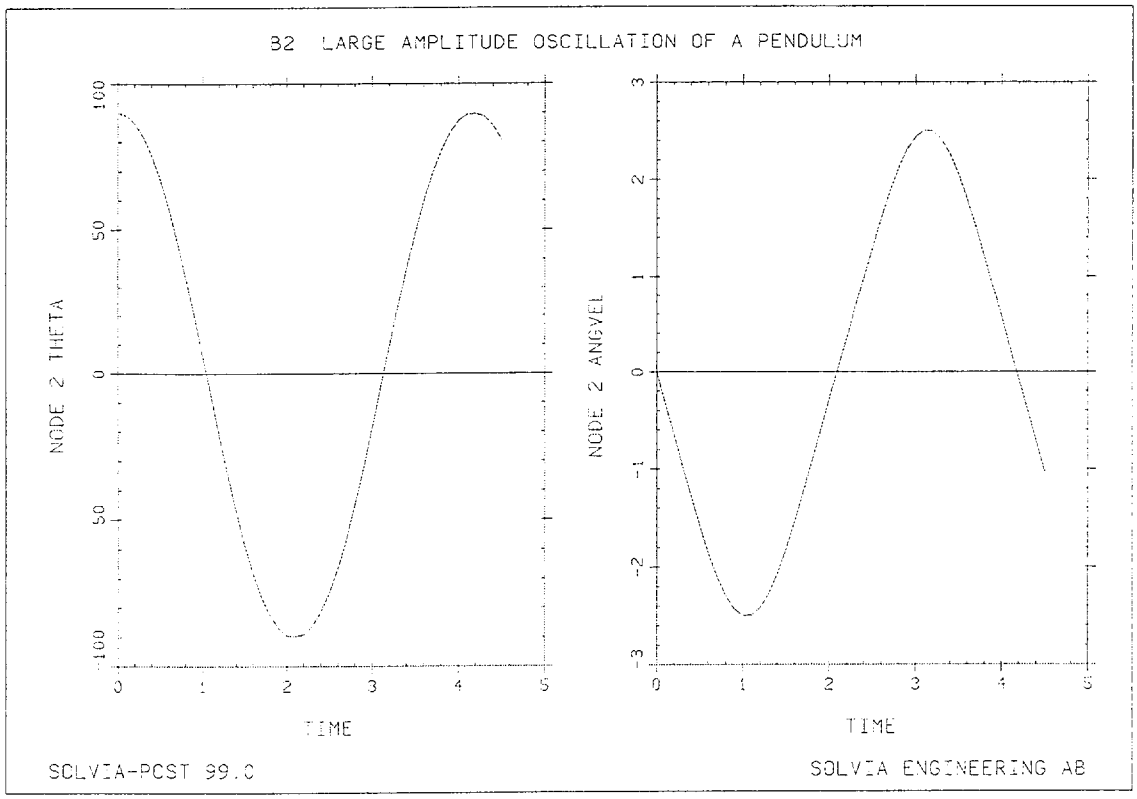
  $$2\pi\sqrt{mL/EA} = 0.1096 \text{ sec}$$

  which is of the same order as the time step. An $E \cdot A$-value of $1.0 \cdot 10^7$ N, for example, would produce spurious axial force oscillations in the element if the time step is kept the same.

- Note that the load vector input by time functions at the start of the solution (t=0) is not used in SOLVIA, see User Hints for Example A52. In this example, an initial acceleration is therefore prescribed at time t=0.

- SOLVIA-POST has the capability to evaluate a user-supplied arithmetic expression based on the nodal or element results. The results are calculated by SOLVIA referring to the global coordinates and a transformation to a cylindrical coordinate system can be done in SOLVIA-POST.


B2 LARGE AMPLITUDE OSCILLATION OF A PENDULUM
ORIGINAL 1.
MAX DISPL. 4.0676
TIME 1
Y
X
1
2
EG 1 E 1 P 1 FORCE-R
300
250
200
150
100
50
0
-50
0
1
2
3
4
5
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B2 LARGE AMPLITUDE OSCILLATION OF A PENDULUM
NODE 2 THETA
100
50
0
-50
-100
0
1
2
3
4
5
TIME
NODE 2 ANGVEL
3
2
1
0
-1
-2
-3
0
1
2
3
4
5
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B2  LARGE AMPLITUDE OSCILLATION OF A PENDULUM'
*
DATABASE  CREATE
MASTER  IDOF=001111 NSTEP=45 DT=0.1
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=LUMPED METHOD=NEWMARK
KINEMATICS  DISPLACEMENTS=LARGE
ITERATION  METHOD=FULL-NEWTON
TOLERANCES  TYPE=F RTOL=1.E-4 RNORM=98.
*
COORDINATES
  1  /  2  3.0443
*
MASSES  /  2  10.
INITIAL  ACCELERATION   /  2  0. -9.8
*
MATERIAL  1  ELASTIC  E=1.E5
*
EGROUP  1  TRUSS
ENODES  /  1  1 2
EDATA   /  1  1.
*
FIXBOUNDARIES  /  1
LOADS  MASSPROPORTIONAL  YFACTOR=1. ACCGRA=-9.8
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B2  LARGE AMPLITUDE OSCILLATION OF A PENDULUM
*
DATABASE  CREATE
TOLERANCES
*
WRITE  FILENAME='b2.lis'
*
SUBFRAME  21
MESH  VIEW=Z NNUMBERS=YES NSYMBOLS=YES ORIGINAL=YES TIME=1.
EHISTORY   ELEMENT=1 POINT=1 KIND=FR
*
NPOINT      NAME=TWO NODE=2
NVARIABLE   NAME=DX  DIRECTION=1
NVARIABLE   NAME=VX  DIRECTION=1 KIND=VELOCITY
NVARIABLE   NAME=VY  DIRECTION=2 KIND=VELOCITY
NVARIABLE   NAME=AX  DIRECTION=1 KIND=ACCELERATION
NVARIABLE   NAME=AY  DIRECTION=2 KIND=ACCELERATION
CONSTANT    NAME=DEG 180
CONSTANT    NAME=L   3.0443
CONSTANT    NAME=PI  3.14159
*
RESULTANT   NAME=THETA  'DEG/PI*ASIN((L+DX)/L)'
RESULTANT   NAME=ANGVEL 'VX/L*COS(ASIN((L+DX)/L))+VY/L*(L+DX)/L'
RESULTANT   NAME=ANGACC 'AX/L*COS(ASIN((L+DX)/L))+AY/L*(L+DX)/L'
*
SUBFRAME  21
RHISTORY  POINTNAME=TWO  RESULTANTNAME=THETA
RHISTORY  POINTNAME=TWO  RESULTANTNAME=ANGVEL
*
RMAX  ZONENAME=N2 RESULTANTNAME=THETA NUMBER=5
RMAX  ZONENAME=N2 RESULTANTNAME=ANGVEL NUMBER=5
RMAX  ZONENAME=N2 RESULTANTNAME=ANGACC NUMBER=5
EMAX
END
```

EXAMPLE B3

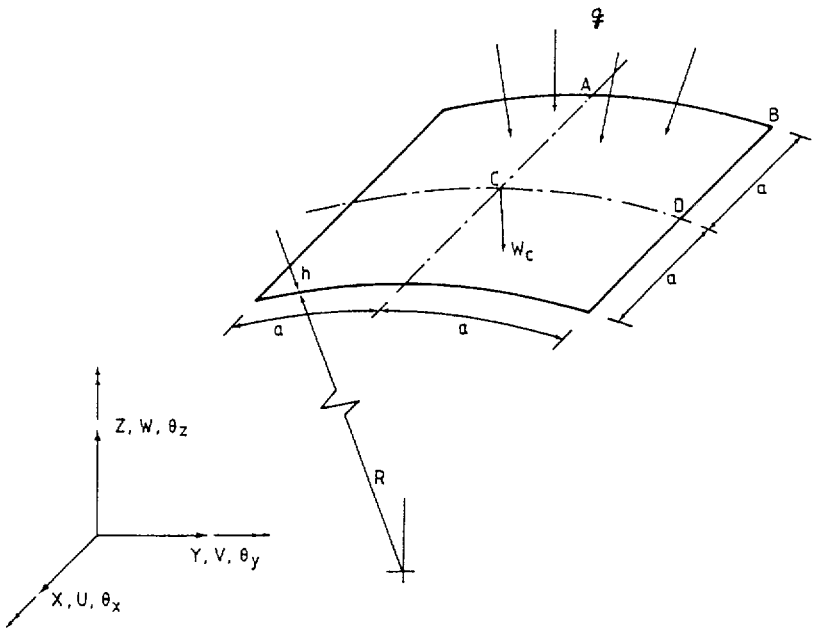
ELASTIC BUCKLING OF A CYLINDRICAL PANEL

Objective

To verify the large displacement behaviour of the PLATE element when subjected to a uniformly distributed loading.

Physical Problem

A cylindrical panel clamped at all four edges, as shown in the figure below, is considered. The panel is subjected to a uniform pressure and the pre- and post-buckling behaviour of the panel is to be determined assuming that the material of the panel remains elastic.



$a = 10.0$ in
$h = 0.125$ in
$R = 100.0$ in
$E = 4.5 \cdot 10^5$ psi
$\nu = 0.3$
$q = 0.4$ psi

Finite Element Model

Using symmetry considerations, only one-quarter of the panel is modeled. The finite element model, shown in the left figure on page B3.2, consists of a 4×4×4 mesh of PLATE elements. The maximum pressure load of 0.4 psi is reached in 20 equal load steps. Large displacements are prescribed in the analysis with stiffness reformation and BFGS equilibrium iterations performed at each load step.

Solution Results

The relation of the central vertical deflection versus the applied loading, as predicted by the finite element model using the input data on page B3.3 is shown in the right figure on page B3.2. The solution of the finite element analysis is in good agreement with the results reported in [1].

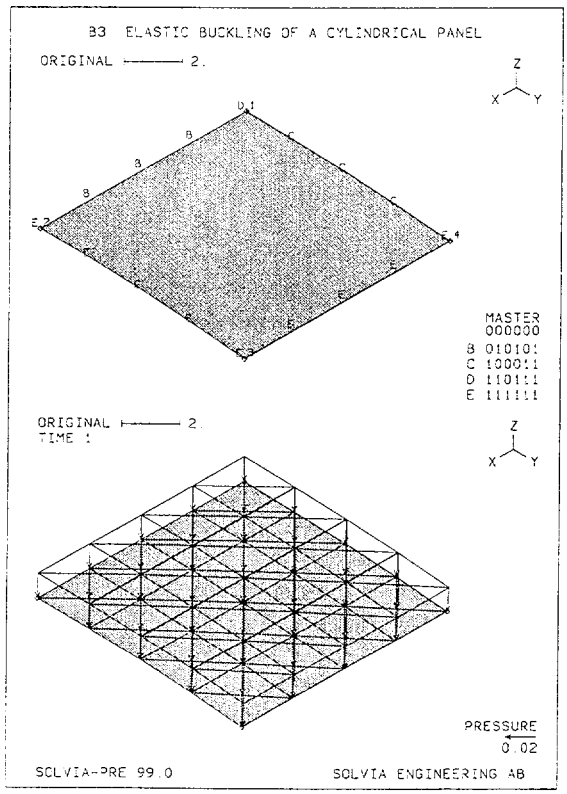
User Hints

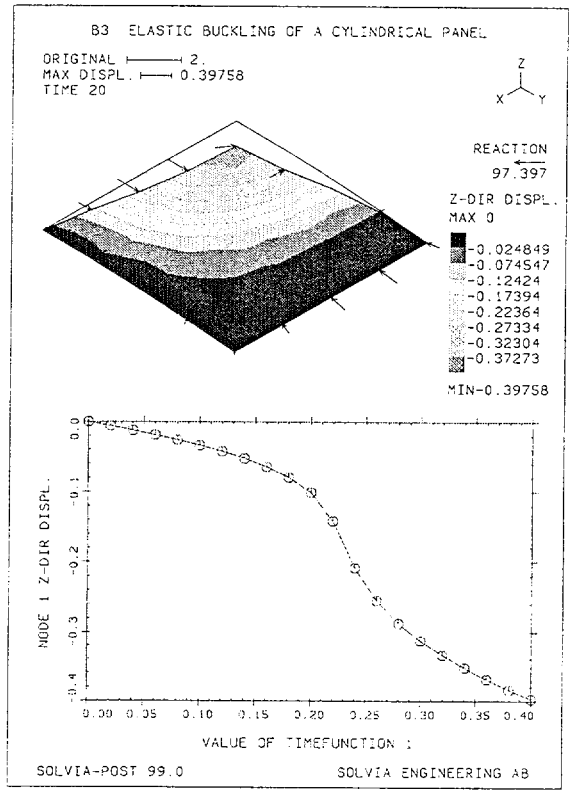
- Note that the program treats this instability analysis as a large deflection problem and does not indicate a bifurcation point. The bifurcation would be indicated by a zero determinant, which in practice means a very small positive or negative pivot, due to round-off.

- This analysis can also be performed very effectively using the automatic load step incrementation method in SOLVIA.

Reference

[1] Dhatt, G., "Instability of Thin Shells by the Finite Element Method", IASS Symp. for Folded Plates and Prismatic Structures, Vienna, 1970.





SOLVIA-PRE input

```
HEADING  'B3  ELASTIC BUCKLING OF A CYLINDRICAL PANEL'
*
DATABASE  CREATE
MASTER  NSTEP=20
KINEMATICS  DISPLACEMENT=LARGE
*
TIMEFUNCTION  1
 0.  0.  /  20. 0.4
*
SYSTEM  1 CYLINDRIC
COORDINATES
 ENTRIES  NODE  R   THETA    XL
           1   100. 90.       0.
           2   100. 90.      10.
           3   100. 84.27042 10.
           4   100. 84.27042  0.
*
MATERIAL  1  ELASTIC  E=4.5E5 NU=0.3
*
EGROUP  1  PLATE
GSURFACE  1 2 3 4  EL1=4 EL2=4 SYSTEM=1
EDATA  /  1  0.125
LOADS  ELEMENT INPUT=ELEMENTS
 1 T 1.  TO  64 T 1.
*
FIXBOUNDARIES  123456 INPUT=LINE  /  2 3  /  3 4
FIXBOUNDARIES  246    INPUT=LINE  /  1 2
FIXBOUNDARIES  156    INPUT=LINE  /  4 1
*
SET  PLOTORIENTATION=PORTRAIT NSYMBOLS=MYNODES
MESH  OUTLINE=YES NNUMBERS=MYNODES BCODE=ALL SUBFRAME=12
MESH  VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B3  ELASTIC BUCKLING OF A CYLINDRICAL PANEL
*
DATABASE  CREATE
WRITE  FILENAME='b3.lis'
*
SET  PLOTORIENTATION=PORTRAIT OUTLINE=YES
MESH  DMAX=1 CONTOUR=DZ ORIGINAL=YES VECTOR=REACTION SUBFRAME=12
*
NHISTORY  NODE=1 DIRECTION=3 XVARIABLE=1 OUTPUT=ALL SYMBOL=1
END
```

EXAMPLE B4

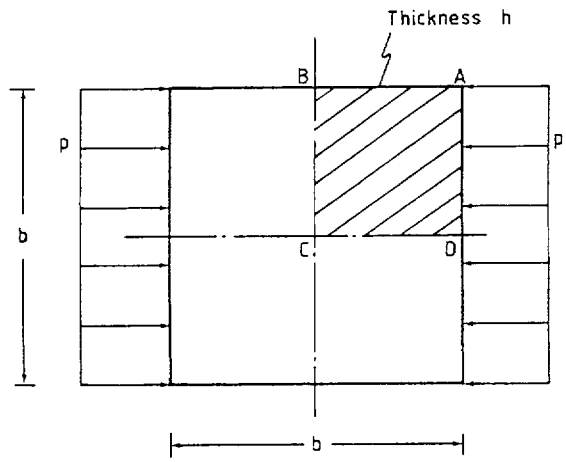
ELASTIC BUCKLING OF AN IMPERFECT SQUARE PLATE

Objective

To verify the large displacement behaviour of the cubic SHELL element when employed in buckling analysis.

Physical Problem

The square plate considered in the analysis is shown in the figure below. The plate is simply supported on all edges and uniform compressive loads are applied at the two opposite ends.

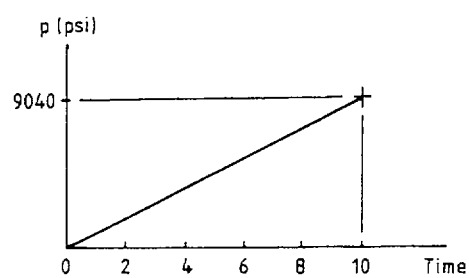


$b = 2.0$ in

$h = 0.01$ in

$E = 1.0 \cdot 10^8$ psi

$\nu = 0.3$



Finite Element Model

The finite element model is shown in the left figure on page B2.2. Using symmetry conditions, only one-quarter of the plate is modeled with one cubic SHELL element. The following initial imperfection in geometry is imposed on the plate:

$$Z(X,Y) = \frac{h}{100}\left(1 - \frac{2X}{b}\right)\left(1 - \frac{2Y}{b}\right)$$

The above equation represents a linear approximation to the first buckling mode shape of the perfect plate. The loads are applied in ten equal load steps and a geometrically nonlinear analysis (option DISPLACEMENTS = LARGE) with stiffness reformation and BFGS equilibrium iterations in each step is performed.

Solution Results

The displacement at the center of the plate versus the applied loading, as predicted by the finite element model using the input data on page B4.3, is shown in the right bottom figure on page B4.2. The analytical buckling load for the perfect plate using [1], p. 352 is given by

$$p_{cr} = 9038 \text{ psi}$$

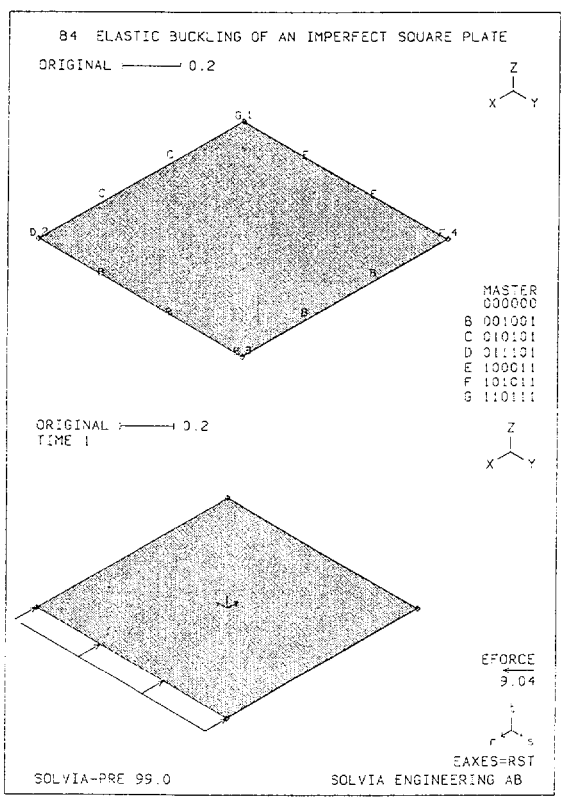
The deformed element mesh is also shown in the right figure on page B4.2.

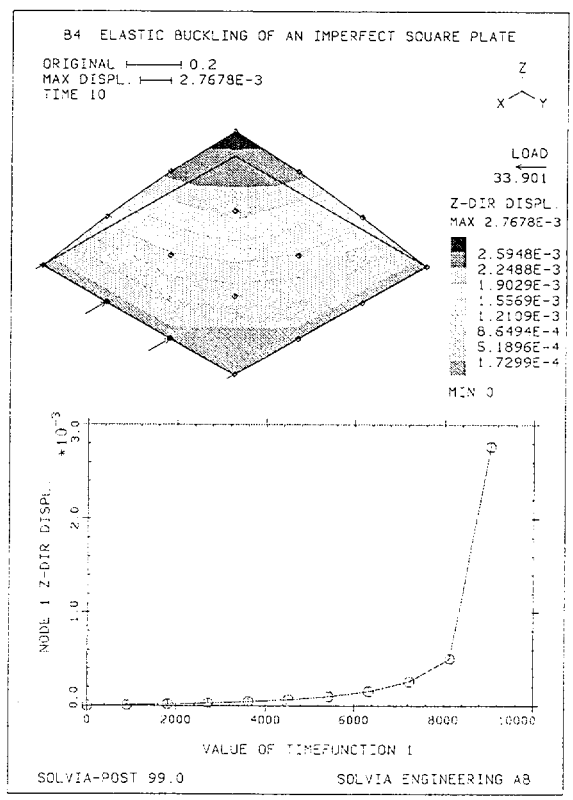
User Hints

- The shape of the initial imperfection in this problem is assumed to be known. In general, the initial imperfection geometry can be automatically generated by SOLVIA as described in Example B44.
- Note that a lower integration order than the default 4×4×2 Gauss integration can produce spurious kinematic modes in the element and, hence, unreliable results.

Reference

[1] Timoshenko, S.P., and Gere, J.M., Theory of Elastic Stability, 2nd Edition, McGraw-Hill, 1961.





SOLVIA-PRE input

```
HEADING  'B4  ELASTIC BUCKLING OF AN IMPERFECT SQUARE PLATE'
*
DATABASE  CREATE
MASTER  NSTEP=10
KINEMATICS  DISPLACEMENTS=LARGE
*
TIMEFUNCTION  1
 0.   0.  /  10.  9040.
*
COORDINATES
 1  0. 0. 1.E-4  /  2  1.  /  3  1. 1.  /  4  0. 1.
*
MATERIAL  1  ELASTIC  E=1.E8 NU=0.3
*
EGROUP  1  SHELL
GSURFACE  1 2 3 4  EL1=1 EL2=1 NODES=16
THICKNESS  1  0.01
LOADS  ELEMENT  TYPE=FORCE INPUT=LINE
 2 3 out -0.01 -0.01
*
FIXBOUNDARIES  36  INPUT=LINE  /  3 4  /  2 3
FIXBOUNDARIES  156 INPUT=LINE  /  4 1
FIXBOUNDARIES  246 INPUT=LINE  /  1 2
*
SET  PLOTORIENTATION=PORTRAIT NSYMBOLS=MYNODES
MESH  NNUMBERS=MYNODES BCODE=ALL SUBFRAME=12
MESH  EAXES=RST VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B4  ELASTIC BUCKLING OF AN IMPERFECT SQUARE PLATE
*
DATABASE  CREATE
WRITE  FILENAME='b4.lis'
*
SET  PLOTORIENTATION=PORTRAIT ORIGINAL=YES
MESH  CONTOUR=DZ NSYMBOLS=YES DMAX=1 VECTOR=LOAD SUBFRAME=12
NHISTORY  NODE=1 DIRECTION=3 SYMBOL=1 XVARIABLE=1 OUTPUT=ALL
END
```

EXAMPLE B5

CANTILEVER BEAM IN CREEP

Objective

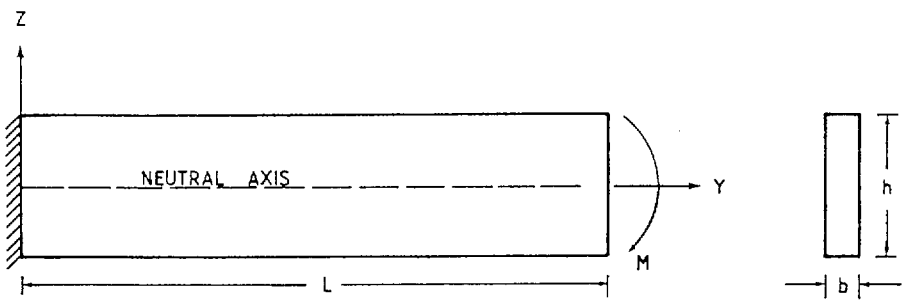
To verify the PLANE STRESS2 element for material-nonlinear-only analysis using the creep material model.

Physical Problem

The cantilever beam, shown in the figure below, is subjected to a constant bending moment applied at the tip. The material of the beam is assumed to obey the uniaxial creep law

$$\varepsilon_c = 6.4 \cdot 10^{-18} \cdot \sigma^{3.15} \cdot t \quad \text{in/in}$$

where σ is measured in psi and t in hours.



$b = 0.3$ in

$h = 4.0$ in

$L = 40.0$ in

$M = 6000.0$ lb − in

$E = 3.0 \cdot 10^7$ psi

$\nu = 0.3$

$\dot{\varepsilon}_C = 6.4 \cdot 10^{-18} \cdot \sigma^{3.15}$ in / in / hr

Finite Element Model

The finite element model considered for the analysis consists of eight 8-node PLANE STRESS2 elements as shown in the figure on page B5.2. By restraining the Y-displacement at the neutral axis, only the portion of the beam above the neutral axis is modeled. For the analysis, a step size of ten hours and twenty time steps with BFGS equilibrium iterations in each step is used. For the integration of the creep response the Euler backward method is used, i.e. the parameter α is set to 1.0, and one subdivision per time step is employed. The finite element stiffness matrices are evaluated using 3×3 Gauss integration.

Solution Results

The input data on pages B5.3 and B5.4 are used in the finite element analysis. The figure on page B5.3 shows the time history response of the bending stress at element 6, integration point 1 (Y = 18.873, Z = 1.887). The theoretical steady-state value at the same location is

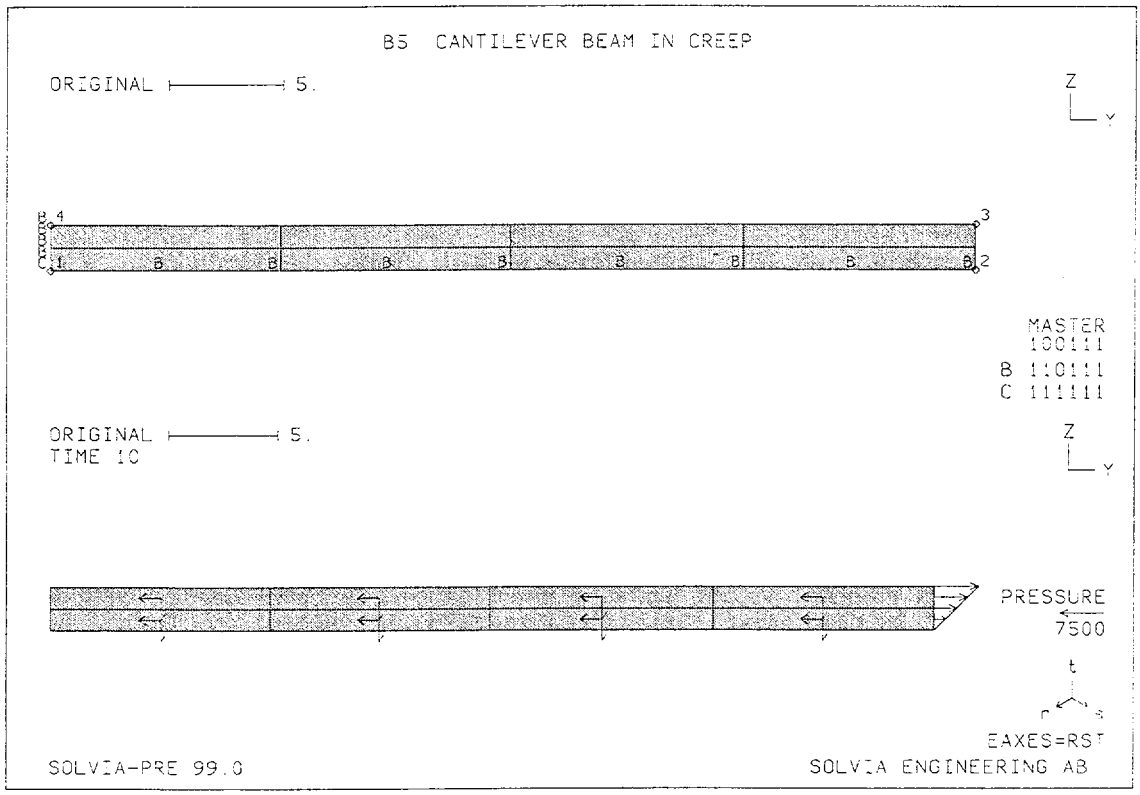
$$\sigma_{yy} = \frac{M}{2b} \frac{2a_1 + 1}{a_1} \left(\frac{h}{2}\right)^{-\frac{2a_1+1}{a_1}} Z^{\frac{1}{a_1}} = 5688\text{psi}$$

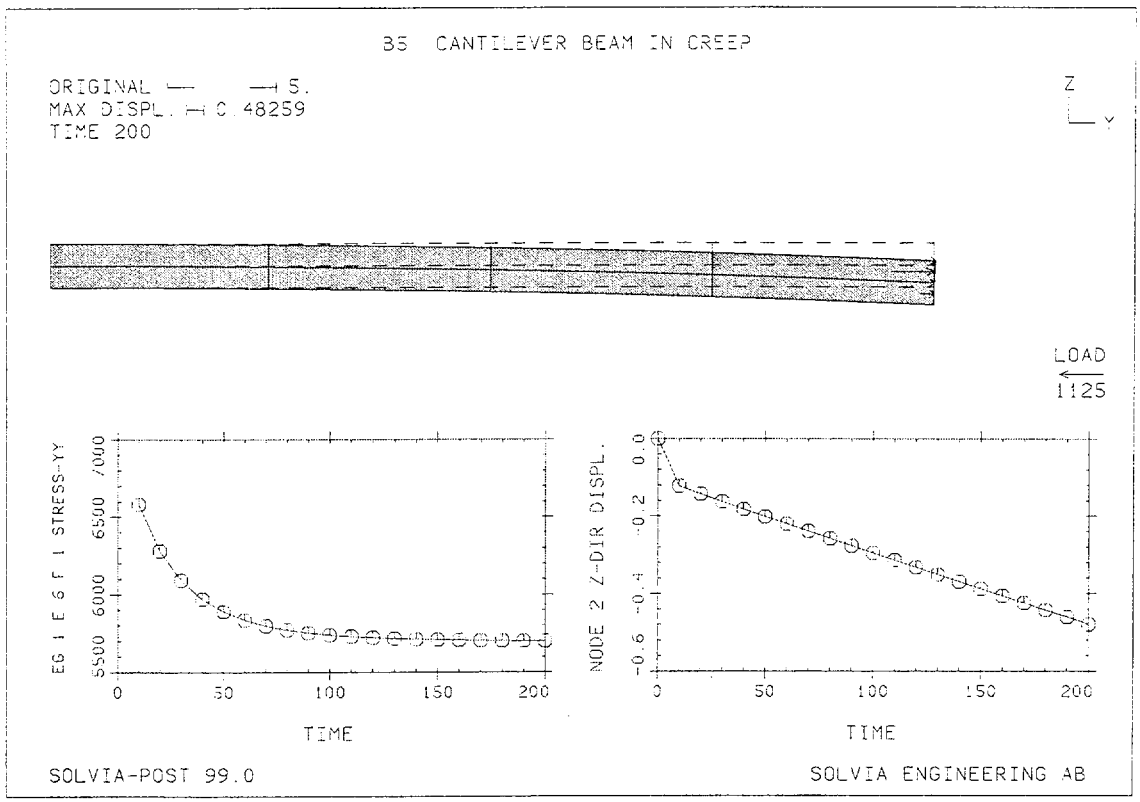
The figure on page B5.3 shows the deformed mesh at the time t=200 hours.

User Hints

- For materially nonlinear analysis it may be important to use higher order Gauss integration and more elements than in linear analysis. In a linear analysis only one 8-node PLANE STRESS element through the thickness of the beam would yield excellent results.

- A variable time step solution of this problem is performed in Example B39.





SOLVIA-PRE input

```
HEADING  'B5  CANTILEVER BEAM IN CREEP'
*
DATABASE  CREATE
MASTER  IDOF=100111 NSTEP=20 DT=10.
TOLERANCES  ETOL=1.E-7
*
TIMEFUNCTION  1
 0. 1.  /  200. 1.
*
COORDINATES
 ENTRIES  NODE  Y    Z
           1     0.   0.
           2    40.   0.
           3    40.   2.
           4     0.   2.
*
INITIAL  TEMPERATURES  TREF=200.
*
MATERIAL  1  PLASTIC-CREEP ISOTROPIC XKCRP=1 TREF=200. ALPHA=1,
             XISUBM=1 A0=6.4E-18 A1=3.15 A2=1.
               0. 3.E7 0.3 3.E7 3.E6 0.
             400. 3.E7 0.3 3.E7 3.E6 0.
*
```

SOLVIA-PRE input (cont.)

```
EGROUP  1  PLANE STRESS2
GSURFACE  1 2 3 4  EL1=4 EL2=2 NODES=8
EDATA  /  1  0.3
LOADS  ELEMENT  INPUT=LINE
 2 3 0. -7500.
*
FIXBOUNDARIES  2 INPUT=LINES  /  1 2  /  1 4
FIXBOUNDARIES  3 INPUT=NODES  / 1
LOADS  TEMPERATURES INPUT=SURFACE
 1 2 3 4  200.
*
SUBFRAME  12
MESH  NSYMBOLS=MYNODES NNUMBERS=MYNODES BCODE=ALL
MESH  EAXES=RST VECTOR=LOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B5  CANTILEVER BEAM IN CREEP
*
DATABASE  CREATE
*
WRITE  FILENAME='b5.lis'
*
SUBFRAME  12
MESH  ORIGINAL=DASHED SMOOTHNESS=YES VECTOR=LOAD
EHISTORY  ELEMENT=6 POINT=1 KIND=SYY SYMBOL=1 OUTPUT=ALL,
          SUBFRAME=2121
NHISTORY  NODE=2 DIRECTION=3 SYMBOL=1 OUTPUT=ALL SUBFRAME=2221
*
NLIST  KIND=REACTION
END
```

EXAMPLE B6

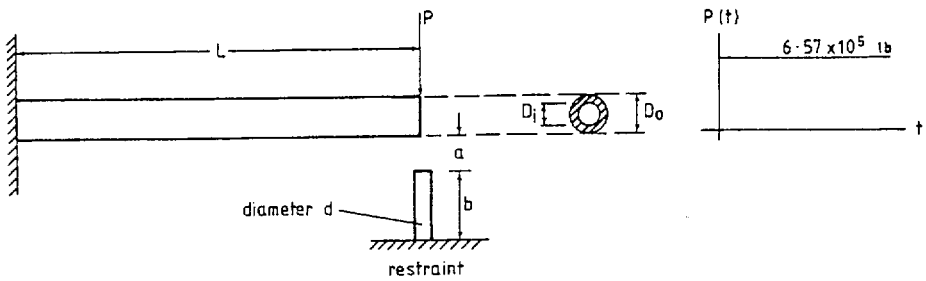
PIPE WHIP ANALYSIS, DIRECT INTEGRATION SOLUTION

Objective

To demonstrate the dynamic behaviour of the BEAM and TRUSS elements in elastic-plastic analysis with gap.

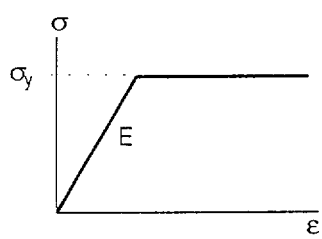
Physical Problem

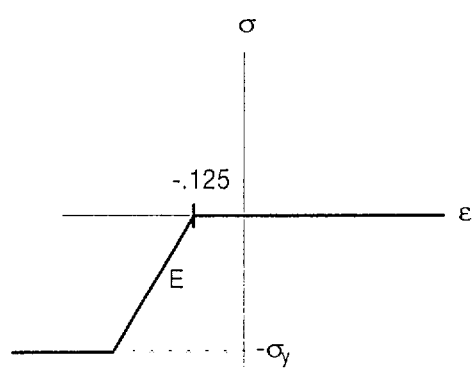
A cantilever pipe, shown in the figure below, is initially at rest and is suddenly subjected to a constant tip load, causing it to impinge onto a restraint.



$D_o = 30.0$ in
$D_i = 27.75$ in
$L = 360.0$ in

$a = 3.0$ in
$b = 21.0$ in
$d = 5.75$ in





Pipe material:

$E = 2.698 \cdot 10^7$ psi

$\nu = 0.3$

$\sigma_y = 2.914 \cdot 10^4$ psi

$\rho = 7.18 \cdot 10^{-4}$ lbs/(g · in^3)

Restraint material:

$E = 2.99 \cdot 10^7$ psi

$\sigma_y = 3.80 \cdot 10^4$ psi

Finite Element Model

The pipe is modeled using six BEAM elements and the pipe material is assumed to be elastic-perfectly-plastic. The restraint is represented by a TRUSS element with elastic-perfectly-plastic material and a gap width of 3 in. The finite element model used for the analysis is shown in the top figure on page B6.3. The Newmark method is used for the step-by-step direct integration and the response is calculated for one thousand time steps using a time step of 0.0001 sec. Full-Newton iterations with line-search are performed in every step using an energy tolerance of $1.0 \cdot 10^{-8}$. A lumped mass matrix assumption is employed in the analysis. The BEAM element stiffness matrices are evaluated using 5×1×5 Newton-Cotes integration.

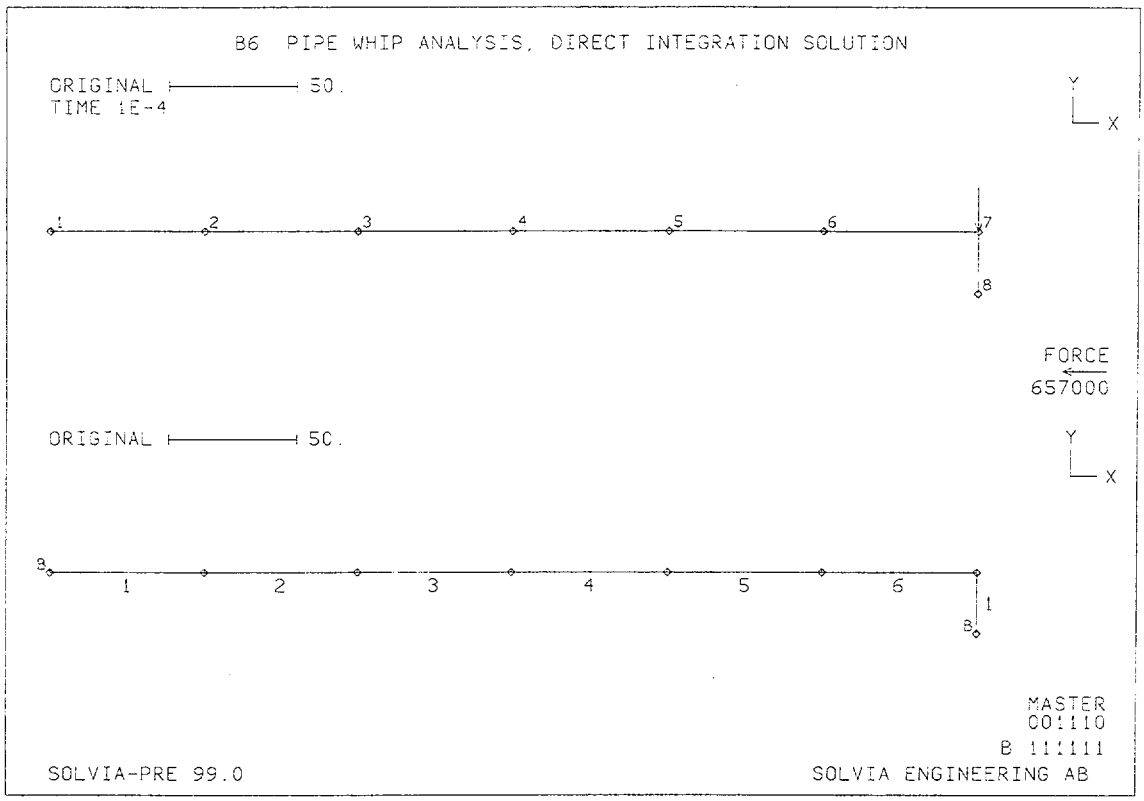
Solution Results

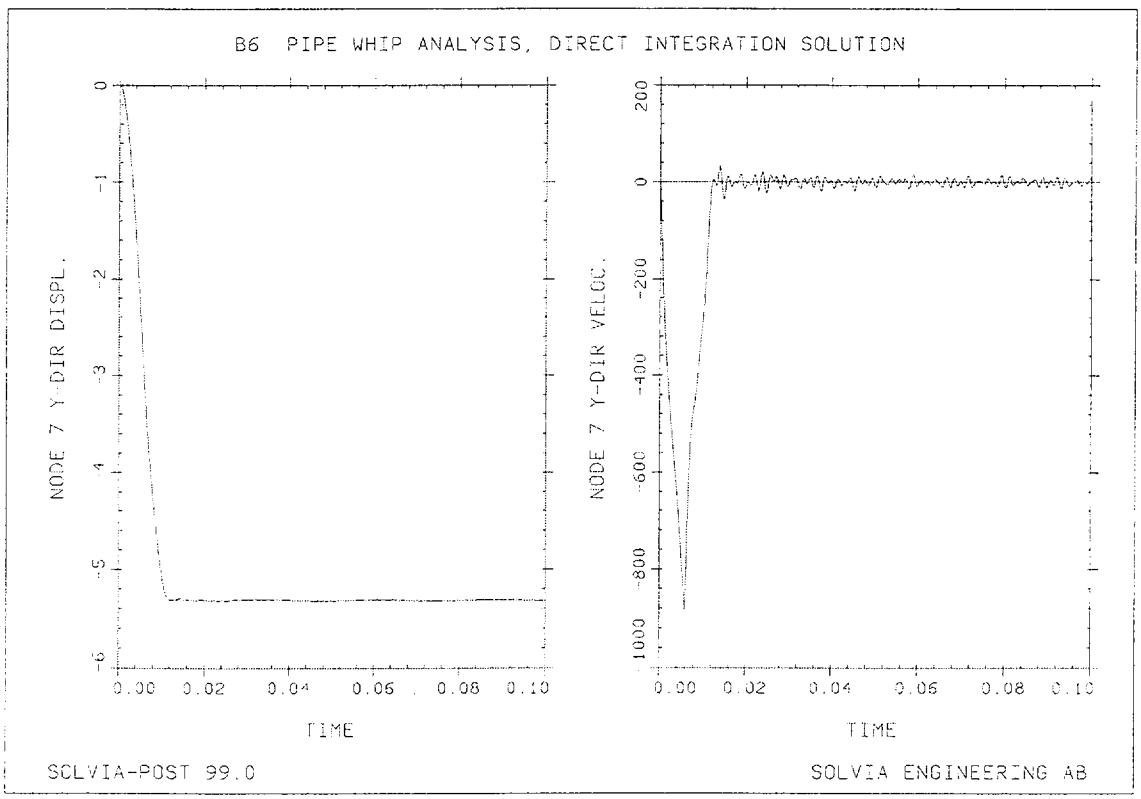
The tip and mid node responses are shown on pages B6.3 and B6.4. The axial stress in the TRUSS element as a function of time and the variation of the axial stress in element 1 of the pipe are shown in figure on page B6.5. The SOLVIA numerical solution is obtained by using the input data on pages B6.5 and B6.6.

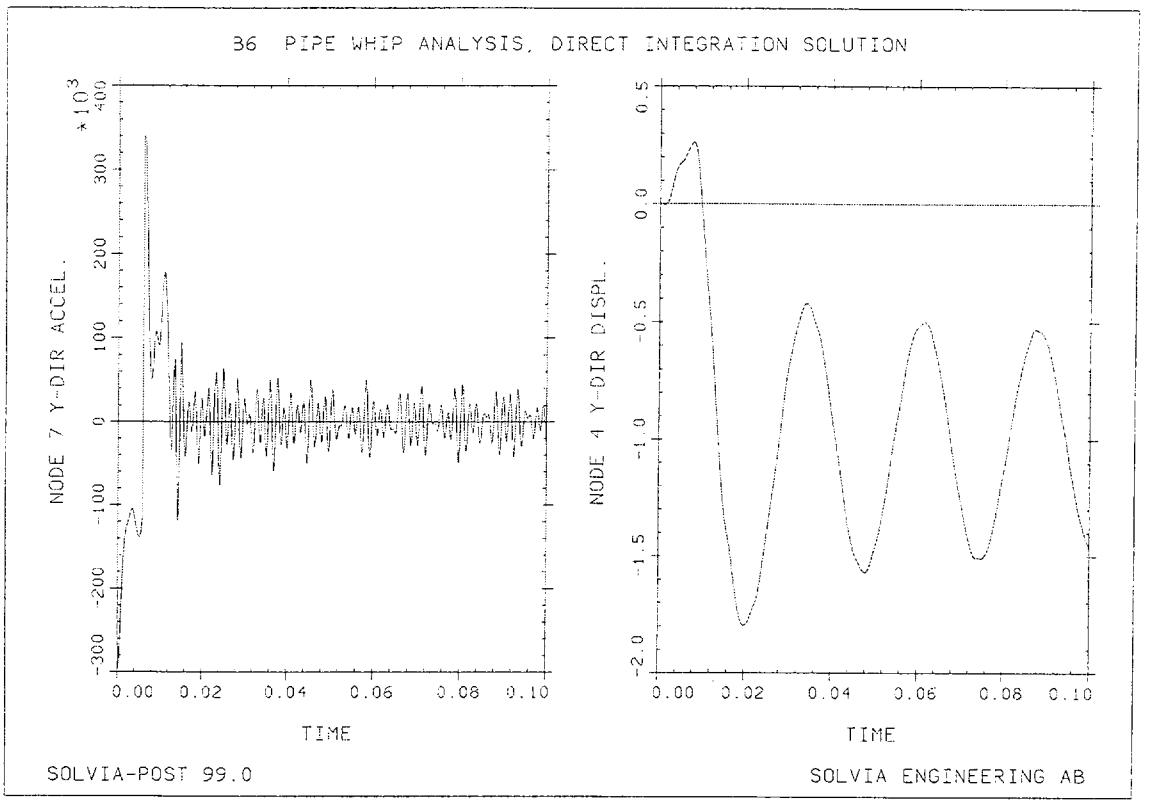
User Hints

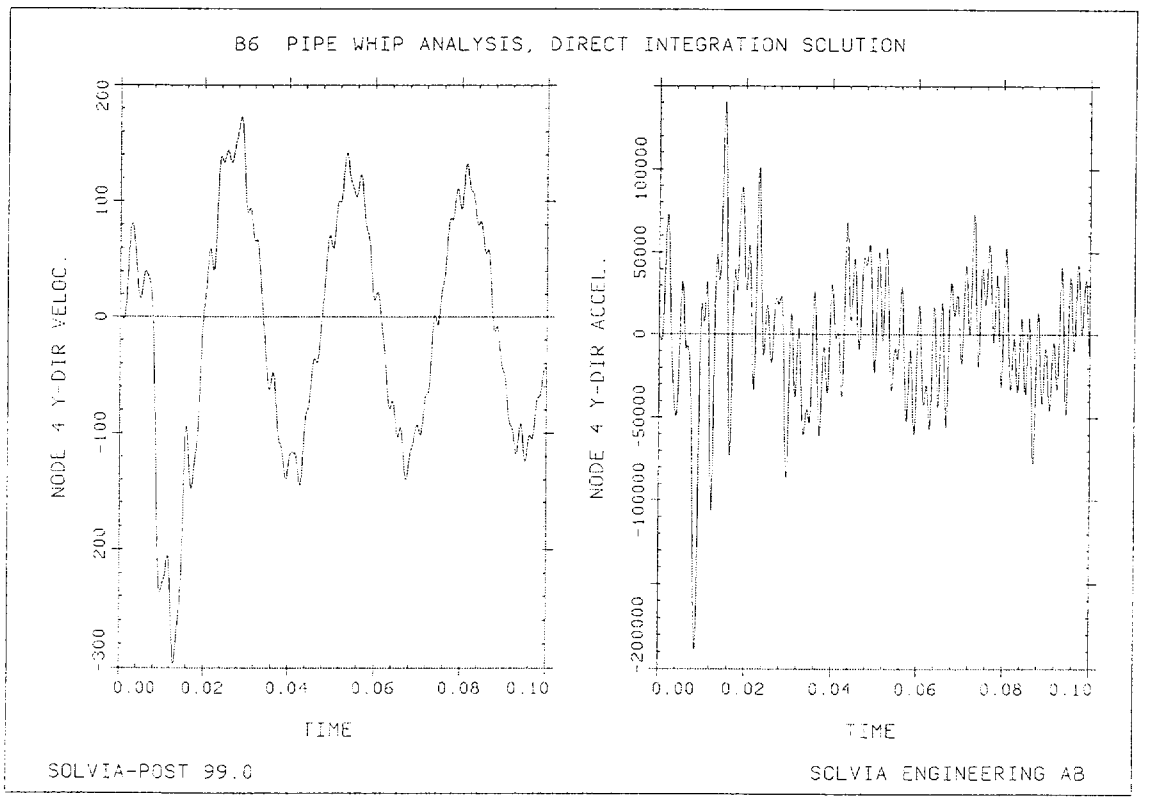
- The characteristics of this problem are that the pipe is very flexible and impinges suddenly onto a relatively stiff restraint. The nonlinearity is due to the gap and the plasticity in the restraint and in the pipe.

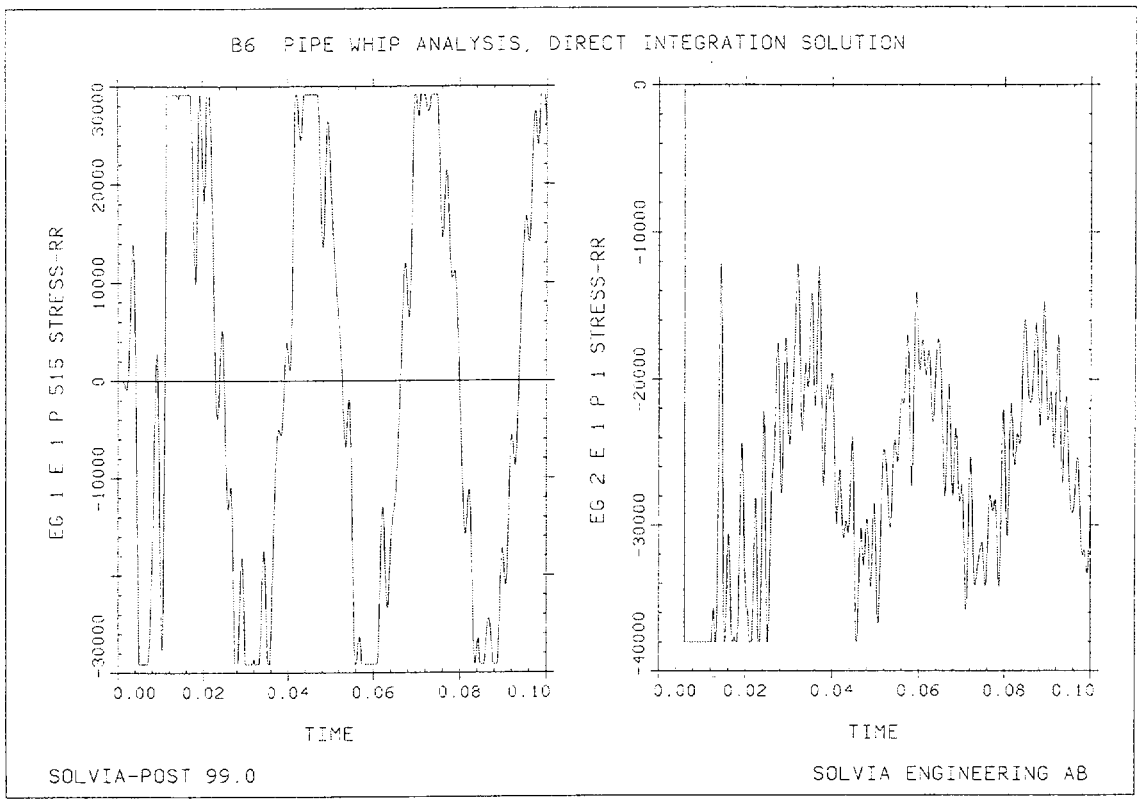
- The restraint can also be modeled using a nonlinear elastic material model and the TRUSS element without the gap option if only the first portion of the response (without unloading from the plastic state) is of interest.


B6 PIPE WHIP ANALYSIS, DIRECT INTEGRATION SOLUTION
ORIGINAL 50.
TIME 1E-4
Y
X
1
2
3
4
5
6
7
8
FORCE
657000
ORIGINAL 50.
Y
X
8
1
2
3
4
5
6
7
8
MASTER
001110
B 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


B6 PIPE WHIP ANALYSIS, DIRECT INTEGRATION SOLUTION
NODE 7 Y-DIR DISPL.
0
-1
-2
-3
-4
-5
-6
0.00
0.02
0.04
0.06
0.08
0.10
TIME
NODE 7 Y-DIR VELOC.
200
0
-200
-400
-600
-800
-1000
0.00
0.02
0.04
0.06
0.08
0.10
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B6 PIPE WHIP ANALYSIS, DIRECT INTEGRATION SOLUTION
NODE 7 Y-DIR ACCEL.
*10^3
400
300
200
100
0
-100
-200
-300
0.00 0.02 0.04 0.06 0.08 0.10
TIME
NODE 4 Y-DIR DISPL.
0.5
0.0
-0.5
-1.0
-1.5
-2.0
0.00 0.02 0.04 0.06 0.08 0.10
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B6 PIPE WHIP ANALYSIS, DIRECT INTEGRATION SOLUTION
NODE 4 Y-DIR VELOC.
200
100
0
-100
-200
-300
0.00 0.02 0.04 0.06 0.08 0.10
TIME
NODE 4 Y-DIR ACCEL.
100000
50000
0
-50000
-100000
-200000
0.00 0.02 0.04 0.06 0.08 0.10
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB



SOLVIA-PRE input

```
HEADING  'B6  PIPE WHIP ANALYSIS, DIRECT INTEGRATION SOLUTION'
*
DATABASE  CREATE
*
MASTER  IDOF=001110 NSTEP=1000 DT=1.E-4
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=LUMPED METHOD=NEWMARK
ITERATION  METHOD=FULL-NEWTON LINE-SEARCH=YES
TOLERANCES  ETOL=1.E-8
*
COORDINATES
 1  TO  7  360.  /  8  360. -24.
*
MATERIAL  1  PLASTIC E=2.698E7 NU=0.3 YIELD=29140 ET=0. DEN=7.18E-4
MATERIAL  2  PLASTIC E=2.990E7 NU=0.  YIELD=38000 ET=0. DEN=0.
*
EGROUP  1  BEAM DIM2  RINT=5 SINT=1 TINT=5 MATERIAL=1 RESULT=TABLE
STRESSTABLE  1  111 115 411 415 511 515
SECTION  1  DIMENSION PIPE DO=30. DI=27.75 ISHEAR=1
ENODES
 1  8 1 2  TO  6  8 6 7
*
```

SOLVIA-PRE input (cont.)

```
EGROUP  2  TRUSS  MATERIAL=2 GAP=YES
ENODES  /  1  8 7
EDATA  /   ENTRIES  EL  AREA  GAPWIDTH
 1  25.967 3.
*
FIXBOUNDARIES  /  1 8
*
LOADS  CONCENTRATED
 7 2 -6.57E5
*
SET  NSYMBOLS=YES VIEW=Z
MESH  NNUMBERS=YES VECTOR=LOAD SUBFRAME=12
MESH  ENUMBERS=YES BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B6  PIPE WHIP ANALYSIS, DIRECT INTEGRATION SOLUTION
*
DATABASE  CREATE
WRITE  FILENAME='b6.lis'
*
SUBFRAME  21
NHISTORY  NODE=7 DIRECTION=2 KIND=DISPLACEMENT
NHISTORY  NODE=7 DIRECTION=2 KIND=VELOCITY
*
SUBFRAME  21
NHISTORY  NODE=7 DIRECTION=2 KIND=ACCELERATION
NHISTORY  NODE=4 DIRECTION=2 KIND=DISPLACEMENT
*
SUBFRAME  21
NHISTORY  NODE=4 DIRECTION=2 KIND=VELOCITY
NHISTORY  NODE=4 DIRECTION=2 KIND=ACCELERATION
*
SUBFRAME  21
EHISTORY  ELEMENT=1 POINT=515 KIND=SRR
EGROUP  2
EHISTORY  ELEMENT=1 POINT=1 KIND=SRR
*
NMAX DIRECTION=2
END
```

EXAMPLE B7

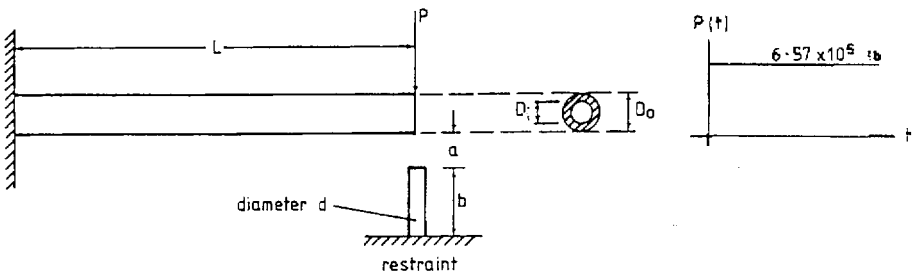
PIPE WHIP ANALYSIS, MODE SUPERPOSITION

Objective

To verify the use of mode superposition in nonlinear analysis.

Physical Problem

A cantilever pipe, shown in the figure below, is initially at rest and is suddenly subjected to a constant tip load, causing it to impinge onto a restraint. The same physical problem is already analyzed in Example B6 using direct integration which serves as a reference solution.



$D_o = 30.0$ in

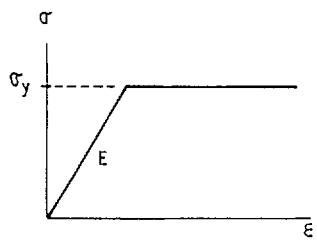
$D_i = 27.75$ in

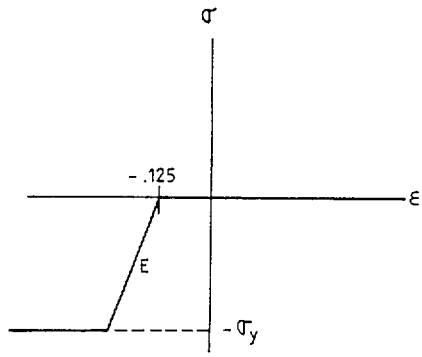
$L = 360.0$ in

$a = 3.0$ in

$b = 21.0$ in

$d = 5.75$ in





Pipe material:

$E = 2.698 \cdot 10^7$ psi

$\nu = 0.3$

$\sigma_y = 2.914 \cdot 10^4$ psi

$\rho = 7.18 \cdot 10^{-4}$ lbs/(g · in^3)

Restraint material:

$E = 2.99 \cdot 10^7$ psi

$\sigma_y = 3.80 \cdot 10^4$ psi

Finite Element Model

The same model as in Example B6 is used, see the top figure of page B7.3. The frequencies and mode shapes are calculated based on the initial configuration using the subspace iteration method. The mode shapes are employed in the nonlinear time integration using the modified Newton method with the energy tolerance $1.0 \cdot 10^{-8}$.

The model has 6 nodes with translational lumped masses in two directions. The 6 lowest bending mode shapes are of interest here. The axial mode shapes give no contribution to the response since the load acts in the transverse direction. The first run is based on 8 modes, which include all the 6 bending modes with frequencies from 8.4 Hz to 553 Hz. The time step is then selected to 10^{-4} sec which gives about 18 steps per highest period.

The second run with 2 modes was performed with the time step $0.25 \cdot 10^{-3}$ sec giving about 80 steps per highest period. The time step is shorter than in a conventional linear modal superposition because of the nonlinearities due to impact and plasticity.

Solution Results

The input data on pages B7.8 and B7.9 is used in the finite element analysis with 8 modes. The shapes of the first 2 modes are shown in the bottom figure on page B7.3.

The tip and mid node responses (displacements, velocities, accelerations) predicted in the analysis are shown in the figures on pages B7.4 and B7.5. The axial stress in the TRUSS element as a function of the axial strain and the variation of the axial stress in element 1 of the pipe are also shown in the bottom figures on page B7.5.

We note that almost the same response as in the reference solution given in Example B6 is obtained. It is not possible to obtain exactly the same solution using modal superposition with a constant set of mode shapes. The actual mode shapes change somewhat during the solution since the stiffness varies due to development of plasticity.

The solution with 2 modes is approximate, see pages B7.6 and B7.7. It displays, however, the main characteristics of the response. The tip response of the pipe and the axial stress response in the restraint are in good agreement with the reference solution.

User Hints

- The mode superposition solution is very effective when only a few modes need to be used to represent the response to sufficient accuracy. The pipe whip example solved here is such a case.

- Note that the mode shapes of the linear structure (corresponding to time 0) are used in the mode superposition solution. The mode shapes satisfy

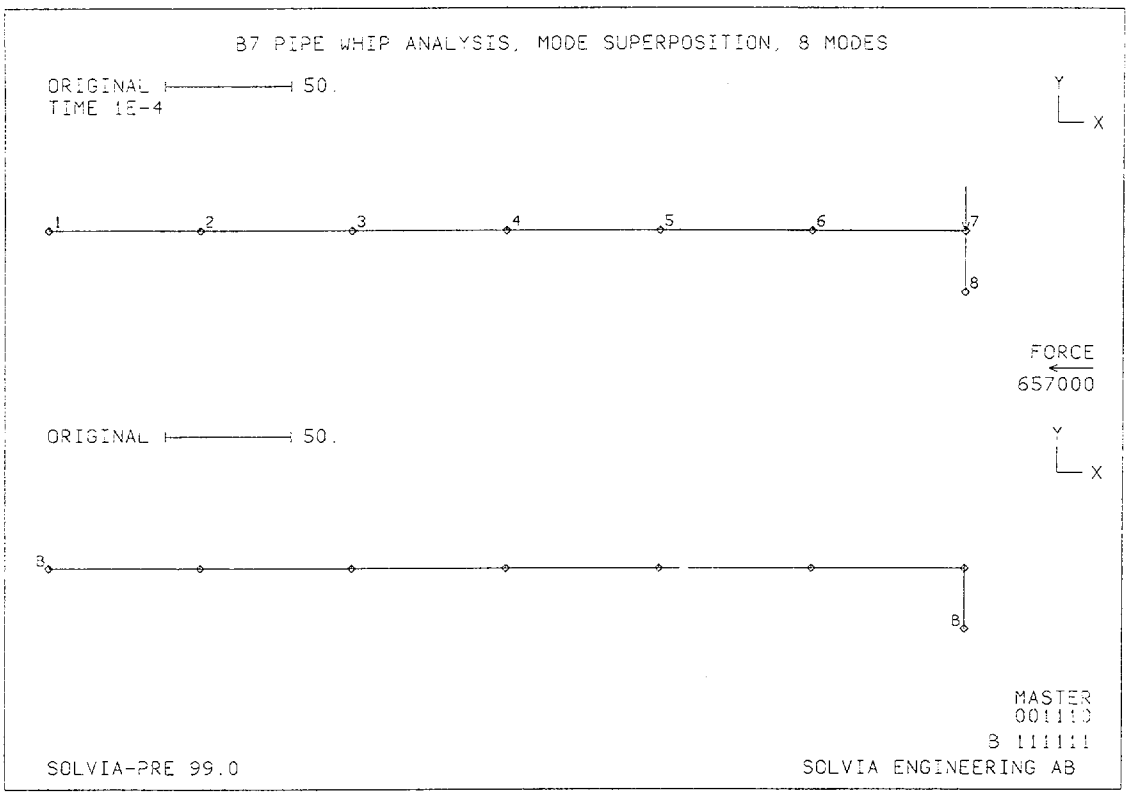
  $$^{0}\mathbf{K}\,\varphi_i = \omega_i^2\,\mathbf{M}\,\varphi_i$$

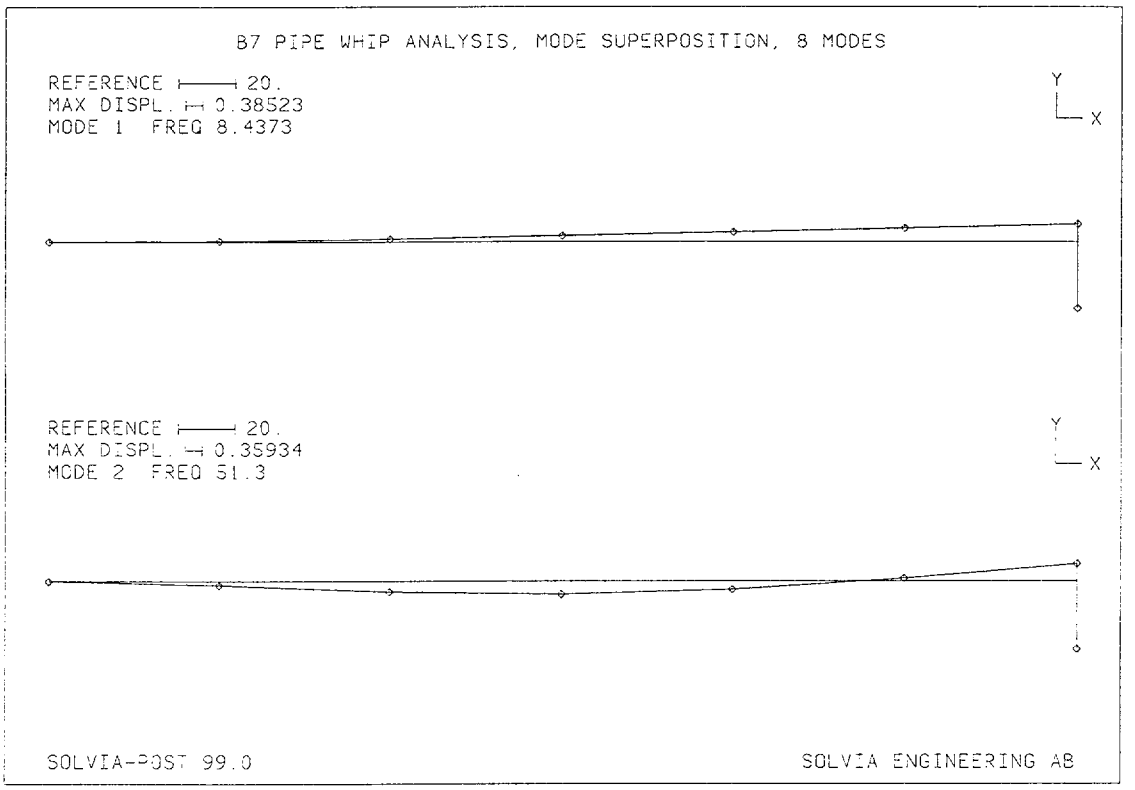
  When 2 modes are employed the response is calculated using

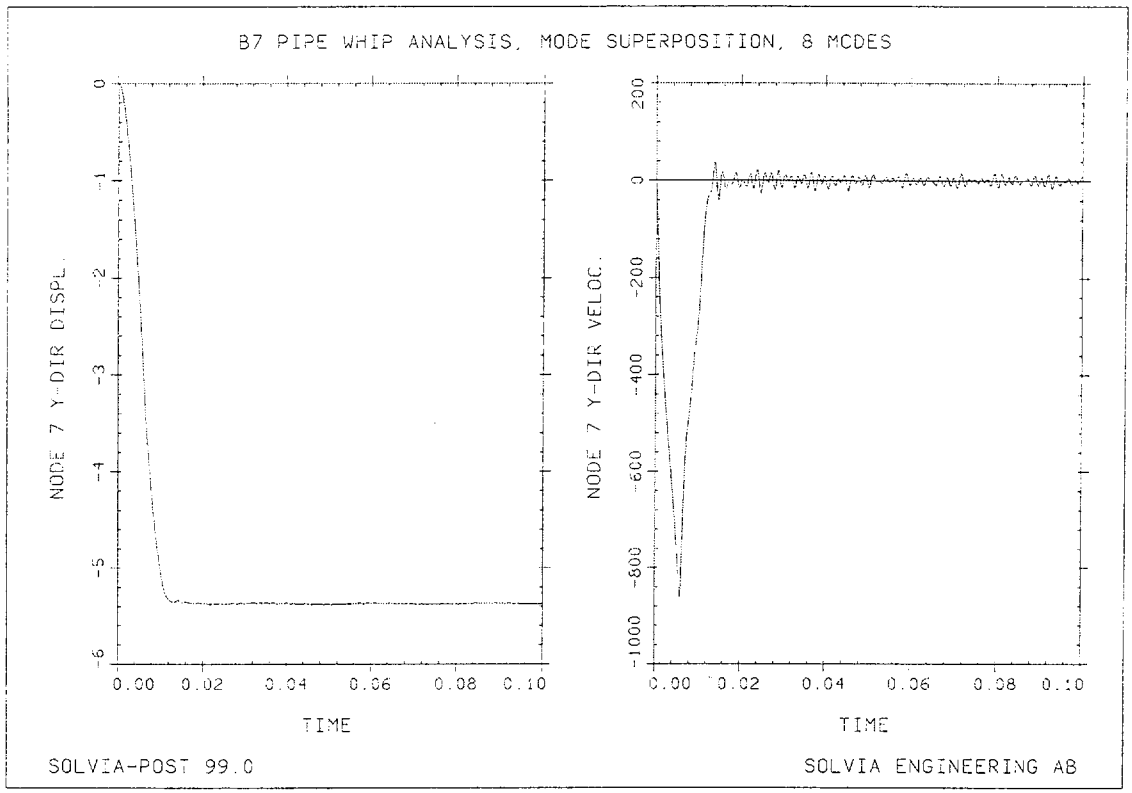
  $$^{t}\mathbf{U} = \sum_{i=1}^{2} \alpha_i(t)\varphi_i$$

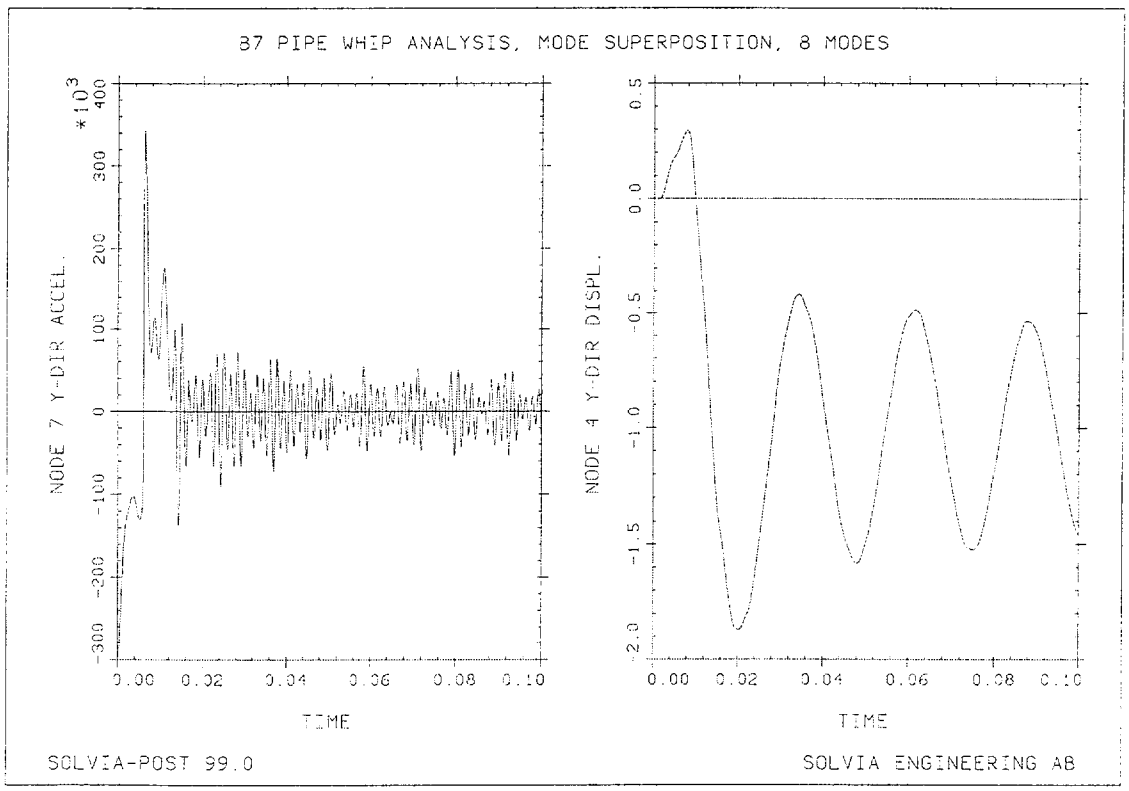
  where $\alpha_i(t)$ are time dependent coefficients evaluated in the solution. In essence, therefore, a two degree-of-freedom finite element model is considered with the displacement assumptions φ_1 and φ_2 but all elastic-plastic nonlinearities of the pipe and the restraint based on the displacement assumption are included in the analysis.

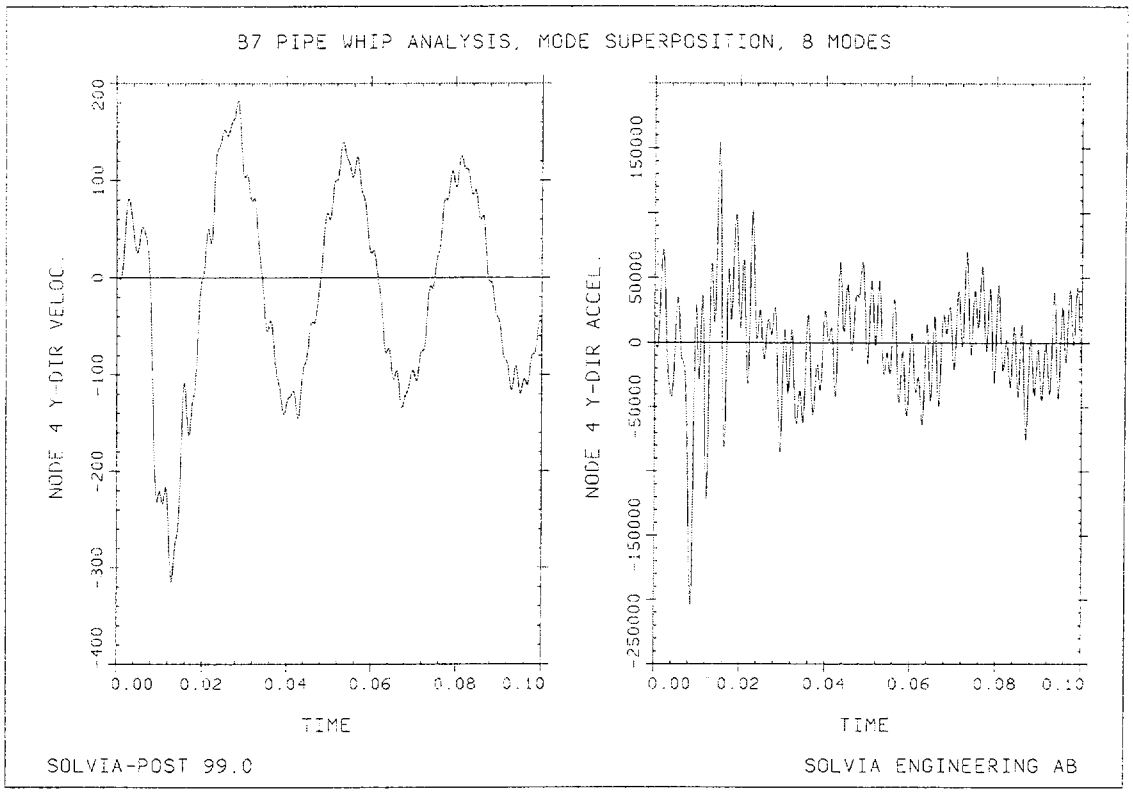
- In nonlinear mode superposition analysis only modified Newton equilibrium iteration is available.

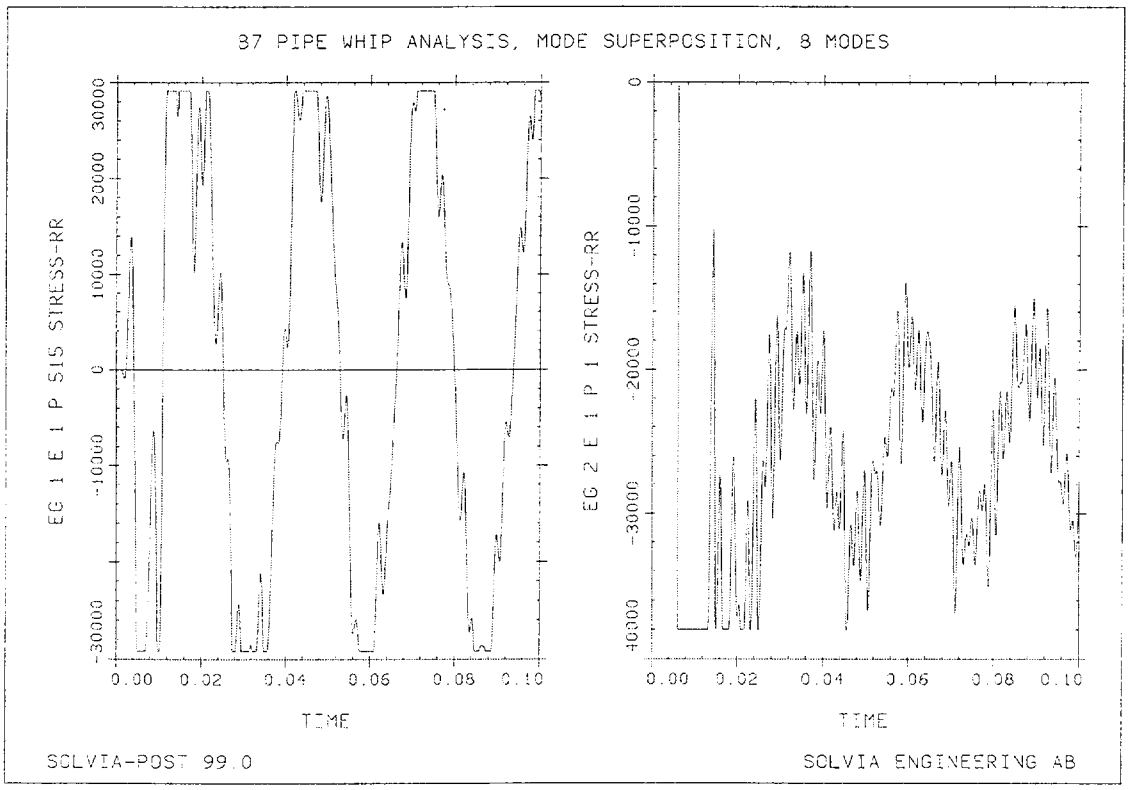

B7 PIPE WHIP ANALYSIS, MODE SUPERPOSITION, 8 MODES
ORIGINAL 50.
TIME 1E-4
Y
X
1
2
3
4
5
6
7
8
FORCE
657000
ORIGINAL 50.
Y
X
1
8
MASTER
001110
3 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

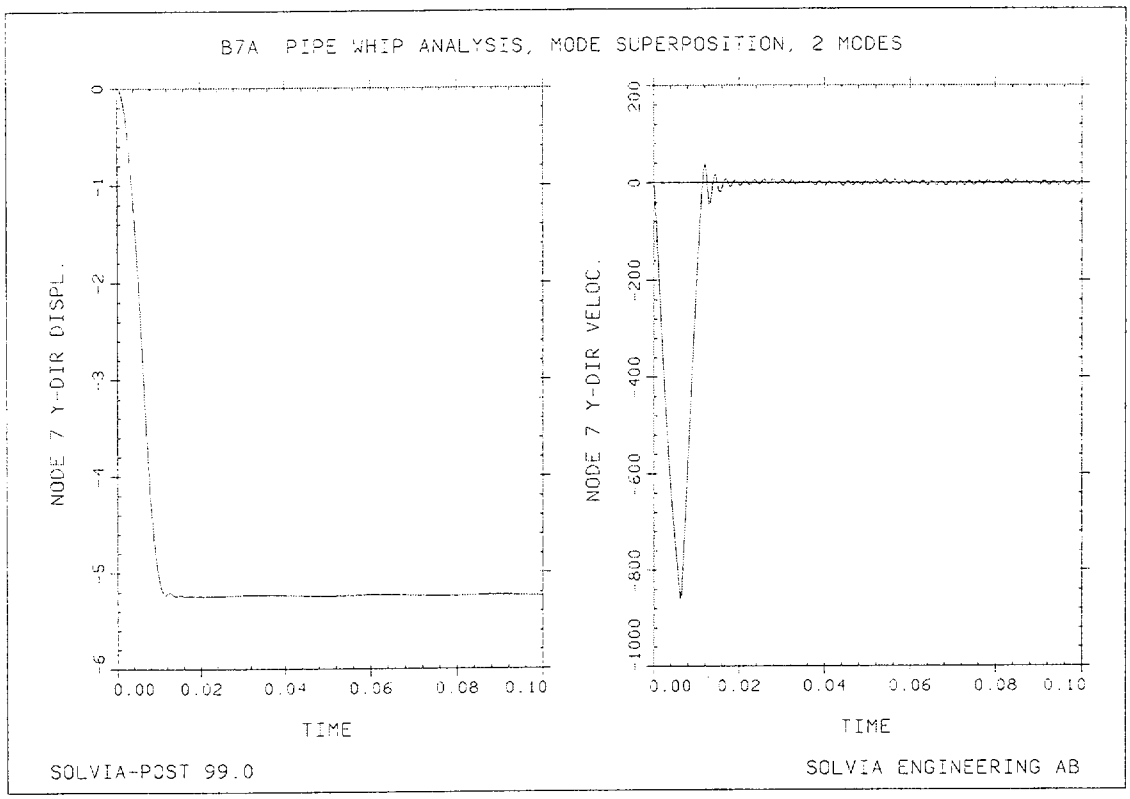

B7 PIPE WHIP ANALYSIS, MODE SUPERPOSITION, 8 MODES
REFERENCE 20.
MAX DISPL. 0.38523
MODE 1 FREQ 8.4373
Y
X
REFERENCE 20.
MAX DISPL. 0.35934
MODE 2 FREQ 51.3
Y
X
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

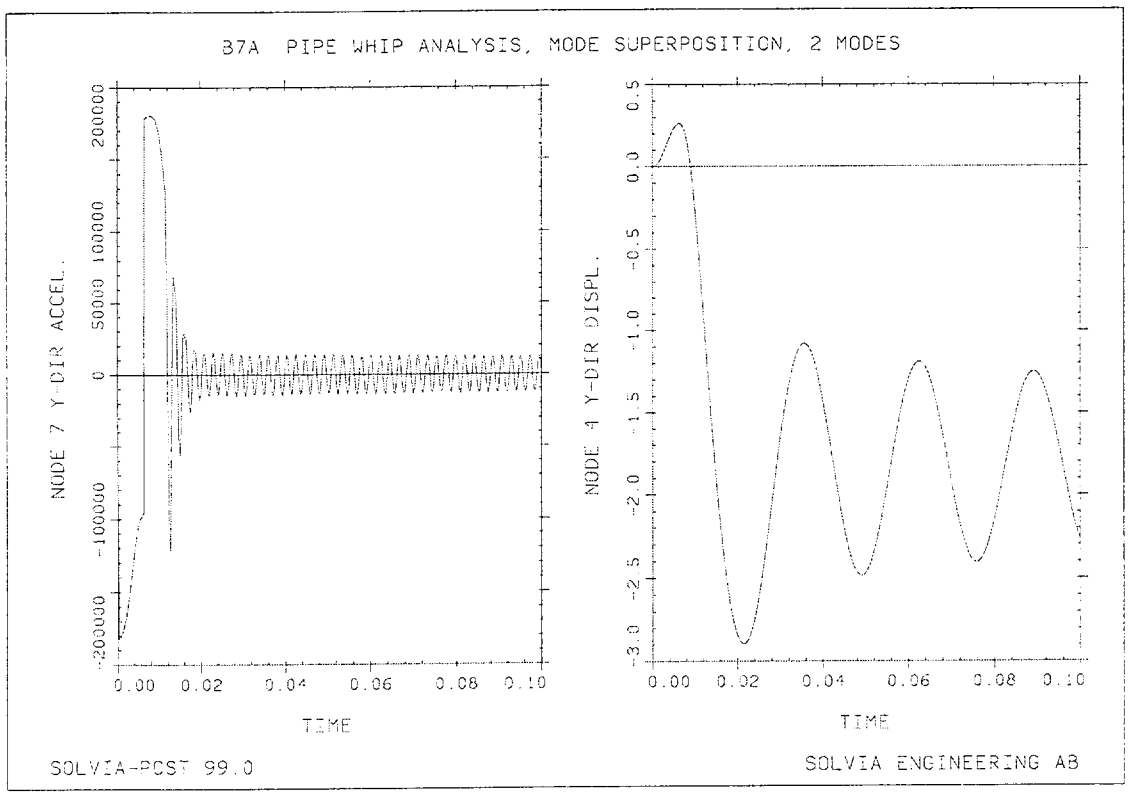

B7 PIPE WHIP ANALYSIS, MODE SUPERPOSITION, 8 MODES
NODE 7 Y-DIR DISPL.
0
-1
-2
-3
-4
-5
-6
0.00 0.02 0.04 0.06 0.08 0.10
TIME
NODE 7 Y-DIR VELOC.
200
0
-200
-400
-600
-800
-1000
0.00 0.02 0.04 0.06 0.08 0.10
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

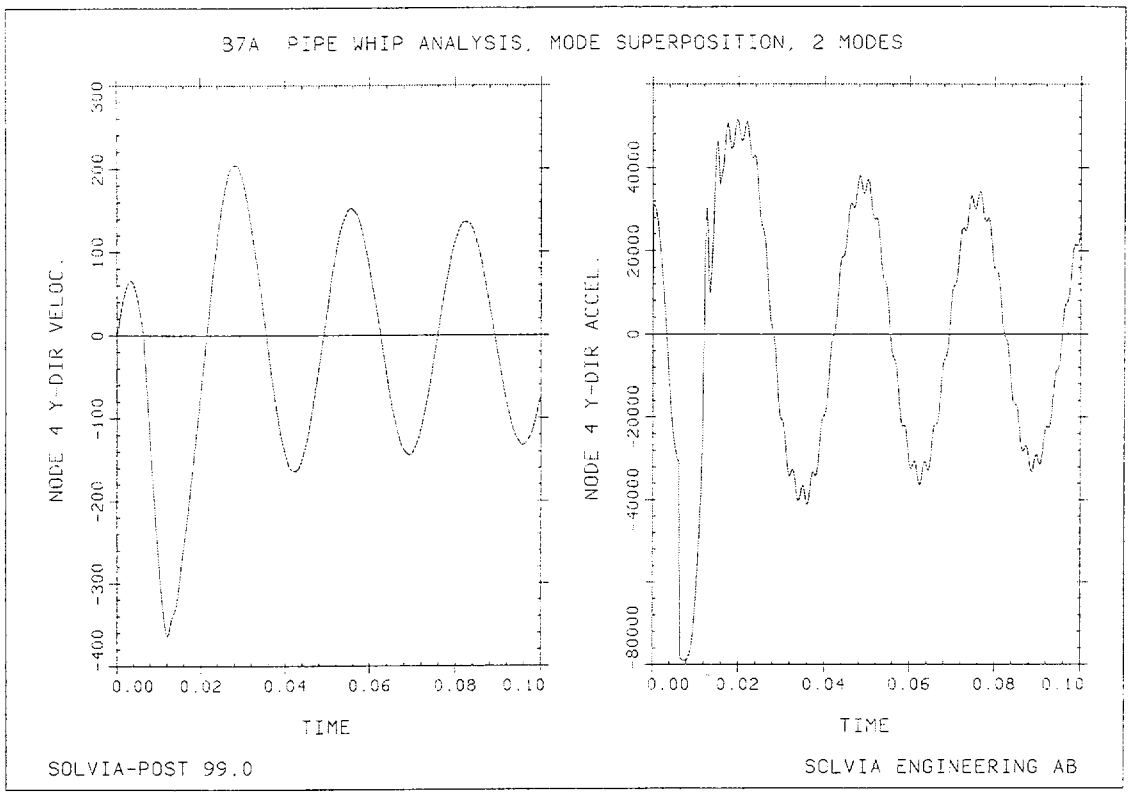

B7 PIPE WHIP ANALYSIS, MODE SUPERPOSITION, 8 MODES
NODE 7 Y-DIR ACCEL.
*10^3
400
300
200
100
0
-100
-200
-300
0.00 0.02 0.04 0.06 0.08 0.10
TIME
NODE 4 Y-DIR DISPL.
0.5
0.0
-0.5
-1.0
-1.5
-2.0
0.00 0.02 0.04 0.06 0.08 0.10
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

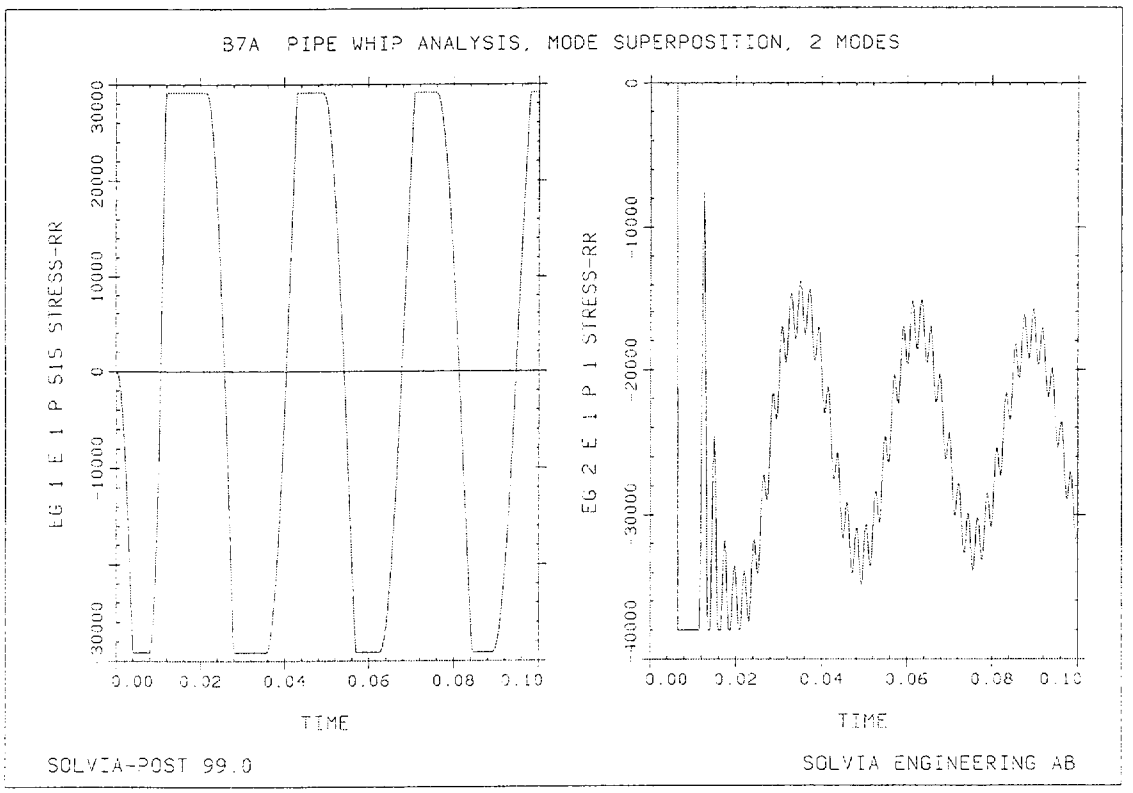

B7 PIPE WHIP ANALYSIS, MODE SUPERPOSITION, 8 MODES
NODE 4 Y-DIR VELOC.
200
100
0
-100
-200
-300
-400
0.00 0.02 0.04 0.06 0.08 0.10
TIME
NODE 4 Y-DIR ACCEL.
150000
50000
0
-50000
-150000
-250000
0.00 0.02 0.04 0.06 0.08 0.10
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B7 PIPE WHIP ANALYSIS, MODE SUPERPOSITION, 8 MODES
EG 1 E 1 P S15 STRESS-RR
30000
20000
10000
0
-10000
-30000
0.00 0.02 0.04 0.06 0.08 0.10
TIME
EG 2 E 1 P 1 STRESS-RR
0
-10000
-20000
-30000
-40000
0.00 0.02 0.04 0.06 0.08 0.10
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B7A PIPE WHIP ANALYSIS, MODE SUPERPOSITION, 2 MODES
NODE 7 Y-DIR DISPL.
0
-1
-2
-3
-4
-5
-6
0.00 0.02 0.04 0.06 0.08 0.10
TIME
NODE 7 Y-DIR VELOC.
200
0
-200
-400
-600
-800
-1000
0.00 0.02 0.04 0.06 0.08 0.10
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B7A PIPE WHIP ANALYSIS, MODE SUPERPOSITION, 2 MODES
NODE 7 Y-DIR ACCEL.
200000
100000
50000
0
-100000
-200000
0.00 0.02 0.04 0.06 0.08 0.10
TIME
NODE 4 Y-DIR DISPL.
0.5
0.0
-0.5
-1.0
-1.5
-2.0
-2.5
-3.0
0.00 0.02 0.04 0.06 0.08 0.10
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B7A PIPE WHIP ANALYSIS, MODE SUPERPOSITION, 2 MODES
NODE 4 Y-DIR VELOC.
NODE 4 Y-DIR ACCEL.
TIME
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B7A PIPE WHIP ANALYSIS, MODE SUPERPOSITION, 2 MODES
EG 1 E 1 P 515 STRESS-RR
EG 2 E 1 P 1 STRESS-RR
TIME
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B7 PIPE WHIP ANALYSIS, MODE SUPERPOSITION, 8 MODES'
*
DATABASE  CREATE
MASTER  IDOF=001110 NSTEP=1000 DT=1.E-4
FREQUENCIES  SUBSPACE-ITERATION NEIG=12 SSTOL=1.E-10
ANALYSIS  TYPE=DYNAMIC MASSMATRIX=LUMPED IMODS=1 NMODES=8
ITERATION  METHOD=MODIFIED-NEWTON
TOLERANCES ETOL=1E-8
*
COORDINATES
 1  TO  7  360.  /  8  360. -24.
*
MATERIAL  1  PLASTIC E=2.698E7 NU=0.3 YIELD=29140 DENSITY=7.18E-4
MATERIAL  2  PLASTIC E=2.990E7 NU=0.  YIELD=38000 ET=0. DEN=0.
*
EGROUP  1  BEAM DIM2 RINT=5 SINT=1 TINT=5 MATERIAL=1 RESULT=TABLE
STRESSTABLE  1  111 115 411 415 511 515
SECTION  1  DIMENSION PIPE DO=30. DI=27.75 ISHEAR=1
ENODES  /  1  8 1 2  TO  6  8 6 7
*
EGROUP  2  TRUSS  MATERIAL=2 GAP=YES
ENODES  /  1  8 7
EDATA  /   ENTRIES  EL  AREA  GAPWIDTH
 1  25.967 3.
*
FIXBOUNDARIES  /  1 8
LOADS  CONCENTRATED
 7 2 -6.57E5
*
MESH  VIEW=Z NSYMBOLS=YES NNUMBERS=YES VECTOR=LOAD SUBFRAME=12
MESH  VIEW=Z NSYMBOLS=YES BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B7  PIPE WHIP ANALYSIS, MODE SUPERPOSITION, 8 MODES
*
DATABASE  CREATE
WRITE  FILENAME='b7.lis'
*
FREQUENCIES
*
SET RESPONSETYPE=VIBRATION ORIGINAL=YES VIEW=Z
MESH NSYMBOL=YES TIME=1 SUBFRAME=12
MESH NSYMBOL=YES TIME=2
*
SUBFRAME  21
NHISTORY  NODE=7 DIRECTION=2 KIND=DISPLACEMENT
NHISTORY  NODE=7 DIRECTION=2 KIND=VELOCITY
*
SUBFRAME  21
NHISTORY  NODE=7 DIRECTION=2 KIND=ACCELERATION
NHISTORY  NODE=4 DIRECTION=2 KIND=DISPLACEMENT
*
SUBFRAME  21
NHISTORY  NODE=4 DIRECTION=2 KIND=VELOCITY
NHISTORY  NODE=4 DIRECTION=2 KIND=ACCELERATION
*
SUBFRAME  21
EHISTORY  ELEMENT=1 POINT=515 KIND=SRR
EGROUP  2
EHISTORY  ELEMENT=1 POINT=1 KIND=SRR
END
```

EXAMPLE B8

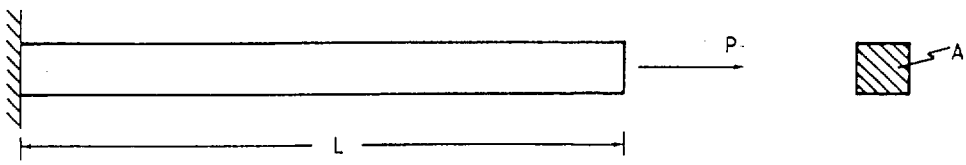
CYCLIC LOADING OF PLASTIC TRUSS, ISOTROPIC HARDENING

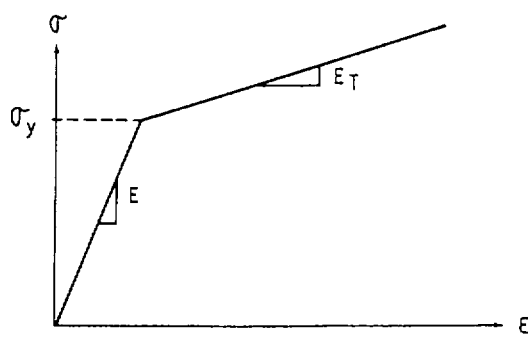
Objective

To verify the behaviour of the 3-node TRUSS element in MNO (Materially-Nonlinear-Only) analysis using the plastic isotropic hardening material model.

Physical Problem

A bar subjected to an axial force as shown in the figure below is considered. The material of the bar is assumed to obey an elastic-plastic material law with isotropic hardening.





$A = 0.01\ m^2$

$L = 1.0\ m$

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

$\sigma_y = 4.0 \cdot 10^{8}\ N/m^2$

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

Finite Element Model

The finite element model consists of one 3-node TRUSS element and the element stiffness matrix is evaluated using 2-point Gauss integration. The applied force is cyclicly varied to obtain plastic response both in tension and compression of the truss. Stiffness reformation and BFGS equilibrium iterations are performed at every time step.

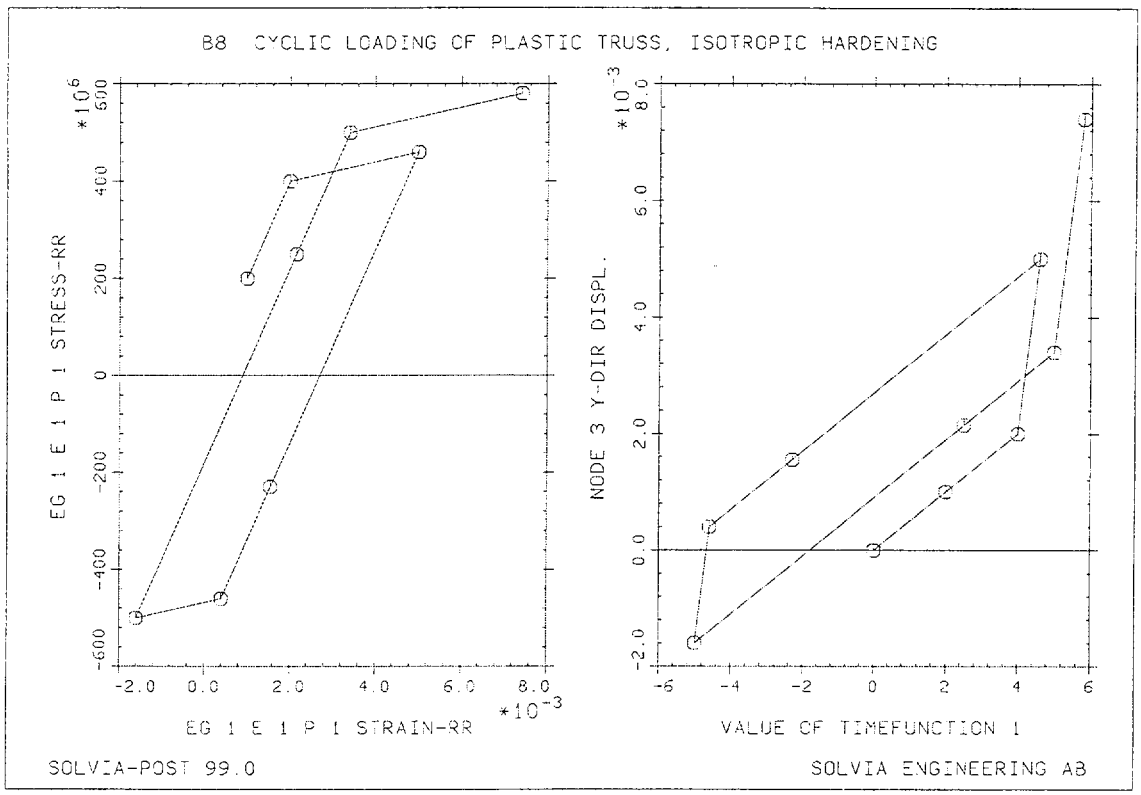
Solution Results

The input data on page B8.3 is used in the finite element analysis. The figure on page B8.2 shows the stress-strain response in the TRUSS element predicted in the analysis as displayed by SOLVIA-POST. A hand calculation shows that the response predicted by SOLVIA agrees with the analytical solution of the model.

User Hints:

- To model this example, of course, only a 2-node truss and 1-point Gauss integration need be employed. The results for the 2 Gauss points used are exactly the same and the midnode displacement is exactly half of the displacement at the end node.

- It is important to use the BFGS or the full Newton method of equilibrium iteration in this example. The modified Newton method of iteration does not converge at unloading in this example from plastic to elastic conditions, and has difficulty to converge when loading from elastic to plastic conditions.



SOLVIA-PRE input

```
HEADING  'B8  CYCLIC LOADING OF PLASTIC TRUSS, ISOTROPIC HARDENING'
*
DATABASE  CREATE
MASTER  IDOF=101111 NSTEP=9
*
TIMEFUNCTION  1
 0.  0.   /  1.  2.0  /  2.  4.0
 3.  4.6  /  4. -2.3  /  5. -4.6
 6. -5.0  /  7.  2.5  /  8.  5.0
 9.  5.8
*
COORDINATES
 1  TO  3  0. 1.
*
MATERIAL  1  PLASTIC ISOTROPIC  E=2.E11 YIELD=4.E8 ET=2.E10
*
EGROUP  1  TRUSS
ENODES  /  1  1 3 2
EDATA  /  1  0.01
*
FIXBOUNDARIES  /  1
LOADS  CONCENTRATED
 3 2 1.E6
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B8 CYLIC LOADING OF PLASTIC TRUSS, ISOTROPIC HARDENING
*
DATABASE  CREATE
*
WRITE  FILENAME='b8.lis'
*
SUBFRAME  21
EXYPLOT  EL=1 POINT=1 XKIND=ERR YKIND=SRR OUTPUT=ALL SYMBOL=1
NHISTORY  NODE=3 DIRECTION=2 XVARIABLE=1 OUTPUT=ALL SYMBOL=1
END
```

EXAMPLE B9

CYCLIC LOADING OF PLASTIC TRUSS, KINEMATIC HARDENING

Objective

To verify the behaviour of the 3-node TRUSS element in MNO (Materially-Nonlinear-Only) analysis using the plastic kinematic hardening material model.

Physical Problem

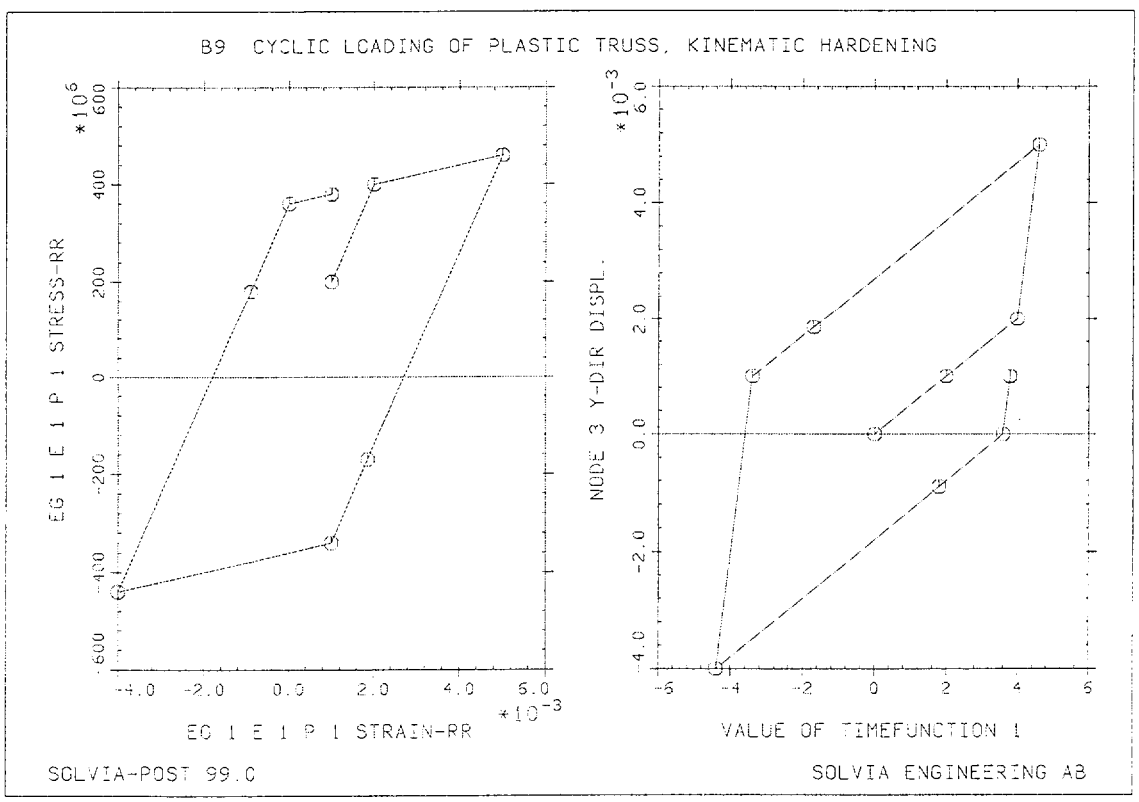
Same as in Example B8.

Finite Element Model

Same as in Example B8 except that the material of the TRUSS element assumes an elastic-plastic kinematic hardening material law and other time function values.

Solution Results

The input data on pages B9.2 is used in the finite element analysis. The figure below shows the stress-strain response at one integration point of the TRUSS element during the load history in the finite element analysis as displayed by SOLVIA-POST. A hand calculation shows that the response predicted by SOLVIA agrees with the analytical solution of the model.



SOLVIA-PRE input

```
HEADING  'B9  CYCLIC LOADING OF PLASTIC TRUSS, KINEMATIC HARDENING'
*
DATABASE  CREATE
MASTER  IDOF=101111 NSTEP=9
*
TIMEFUNCTION  1
 0.  0.   /  1.  2.0  /  2.  4.0
 3.  4.6  /  4. -1.7  /  5. -3.4
 6. -4.4  /  7.  1.8  /  8.  3.6
 9.  3.8
*
COORDINATES
 1  TO  3  0. 1.
*
MATERIAL  1  PLASTIC KINEMATIC  E=2.E11 YIELD=4.E8 ET=2.E10
*
EGROUP  1  TRUSS
ENODES  /  1  1 3 2
EDATA  /  1  0.01
*
FIXBOUNDARIES  /  1
LOADS  CONCENTRATED
 3 2 1.E6
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B9  CYLIC LOADING OF PLASTIC TRUSS, KINEMATIC HARDENING
*
DATABASE  CREATE
*
WRITE  FILENAME='b9.lis'
*
SUBFRAME  21
EXYPLOT  EL=1 POINT=1 XKIND=ERR YKIND=SRR OUTPUT=ALL SYMBOL=1
NHISTORY  NODE=3 DIRECTION=2 XVARIABLE=1 OUTPUT=ALL SYMBOL=1
END
```

EXAMPLE B10

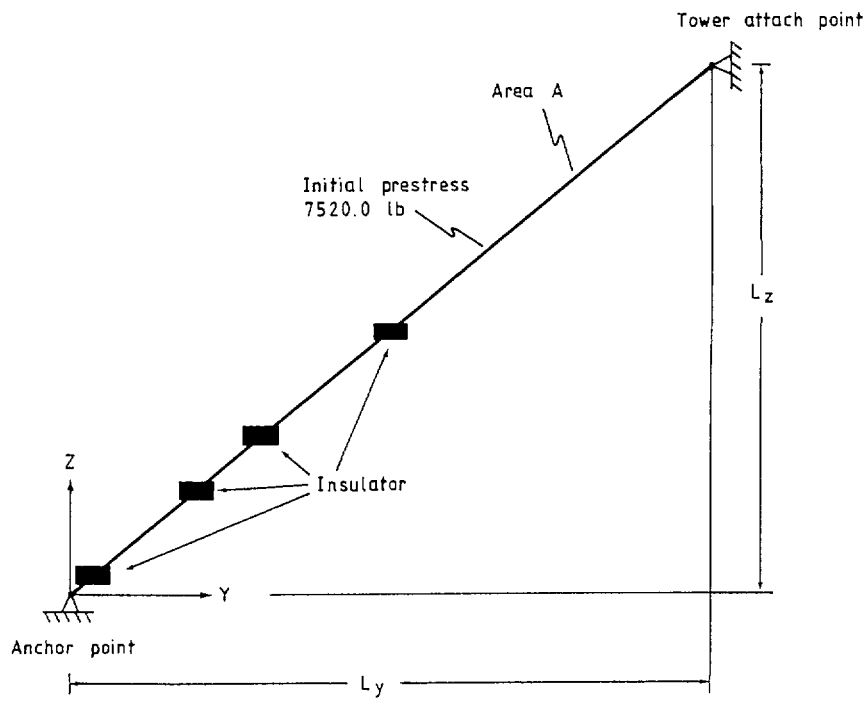
CABLE UNDER GRAVITY LOAD

Objective

To verify the large displacement behaviour of the TRUSS element and demonstrate the modeling of analyses of cables.

Physical Problem

A cable stretched between a ground anchor point and a tower attach point, see figure below, is considered. The cable has a cluster of 6 insulators attached to a point in the middle portion and 3 insulators attached to the lower portion. The weight of each insulator is 510 lbs. The initial prestress of the cable is 7520 lbs and the self weight is 0.106667 lb/in.



$A = 0.361 \text{ in}^2$

$L_y = 8191.20 \text{ in}$

$L_z = 7312.05 \text{ in}$

$E = 1.9 \cdot 10^7 \text{ psi}$

$\gamma = 0.295476 \text{ lb/in}^3$

$\gamma A = 0.106667 \text{ lb/in}$

Finite Element Model

The figure on page B10.3 shows the finite element model. It consists of twelve 2-node TRUSS elements. The insulators are attached as concentrated masses at the nodes 2, 4, 6 and 8. The material model for the TRUSS elements is linear elastic and a large displacement analysis is prescribed.

Solution Results

The input to SOLVIA-PRE and SOLVIA-POST are shown on pages B10.5 and B10.6. The Full Newton method of equilibrium iteration is used. The energy tolerance is set to $ETOL = 1.0 \cdot 10^{-5}$. The gravity loading is applied in 5 equal load steps. The calculated deformed configuration and the displacements for node 8 in the Y- and Z-direction are given in figures on page B10.4. We note that the cable stiffens as the gravity load is increased. For analytical verification we select the following results as typical:

Node	Displacements (inch)		Deformed Coordinates (inch)	
	y	z	y	z
7	215.746	-248.196	3421.8026	2613.7601
8	242.152	-275.898	4132.1693	3196.6091
9	199.365	-226.344	4949.6183	4014.0711

Element	Truss Internal Force (lbs)
7	22843.5
8	24975.0

Using the above solution the following internal TRUSS element forces can be calculated theoretically:

Element 7:

$$F_7 = F_p + \left(\frac{l_7}{L_7} - 1\right) EA = 22844 \text{ (lbs)}$$

where

F_p = initial prestress,

l_7 = final length,

L_7 = original length,

Element 8:

$$F_8 = F_p + \left(\frac{l_8}{L_8} - 1\right) EA = 24975 \text{ (lbs)}$$

The fifth digit is uncertain due to the number of digits in the listed results. Considering this accuracy it is concluded that the calculated values for F_7 and F_8 agree with the corresponding SOLVIA results.

By projecting the SOLVIA calculated internal forces for elements 7 and 8 onto the vertical (z) axis it can be seen that equilibrium is satisfied in the vertical direction for node 8.

User Hints

- The initial prestress in the TRUSS elements is important since it provides stiffness in the lateral direction for the elements in a large displacement analysis. Without the prestress the stiffness matrix in this example would be singular and no solution would be possible. See also Example B51.

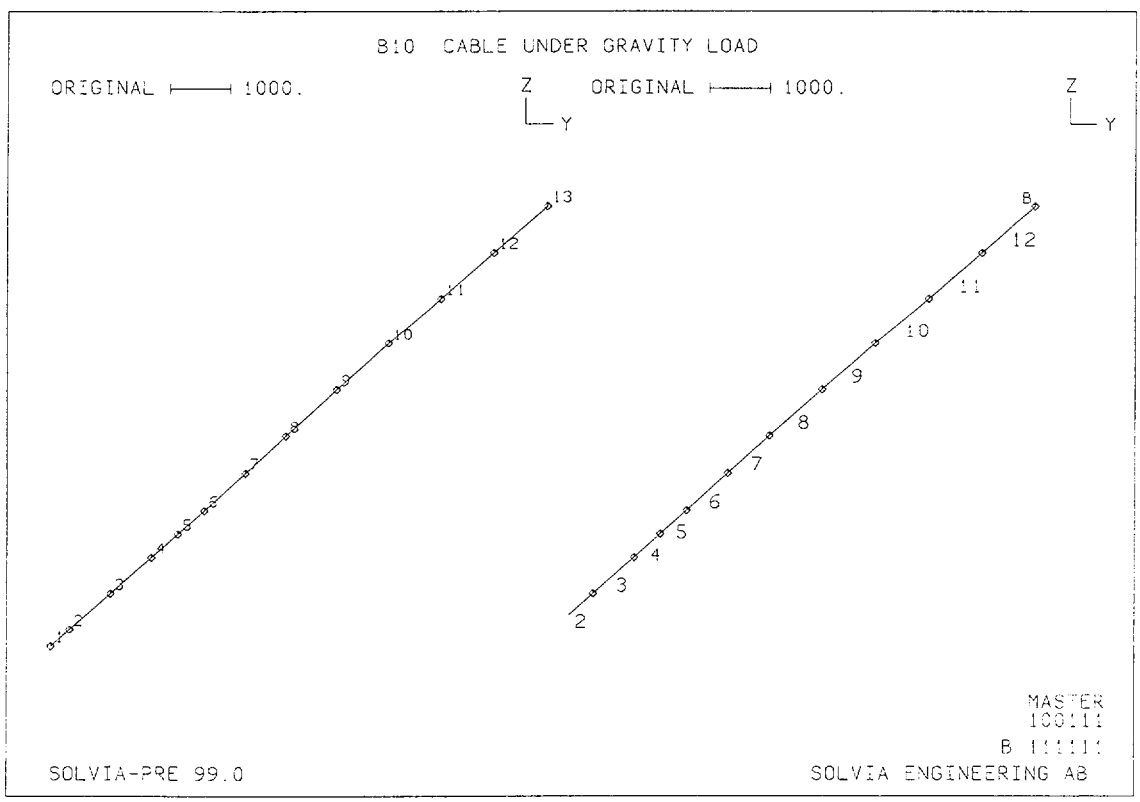
- The steady-state solution for gravity loading is considered a static load case in SOLVIA. The lumped masses of the model are assembled and only used to calculate the corresponding concentrated weight forces (using the acceleration of gravity).

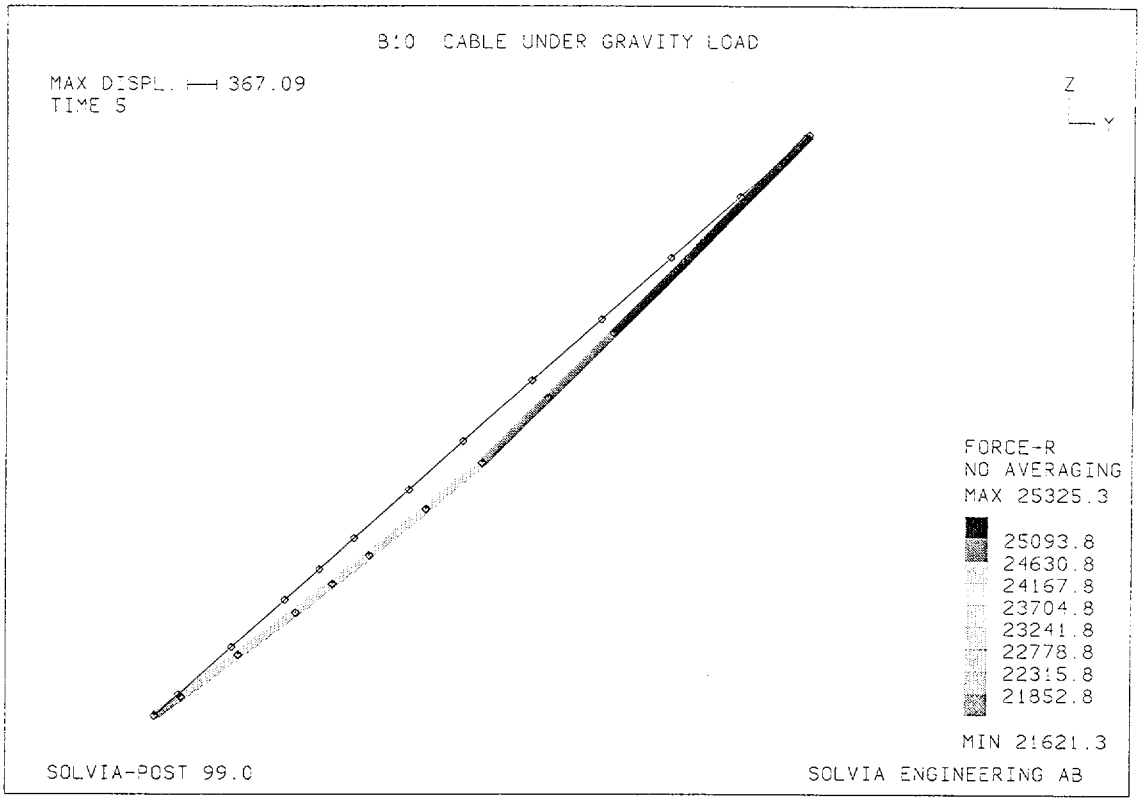
- The internal forces and the stresses in the TRUSS elements are calculated using the small strain assumption, but large displacements and rotations are included in the large displacement formulation.

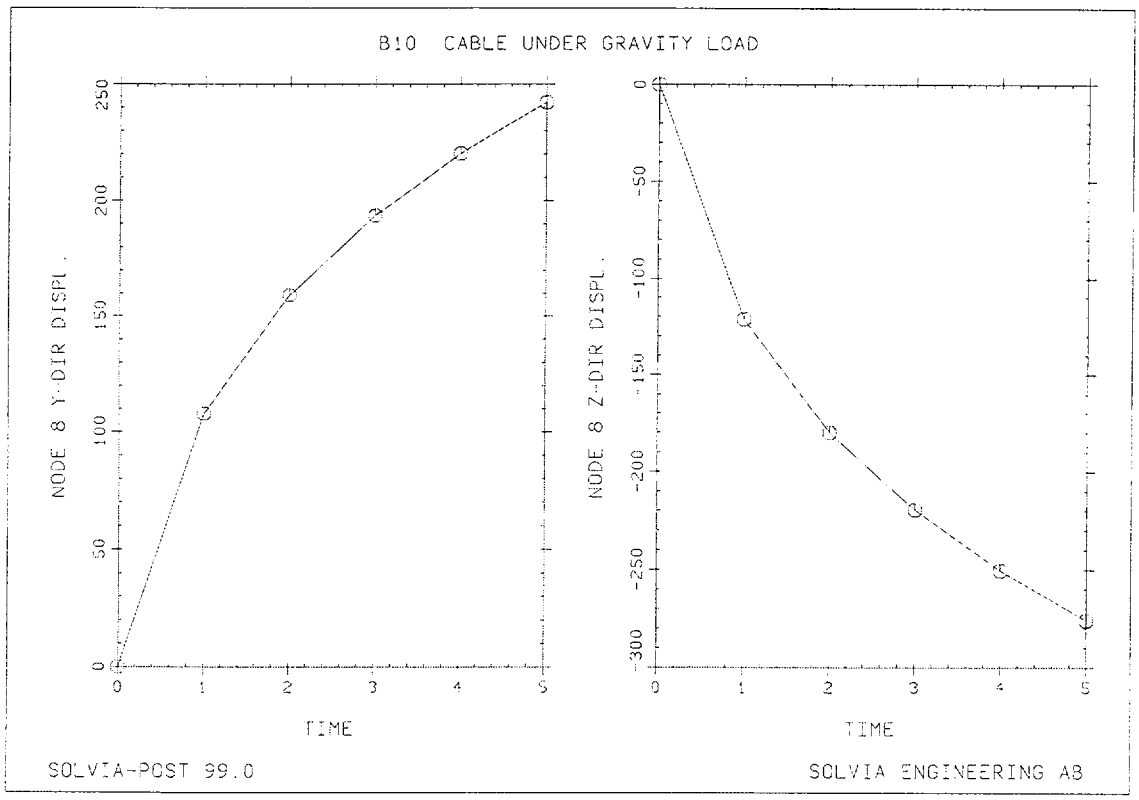
  Hence, here the area of a TRUSS element remains constant and the element strain is equal to

  $$\frac{(\text{final length}) - (\text{original length})}{(\text{original length})}$$

- In a stiffening problem it may be necessary to use force tolerances in combination with the energy tolerance during the equilibrium iterations. Small variations in displacement increments may give relatively large variations in unbalance forces.




B10 CABLE UNDER GRAVITY LOAD
MAX DISPL. 367.09
TIME 5
Z
Y
FORCE-R
NO AVERAGING
MAX 25325.3
25093.8
24630.8
24167.8
23704.8
23241.8
22778.8
22315.8
21852.8
MIN 21621.3
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B10 CABLE UNDER GRAVITY LOAD
NODE 8 Y-DIR DISPL.
0
50
100
150
200
250
0 1 2 3 4 5
TIME
NODE 8 Z-DIR DISPL.
0
-50
-100
-150
-200
-250
-300
0 1 2 3 4 5
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEAD  'B10  CABLE UNDER GRAVITY LOAD'
*
DATABASE  CREATE
MASTER  IDOF=100111 NSTEP=5
ANALYSIS  TYPE=STATIC MASSMATRIX=LUMPED
KINEMATICS  DISPLACEMENT=LARGE
TOLERANCES  EF ETOL=1.E-5 RTOL=1.E-2 RNORM=500.
ITERATION  METHOD=FULL-NEWTON LINE-SEARCH=YES
*
TIMEFUNCTION  1
 0. 0.  /  5. 1.
*
COORDINATES
 ENTRIES  NODE    Y          Z
           1      0.         0.
           2    306.049   273.201
           3    977.988   873.022
           4   1649.927  1472.843
           5   2086.012  1862.123
           6   2522.097  2251.404
           7   3206.057  2861.956
           8   3890.017  3472.507  TO
          13   8191.200  7312.050
MASSES
 2  0.  510.   510.
 4  0.  510.   510.
 6  0.  510.   510.
 8  0. 3060. 3060.
*
MATERIAL  1  ELASTIC E=1.9E7 DENSITY=0.295476
*
EGROUP  1  TRUSS
ENODES  /  1  1 2  TO  12  12 13
EDATA
 ENTRIES  EL AREA  INIT-STRAIN
          1  0.361 1.09637E-3  TO  12 0.361 1.09637E-3
*
FIXBOUNDARIES  /  1 13
LOADS  MASSPROPORTIONAL  ZFACTOR=-1. ACCGRA=1.
*
MESH  VIEW=X NSYMBOLS=YES NNUMBERS=YES SUBFRAME=21
MESH  VIEW=X NSYMBOLS=YES ENUMBERS=YES BCODE=ALL GSCALE=OLD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B10  CABLE UNDER GRAVITY LOAD
*
DATABASE  CREATE
WRITE  FILENAME='b10.lis'
*
MASS-PROPERTIES
*
SET  VIEW=X
CONTOUR  AVERAGE=NO
MESH  NSYMBOLS=YES CONTOUR=FR
MESH  ORIGINAL=YES DEFORMED=NO TEXT=NO AXES=NO GSCALE=OLD,
      NSYMBOLS=YES SUBFRAME=OLD
*
SUBFRAME  21
NHISTORY  NODE=8 DIRECTION=2 SYMBOL=1
NHISTORY  NODE=8 DIRECTION=3 SYMBOL=1
*
NLIST
ELIST
GLIST
*
SUMMATION  KIND=LOAD
SUMMATION  KIND=REACTION DETAILS=YES
END
```

EXAMPLE B11

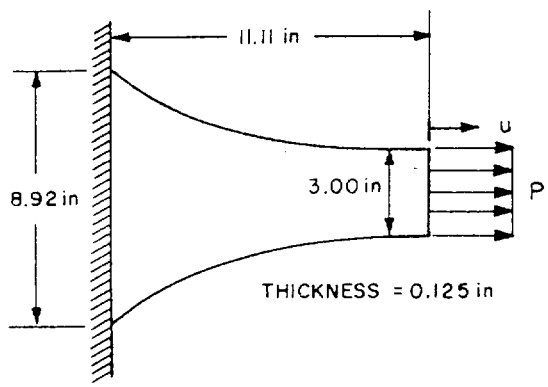
RUBBER SHEET IN LARGE STRAIN ANALYSIS

Objective

To verify the large displacement and large strain behaviour of the PLANE STRESS2 element employing the rubber model.

Physical Problem

A rubber sheet subjected to a uniform end load, see figure below, is considered. The material is assumed to be of the Mooney-Rivlin type, for which experiments by Iding [1, 2] gave $C_1 = 21.605$ lb./in.^{2}, $C_2 = 15.747$ lb./in.^{2}



RUBBER SHEET

Finite Element Model

By symmetry only one half of the rubber strip need be modeled, see figure on page B11.2. Thirtythree 4-node PLANE STRESS2 elements are used and the end loading is applied as concentrated forces.

Solution Results

The final total end load of 41.80 lb. is reached in 4 equal load steps using modified Newton iteration and stiffness reformation for every step. The tolerance is ETOL=1.E-5. The input data used are shown on pages B11.4 and B11.5.

Note that the scale for the displacements is the same as for the geometry. The end displacements corresponding to the load steps are shown in the top figure on page B11.3 and good agreement with experimental results by Iding is obtained. A contour plot of the maximum principal stretch and a vector plot of the principal stretches in the YZ-plane are also shown in the figure.

The stress in the Y-direction is shown as a function of the Green-Lagrange strain for element 11, integration point 1 in the left bottom figure on page B11.3. The stress distributions at sections A-A and B-B are shown in the right bottom figure on page B11.3, respectively. These stress distributions are also shown as displayed by SOLVIA-POST.

User Hints

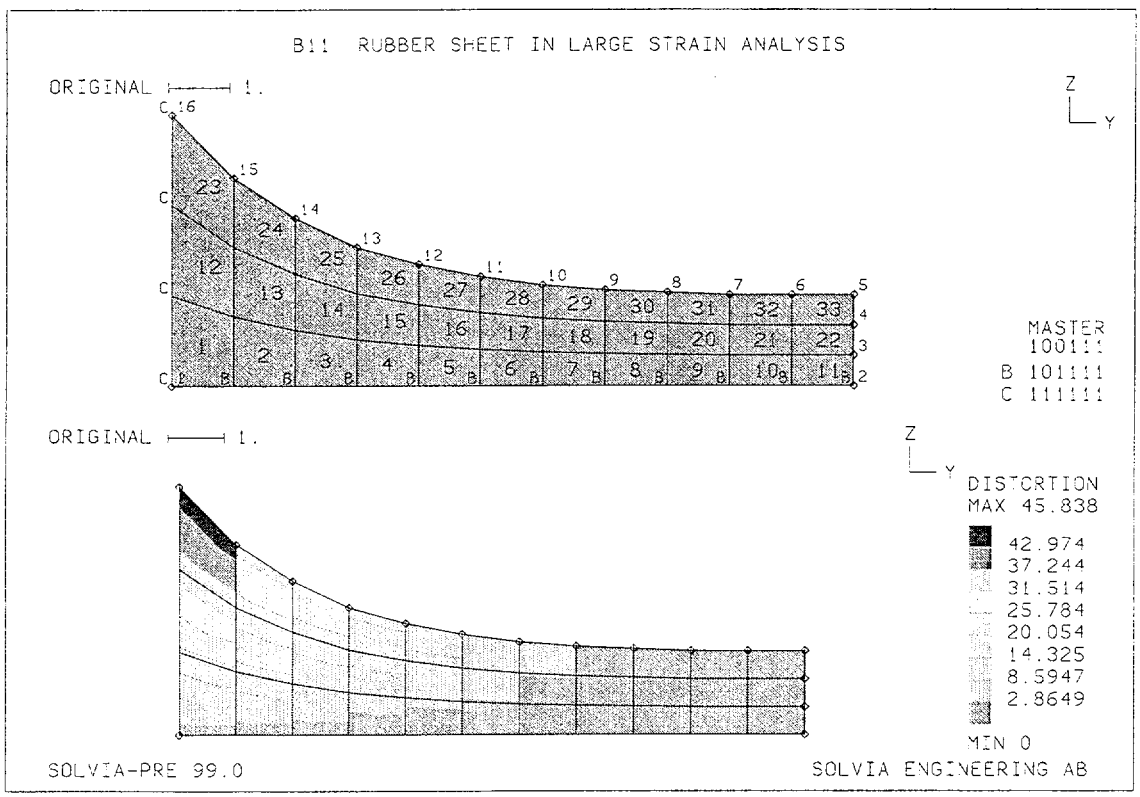
- The modified Newton iteration method is quite economical to use in this example, since nonlinearities monotonically soften the structure. However, the nonlinearities are very significant.

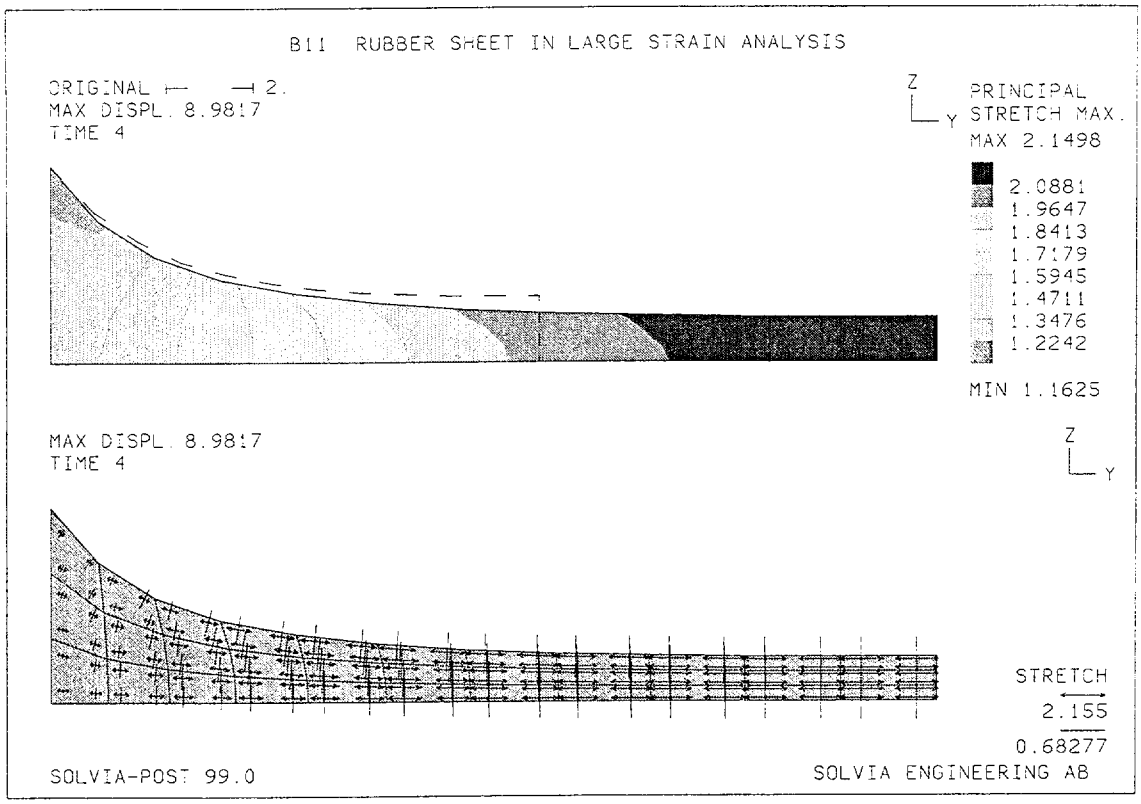
- Using 8-node PLANE STRESS2 elements much less elements could be used in the model for an accurate response prediction.

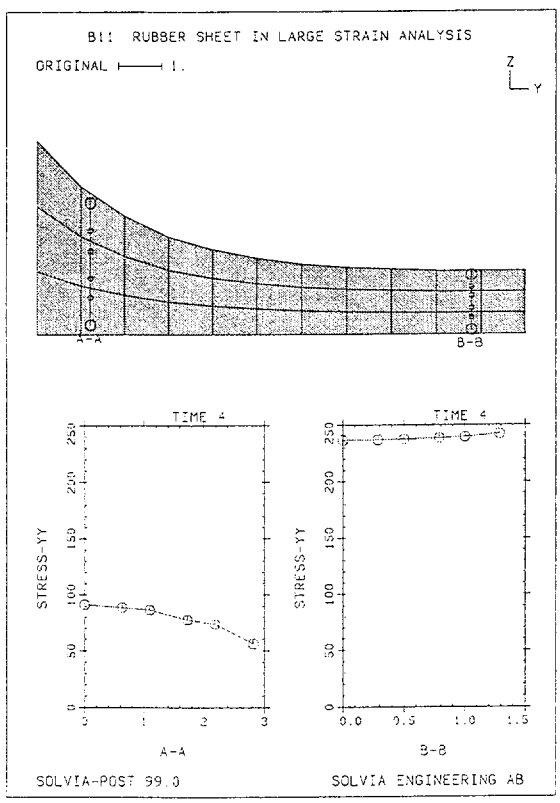
References

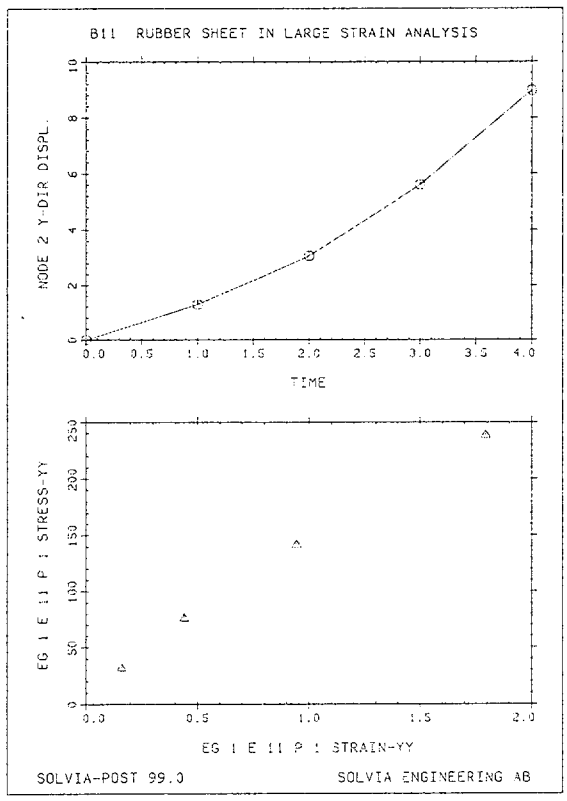
[1] Iding, R.H., "Identifications of Nonlinear Material by Finite Element Methods", Report UC SESM 73-4, Department of Civil Engineering, University of California, Berkeley, Jan., 1973.

[2] "NONSAP - A Structural Analysis Program for Static and Dynamic Response of Nonlinear Systems", Report UC SESM 74-3, Department of Civil Engineering, University of California, Berkeley, February 1974.




B11 RUBBER SHEET IN LARGE STRAIN ANALYSIS
ORIGINAL 2.
MAX DISPL. 8.9817
TIME 4
PRINCIPAL STRETCH MAX.
MAX 2.1498
2.0881
1.9647
1.8413
1.7179
1.5945
1.4711
1.3476
1.2242
MIN 1.1625
MAX DISPL. 8.9817
TIME 4
STRETCH
2.155
0.68277
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B11 RUBBER SHEET IN LARGE STRAIN ANALYSIS
ORIGINAL 1.
A-A
B-B
TIME 4
STRESS-YY
A-A
TIME 4
STRESS-YY
B-B
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B11 RUBBER SHEET IN LARGE STRAIN ANALYSIS
NODE 2 Y-DIR DISPL.
TIME
EG 1 E 11 P 1 STRESS-YY
EG 1 E 11 P 1 STRAIN-YY
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B11  RUBBER SHEET IN LARGE STRAIN ANALYSIS'
*
DATABASE  CREATE
MASTER  IDOF=100111 NSTEP=4
KINEMATICS  DISPLACEMENTS=LARGE STRAINS=LARGE
ITERATION   METHOD=MODIFIED-NEWTON LINE-SEARCH=YES
TOLERANCES  ETOL=1.E-5
*
TIMEFUNCTION  1
 0. 0.  /  4. 20.9
*
COORDINATES  /  ENTRIES  NODE  Y  Z
  1  /  2  11.11  TO  5  11.11 1.5  TO  7  9.09 1.5
  8  8.08 1.54  /  9  7.07 1.58  / 10  6.06 1.66
 11  5.05 1.8   / 12  4.04 2.0   / 13  3.03 2.28
 14  2.02 2.76  / 15  1.01 3.42  / 16  0.   4.46
*
LINE  NODES  2 5  /  3 4
LINE  NODES  5 16  /  6  TO  15
*
MATERIAL  1 RUBBER  C1=21.605 C2=15.747
*
EGROUP  1 PLANE STRESS2
GSURFACE  1 2 5 16  EL1=11 EL2=3 NODES=4
EDATA  /  1  0.125
*
FIXBOUNDARIES  23 INPUT=LINE  /  1 16
FIXBOUNDARIES  3  INPUT=LINE  /  1 2
LOADS  CONCENTRATED
 2  2 .16666667
 3  2 .33333333
 4  2 .33333333
 5  2 .16666667
*
SET  NSYMBOLS=MYNODES
SUBFRAME  12
MESH  NNUMBERS=MYNODES ENUMBERS=YES BCODE=ALL
MESH  CONTOUR=DISTORTION
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B11  RUBBER SHEET IN LARGE STRAIN ANALYSIS
*
DATABASE  CREATE
*
WRITE  FILENAME='b11.lis'
*
SUBFRAME  12
MESH  ORIGINAL=DASHED CONTOUR=LPMAX OUTLINE=YES
MESH  VECTOR=STRETCH
*
EPLINE  NAME=A-A
 2 4 3  STEP 11 TO  24 4 3
EPLINE  NAME=B-B
 10 2 1  STEP 11 TO  32 2 1
*
AXIS  ID=1 VMIN=0 VMAX=250 LABEL='STRESS-YY'
*
SET  PLOTORIENTATION=PORTRAIT
FRAME
MESH  PLINES=ALL SUBFRAME=1122
ELINE  LINENAME=A-A KIND=SYY SYMBOL=1 YAXIS=1 SUBFRAME=2121
ELINE  LINENAME=B-B KIND=SYY SYMBOL=1 YAXIS=1 SUBFRAME=2221
SUBFRAME 12
NHISTORY  NODE=2 DIRECTION=2 SYMBOL=1
EXYPLOT  ELEMENT=11 POINT=1 XKIND=EYY YKIND=SYY SYMBOL=-2
*
ZONE  NAME=BB INPUT=ELEMENTS  /  10 21 32
SET  ZONENAME=BB
NLIST
ELIST
ELIST SELECT=STRETCH
EMAX SELECT=STRETCH
GLIST E32
GLIST E32 TSTART=0
END
```

EXAMPLE B12

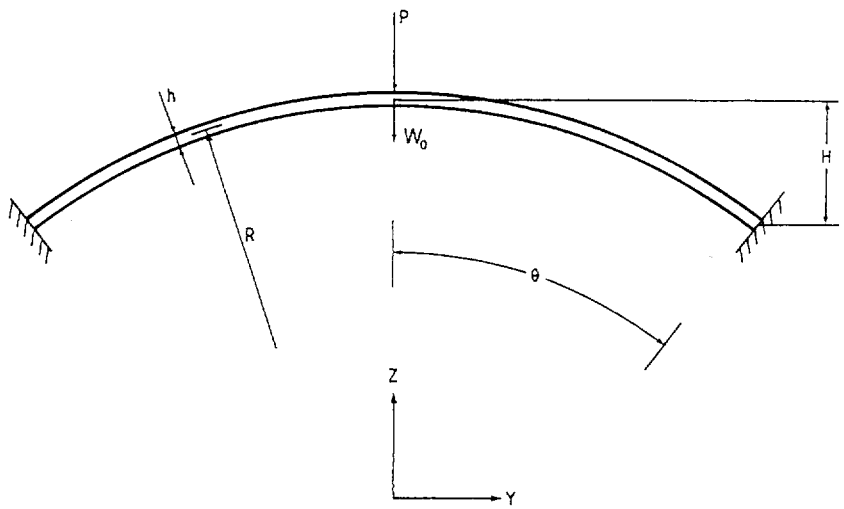
LARGE DISPLACEMENT ANALYSIS OF A SPHERICAL SHELL

Objective

To verify the PLANE AXISYMMETRIC element in large displacements.

Physical Problem

A spherical shell subjected to a concentrated apex load, see figure below, is considered. This problem is also analyzed in Example B65 using the AUTOMATIC-ITERATION method.



$h = 0.01576$ in

$H = 0.0859$ in

$R = 4.76$ in

$\theta = 10.9^{\circ}$

$E = 1.0 \cdot 10^7$ psi

$\nu = 0.3$

Finite Element Model

Due to symmetry only one-half of the spherical shell is modelled using ten 8-node PLANE AXISYMMETRIC elements as shown in the figure on page B12.2. The nodes at the apex can only slide in the vertical (Z) direction. At the built-in end the top and bottom nodes can slide in the c-direction of the skew system and the midnode is fixed in b- and c-direction.

Solution Results

The input data used in the analysis is shown on pages B12.4 and B12.5. The final total apex load is 100 lbs which is reached in 20 equal load steps using Full-Newton iterations.

The deformed mesh for load step 20 is shown in the top figure on page B12.3. The central deflection as function of the applied apex load for 1 radian is shown in the left bottom figure on page B12.3. The history of the element stress σ_{rr} at integration point 7, which is closest to the bottom surface and to the built-in end, is shown in the same figure. The element r-axis is in the θ-direction of the shell.

The apex point load as a function of the deflection ratio W_0 / H is shown in the right bottom figure on page B12.3.

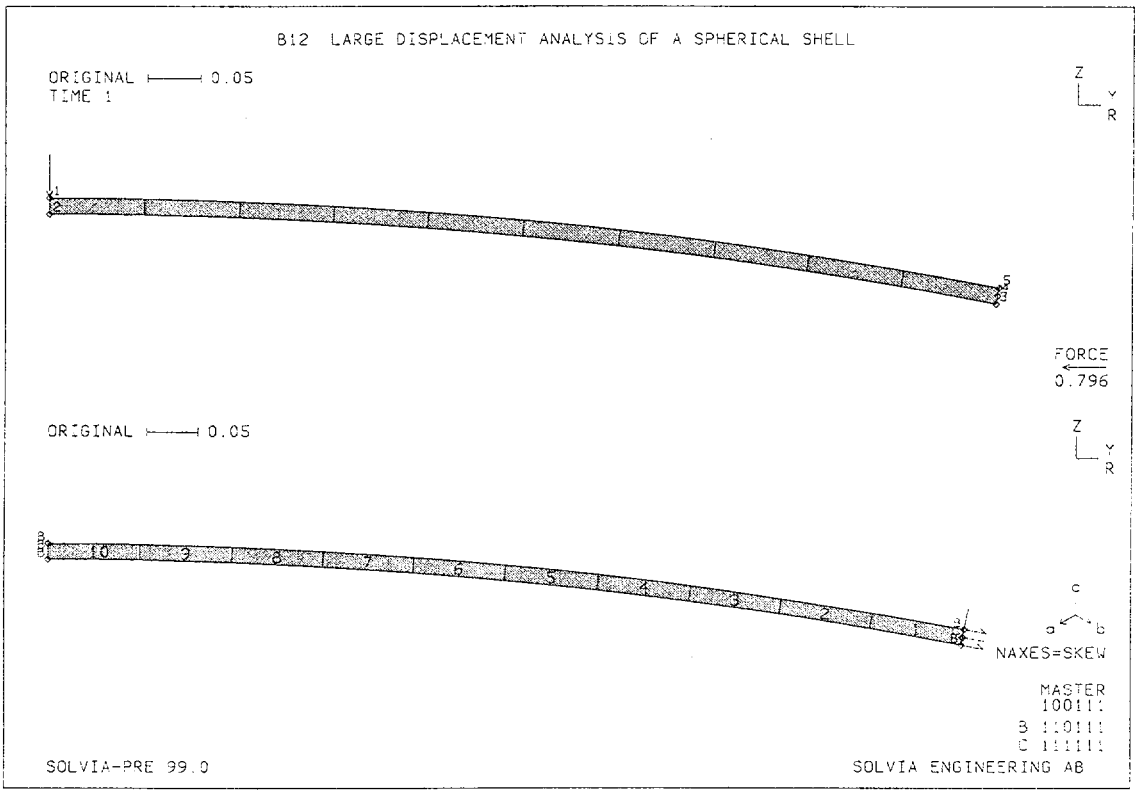
User Hints

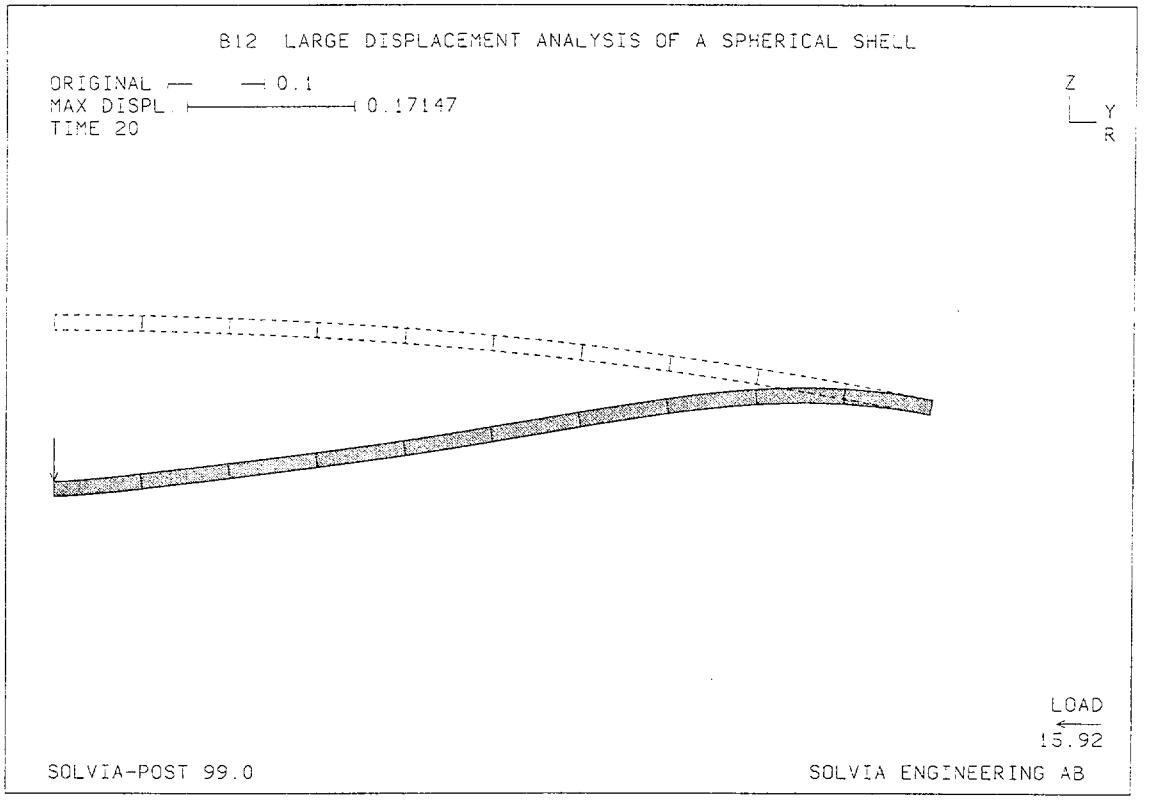
- The PLANE AXISYMMETRIC element extends 1 radian in the circumferential direction. The total apex load of 100 lbs applied to the actual shell is, therefore, represented by a load of $100/2\pi$ applied to the model.

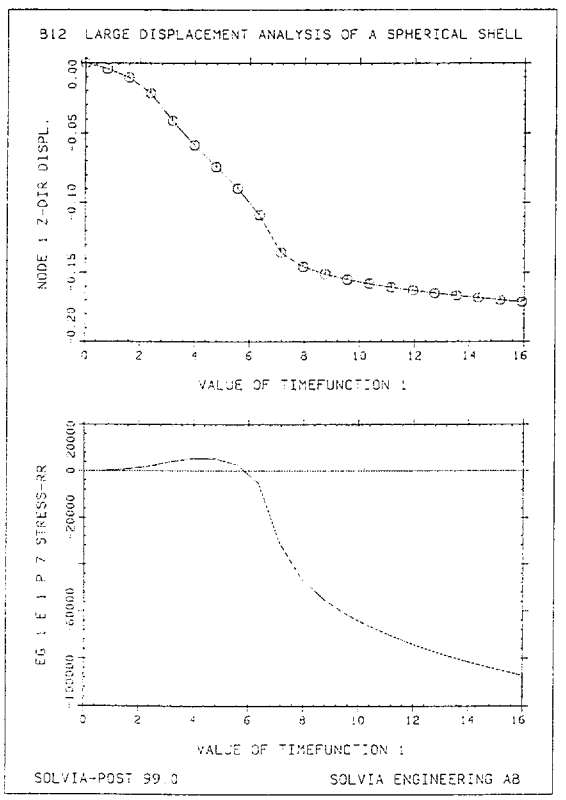
References

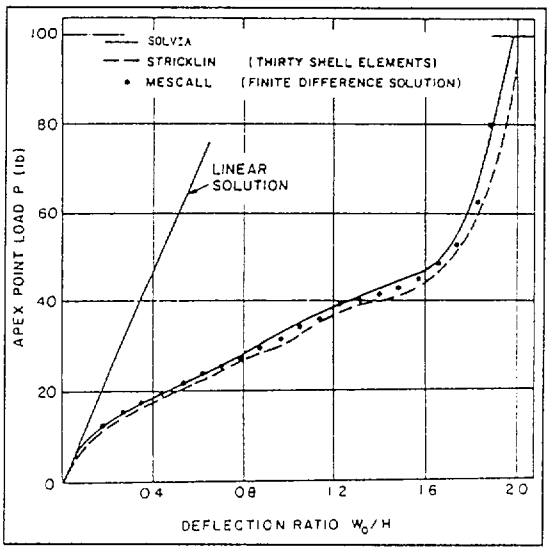
[1] Stricklin, J.A., "Geometrically Nonlinear Static and Dynamic Analysis of Shells of Revolution", High Speed Computing of Elastic Structures, Proceedings of the Symposium of IUTAM, Univ. of Liege, August, 1970.

[2] Mescall, J.F., "Large Deflections of Spherical Shells Under Concentrated Loads", J. App. Mech., Vol. 32, pp. 936-938, 1965.




B12 LARGE DISPLACEMENT ANALYSIS OF A SPHERICAL SHELL
ORIGINAL 0.1
MAX DISPL 0.17147
TIME 20
Z
Y
R
LOAD
15.92
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B12 LARGE DISPLACEMENT ANALYSIS OF A SPHERICAL SHELL
NODE 1 Z-DIR DISPL.
VALUE OF TIMEFUNCTION 1
EG 1 E 1 P 7 STRESS-RR
VALUE OF TIMEFUNCTION 1
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


SOLVIA
STRICKLIN (THIRTY SHELL ELEMENTS)
MESCALL (FINITE DIFFERENCE SOLUTION)
LINEAR SOLUTION
APEX POINT LOAD P (lb)
DEFLECTION RATIO W_0/H

SOLVIA-PRE input

```
HEADING  'B12  LARGE DISPLACEMENT ANALYSIS OF A SPHERICAL SHELL'
*
DATABASE  CREATE
MASTER  IDOF=100111 NSTEP=20
KINEMATICS  DISPLACEMENT=LARGE
TOLERANCES  TYPE=F RTOL=0.001 RNORM=15.92
ITERATION  FULL-NEWTON
*
TIMEFUNCTION  1
 0. 0.  /  20. 15.92
*
SYSTEM  1 CYLINDRICAL
COORDINATES
 ENTRIES  NODE  R       THETA
            1  4.76788  90.
            2  4.75212  90.
            3  4.75212  79.10  TO
            5  4.76788  79.10
*
SKEWSYSTEM  EULERANGLES
 1  -10.9
NSKEWS  /  3 1  TO  5 1
*
MATERIAL  1  ELASTIC E=1.E7 NU=0.3
*
EGROUP  1  PLANE AXISYMMETRIC
GSURFACE  5 1 2 3  EL1=10 EL2=1 NODES=8 SYSTEM=1
*
FIXBOUNDARIES  2 INPUT=LINES  /  1 2  /  3 5
FIXBOUNDARIES  3 INPUT=NODES  /  4
LOADS  CONCENTRATED
 1 3 -1.
*
SET  NSYMBOLS=MYNODES HEIGHT=0.25 SMOOTHNESS=YES
MESH  NNUMBERS=MYNODES VECTOR=LOAD SUBFRAME=12
MESH  NAXES=SKEW ENUMBERS=YES BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B12  LARGE DISPLACEMENT ANALYSIS OF A SPHERICAL SHELL
*
DATABASE  CREATE STRESSREFERENCE=ELEMENT
*
WRITE  FILENAME='b12.lis'
*
MESH  ORIGINAL=DASHED SMOOTHNEES=YES VECTOR=LOAD
*
SET  PLOTORIENTATION=PORTRAIT
SUBFRAME  12
NHISTORY  NODE=1 DIRECTION=3 XVARIABLE=1 SYMBOL=1 OUTPUT=ALL
EHISTORY  ELEMENT=1 POINT=7 KIND=SRR XVARIABLE=1 OUTPUT=ALL
END
```

EXAMPLE B13

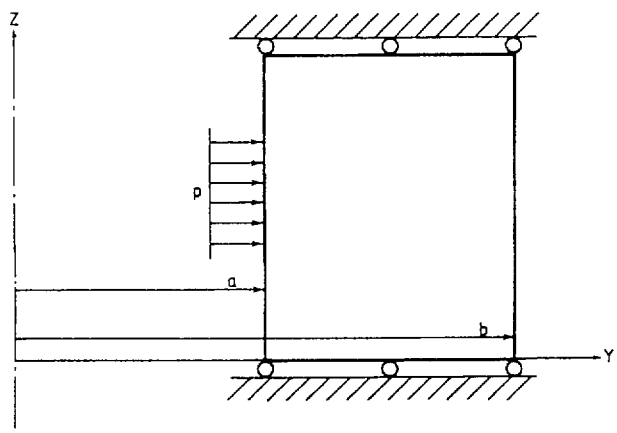
ELASTIC-PLASTIC ANALYSIS OF A THICK-WALLED CYLINDER

Objective

To verify the PLANE AXISYMMETRIC element for material nonlinear analysis using the thermo-elastic-plastic material model subjected to loading, unloading and reloading conditions.

Physical Problem

A section of a very long thick-walled cylinder under internal pressure loading at constant temperature, see figure below, is considered. The material is elastic - perfectly plastic.



Geometry: a = 0.01 m, b = 0.02 m

Elastic - perfectly plastic material
Plane strain conditions
von Mises yield condition

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

$\nu = 0.3$

$E_T = 0.0$ N/m^2

$\alpha = 0.0$ 1/°C

$\sigma_y = 400.0 \cdot 10^6$ N/m^2 at 20°C

$\sigma_y = 352.4 \cdot 10^6$ N/m^2 at 50°C

$\sigma_y = 320.6 \cdot 10^6$ N/m^2 at 150°C

$\sigma_y = 293.8 \cdot 10^6$ N/m^2 at 250°C

Finite Element Model

The model is shown in the figure on page B13.2. Ten 8-node PLANE AXISYMMETRIC elements are employed for the unit length of cylinder considered. The axial strain is assumed to be zero.

Solution Results

The input data is shown on pages B13.4 and B13.5. The internal pressure is increased under constant temperature of 20°C to a maximum value of $311.78 \cdot 10^6$ N/m^2 with an unloading-reloading sequence at $277.14 \cdot 10^6$ N/m^2. The stiffness is reformed at every load step. The BFGS method of equilibrium iteration (default) is employed with tolerance ETOL = 10^{-6}.

The radial displacement solution for the nodes at the outer radius and the stress distribution through the wall at the pressure $288.22 \cdot 10^6$ N/m^2 are shown in the figures on page B13.3. Excellent agreement with the solution by Hodge and White [1] is achieved.

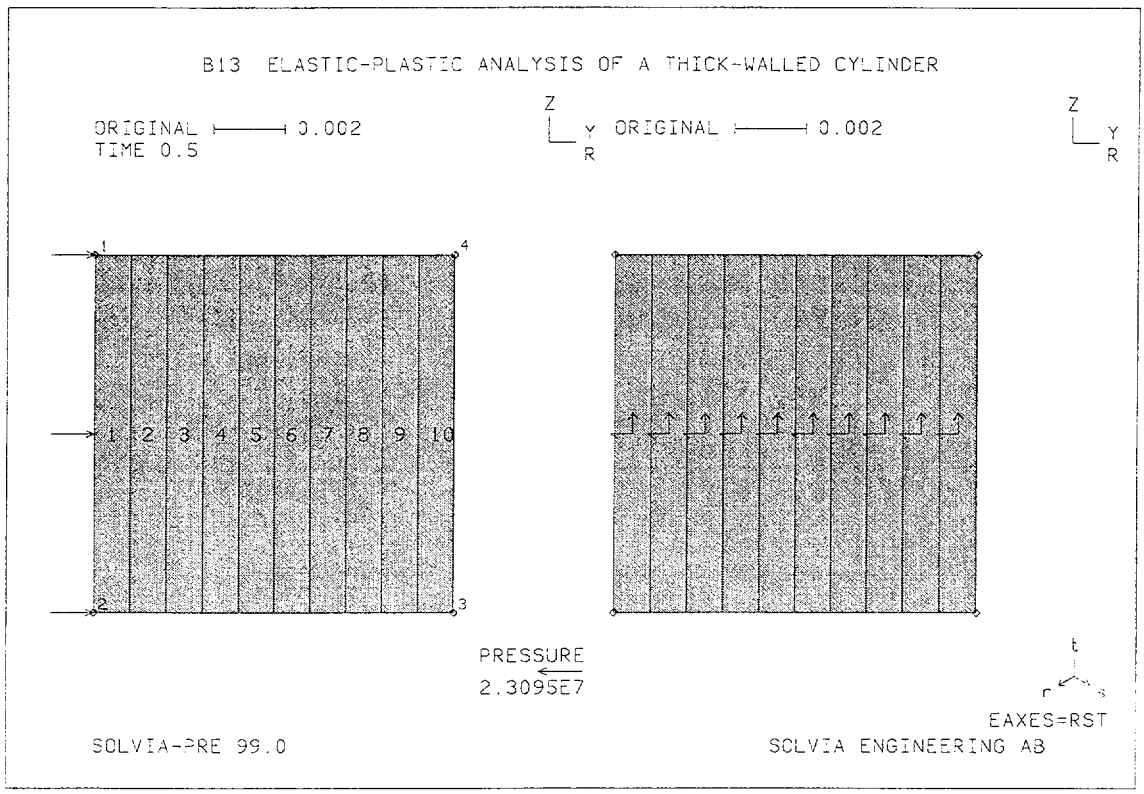
User Hints

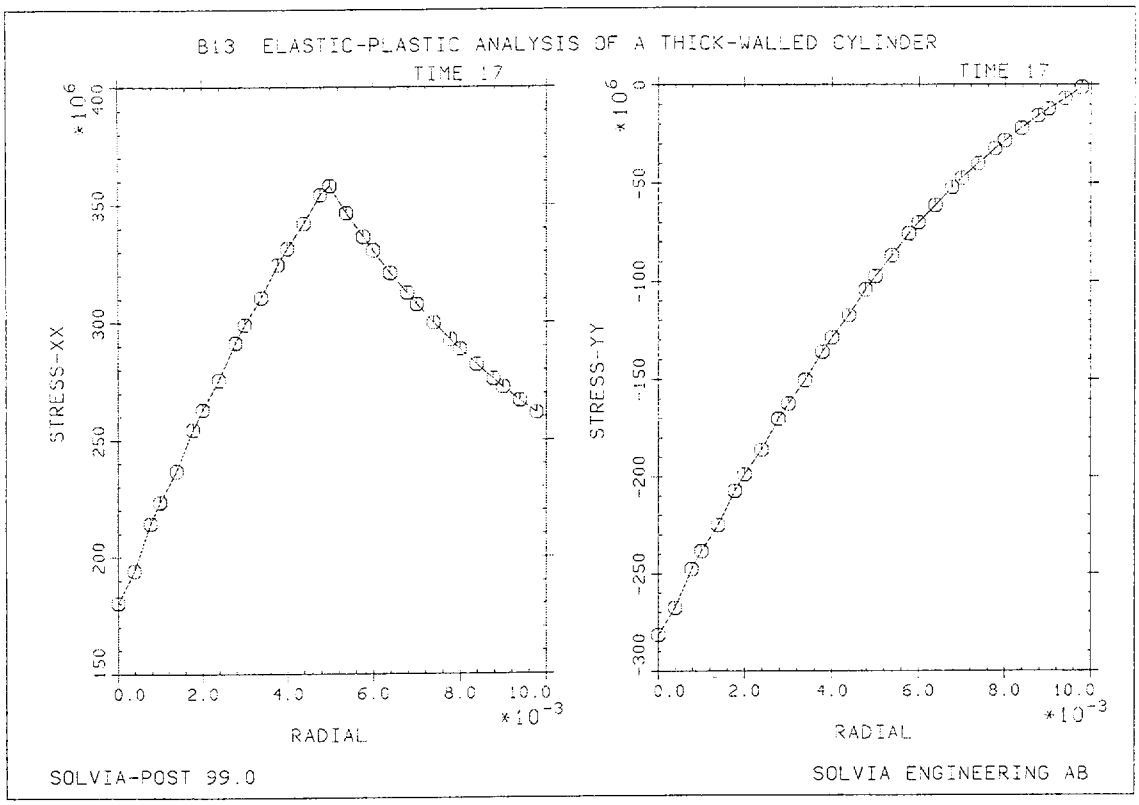
- A drastic change in stiffness for the plastic elements occurs at the instant of unloading and solution difficulties can develop if the modified Newton method of iteration is used. However, the BFGS method of iteration, which is the default iteration method, can handle unloading situations successfully and is, therefore, employed.

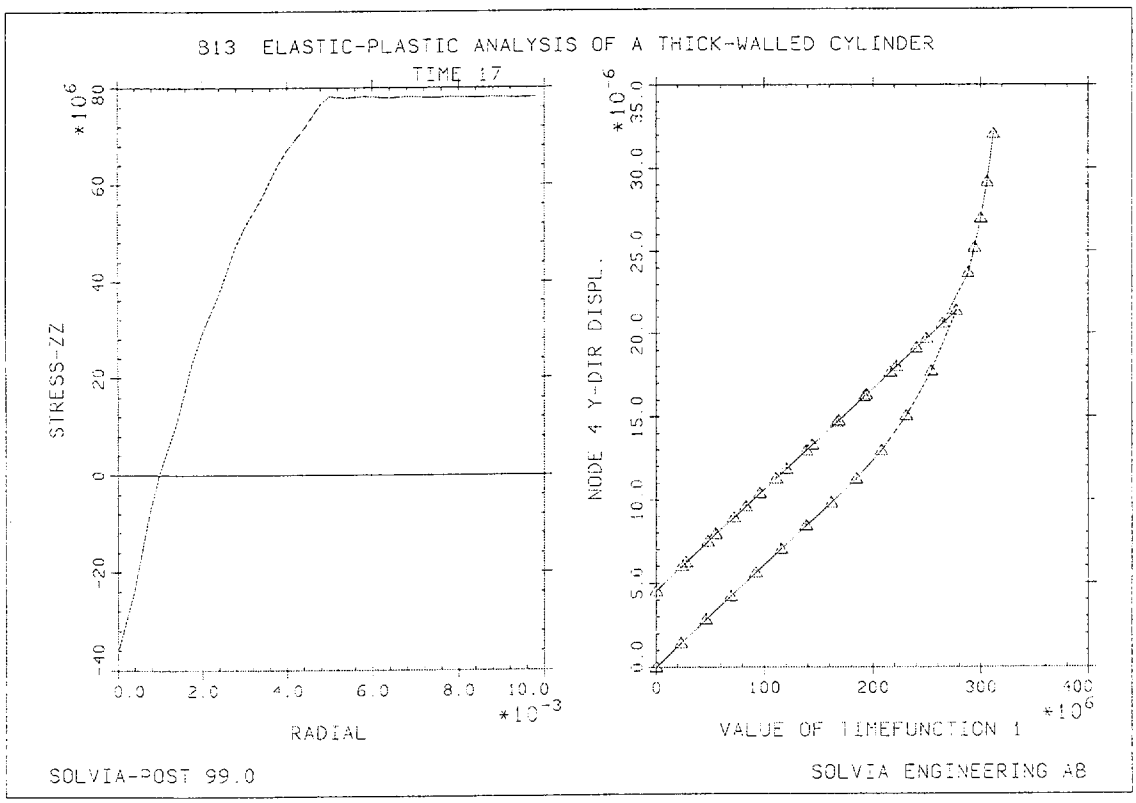
- The thermo-elastic-plastic material model can be employed for elasto-plastic analysis (α=0, $\Delta\theta$=0) as shown here, but the bilinear elastic-plastic material would be more efficient.

Reference

[1] Hodge, P.E. and White, S.H., "A Quantitative Comparison of Flow and Deformation Theories of Plasticity", Journal of Applied Mechanics, Vol. 17, pp. 180-184, 1950.




B13 ELASTIC-PLASTIC ANALYSIS OF A THICK-WALLED CYLINDER
TIME 17
STRESS-XX
*10^6
RADIAL
*10^-3
TIME 17
STRESS-YY
*10^6
RADIAL
*10^-3
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B13 ELASTIC-PLASTIC ANALYSIS OF A THICK-WALLED CYLINDER
TIME 17
STRESS-ZZ
*10^6
RADIAL
*10^-3
NODE 4 Y-DIR DISPL.
*10^-6
VALUE OF TIMEFUNCTION 1
*10^6
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B13  ELASTIC-PLASTIC ANALYSIS OF A THICK-WALLED CYLINDER'
*
DATABASE  CREATE
MASTER  IDOF=100111 NSTEP=38 DT=0.5
TOLERANCES  ETOL=1.E-6
*
TIMEFUNCTION  1
  0.   0.     /  6. 277.14E6 / 11.  0.
 17. 288.22E6 / 19. 311.78E6
*
COORDINATES
 ENTRIES  NODE   Y      Z
           1     0.01  0.01
           2     0.01  0.
           3     0.02  0.
           4     0.02  0.01
*
INITIAL  TEMPERATURE  TREF=20.
*
MATERIAL  1  THERMO-PLASTIC  TREF=20
  20  2.0E11  0.3  400.0E6  0.  0.
  50  2.0E11  0.3  352.4E6  0.  0.
 150  2.0E11  0.3  320.6E6  0.  0.
 250  2.0E11  0.3  293.8E6  0.  0.
*
EGROUP  1  PLANE AXISYMMETRIC
GSURFACE  1 2 3 4  EL1=1 EL2=10 NODES=8
LOADS  ELEMENT  INPUT=LINE
 1 2  1.
*
FIXBOUNDARIES  3 INPUT=LINE  /  2 3  /  1 4
LOADS  TEMPERATURE  TREF=20.
*
SET  NSYMBOLS=MYNODES MARGIN=2.0
MESH  NNUMBERS=MYNODES ENUMBERS=YES VECTOR=LOAD SUBFRAME=21
MESH  EAXES=RST GSCALE=OLD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B13  ELASTIC-PLASTIC ANALYSIS OF A THICK-WALLED CYLINDER
*
DATABASE  CREATE
TOLERANCES
*
WRITE  FILENAME='b13.lis'
*
EPLINE  NAME=RADIAL
 1 6 5 4  TO  10 6 5 4
*
SUBFRAME  21
ELINE  LINENAME=RADIAL KIND=SXX TIME=17 OUTPUT=ALL SYMBOL=1
ELINE  LINENAME=RADIAL KIND=SYY TIME=17 OUTPUT=ALL SYMBOL=1
*
SUBFRAME  21
ELINE  LINENAME=RADIAL KIND=SZZ TIME=17 OUTPUT=ALL
NHISTORY  NODE=4 DIRECTION=2 XVARIABLE=1 SYMBOL=2 OUTPUT=ALL
END
```

EXAMPLE B14

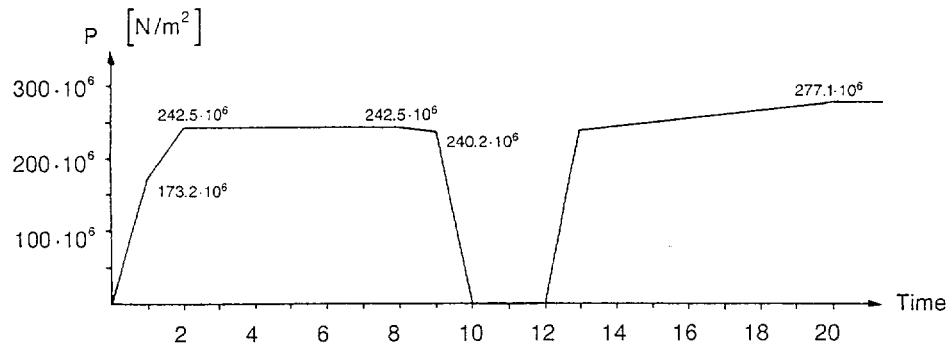
THERMO-PLASTIC ANALYSIS OF A THICK-WALLED CYLINDER

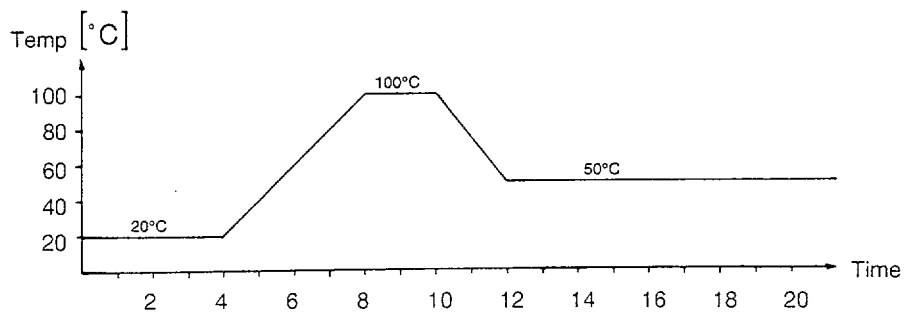
Objective

To demonstrate the thermo-elastic-plastic analysis capability of the PLANE AXISYMMETRIC element.

Physical Problem

The physical problem is the same as in Example B13 shown on page B13-1, but the cylinder is now subjected to the pressure and uniform temperature load variation shown in figure below.





Finite Element Model

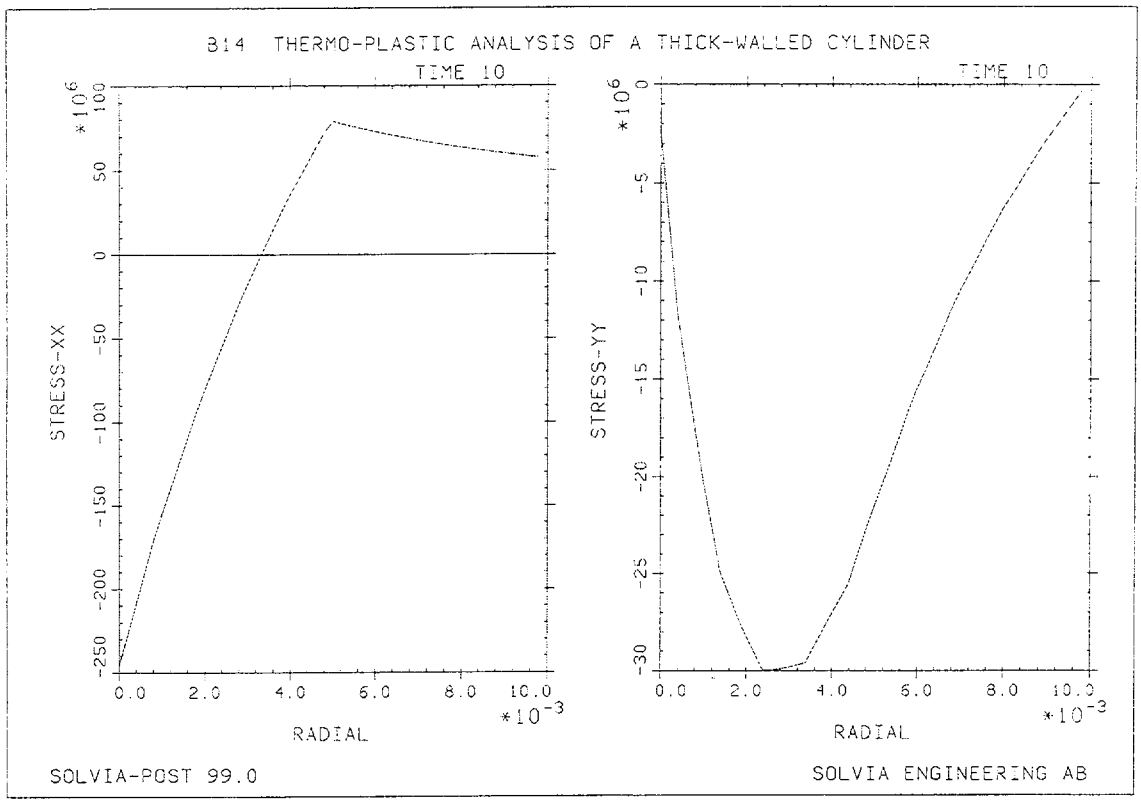
The model is the same as shown on page B13-2.

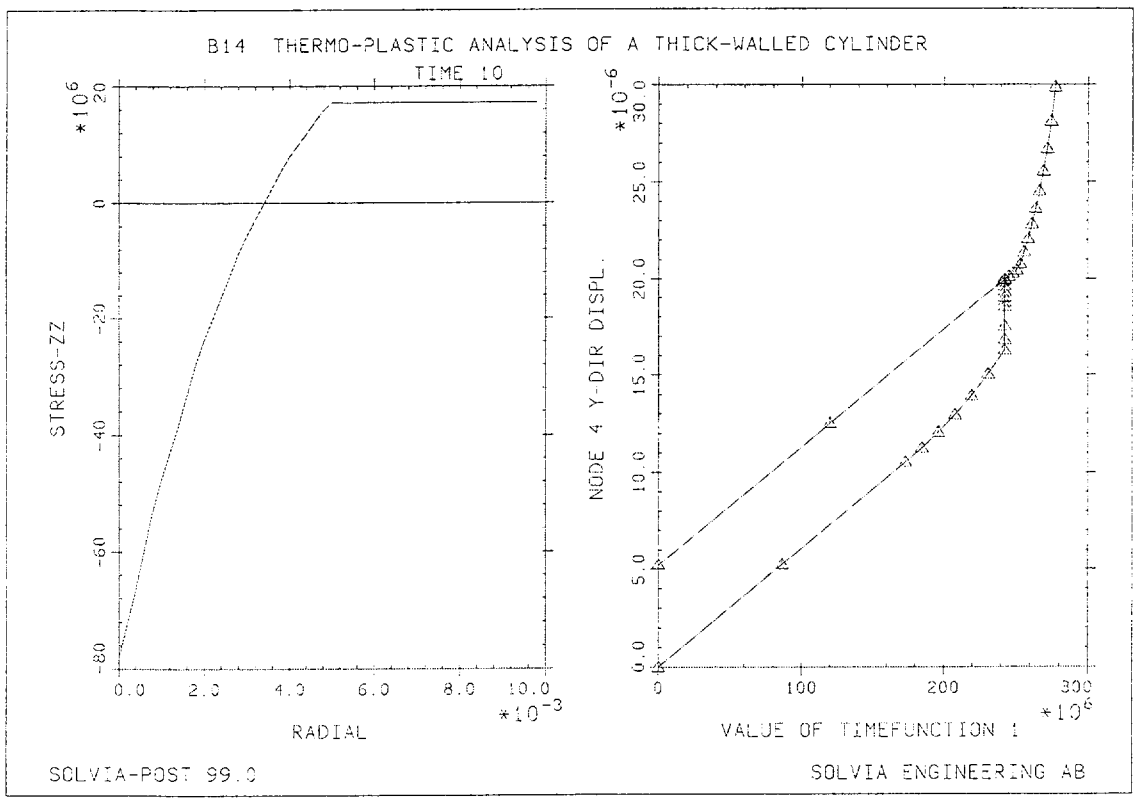
Solution Results

The input data is shown on pages B14.3 and B14.4. The same solution procedure as for the previous Example B13 is used. The radial displacement response for the nodes on the outer radius and the residual stresses at time=10 are shown on page B14.2.

User Hints

- The nodal temperatures can be specified directly by the user (as in this example) or they can be read from a file, which may have been generated in a temperature analysis using SOLVIA-TEMP. Example B38 shows how a combined thermal analysis (using SOLVIA-TEMP) and a thermal stress analysis (using SOLVIA) may be carried out.


B14 THERMO-PLASTIC ANALYSIS OF A THICK-WALLED CYLINDER
TIME 10
STRESS-XX
*10^6
100
50
0
-50
-100
-150
-200
-250
0.0
2.0
4.0
6.0
8.0
10.0
*10^-3
RADIAL
TIME 10
STRESS-YY
*10^6
0
-5
-10
-15
-20
-25
-30
0.0
2.0
4.0
6.0
8.0
10.0
*10^-3
RADIAL
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B14 THERMO-PLASTIC ANALYSIS OF A THICK-WALLED CYLINDER
TIME 10
STRESS-ZZ
*10^6
20
0
-20
-40
-60
-80
0.0
2.0
4.0
6.0
8.0
10.0
*10^-3
RADIAL
NODE 4 Y-DIR DISPL.
*10^-6
30.0
25.0
20.0
15.0
10.0
5.0
0.0
0
100
200
300
*10^6
VALUE OF TIMEFUNCTION 1
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B14  THERMO-PLASTIC ANALYSIS OF A THICK-WALLED CYLINDER'
*
DATABASE  CREATE
MASTER  IDOF=100111 NSTEP=40 DT=0.5
ITERATION  METHOD=BFGS
TOLERANCES  ETOL=1.E-3
*
TIMEFUNCTION  1
  0.  0.      /   1. 173.2E6  /   4. 242.5E6  /   8. 242.5E6
  9. 240.2E6  /  10. 0.       /  12.  0.      /  13. 240.2E6
 20. 277.1E6  /  25. 277.1E6
*
TIMEFUNCTION  2
  0.  20.  /   4.  20.  /   8. 100.
 10. 100.  /  12.  50.  /  25.  50.
*
COORDINATES  /  ENTRIES  NODE  Y  Z
 1  0.01 0.01  /  2  0.01  /  3  0.02  /  4  0.02 0.01
*
MATERIAL  1  THERMO-PLASTIC  TREF=20
 20  2.0E11  0.3  400.0E6  0.  0.
 50  2.0E11  0.3  352.4E6  0.  0.
150  2.0E11  0.3  320.6E6  0.  0.
200  2.0E11  0.3  293.8E6  0.  0.
*
EGROUP  1  PLANE AXISYMMETRIC
GSURFACE  1 2 3 4  EL1=1 EL2=10 NODES=8
LOADS  ELEMENT  INPUT=LINE
 1 2  1. 1. 1
*
FIXBOUNDARIES  3 INPUT=LINE  /  2 3  /  1 4
 2 3 / 1 4
INITIAL  TEMPERATURES  TREF=0. INPUT=SURFACE
 1 2 3 4  20.
LOADS  TEMPERATURE  INPUT=SURFACE
 1 2 3 4  1. 1. 1. 1.  2
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B14  THERMO-PLASTIC ANALYSIS OF A THICK-WALLED CYLINDER
*
DATABASE  CREATE
*
WRITE  FILENAME='b14.lis'
*
EPLINE  NAME=RADIAL
 1  6 5 4  TO  10  6 5 4
*
SUBFRAME  21
ELINE  RADIAL KIND=SXX TIME=10 OUTPUT=ALL
ELINE  RADIAL KIND=SYY TIME=10 OUTPUT=ALL
*
SUBFRAME  21
ELINE  RADIAL KIND=SZZ TIME=10 OUTPUT=ALL
NHISTORY  NODE=4 DIRECTION=2 XVARIABLE=1 SYMBOL=2 OUTPUT=ALL
*
ELIST
SUMMATION  KIND=REACTION
END
```

EXAMPLE B15

CYCLIC CREEP ANALYSIS OF A THICK-WALLED CYLINDER

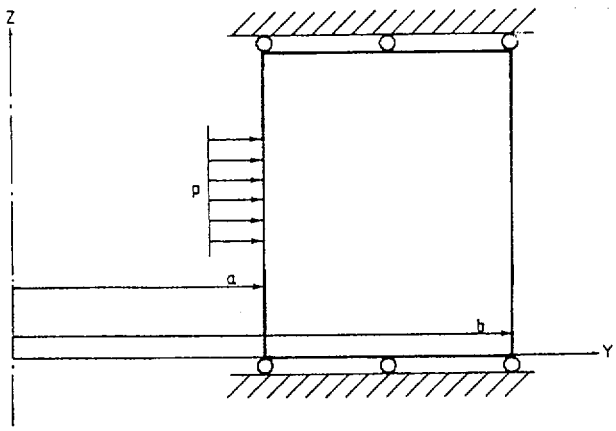
Objective

To verify the creep material model when employed with PLANE AXISYMMETRIC elements.

Physical Problem

The figure below shows the thick-walled cylinder which is to be analyzed for cyclic internal pressure. The pressure is varied as shown in the figure. Plane strain conditions are assumed. The material of the cylinder is assumed to obey a uniaxial creep law of the form

$$\varepsilon_c = 6.4 \cdot 10^{-18} \cdot \sigma^{4.4} \cdot t$$



Plane strain conditions

$E = 2.0 \cdot 10^7$ lb/in^2

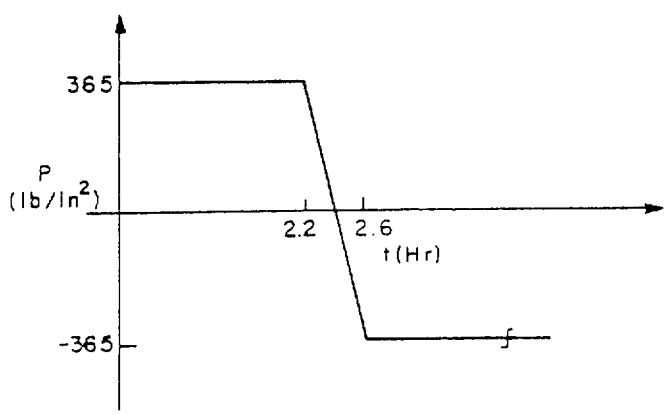
$\nu = 0.3$

Temperature 100°F

Geometry

a = 0.16 in

b = 0.25 in



Finite Element Model

The model is shown in the top figure on page B15.3. It consists of eight PLANE AXISYMMETRIC 8-node elements.

Solution Results

The input data is shown on pages B15.4 and B15.5. The BFGS method of equilibrium iteration is used with tolerance ETOL = 0.0001 and the stiffness matrix is reformed at every step. The creep rate equation is integrated using 10 subdivisions and the Euler forward method.

The left bottom figure on page B15.3 shows the effective stress at the element integration points closest to the inside and outside surfaces of the cylinder. The solution results compare well with the steady state solution reported by Bailey [1].

The right bottom figure on page B15.3 shows the von Mises stress distribution in the cylinder in the last solution step and the radial displacement history of the inner cylinder radius.

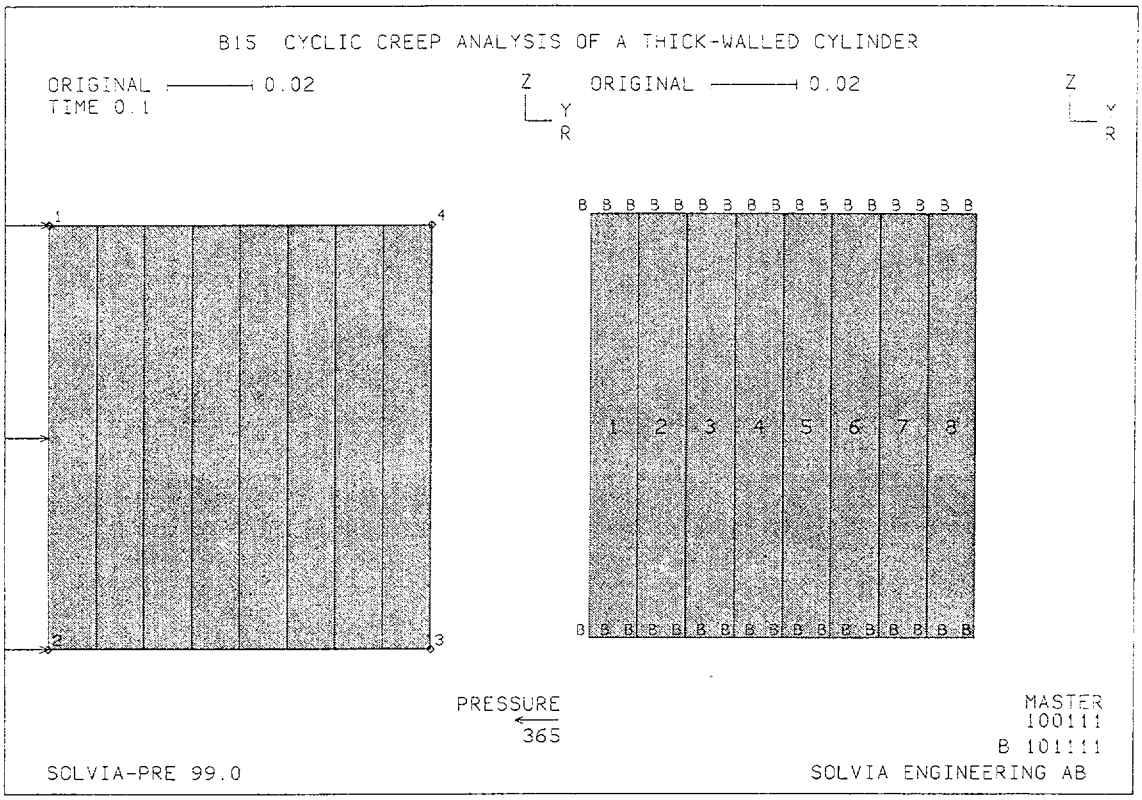
User Hints

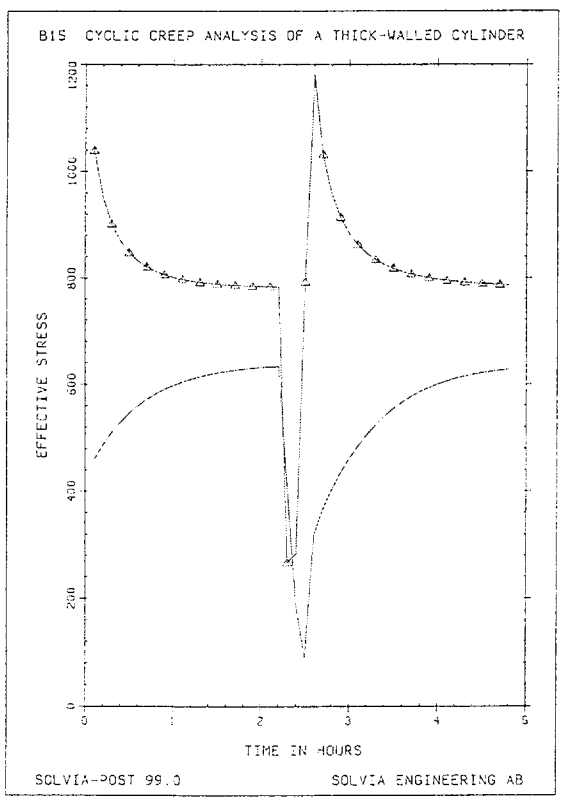
- The explicit method of integrating the creep equation requires a subdivision of the time step into sufficiently small increments for stability reasons. The implicit integration schemes can display considerably better stability characteristics, but then also iterations are used at the integration points for the creep strains.

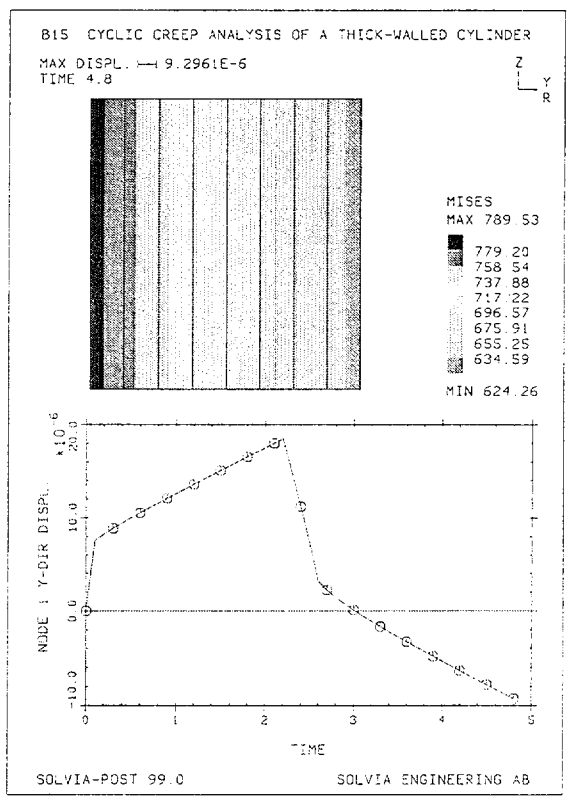
- The time step Δt is here selected using the "rule of thumb" that the maximum creep strain increment must be smaller than $1 / a_1 = 1 / 4.4$ of the current elastic strain.

Reference

[1] Finnie, I., and Heller W.R., Creep of Engineering Materials, McGraw-Hill, 1959, p. 208.


B15 CYCLIC CREEP ANALYSIS OF A THICK-WALLED CYLINDER
ORIGINAL 0.02
TIME 0.1
ORIGINAL 0.02
PRESSURE
365
MASTER
100111
B 101111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


B15 CYCLIC CREEP ANALYSIS OF A THICK-WALLED CYLINDER
EFFECTIVE STRESS
TIME IN HOURS
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B15 CYCLIC CREEP ANALYSIS OF A THICK-WALLED CYLINDER
MAX DISPL. 9.2961E-6
TIME 4.8
MISES
MAX 789.53
779.20
758.54
737.88
717.22
696.57
675.91
655.25
634.59
MIN 624.26
NODE 1 Y-DIR DISPL.
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B15  CYCLIC CREEP ANALYSIS OF A THICK-WALLED CYLINDER'
*
DATABASE  CREATE
MASTER  IDOF=100111 NSTEP=48 DT=0.1
TOLERANCES  ETOL=1.E-4
ITERATION  METHOD=BFGS
*
TIMEFUNCTION  1
 0.   365.  /   2.2  365.
 2.6 -365.  /  10.  -365.
TIMEFUNCTION  2
 0. 1.  /  100. 1.
*
COORDINATES  /  ENTRIES  NODE  Y  Z
 1  .16 .1  /  2  .16  /  3  .25  /  4  .25 .1
*
INITIAL  TEMPERATURES  TREF=100.
*
MATERIAL  1  PLASTIC-CREEP  XKCRP=1 TREF=100. ALPHA=0.,
             A0=6.4E-18 A1=4.4 A2=1.
              70. 2.E7 0.3 8.E7 2.E6 0.
             100. 2.E7 0.3 8.E7 2.E6 0.
*
EGROUP  1  PLANE AXISYMMETRIC
GSURFACE  1 2 3 4  EL1=1 EL2=8 NODES=8
LOADS  ELEMENT  INPUT=LINE
 1 2  1. 1. 1
*
FIXBOUNDARIES  3 INPUT=LINE  /  2 3  /  1 4
LOADS  TEMPERATURES  INPUT=SURFACE
 1 2 3 4  100. 100. 100. 100. 2
*
MESH  NSYMBOLS=MYNODES NNUMBERS=MYNODES VECTOR=LOAD SUBFRAME=21
MESH  ENUMBERS=YES BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B.15  CYCLIC CREEP ANALYSIS OF A THICK-WALLED CYLINDER
*
DATABASE  CREATE
WRITE  FILENAME='b15.lis'
*
AXIS  ID=1 VMIN=0. VMAX=4.8 LABEL='TIME IN HOURS'
AXIS  ID=2 VMIN=0. VMAX=1200. LABEL='EFFECTIVE STRESS'
*
SET  PLOTORIENTATION=PORTRAIT
EHISTORY  ELEMENT=1 POINT=9 KIND=MISES XAXIS=1 YAXIS=2,
          SYMBOL=2 SSKIP=1
EHISTORY  ELEMENT=8 POINT=1 KIND=MISES XAXIS=-1 YAXIS=-2,
          SUBFRAME=OLD OUTPUT=ALL
*
MESH  CONTOUR=MISES SUBFRAME=12
NHISTORY  NODE=1 DIRECTION=2 OUTPUT=ALL SYMBOL=1 SSKIP=2
END
```

EXAMPLE B16

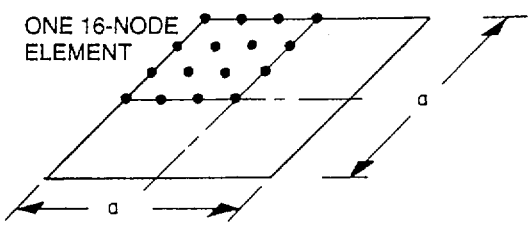
LARGE DISPLACEMENT ANALYSIS OF A PLATE

Objective

To verify the SHELL element for large displacement analysis.

Physical Problem

A simply supported square plate subjected to uniform pressure load, see figure below, is considered. The plate material is linear elastic.



E = Young modulus = 10^7 psi

ν = Poisson ratio = $\sqrt{0.1}$

h = Plate thickness = 0.12 in

a = Plate width = 24 in

q = Uniform applied pressure psi

$4 \times 4 \times 2$ Gauss integration

Finite Element Model

Because of symmetry conditions only 1/4 of the plate need be modeled, see top figure on page B16.3. One 16-node SHELL element is used. Constraint equations are used to model uniform in-plane edge displacements.

Solution Results

The input data is shown on page B16.4. The loading is applied in 22 load steps up to the final pressure $q = 3.125$ psi. For every load step the stiffness is reformed and the BFGS method of equilibrium iteration is employed with tolerance ETOL = 1.10^{-8}.

The computed center deflection ratio w/h as a function of the load parameter K is shown in the figure on page B16.2. The computed displacement response agrees very closely with the solution given by Levy [1]. The deformed shape corresponding to the final load step and center deflection as a function of the pressure are shown in the left bottom figure on page B16.3.

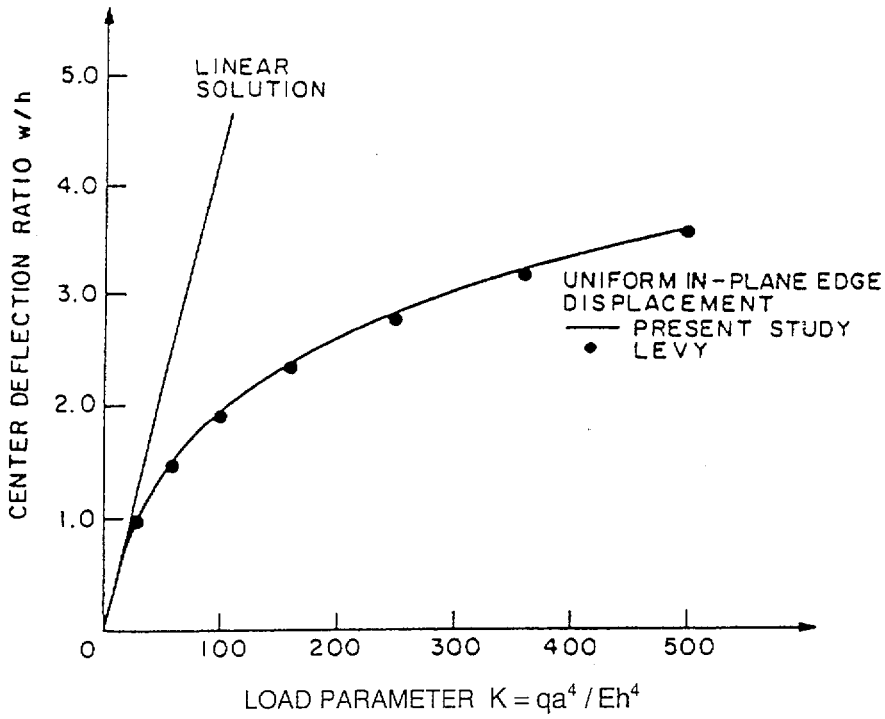
A contour plot of the vertical displacement and a vector plot of the principal stresses are shown in the right bottom figure on page B16.3.

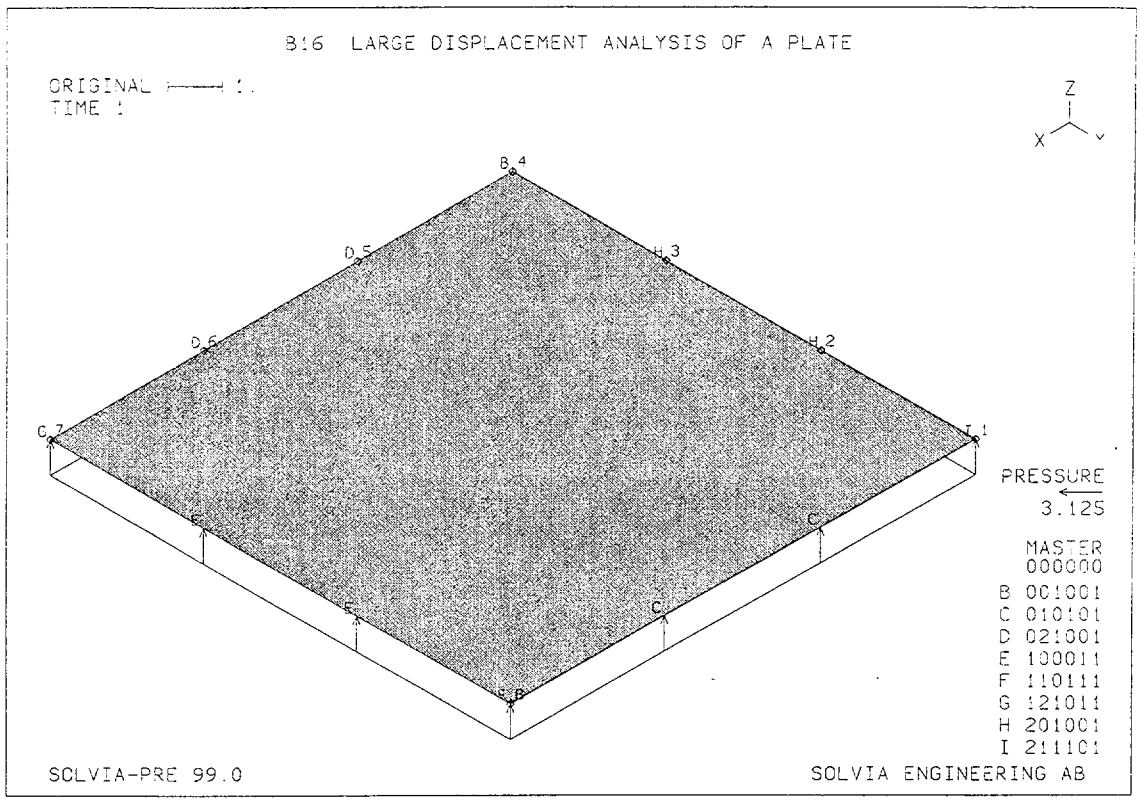
User Hints

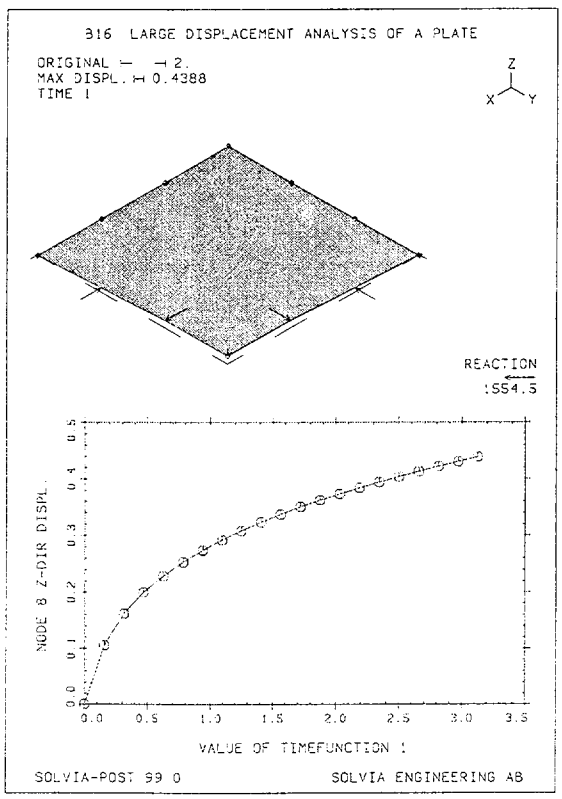
- The large displacement response of shells generally involves severe nonlinearities because the structure, initially only in bending action, is subjected to increasing membrane action with increasing load. This results into a stiffening effect. The BFGS method is generally quite effective for these types of problems. If difficulties are encountered in the convergence of iteration, then the powerful full Newton iteration with line searches can be employed.

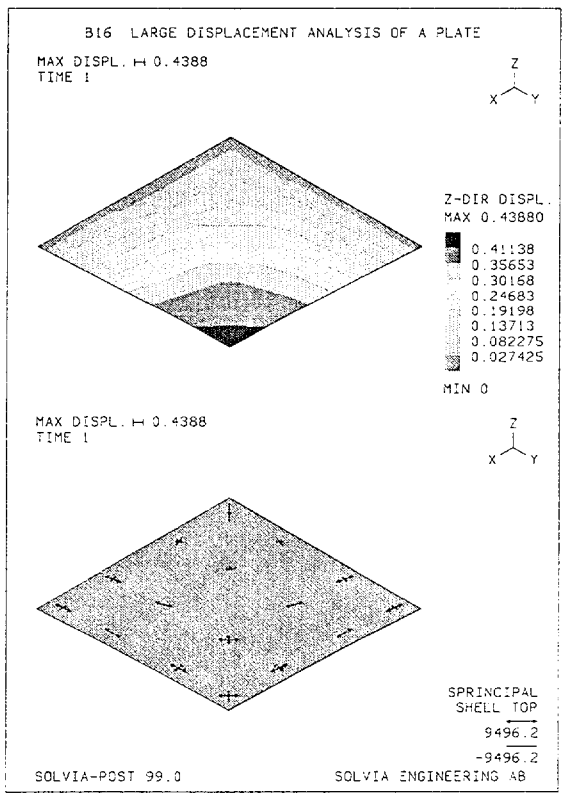
Reference

[1] Levy, S., "Bending of Rectangular Plates with Large Deflections", Technical Notes, National Advisory Committee for Aeronautics, No. 846, 1942.




B16 LARGE DISPLACEMENT ANALYSIS OF A PLATE
ORIGINAL
TIME 1
PRESSURE
3.125
MASTER
000000
B 001001
C 010101
D 021001
E 100011
F 110111
G 121011
H 201001
I 211101
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


B16 LARGE DISPLACEMENT ANALYSIS OF A PLATE
ORIGINAL
MAX DISPL. 0.4388
TIME 1
REACTION
1554.5
NODE 8 Z-DIR DISPL.
VALUE OF TIMEFUNCTION 1
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B16 LARGE DISPLACEMENT ANALYSIS OF A PLATE
MAX DISPL. 0.4388
TIME 1
Z-DIR DISPL.
MAX 0.43880
0.41138
0.35653
0.30168
0.24683
0.19198
0.13713
0.082275
0.027425
MIN 0
MAX DISPL. 0.4388
TIME 1
SPRINCIPAL
SHELL TOP
9496.2
-9496.2
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B16  LARGE DISPLACEMENT ANALYSIS OF A PLATE'
*
DATABASE  CREATE
MASTER  NSTEP=20 DT=0.05
KINEMATICS  DISPLACEMENTS=LARGE
TOLERANCES  ETOL=1.E-8
ITERATION  METHOD=BFGS
*
TIMEFUNCTION  1
  0. 0.      /  1. 3.125
*
COORDINATES
 1  0. 12.  TO  4  TO  7  12.  /  8  12. 12.
*
MATERIAL  1  ELASTIC  E=1.E7 NU=0.316228
*
EGROUP  1 SHELL
GSURFACE  1 4 7 8  EL1=1 EL2=1 NODES=16
THICKNESS  1  0.12
LOADS  ELEMENT
 1 -T 1.
*
FIXBOUNDARIES  36   INPUT=LINE  /  1 4  /  4 7
FIXBOUNDARIES  156  INPUT=LINE  /  7 8
FIXBOUNDARIES  246  INPUT=LINE  /  8 1
CONSTRAINTS
 1 1 4 1 1.  TO  3 1 4 1 1.
 5 2 4 2 1.  TO  7 2 4 2 1.
*
MESH  NSYMBOLS=MYNODES NNUMBERS=MYNODES VECTOR=LOAD,
  BCODE=ALL TIME=1.0
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B16  LARGE DISPLACEMENT ANALYSIS OF A PLATE
*
DATABASE  CREATE
WRITE  FILENAME='b16.lis'
*
SET  PLOTORIENTATION=PORTRAIT
SUBFRAME  12
MESH  ORIGINAL=DASHED NSYMBOLS=MYNODES VECTOR=REACTION
NHISTORY  NODE=8 DIRECTION=3 XVARIABLE=1 SYMBOL=1 OUTPUT=ALL
*
SUBFRAME  12
MESH  CONTOUR=DZ
MESH  VECTOR=SPRINCIPAL
*
SUMMATION  KIND=REACTION
SUMMATION  KIND=LOAD
END
```

EXAMPLE B17

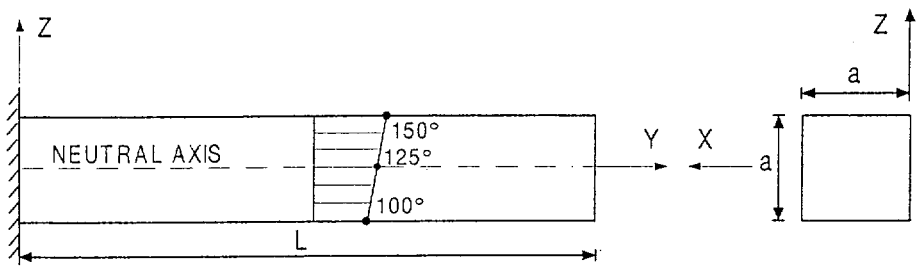
THERMO-ELASTIC ANALYSIS OF A CANTILEVER BEAM

Objective

To verify the thermo-elastic material model when used with SOLID elements.

Physical Problem

A cantilever beam is subjected to a linearly varying temperature distribution in the Z-direction, see figure below. No other loads are applied to the beam. The same physical problem is also analyzed in Example B38, but using temperatures calculated by SOLVIA-TEMP.



70°F	**200°F**	L = 6 in.
$E = 3.0 \cdot 10^7$ lb/in^2	$E = 3.0 \cdot 10^7$ lb/in^2	a = 1 in.
$\nu = 0.3$	$\nu = 0.3$	
$\alpha = 6.0 \cdot 10^{-6}$ in/in/°F	$\alpha = 6.0 \cdot 10^{-6}$ in/in/°F	

Reference temperature = 125°F

Finite Element Model

Since the problem is anti-symmetric with respect to the neutral axis of the beam, only the portion above the neutral axis is modeled, see figure on page B17.2. Three 20-node SOLID elements are used.

Solution Results

The input data is shown on pages B17.3 and B17.4. The analysis is carried out for material nonlinearities only. The analytical solution for the vertical displacement component using the theory given, for example, in [1], is

$$w = -Y^2 \cdot 1.5 \cdot 10^{-4} \text{(in)}$$

The vertical displacement component at the free end of the beam, Y = 6 (in), is:

Theory	SOLVIA (node 1)
$-5.400 \cdot 10^{-3}$	$-5.400 \cdot 10^{-3}$

An excellent agreement can be observed. The deformed mesh and contour lines of temperature are shown in the figure on page B17.3. Note that the maximum displacement given in the figure is the maximum total displacement, not the maximum component displacement.

User Hints

- The thermo-elastic material model allows Young's modulus E, Poisson's ratio ν and the mean coefficient of thermal expansion α to vary as a function of temperature. The material model is considered nonlinear since the resulting stiffness matrix, in general, is dependent on the applied temperature loading. However, no equilibrium iterations are necessary except in analysis with one (or both) of the two conditions:

  1) The stiffness matrix is not reformed in every step.

  2) The model accounts for large displacement effects.

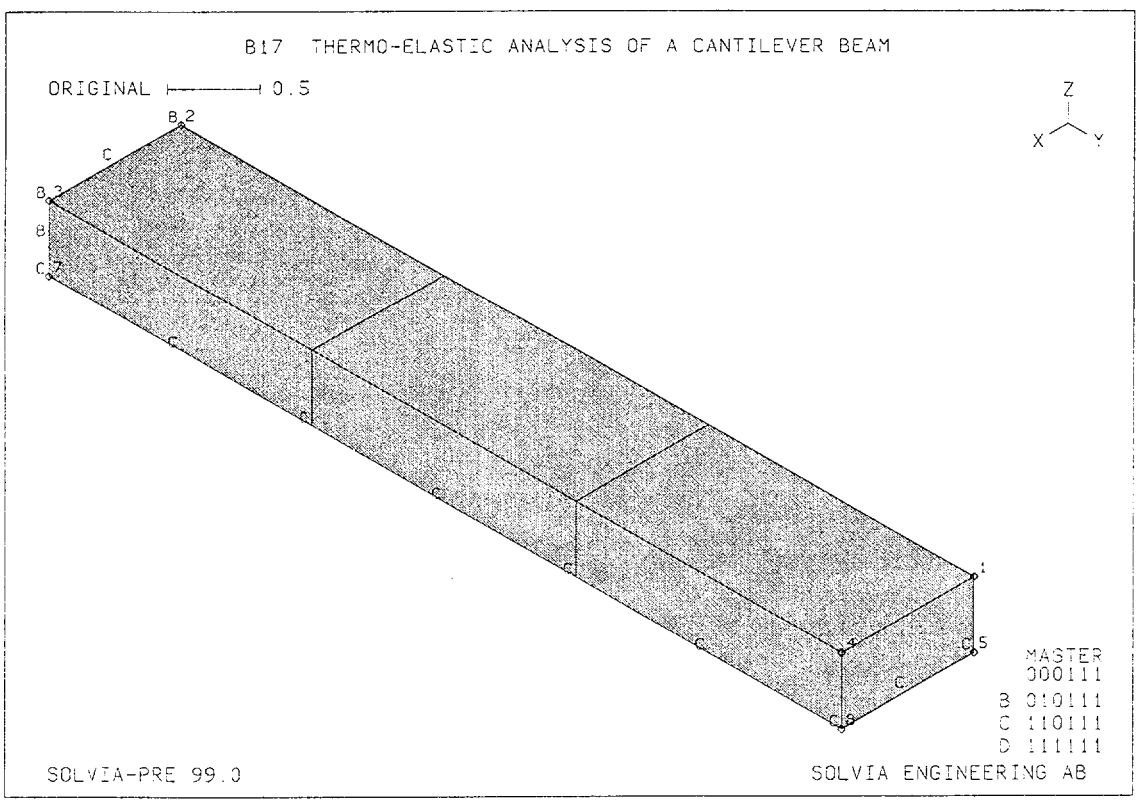
- The reference temperature TREF is the temperature for which the thermal stresses are zero. Note that TREFN serves as the reference temperature for the mean value of the coefficient of thermal expansion α.

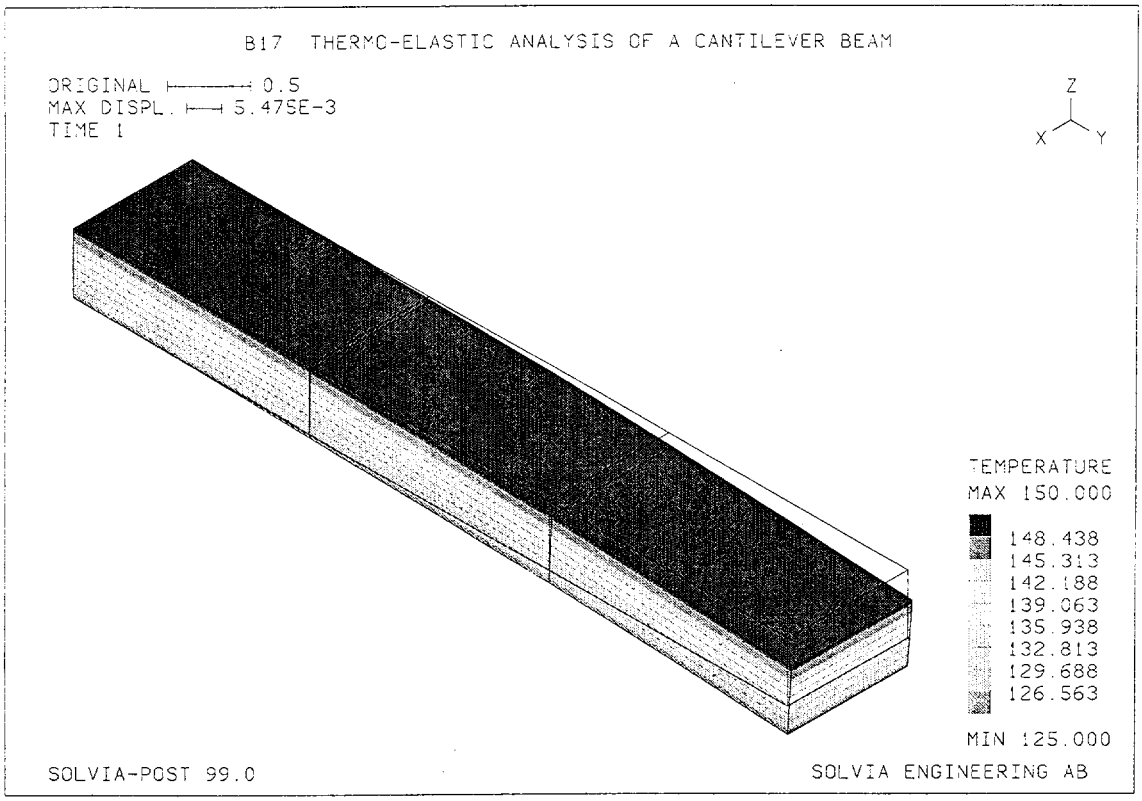
- Since the theoretical displacement solution varies quadratically in the X- and Y-directions, the 20-node SOLID element can describe the displacement field exactly. Therefore, since the beam is free to bend in both the X- and Y-directions, the stresses are zero everywhere.

- The problem can also be modeled using BEAM or ISOBEAM elements since these elements accept thermal gradient loadings in the transverse directions.

Reference

[1] Boley, B.A., and Weiner, J.H., Theory of Thermal Stresses, John Wiley and Sons, 1960, pp. 307-314.





SOLVIA-PRE input

```
HEADING  'B17  THERMO-ELASTIC ANALYSIS OF A CANTILEVER BEAM'
*
DATABASE  CREATE
MASTER  IDOF=000111
COORDINATES
 1  0. 6. 0.5  /  2  0. 0. 0.5
 3  1. 0. 0.5  /  4  1. 6. 0.5
 5  0. 6. 0.   /  6  0. 0. 0.
 7  1. 0. 0.   /  8  1. 6. 0.
*
MATERIAL  1  THERMO-ELASTIC  TREF=125.
              70  3.E7  0.3  6.E-6
             200  3.E7  0.3  6.E-6
*
EGROUP  1  SOLID
GVOLUME  1 2 3 4 5 6 7 8  EL1=3 EL2=1 EL3=1 NODES=20
*
FIXBOUNDARIES  12  INPUT=SURFACE  /  2 3 7 6
FIXBOUNDARIES  3   INPUT=LINE  /  6 7
FREEBOUNDARIES 13  INPUT=LINE  /  2 6  /  3 7
FIXBOUNDARIES  12  INPUT=SURFACE  /  5 6 7 8
LOADS  TEMPERATURE  TREFN=125 INPUT=ZONE INTERPOLATION=Z,
 COORD1=0. V1=0. COORD2=0.5 V2=25 ZONE1=WHOLE
*
MESH  NSYMBOLS=MYNODES NNUMBERS=MYNODES BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B17  THERMO-ELASTIC ANALYSIS OF A CANTILEVER BEAM
*
DATABASE  CREATE
*
WRITE  FILENAME='b17.lis'
*
MESH  CONTOUR=TEMPERATURE ORIGINAL=YES
*
NLIST  ZONENAME=EL1
END
```

EXAMPLE B18

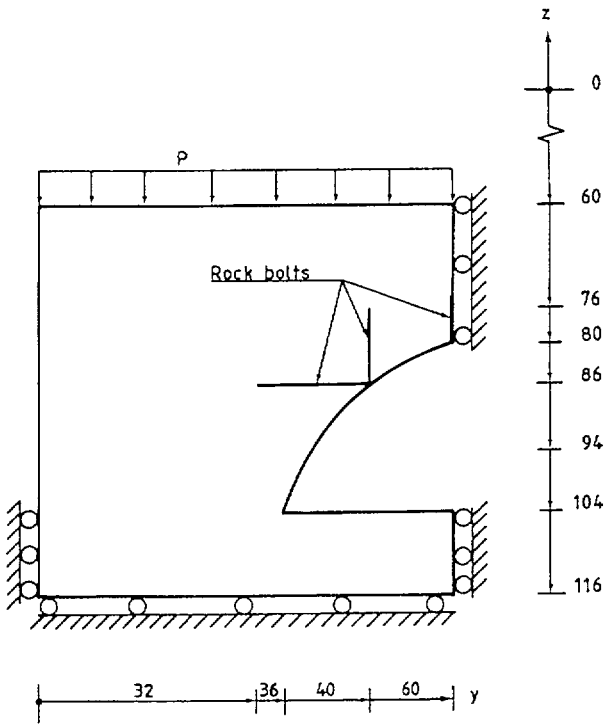
ANALYSIS OF AN UNDERGROUND OPENING

Objective

To demonstrate the use of the curve description material model in a simplified analysis of an underground opening.

Physical Problem

A tunnel through a rock mass, see figure below, is considered. The walls of the tunnel are reinforced with rock bolts. The rock material is assumed to have constant Young's modulus and Poisson's ratio.



Rock material (E and ν constant)

$E = 43200$ kip/ft^2

$\nu = 0.$

Curve description material model

Material weight density for tensile failure option 0.12 kip/ft^3

Normal stiffness reduction factor 1.10^{-3}

Shear stiffness reduction factor $9.9 \cdot 10^{-4}$

Bulk modulus 14400 kip/ft^2

Loading shear modulus 21600 kip/ft^2

Rock bolts

$E = 4.32 \cdot 10^6$ kip/ft^2

Yield = 14400 kip/ft^2

ET = 0.

Area = 0.0021 ft^2

Initial strain 1.10^{-3}

Finite Element Model

The model is shown in the left top figure on page B18.3. The rock is modeled using 8-node PLANE STRAIN elements and the rock bolts by 2-node TRUSS elements. The pressure due to the overburden rock material is simulated by element pressure loads. The curve description model is employed to describe the rock material. The tensile failure option is used with material density $\gamma = 0.12$ kip/ft^3, and shear reduction factor = 0.00099. Both the tensile stress and the shear stress are, therefore, reduced to zero for a crack, i.e. when any principal tensile stress exceeds the in-situ gravity stress at an integration point.

Solution Results

The input data is shown on pages B18.4 and B18.5. A materially-nonlinear-only analysis is carried out in eleven steps using the full Newton iteration with line search. The final overburden pressure reached in the analysis is 30 kip/ft^2 and the corresponding distribution of Tresca effective stress and of the hydrostatic pressure (negative mean stress) are shown in the bottom figure on page B18.3.

The displacement histories at two characteristic locations are shown in the right top figure on page B18.3.

User Hints

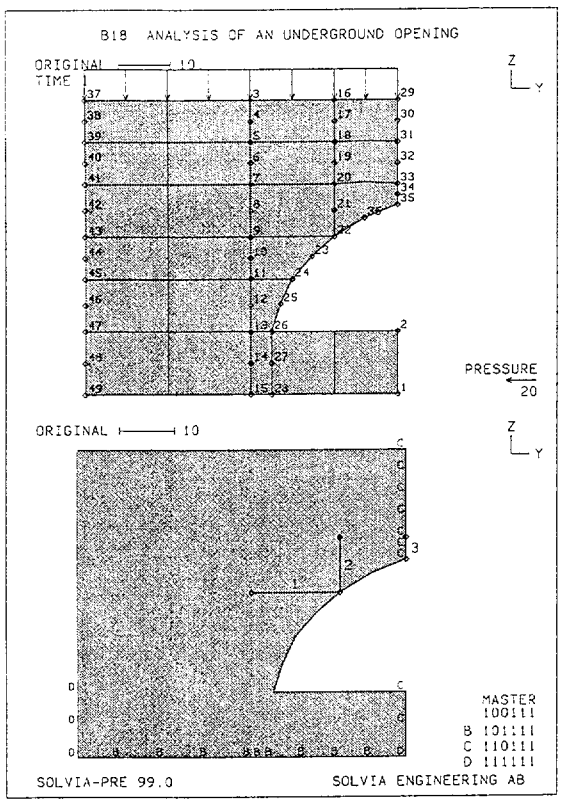
- When cracks form in a finite element model using the curve description material model (or the concrete model) the stiffness generally changes drastically. Depending on the in-situ gravity pressure the change in stresses associated with the formation of a crack may also be significant. These effects generally result into a considerable redistribution of stresses due to cracking.

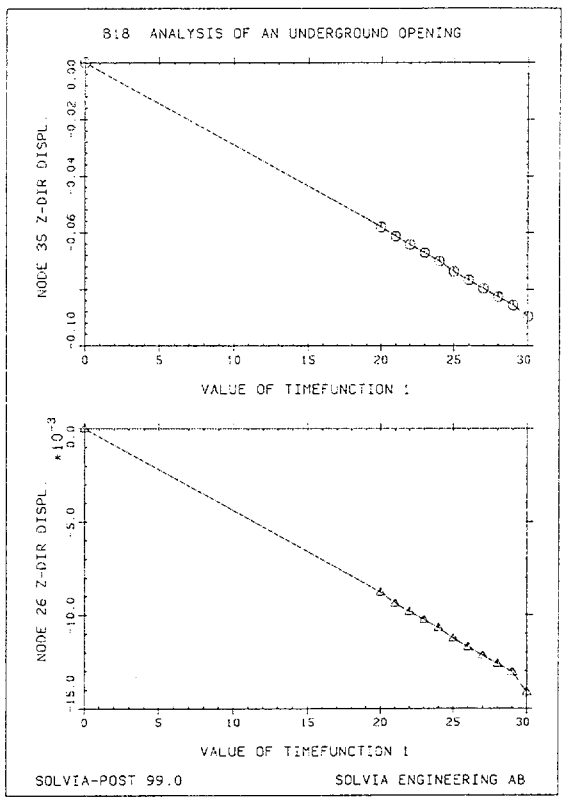
  It is, therefore, important to take small enough load steps whenever cracks are forming in order to avoid solution difficulties.

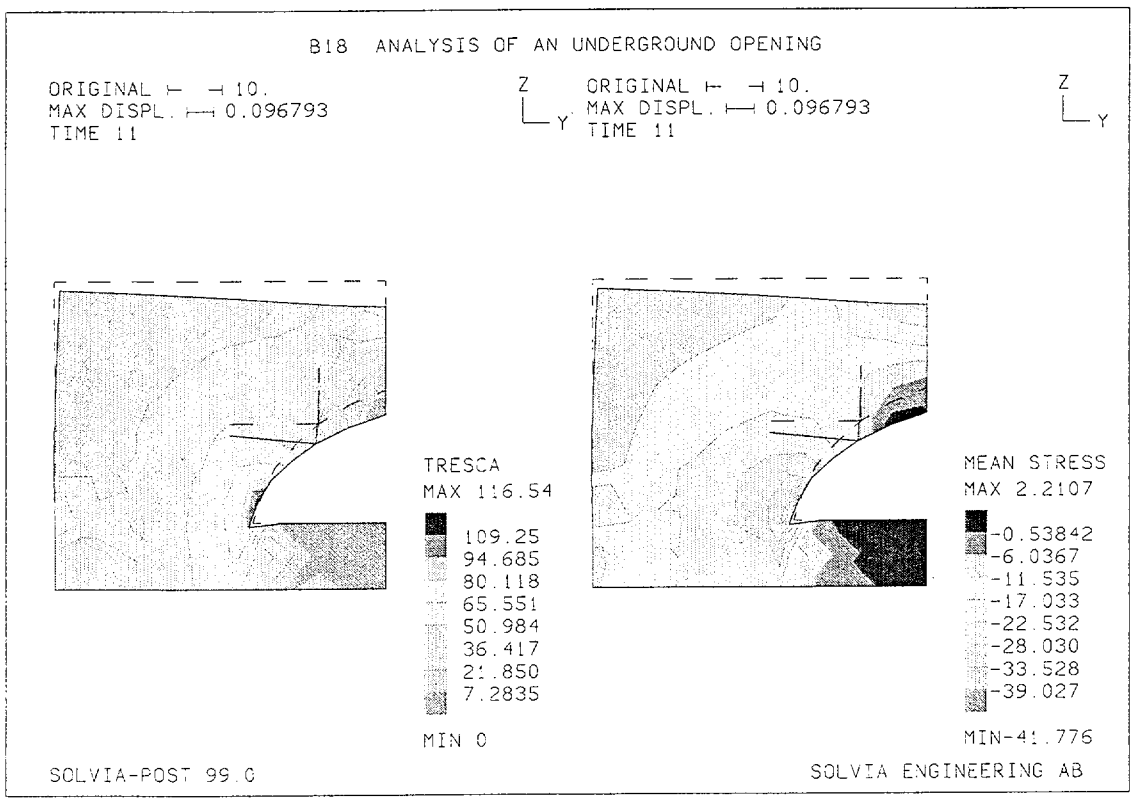
- Note that the in-situ gravity pressure p_i at the nodes is calculated using

  $$p_i = \gamma \cdot Z_i$$

  where Z_i is the nodal Z-coordinate. The stress corresponding to this gravity pressure is not included in the calculated stresses which are output from SOLVIA. It is, however, taken into account when the bulk and shear moduli are determined and in the criteria for cracking.


B18 ANALYSIS OF AN UNDERGROUND OPENING
ORIGINAL 10
TIME 1
PRESSURE
20
ORIGINAL 10
MASTER
100111
B 101111
C 110111
D 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


B18 ANALYSIS OF AN UNDERGROUND OPENING
NODE 35 Z-DIR DISPL.
VALUE OF TIMEFUNCTION 1
NODE 26 Z-DIR DISPL.
VALUE OF TIMEFUNCTION 1
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B18 ANALYSIS OF AN UNDERGROUND OPENING
ORIGINAL 10.
MAX DISPL. 0.096793
TIME 11
ORIGINAL 10.
MAX DISPL. 0.096793
TIME 11
TRESCA
MAX 116.54
109.25
94.685
80.118
65.551
50.984
36.417
21.850
7.2835
MIN 0
MEAN STRESS
MAX 2.2107
-0.53842
-6.0367
-11.535
-17.033
-22.532
-28.030
-33.528
-39.027
MIN-41.776
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B18  ANALYSIS OF AN UNDERGROUND OPENING'
*
DATABASE  CREATE
MASTER  IDOF=100111 NSTEP=11
ITERATION  FULL-NEWTON LINE-SEARCH=YES
TIMEFUNCTION  1
 0. 0. / 1. 20. / 11. 30.
*
COORDINATES  / ENTRIES  NODE  Y  Z
 1  60. -116. / 2  60. -104. / 3  32. -60. TO 7  32. -76. TO
 9  32. -86. TO 11  32. -94. TO 13  32. -104. TO 15  32. -116.
16  48. -60. TO 20  48. -76. TO 22  48. -86. / 23  43.75 -89.75
24  40. -94. / 25  37.75 -98.75 / 26  36. -104. TO 28  36. -116.
29  60. -60. TO 33  60. -76. TO 35  60. -80. / 36  53.75 -82.5
NGENERATION  NSTEP=34 YSTEP=-32.  /  3 TO 15
*
LINE  NODES  3  15  /   4 TO 14
LINE  NODES  16 28  /  17 TO 27
LINE  NODES  16 22  /  17 TO 21
LINE  NODES  29 35  /  30 TO 34
LINE  NODES  37 49  /  38 TO 48
LINE  NODES  35 22  /  36
*
MATERIAL  1  PLASTIC  E=4.32E6 YIELD=1.44E4 ET=0.
MATERIAL  2  CURVE-DESCRIPTION  ICRACK=2 GAMMA=0.12 STIFAC=1.E-3,
             SHEFAC=9.9E-4
  .0   1.44E4  1.44E4  2.16E4  /  .01  1.44E4  1.44E4  2.16E4
  .05  1.44E4  1.44E4  2.16E4  /  .1   1.44E4  1.44E4  2.16E4
  .5   1.44E4  1.44E4  2.16E4  / 1.    1.44E4  1.44E4  2.16E4
*
EGROUP  2  PLANE STRAIN  MATERIAL=2
GSURFACE  37 49 15 3  EL1=6 EL2=2 NODES=8
GSURFACE  3  15 28 16 EL1=6 EL2=1 NODES=8
GSURFACE  16 22 35 29 EL1=3 EL2=1 NODES=8
GSURFACE  26 28 1  2  EL1=1 EL2=2 NODES=8
LOADS  ELEMENT INPUT=LINES
37 29 1.0
*
EGROUP  1  TRUSS  MATERIAL=1
ENODES  /  1  22 9  /  2  22 20  /  3  35 33
EDATA  /  1 2.1E-3 1.E-3  TO  3 2.1E-3 1.E-3
*
FIXBOUNDARIES  23 INPUT=NODES  /  47  TO  49
FIXBOUNDARIES  3  INPUT=LINES  /  49 15  /  15 28  /  28 1
FIXBOUNDARIES  2  INPUT=LINES  /   1 2   /  35 29
*
SET  PLOTORIENTATION=PORTRAIT NSYMBOLS=MYNODES VIEW=X
MESH  NNUMBERS=MYNODES VECTOR=LOAD SUBFRAME=12
MESH  ENUMBERS=EG1 BCODE=ALL OUTLINE=YES NSYMBOL=EG1
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B18  ANALYSIS OF AN UNDERGROUND OPENING
*
DATABASE  CREATE
WRITE  FILENAME='b18.lis'
*
SET  PLOTORIENTATION=PORTRAIT
SUBFRAME  12
NHISTORY  NODE=35 DIRECTION=3 XVARIABLE=1 SYMBOL=1 OUTPUT=ALL
NHISTORY  NODE=26 DIRECTION=3 XVARIABLE=1 SYMBOL=2 OUTPUT=ALL
*
SET  VIEW=X ORIGINAL=DASHED OUTLINE=YES
SET  PLOTORIENTATION=LANDSCAPE
MESH  CONTOUR=TRESCA SUBFRAME=21
MESH  CONTOUR=SMEAN
*
END
```

EXAMPLE B19

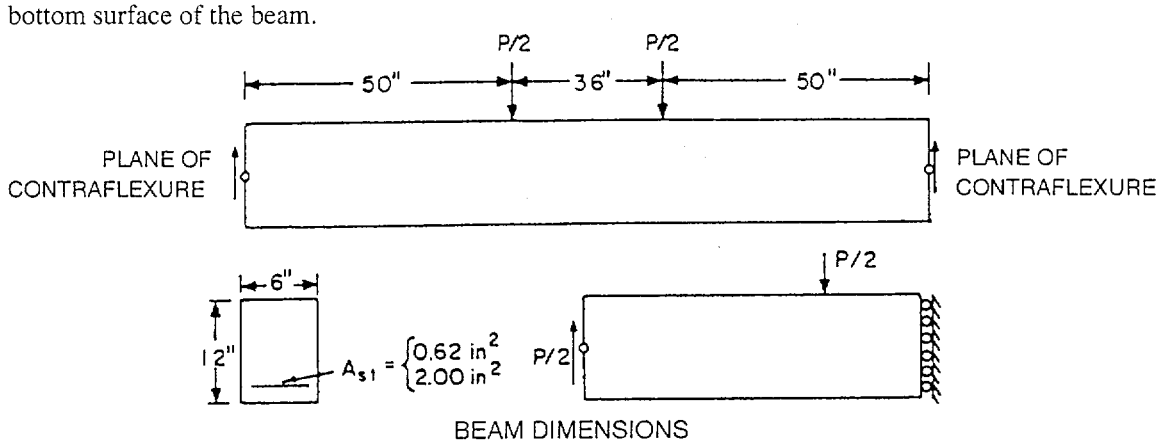
REINFORCED CONCRETE BEAM, PLANE STRESS

Objective

To demonstrate the concrete material model in plane stress analysis and the modeling of reinforced concrete.

Physical Problem

A simply supported reinforced concrete beam subjected to two symmetrically applied concentrated loads as shown in the figure below is considered. The steel reinforcement is located 2.06 in. from the bottom surface of the beam.



Material parameters

CONCRETE:		
Density	$0.2172 \cdot 10^{-3}$	$lbf \cdot s^2/in^4$
Initial tangent modulus	3060	ksi
Poisson's ratio	0.2	
Uniaxial cut-off tensile strength	0.458	ksi
Uniaxial maximum compressive stress (SIGMAC)	-3.74	ksi
Compressive strain at SIGMAC	-0.002	in/in
Uniaxial ultimate compressive stress	-3.225	ksi
Uniaxial ultimate compressive strain	-0.003	in/in
STEEL:		
Density	$0.7339 \cdot 10^{-3}$	$lbf \cdot s^2/in^4$
Young's modulus	30000	ksi
Initial yield stress	44	ksi
Strain hardening modulus	300	ksi

Finite Element Model

The finite element model used is shown in the figures on page B19.4. Using symmetry conditions only one-half of the structure need to be considered. Forty 8-node PLANE STRESS elements are used to model the concrete of the beam and ten 3-node TRUSS elements are used to model the reinforce-

ment. In addition, three vertical TRUSS elements are used at the left end support of the beam to distribute the support force. The material properties of the concrete are idealized using the concrete material model. The material of the steel bars is modeled as elastic-plastic with strain hardening. The solution is obtained by using the AUTOSTEP method and full Newton iteration with line searches.

Solution Results

The input data on pages B19.9 and B19.10 is used in the finite element analysis.

Krahl et al. [1] analyzed the development of cracks in reinforced concrete beams subjected to pure moment. Using a simplified stress-strain law for the concrete and excluding the tension stiffening effect after tensile cracking the critical (collapse) load was found to be approximately 10 kips for the beam in this example with reinforcement area $0.62\ \text{in}^2$. Suidan and Schnobrich [2] presented a finite element solution of the same beam using the simplified stress-strain law for the concrete.

The SOLVIA solution results can be seen on pages B19.5 and B19.6. The collapse load of the beam is calculated by the model to be 10.5 kips. The SOLVIA solution compares well with the results found in [1] and [2].

The top figure on page B19.5 shows the formation of cracks as contour and vector plots for the last load step of the analysis. The bottom figures on page B19.5 show the midspan deflection of the beam as a function of the load together with typical stress-strain curves for the concrete material in tension and compression and the elastic-plastic response curve from the steel reinforcement.

The figures on page B19.6 show the stress and strain variations along typical lines in the concrete and along the reinforcement bar. It can be seen that the tensile stresses along the line SECTION are small due to cracking. The distribution of axial strain along the line BAR in the reinforcement and along the line ABOVE in the concrete shows a concentration of strain in the line portion located under the load application.

An analysis was also carried out for the beam when the reinforcement area is $A_{st} = 2.0\,\text{in}^2$ and corresponding results are shown on pages B19.7 and B19.8. The analysis is stopped because of crushing of the concrete under the load application. Since the reinforcement remains elastic strain concentration due to cracking is avoided.

User Hints

- The solution of concrete problems can be difficult, due to the sudden nonlinearities that take place as a result of cracking and crushing of the material. The overall structural nonlinearities are more pronounced when only small amounts of steel reinforcements are used in the structure.

- When the structure is lightly reinforced the effect of the nonlinearities can be sudden and significant strain redistributions can occur for each load increment. Note that cracking is associated with a negative slope in the so-called tension stiffening portion of the uniaxial stress and strain relation for concrete. If enough cracking takes place in a structure the total tangent stiffness matrix may also be negative so that local maxima (and minima) may occur in the force-displacement response. Conventional iteration methods can then not be used and the AUTOMATIC-ITERATION method need then be tried, unless a remodelling is performed.

- The redistribution of strain during cracking can be mesh and path dependent. While some integration points may show strain concentration, thus in simplified words move along the uniaxial stress and strain curve with decreasing stress for increasing strain (the tension stiffening portion), other integration points may unload with decreasing stress and strain. This possibility of different stress unloading paths when cracking has occurred can cause numerical difficulties.

- In reinforced concrete analysis it may in many cases be important to make a relatively coarse mesh so that the overall structural behaviour is described rather than all the details. The amount of details in the complex world of reinforced concrete analysis can be overwhelming and can, if included in the model, very well make the solution difficulties insurmountable in practice.

- It is often of interest to verify the concrete material parameters by running uniaxial tension and compression tests using a single element and compare the produced stress-strain curves with experimental curves, if available. It is particularly important to verify the effect of the chosen values for σ_c / ε_c and σ_u / ε_u on the uniaxial compression curve.

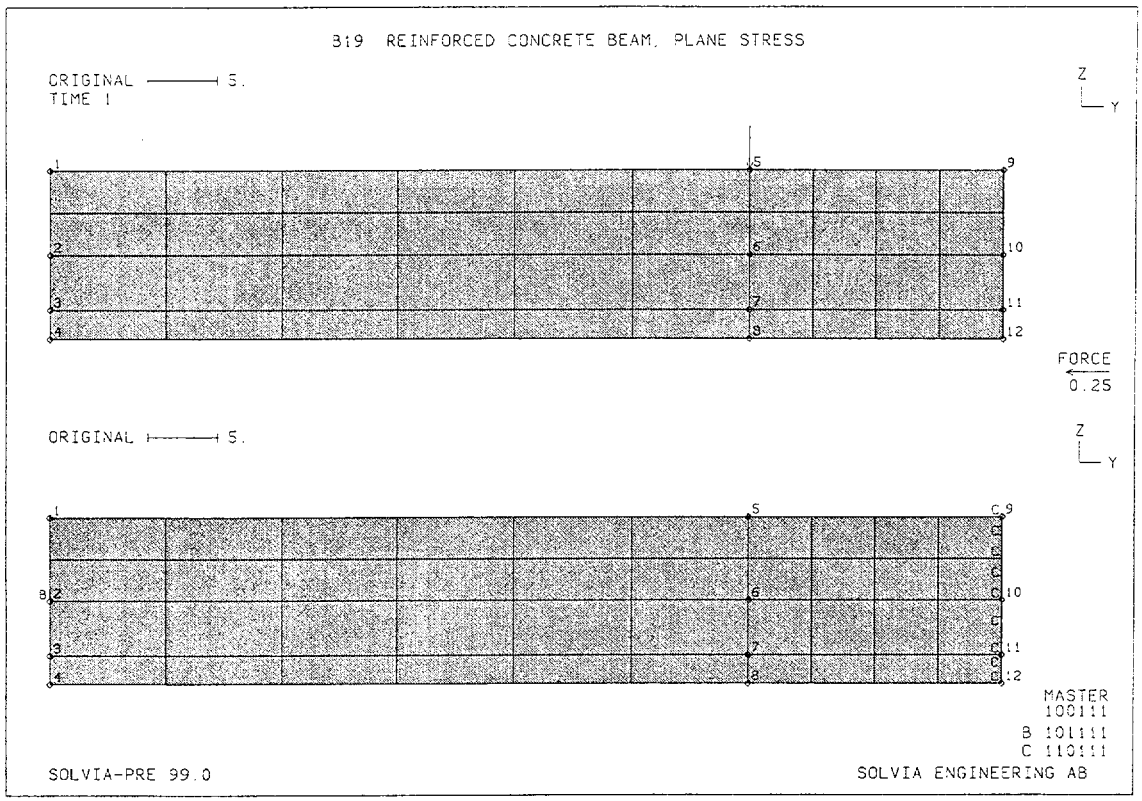
- Convergence of the equilibrium iterations is often improved by specifying as large values for KAPPA, SHEFAC and the equilibrium tolerances as are technically reasonable.

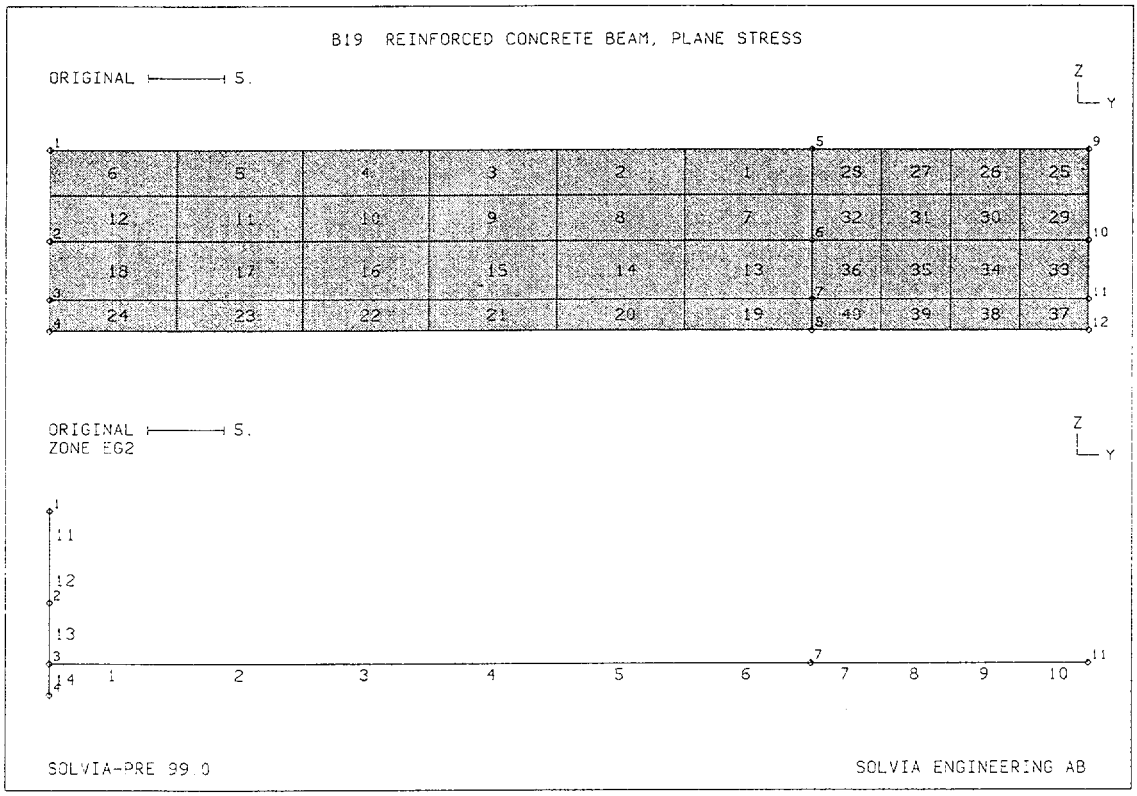
- In case of severe difficulties in obtaining a converged solution one may often gain insight in the behaviour of the model by running an analysis with very small load steps and without equilibrium iterations, see command EQUILIBRIUM-ITERATION in the SOLVIA-PRE Users Manual.

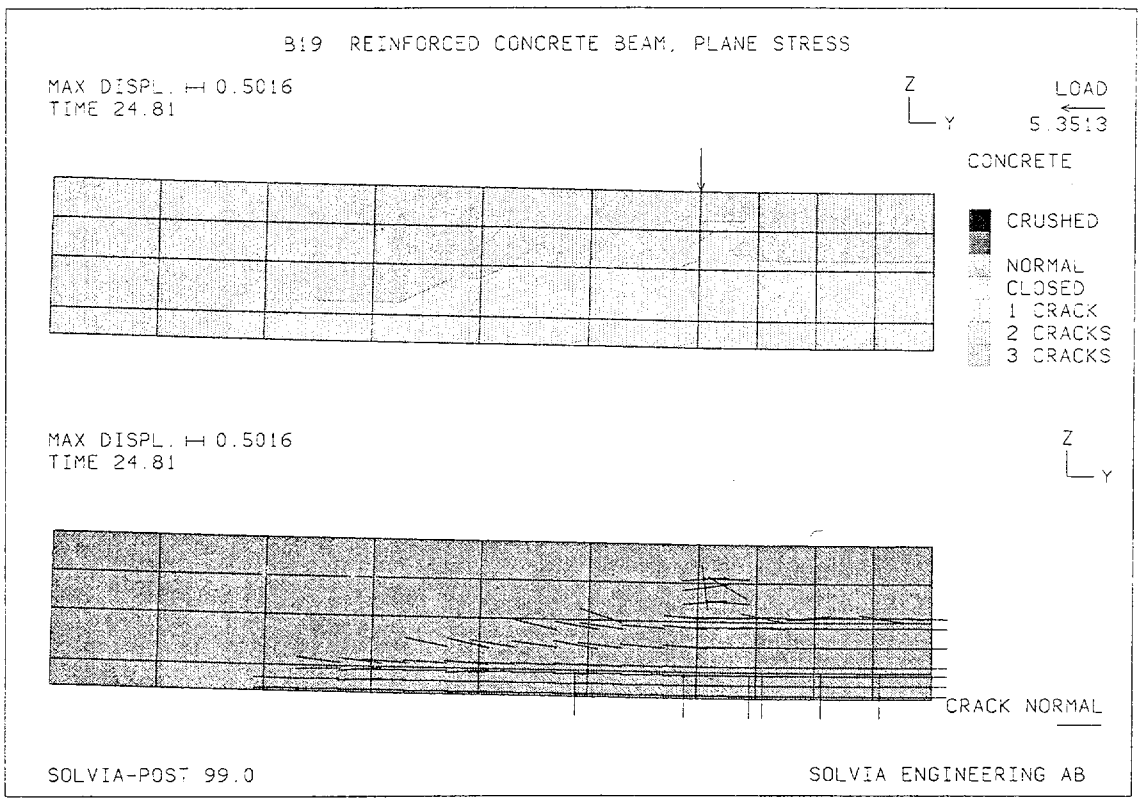
References

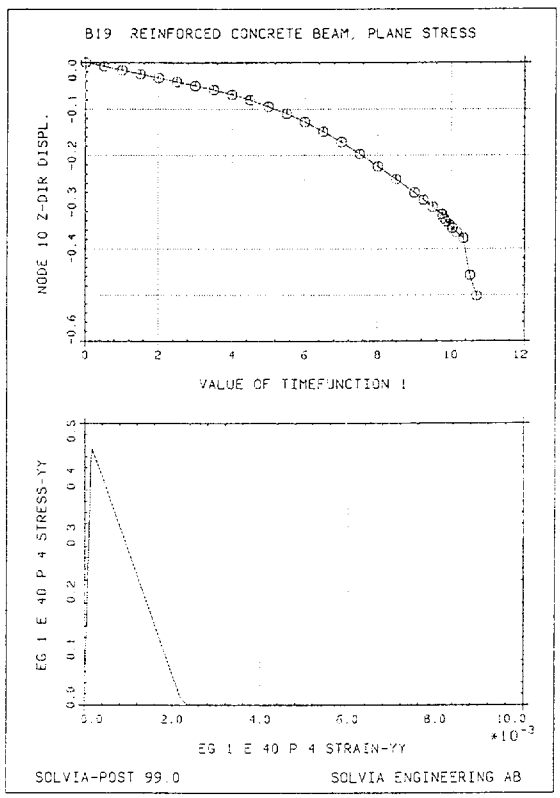
[1] Krahl, N.W., Khachaturian, M., and Seiss, C.P., "Stability of Tensile Cracks in Concrete Beams", ASCE, J. Struct. Div., Feb. 1967, pp. 235-254.

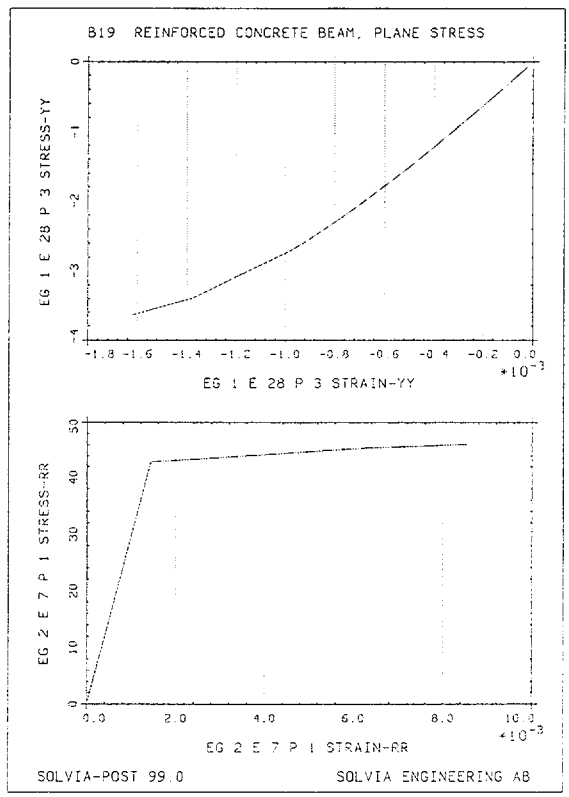
[2] Suidan, M., and Schnobrich, W., "Finite Element Analysis of Reinforced Concrete", ASCE, J. Struct. Div., Vol. 99, No. ST10, Oct. 1973.

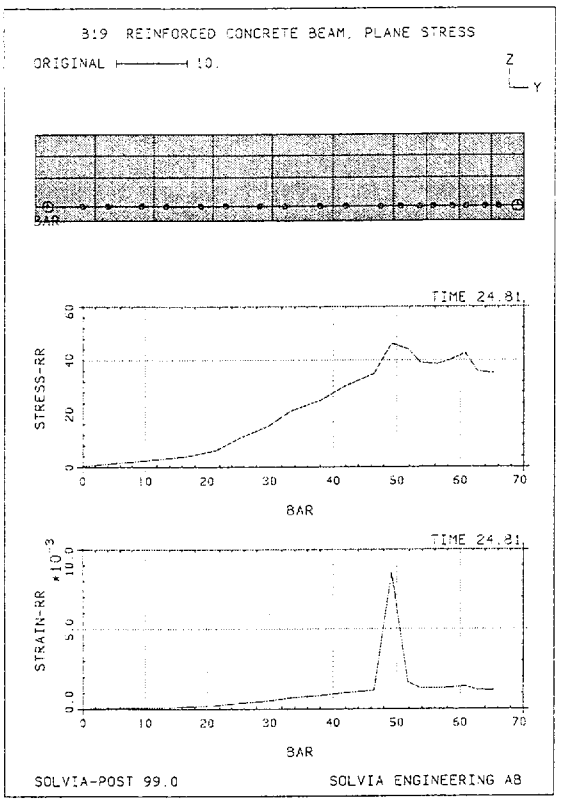

B19 REINFORCED CONCRETE BEAM, PLANE STRESS
ORIGINAL 5.
TIME 1
FORCE
0.25
ORIGINAL 5.
MASTER
100111
B 101111
C 110111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

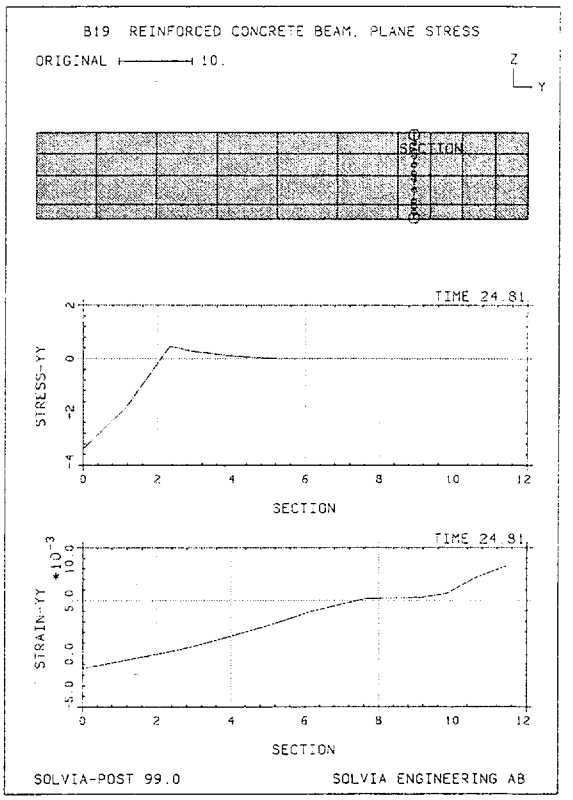

B19 REINFORCED CONCRETE BEAM, PLANE STRESS
ORIGINAL 5.
ORIGINAL 5.
ZONE EG2
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

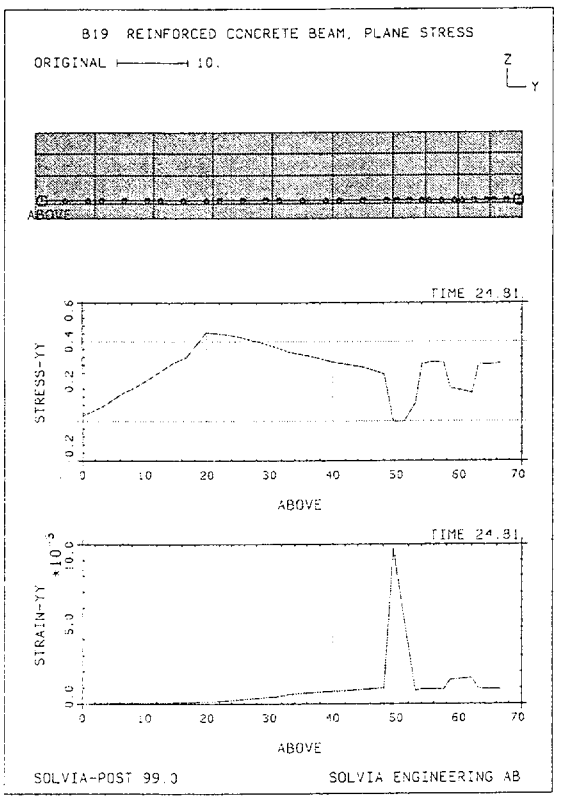

B19 REINFORCED CONCRETE BEAM, PLANE STRESS
MAX DISPL. = 0.5016
TIME 24.81
Z
Y
LOAD
5.3513
CONCRETE
CRUSHED
NORMAL
CLOSED
1 CRACK
2 CRACKS
3 CRACKS
MAX DISPL. = 0.5016
TIME 24.81
Z
Y
CRACK NORMAL
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

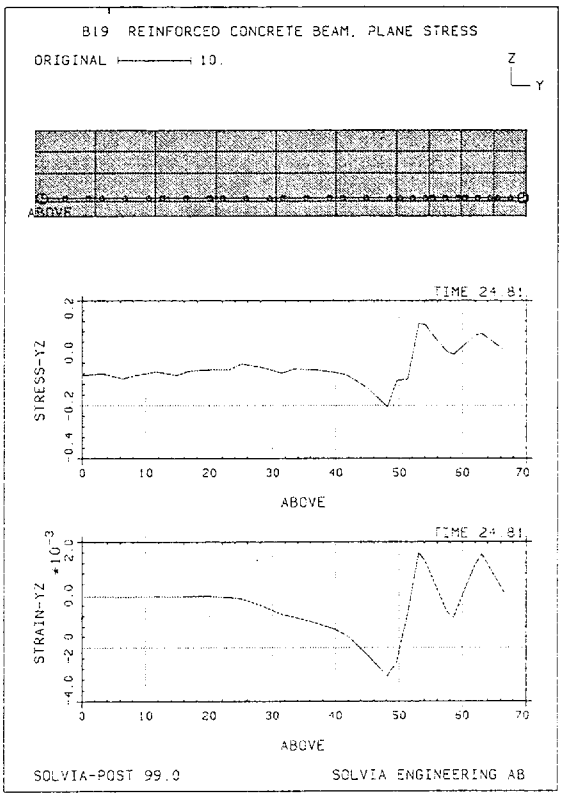

B19 REINFORCED CONCRETE BEAM, PLANE STRESS
NODE 10 Z-DIR DISPL.
VALUE OF TIMEFUNCTION 1
EG 1 E 40 P 4 STRESS-YY
EG 1 E 40 P 4 STRAIN-YY
*10^-3
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

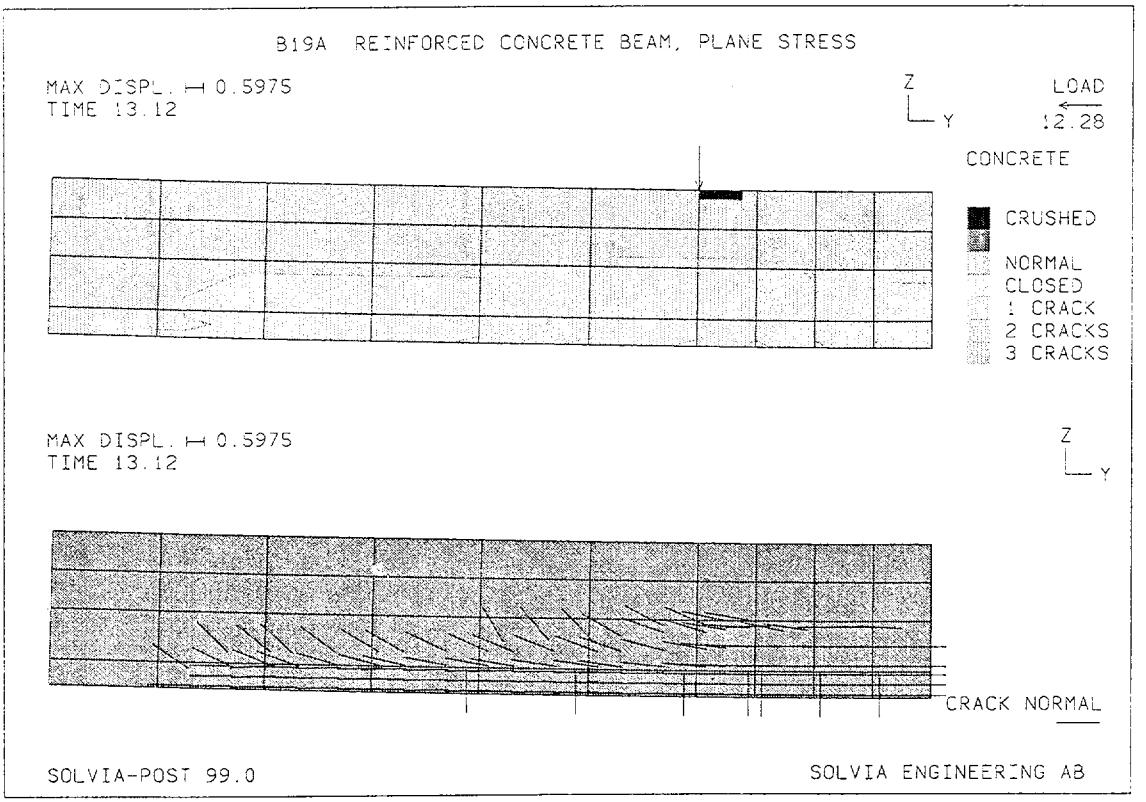

B19 REINFORCED CONCRETE BEAM, PLANE STRESS
EG 1 E 28 P 3 STRESS-YY
EG 1 E 28 P 3 STRAIN-YY
*10^-3
EG 2 E 7 P 1 STRESS-RR
EG 2 E 7 P 1 STRAIN-RR
*10^-3
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

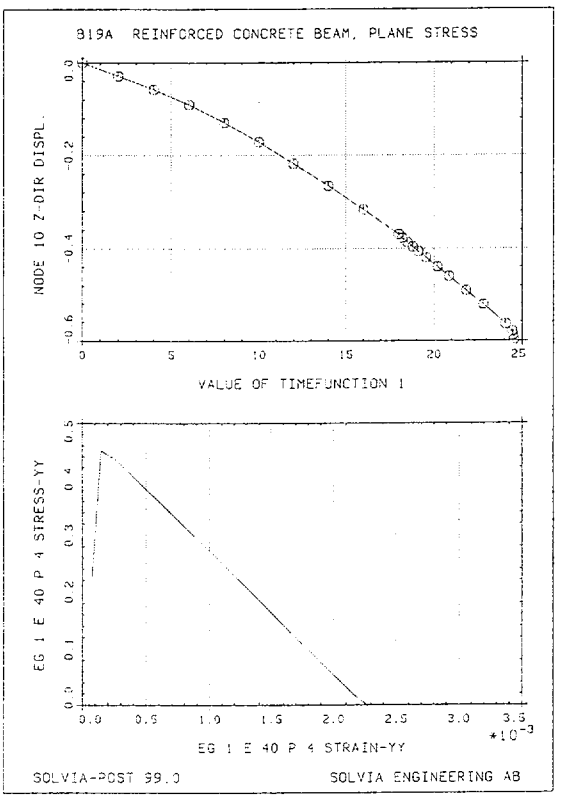

B19 REINFORCED CONCRETE BEAM, PLANE STRESS
ORIGINAL 10.
Z
Y
BAR
TIME 24.81
STRESS-RR
BAR
TIME 24.81
STRAIN-RR
BAR
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

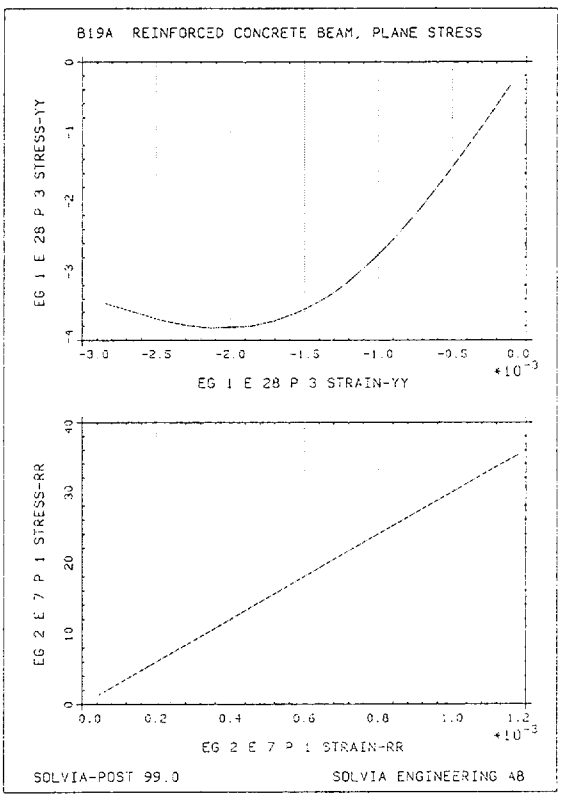

B19 REINFORCED CONCRETE BEAM, PLANE STRESS
ORIGINAL 10.
Z
Y
SECTION
TIME 24.81
STRESS-YY
SECTION
TIME 24.81
STRAIN-YY
SECTION
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

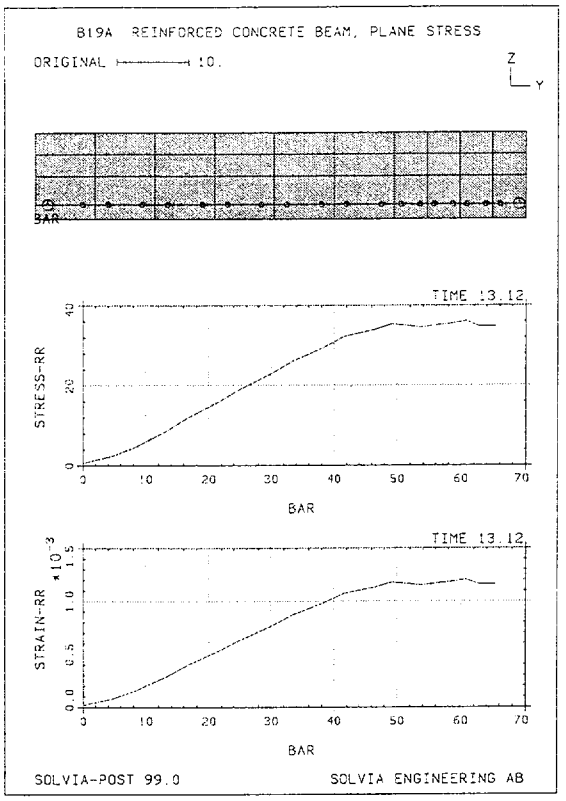

B19 REINFORCED CONCRETE BEAM, PLANE STRESS
ORIGINAL 10.
Z
Y
ABOVE
TIME 24.81
STRESS-YY
ABOVE
TIME 24.81
STRAIN-YY
ABOVE
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

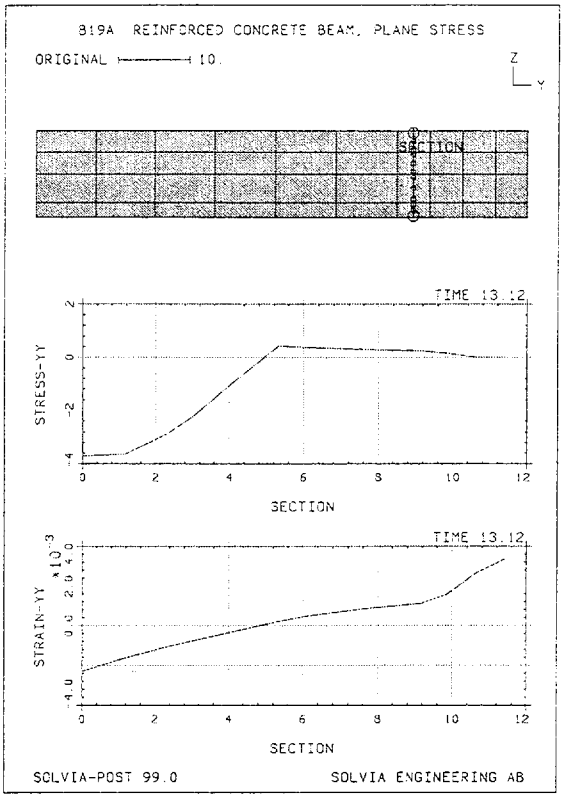

B19 REINFORCED CONCRETE BEAM, PLANE STRESS
ORIGINAL 10.
Z
Y
ABOVE
TIME 24.81
STRESS-YZ
ABOVE
TIME 24.81
STRAIN-YZ
ABOVE
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

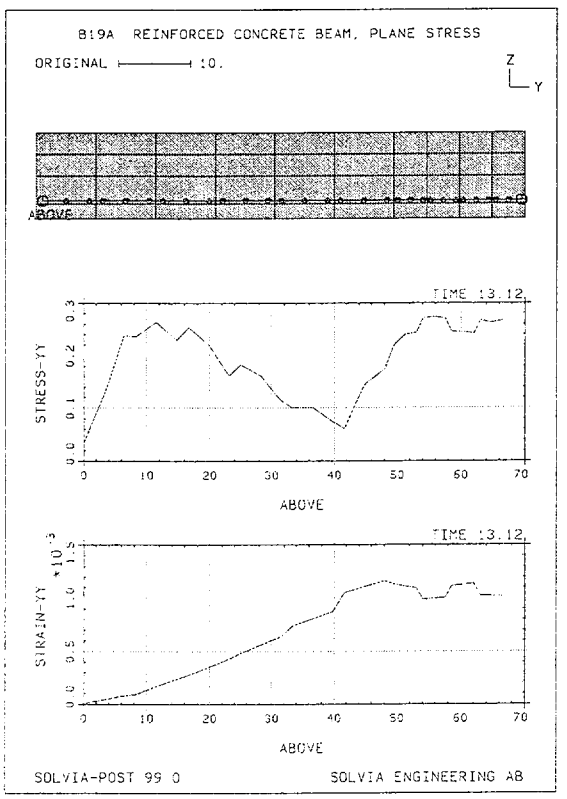

B19A REINFORCED CONCRETE BEAM, PLANE STRESS
MAX DISPL. = 0.5975
TIME 13.12
Z
Y
LOAD
12.28
CONCRETE
CRUSHED
NORMAL
CLOSED
1 CRACK
2 CRACKS
3 CRACKS
MAX DISPL. = 0.5975
TIME 13.12
Z
Y
CRACK NORMAL
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

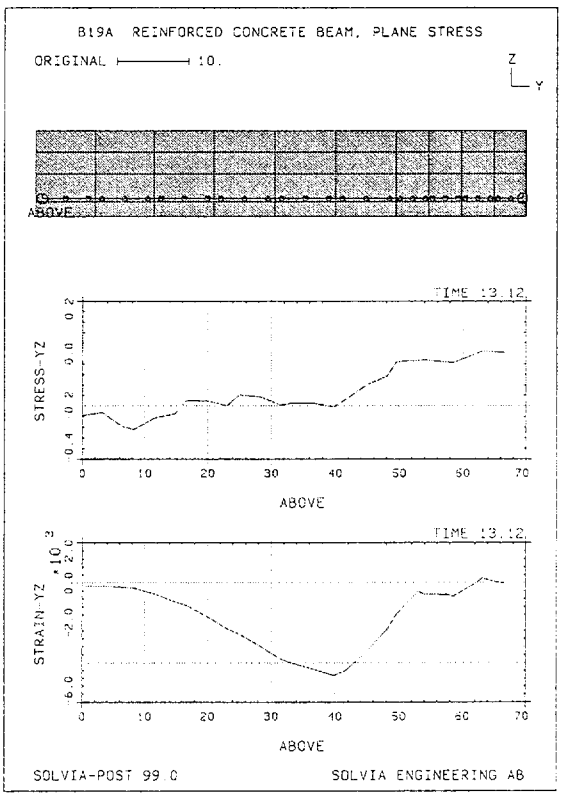

B19A REINFORCED CONCRETE BEAM, PLANE STRESS
NODE 10 Z-DIR DISPL.
VALUE OF TIMEFUNCTION 1
EG 1 E 40 P 4 STRESS-YY
EG 1 E 40 P 4 STRAIN-YY
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B19A REINFORCED CONCRETE BEAM, PLANE STRESS
EG 1 E 28 P 3 STRESS-YY
EG 1 E 28 P 3 STRAIN-YY
EG 2 E 7 P 1 STRESS-RR
EG 2 E 7 P 1 STRAIN-RR
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B19A REINFORCED CONCRETE BEAM, PLANE STRESS
ORIGINAL 10.
TIME 13.12
STRESS-RR
BAR
TIME 13.12
STRAIN-RR
BAR
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B19A REINFORCED CONCRETE BEAM, PLANE STRESS
ORIGINAL 10.
SECTION
TIME 13.12
STRESS-YY
SECTION
TIME 13.12
STRAIN-YY
SECTION
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B19A REINFORCED CONCRETE BEAM, PLANE STRESS
ORIGINAL 10.
ABOVE
TIME 13.12
STRESS-YY
ABOVE
TIME 13.12
STRAIN-YY
ABOVE
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B19A REINFORCED CONCRETE BEAM, PLANE STRESS
ORIGINAL 10.
ABOVE
TIME 13.12
STRESS-YZ
ABOVE
TIME 13.12
STRAIN-YZ
ABOVE
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B19A  REINFORCED CONCRETE BEAM, PLANE STRESS'
*
DATABASE  CREATE
MASTER  IDOF=100111 NSTEP=40 DT=1
ITERATION  METHOD=FULL-NEWTON LINE-SEARCH=YES
AUTO-STEP  DTMIN=1.E-5 DTMAX=1. ITELOW=7 ITEHIGH=15,
           FDECREASE=0.1 FRESTART=0.1 TMAX=24
TOLERANCES  TYPE=F RNORM=10 RTOL=0.05 ITEMAX=30
*
TIMEFUNCTION  1
 0. 0.  /  12. 24.  /  24. 30.
*
COORDINATES  /  ENTRIES  NODE  Y  Z
 1   0. 12.  /  2   0. 6.  /  3   0. 2.06  /  4   0.
 5  50. 12.  /  6  50. 6.  /  7  50. 2.06  /  8  50.
 9  68. 12.  / 10  68. 6.  / 11  68. 2.06  / 12  68.
*
MATERIAL  1  CONCRETE E0=3.06E3 NU=0.2 SIGMAT=0.458 SIGMAC=-3.74,
             EPSC=-0.002 SIGMAU=-3.225 EPSU=-0.003 BETA=0.75,
             KAPPA=15 STIFAC=0.0001 SHEFAC=0.5
MATERIAL  2  PLASTIC E=3.E4 YIELD=44 ET=300.
*
EGROUP  1  PLANE STRESS2  MATERIAL=1
GSURFACE   5 1 2  6  EL1=6 EL2=2 NODES=8
GSURFACE   6 2 3  7  EL1=6 EL2=1 NODES=8
GSURFACE   7 3 4  8  EL1=6 EL2=1 NODES=8
GSURFACE   9 5 6 10  EL1=4 EL2=2 NODES=8
GSURFACE  10 6 7 11  EL1=4 EL2=1 NODES=8
GSURFACE  11 7 8 12  EL1=4 EL2=1 NODES=8
EDATA  /  1  6.
*
EGROUP  2  TRUSS  MATERIAL=2
GLINE  3  7  EL=6 NODES=3
GLINE  7 11  EL=4 NODES=3
GLINE  1  2  EL=2 NODES=3
GLINE  2  3  EL=1 NODES=3
GLINE  3  4  EL=1 NODES=3
EDATA  /  1  2.0
*
FIXBOUNDARIES  2 INPUT=LINES  /  9 10  /  10 11  /  11 12
FIXBOUNDARIES  3 INPUT=NODES  /  2
LOADS  CONCENTRATED
 5 3 -0.5
*
SET  NSYMBOLS=MYNODES NNUMBERS=MYNODES VIEW=X HEIGHT=0.25
MESH  VECTOR=LOAD SUBFRAME=12
MESH  BCODE=ALL
MESH  ENUMBERS=EG1 SUBFRAME=12
MESH  EG2 ENUMBERS=YES
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B19  REINFORCED CONCRETE BEAM, PLANE STRESS
*
DATABASE  CREATE
WRITE  FILENAME='b19a.lis'
*
SET  VIEW=X DIAGRAM=GRID
MESH  VECTOR=LOAD CONTOUR=CONCRETE SUBFRAME=12
MESH  VECTOR=CRACK-NORMALS
*
SET  PLOTORIENTATION=PORTRAIT
SUBFRAME  12
NHISTORY NODE=10 DIRECTION=3 SYMBOL=1 XVARIABLE=1 OUTPUT=ALL
EXYPLOT  ELEMENT=40 POINT=4 XKIND=EYY YKIND=SYY
*
EXYPLOT  ELEMENT=28 POINT=3 XKIND=EYY YKIND=SYY SUBFRAME=12
EGROUP  2
EXYPLOT  ELEMENT=7 POINT=1 XKIND=ERR YKIND=SRR
*
EPLINE BAR
1 1 2  TO 10 1 2
MESH PLINE=BAR SUBFRAME=13
ELINE BAR KIND=SRR
ELINE BAR KIND=ERR
*
EGROUP 1
EPLINE SECTION
28 6 5 4 STEP 4 TO 40 6 5 4
MESH PLINE=SECTION SUBFRAME=13
ELINE SECTION KIND=SYY
ELINE SECTION KIND=EYY
*
EPLINE ABOVE
18 1 4 7 STEP -1 TO 13 1 4 7
36 1 4 7 STEP -1 TO 33 1 4 7
MESH PLINE=ABOVE SUBFRAME=13
ELINE ABOVE KIND=SYY
ELINE ABOVE KIND=EYY
MESH PLINE=ABOVE SUBFRAME=13
ELINE ABOVE KIND=SYZ
ELINE ABOVE KIND=EYZ
*
EGROUP  1 / ZONE  NAME=CONCRETE INPUT=ELEMENTS  /  28 33
EGROUP  2 / ZONE  NAME=STEEL INPUT=ELEMENTS  /  1  TO  10
*
ELIST  ZONENAME=CONCRETE SELECT=CONCRETE
ELIST  ZONENAME=CONCRETE SELECT=BASICS
ELIST  ZONENAME=STEEL
END
```

EXAMPLE B20

CONCRETE MATERIAL CURVES IN COMPRESSION

Objective

To verify the uniaxial and biaxial compression characteristics of the SOLVIA concrete material model.

Physical Problem

In a classical paper by Kupfer et al. [1], concrete specimens of dimension $20 \times 20 \times 5$ cm were subjected to stress combinations in the regions of biaxial compression, compression-tension and biaxial tension. In this example the SOLVIA concrete material model is compared with the test results of Kupfer et al. in the biaxial compression and compression-tension regions including uniaxial compression.

Finite Element Model

One 4 node PLANE STRESS2 element is used for the uniaxial case and one 8 node SOLID element is used in the other analyses. The biaxial loading is applied as element pressure in the global Y- and Z-directions. The X-direction is stress-free.

Kupfer et al. analyzed three types of concrete with an unconfined uniaxial compressive strength of 190, 315 and 590 kp/cm^2.

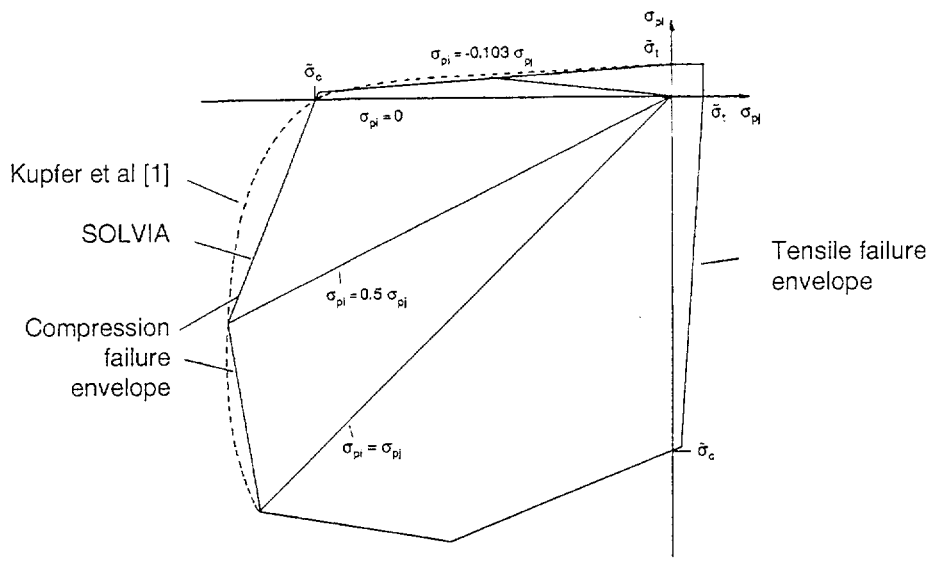
The material data used in this example is based on the experimental results for the concrete with the compressive strength 315 kp/cm^2. All material parameters are shown on the pages with input data and include:

$\tilde{\sigma}_c = -315$ kp/cm^2	uniaxial compressive strength
$\tilde{\varepsilon}_c = -0.0021$	adjusted uniaxial strain value corresponding to the compressive strength
$\tilde{\sigma}_u = -280$ kp/cm^2	ultimate compressive stress
$\tilde{\varepsilon}_u = -0.0031$	corresponding ultimate strain
$E_o = 325000$ kp/cm^2	Young's modulus
$\nu = 0.2$	Poisson's ratio
$\tilde{\sigma}_t = 28.35$ kp/cm^2	uniaxial tensile strength $(-0.09 \cdot \tilde{\sigma}_c)$

The parameters of the failure surface in compression when $\sigma_{xx} = 0$ are:

SP311=1.0	$\sigma_{zz} / \tilde{\sigma}_c$ when $\sigma_{yy} = 0$
SP312=1.24	$\sigma_{zz} / \tilde{\sigma}_c$ when $\sigma_{yy} = 0.5 \cdot \sigma_{zz}$
SP313=1.15	$\sigma_{zz} / \tilde{\sigma}_c$ when $\sigma_{yy} = \sigma_{zz}$

The experimental failure envelope and the failure envelope used in SOLVIA are shown below together with the four stress load paths used in the calculations.



The Automatic-Iteration method is used to trace the pre- and post-crushing behaviour of the concrete model. Force tolerance is used in the equilibrium iterations with RNORM = 1 and RTOL = 0.001. The finite element model is shown in the top left figure on pages B20.4 to B20.7 for the analysis cases.

Solution Results

The first analysis is the uniaxial case with loading only in the Z-direction and the input data used is shown on pages B20.8 and B20.9 . A plane stress finite element model, see top left figure on page B20.4, was employed but the 3D model with a SOLID element would yield the same results. The stress response in the load direction as a function of the strain in the same direction and the strain in the transverse direction are shown in the top right figure on page B20.4. The symbols show the corresponding experimental results from [1].

The stress in the load direction is also shown as a function of the volumetric strain $e_{xx} + e_{yy} + e_{zz}$ (left bottom figure). The agreement with the experimental results of [1] shown as symbols is good up to 80-90 % of the concrete compressive strength. At this load level microcracks start to form in the experiment which is accompanied with a significant increase of the volumetric strain.

The strain in the load direction as a function of the volumetric strain is shown in the right bottom figure of page B20.4 together with the corresponding experimental values. A constant Poisson's value gives a linear relationship as obtained in the analysis. In the experiment the relationship is linear up to the load level 80-90% of the compressive strength where microcracks start to develop.

In the analysis case B20A equal biaxial compressive loading is used, see the input data on pages B20.10 and B20.11 and the figures on page B20.5. The top right figure shows σ_{zz} as a function of the strain e_{zz} and e_{xx}. The X-direction has no applied pressure. The strain response in the Y-direction is equal to the strain response in the Z-direction, as it should, see the left bottom figure. The volumetric strain response is shown in the right bottom figure. In all three figures the corresponding experimental

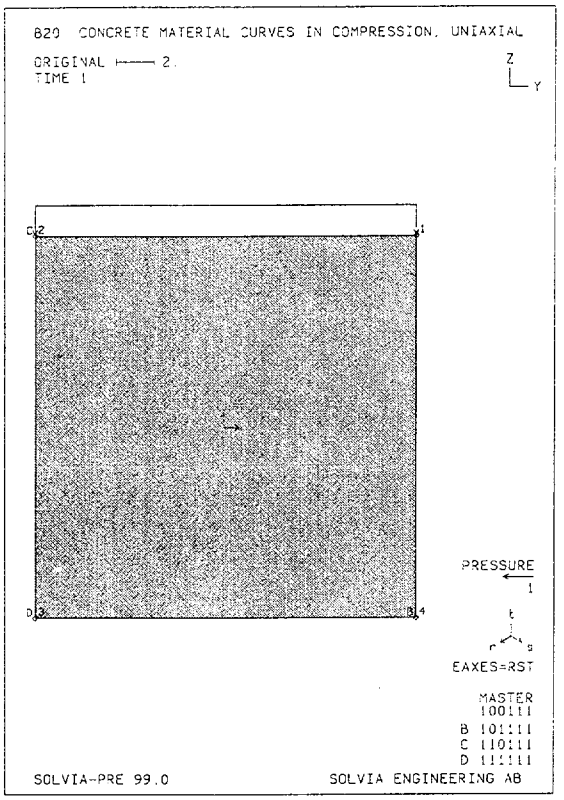
results are shown as symbols and the response earlier calculated in the uniaxial case is also shown for comparison. Good agreement with the experimental values can be seen up to 80-90% of the compressive strength. Higher load levels result again in a significant increase in the experimental volumetric strain which cannot be reproduced since Poisson's ratio is constant in the analysis.

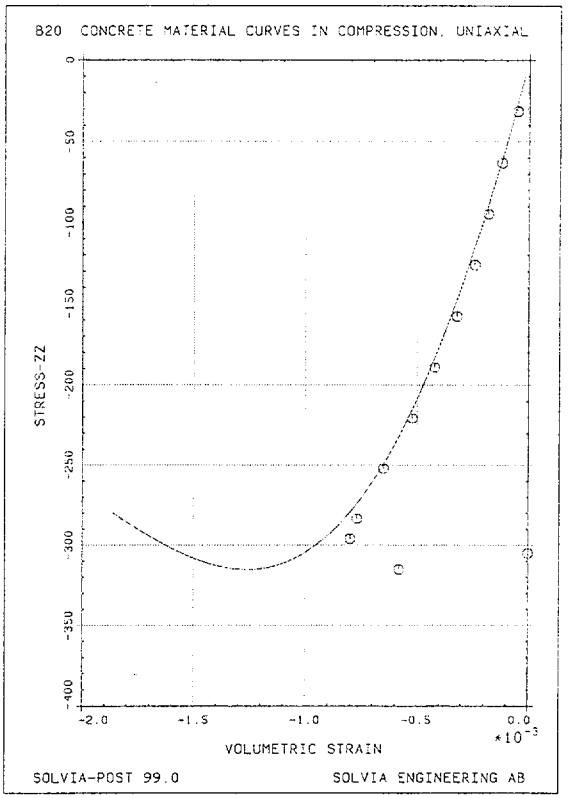
In the analysis case B20B the compressive loading in the Y-direction is 0.52 times the loading in the Z-direction, see the input data on pages B20.12 and B20.13 and the response figures on page B20.6. The strain responses e_{zz} and e_{xx} are shown in the top right figure and e_{zz} and e_{yy} in the left bottom figure together with the corresponding experimental results plotted as symbols. The volumetric strain response is shown in the right bottom figure. The corresponding uniaxial strain responses as earlier calculated are shown in the three figures for comparison. Good agreement between the calculated and the experimental values can again be seen up to the load level of 80-90% of the compressive strength.

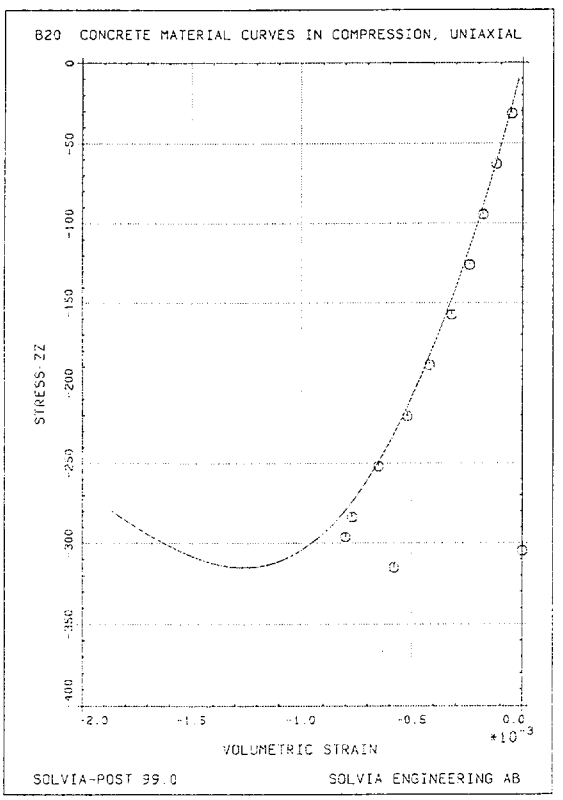
In the analysis case B20C the Y-direction load is -0.103 times the compressive loading in the Z-direction. The Y-direction load is then in tension. The input data is shown on pages B20.14 and B20.15 and the response figures on page B20.7. The experimental results are marked as symbols in the figures and the uniaxial curves have been included for comparison. The calculations stopped when the failure surface was reached at $\sigma_{yy} = 17.1$ kp / cm^2 and $\sigma_{zz} = -166.3$ kp / cm^2. Failure was reached in the experiment when $\sigma_{zz} = -194$ kp / cm^2.

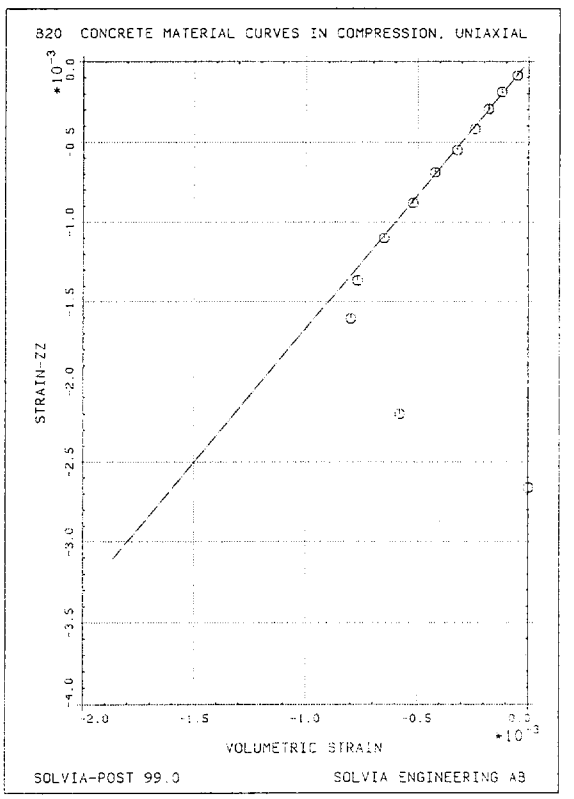
Reference

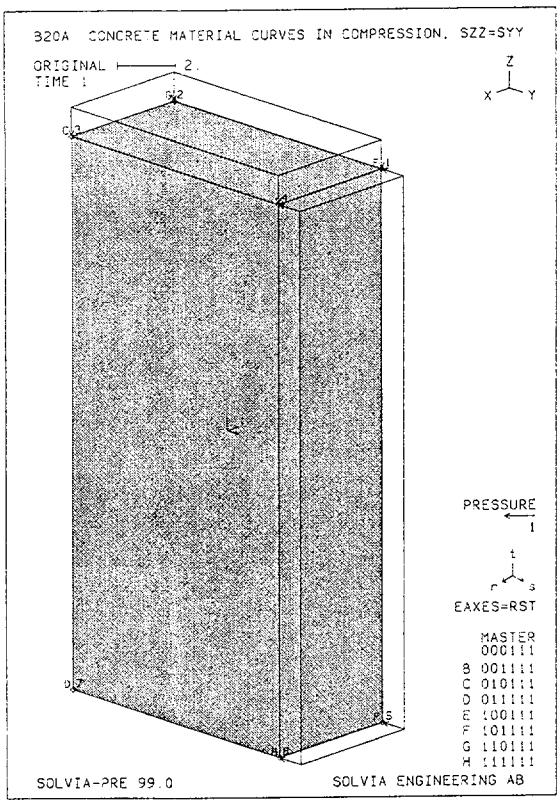
[1] Kupfer, H., Hilsdorf, H.K. and Rusch, H., "Behaviour of Concrete Under Biaxial Stress", Title No. 66-52, ACI Journal, Aug. 1969.

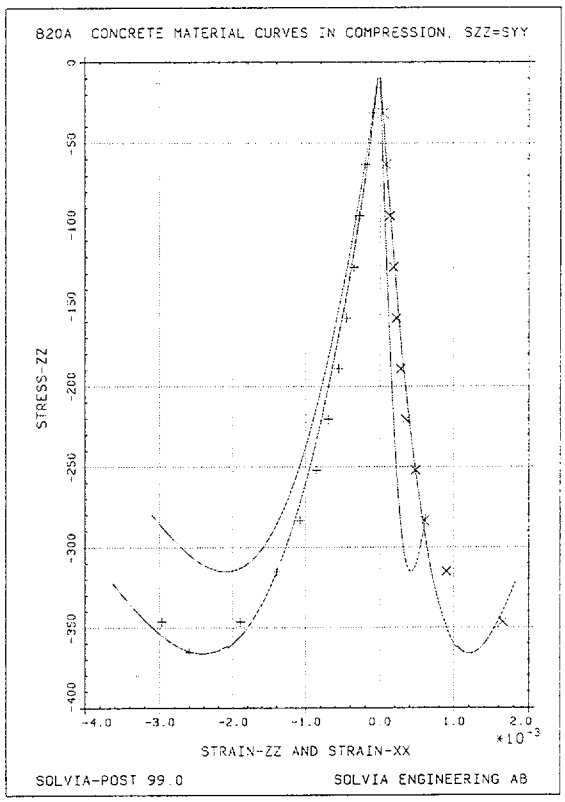

B20 CONCRETE MATERIAL CURVES IN COMPRESSION, UNIAXIAL
ORIGINAL 2.
TIME 1
Z
Y
PRESSURE
1
EAXES=RST
MASTER
100111
B 101111
C 110111
D 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

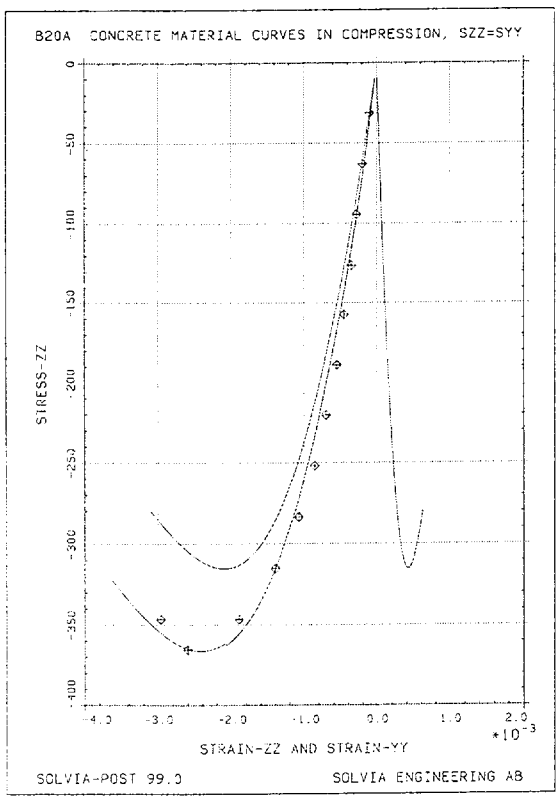

B20 CONCRETE MATERIAL CURVES IN COMPRESSION, UNIAXIAL
STRESS-ZZ
VOLUMETRIC STRAIN
*10^-3
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

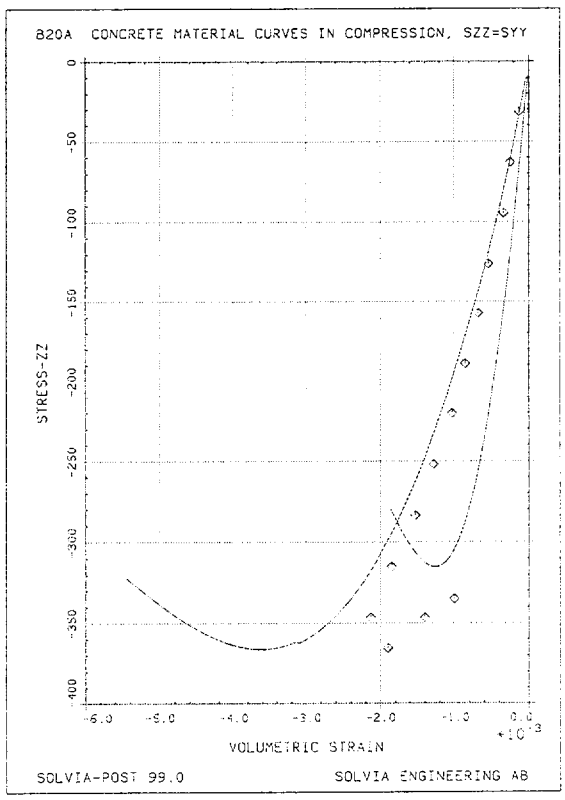

B20 CONCRETE MATERIAL CURVES IN COMPRESSION, UNIAXIAL
STRESS-ZZ
VOLUMETRIC STRAIN
*10^-3
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

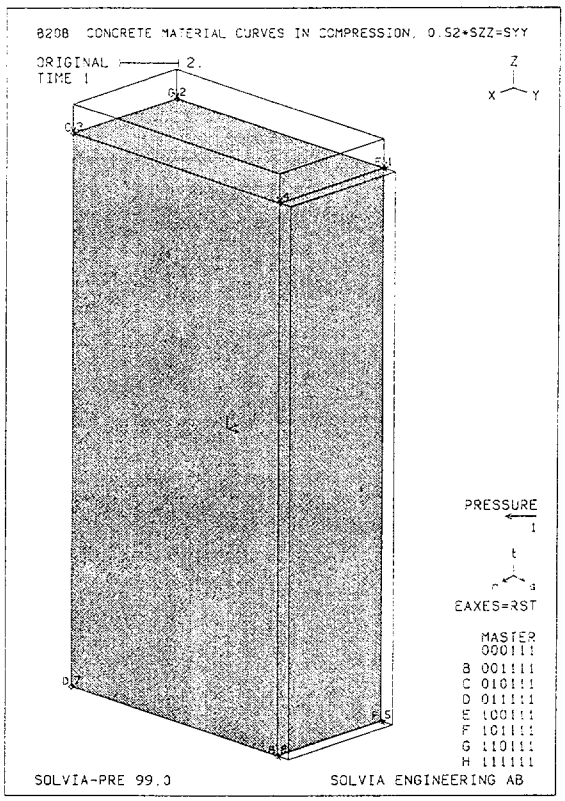

B20 CONCRETE MATERIAL CURVES IN COMPRESSION, UNIAXIAL
STRAIN-ZZ
VOLUMETRIC STRAIN
*10^-3
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

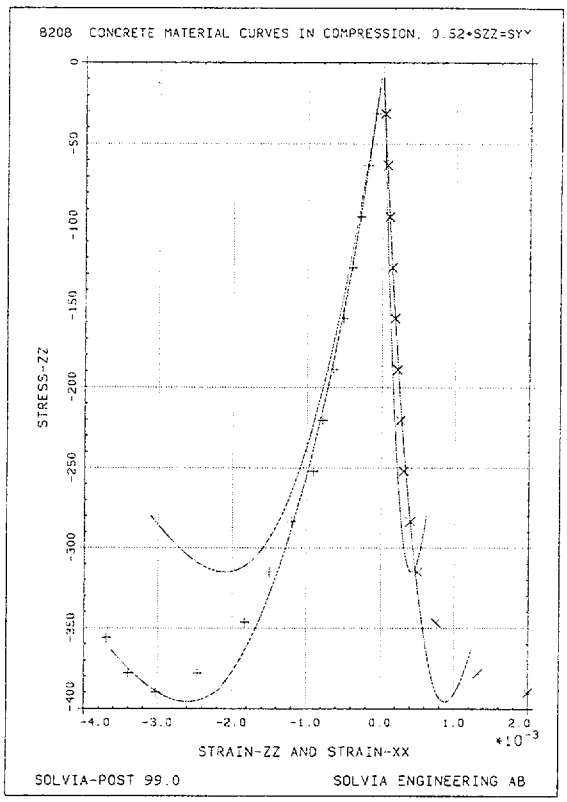

B20A CONCRETE MATERIAL CURVES IN COMPRESSION, SZZ=SYY
ORIGINAL
TIME 1
PRESSURE
EAXES=RST
MASTER
000111
B 001111
C 010111
D 011111
E 100111
F 101111
G 110111
H 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

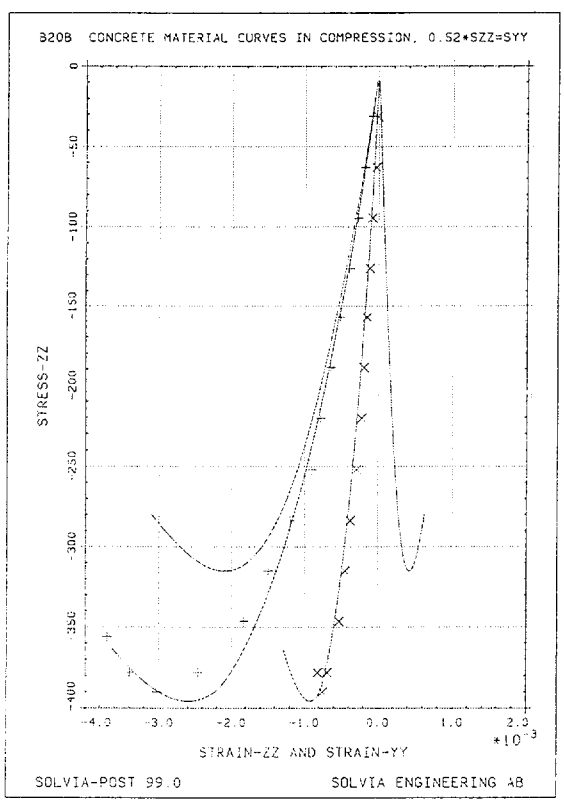

B20A CONCRETE MATERIAL CURVES IN COMPRESSION, SZZ=SYY
STRESS-ZZ
STRAIN-ZZ AND STRAIN-XX
*10^-3
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

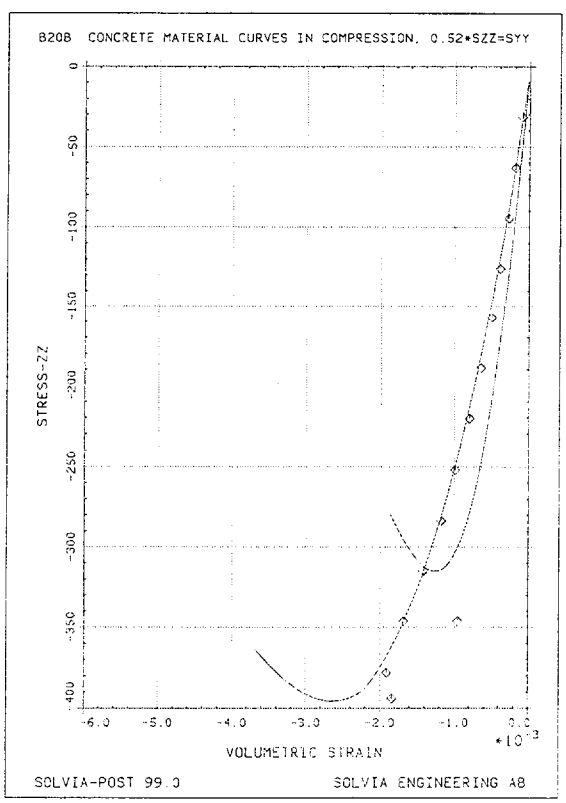

B20A CONCRETE MATERIAL CURVES IN COMPRESSION, SZZ=SYY
STRESS-ZZ
STRAIN-ZZ AND STRAIN-YY
*10^-3
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

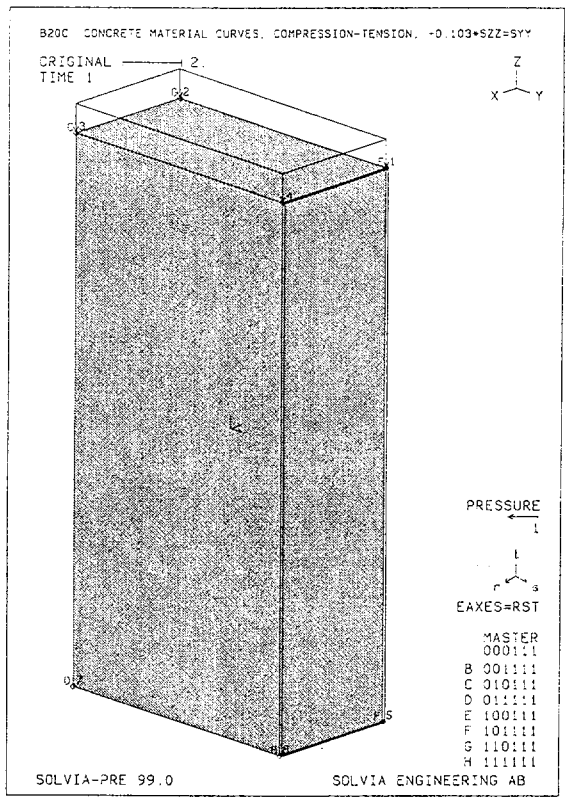

B20A CONCRETE MATERIAL CURVES IN COMPRESSION, SZZ=SYY
STRESS-ZZ
VOLUMETRIC STRAIN
*10^-3
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

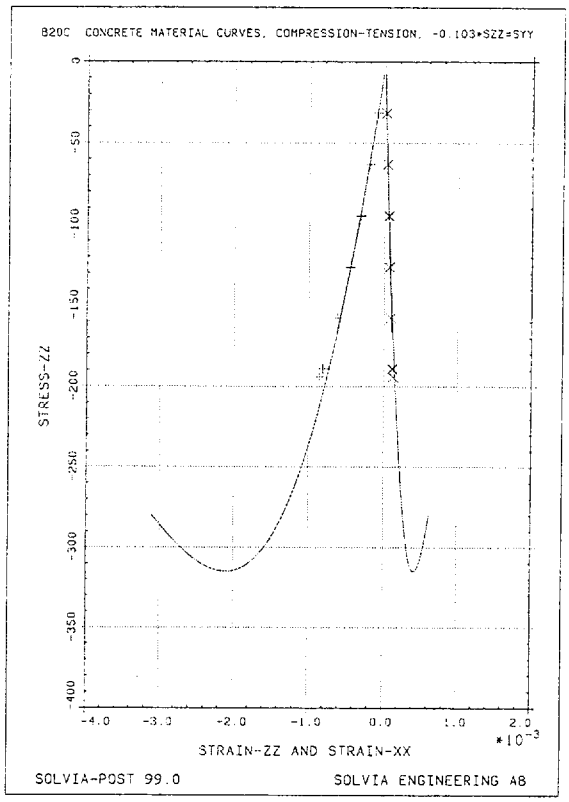

B20B CONCRETE MATERIAL CURVES IN COMPRESSION, 0.52*SZZ=SYY
ORIGINAL
TIME 1
PRESSURE
EAXES=RST
MASTER
000111
B 001111
C 010111
D 011111
E 100111
F 101111
G 110111
H 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB

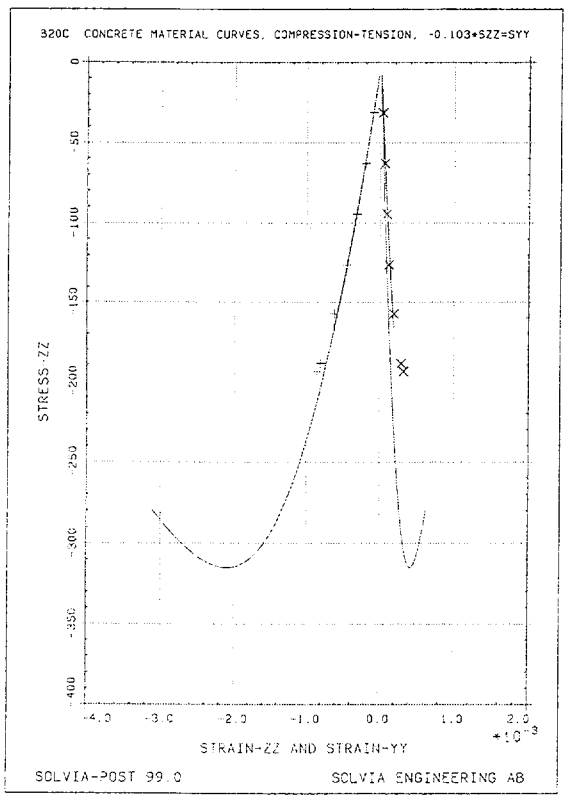

B20B CONCRETE MATERIAL CURVES IN COMPRESSION, 0.52*SZZ=SYY
STRESS-ZZ
STRAIN-ZZ AND STRAIN-XX
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B20B CONCRETE MATERIAL CURVES IN COMPRESSION, 0.52*SZZ=SYY
STRESS-ZZ
STRAIN-ZZ AND STRAIN-YY
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B20B CONCRETE MATERIAL CURVES IN COMPRESSION, 0.52*SZZ=SYY
STRESS-ZZ
VOLUMETRIC STRAIN
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B20C CONCRETE MATERIAL CURVES, COMPRESSION-TENSION, -0.103*SZZ=SYY
ORIGINAL
TIME 1
PRESSURE
EAXES=RST
MASTER
000111
B 001111
C 010111
D 011111
E 100111
F 101111
G 110111
H 111111
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


B20C CONCRETE MATERIAL CURVES, COMPRESSION-TENSION, -0.103*SZZ=SYY
STRESS-ZZ
STRAIN-ZZ AND STRAIN-XX
*10^-3
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B20C CONCRETE MATERIAL CURVES, COMPRESSION-TENSION, -0.103*SZZ=SYY
STRESS-ZZ
STRAIN-ZZ AND STRAIN-YY
*10^-3
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B20  CONCRETE MATERIAL CURVES IN COMPRESSION, UNIAXIAL'
*
DATABASE  CREATE
MASTER  IDOF=100111 NSTEP=200 DT=1.
AUTOMATIC-ITERATION  NODE=1 DIRECTION=3 DISPLACEMENT=-6.E-4,
                     DISPMAX=0.062 CONTINUATION=YES ALFA=0.2
TOLERANCES  TYPE=F RNORM=1 RTOL=0.001
PRINTOUT MIN
*
COORDINATES  /  ENTRIES  NODE  Y  Z
 1  20. 20.  /  2  0. 20.  /  3  /  4  20.
*
MATERIAL  1  CONCRETE E0=325000 NU=0.2 SIGMAT=28.35,
             SIGMAC=-315 EPSC=-0.0021 SIGMAU=-280 EPSU=-0.0031,
             BETA=0.5 KAPPA=15. STIFAC=0.0001 SHEFAC=0.5,
             SP311=1.0 SP312=1.24 SP313=1.15
*
EGROUP  1  PLANE STRESS2
ENODES
 1  1 2 3 4
EDATA  /  1 5.
LOADS  ELEMENT  /  1 S 1. 1.
*
FIXBOUNDARIES  2  /  2 3
FIXBOUNDARIES  3  /  3 4
*
SET PLOTORIENTATION=PORTRAIT NSYMBOLS=YES BCODE=ALL
MESH NNUMBERS=YES VECTOR=LOAD EAXES=RST
*
SOLVIA
*
END
```

SOLVIA-POST input

```
*  B20  CONCRETE MATERIAL CURVES IN COMPRESSION, UNIAXIAL
*
DATABASE  CREATE
WRITE  FILENAME='b20.lis'
SET  PLOTORIENTATION=PORTRAIT DIAGRAM=GRID
SET  NEWPAGE=NO LINESPERPAGE=10000
*
USERCURVE  1  SORT=NO / READ  B20EX0E1.DAT
USERCURVE  2  SORT=NO / READ  B20EX0E3.DAT
USERCURVE  3  SORT=NO / READ  B20EX0VS.DAT
USERCURVE  4  SORT=NO / READ  B20EX0VE.DAT
AXIS  1  VMIN=-0.004 VMAX=0.002 LABEL='STRAIN-ZZ AND STRAIN-YY'
AXIS  2  VMIN=-400.  VMAX=0.    LABEL='STRESS-ZZ'
AXIS  3  VMIN=-0.002 VMAX=0.    LABEL='VOLUMETRIC STRAIN'
AXIS  4  VMIN=-0.004 VMAX=0.    LABEL='STRAIN-ZZ'
*
EXYPLOT  EL=1 POINT=1 XKIND=EZZ YKIND=SZZ XAXIS=1 YAXIS=2 OUTPUT=ALL
EXYPLOT  EL=1 POINT=1 XKIND=EYY YKIND=SZZ XAXIS=-1 YAXIS=-2
OUTPUT=ALL SUBF=OLD
PLOT  USERCURVE  1 XAXIS=-1 YAXIS=-2 SYMBOL=-1 SUBFRAME=OLD
PLOT  USERCURVE  2 XAXIS=-1 YAXIS=-2 SYMBOL=-2 SUBFRAME=OLD
*
EPOINT  NAME=P1 ELEMENT=1 POINT=1
EVARIABLE  NAME=EX TYPE=PLANE KIND=EXX
EVARIABLE  NAME=EY TYPE=PLANE KIND=EYY
EVARIABLE  NAME=EZ TYPE=PLANE KIND=EZZ
EVARIABLE  NAME=SZ TYPE=PLANE KIND=SZZ
RESULTANT  NAME=VOLUME STRING='EX + EY + EZ'
RESULTANT  NAME=STRESS STRING='SZ'
RESULTANT  NAME=STRAIN STRING='EZ'
*
RXYPLOT  XPOINTNAME=P1 XRESULTANTNAME=VOLUME YPOINTNAME=P1,
         YRESULTANTNAME=STRESS XAXIS=3 YAXIS=2 OUTPUT=ALL
PLOT  USERCURVE  3 XAXIS=-3 YAXIS=-2 SYMBOL=-1 SUBFRAME=OLD
*
RXYPLOT  XPOINTNAME=P1 XRESULTANTNAME=VOLUME YPOINTNAME=P1,
         YRESULTANTNAME=STRAIN XAXIS=3 YAXIS=4
PLOT  USERCURVE  4 XAXIS=-3 YAXIS=-4 SYMBOL=-1 SUBFRAME=OLD
END
```

SOLVIA-PRE input

```
HEADING  'B20A  CONCRETE MATERIAL CURVES IN COMPRESSION, SZZ=SYY'
*
DATABASE  CREATE
MASTER  IDOF=000111 NSTEP=100 DT=1.
AUTOMATIC-ITERATION  NODE=1 DIRECTION=3 DISPLACEMENTS=-5.E-4,
                     DISPMAX=0.072 CONTINUATION=YES ALFA=0.8
TOLERANCES  TYPE=F RNORM=1 RTOL=0.001
*
COORDINATES
 1  0 10 20  /  2  0 0 20  /  3  5 0 20  /  4  5 10 20
 5  0 10  0  /  6  0 0  0  /  7  5 0  0  /  8  5 10  0
*
MATERIAL  1  CONCRETE E0=325000 NU=0.2 SIGMAT=28.35,
             SIGMAC=-315 EPSC=-0.0021 SIGMAU=-280 EPSU=-0.0031,
             BETA=0.5 KAPPA=15. STIFAC=0.0001 SHEFAC=0.5,
             SP311=1.0 SP312=1.24 SP313=1.15
*
EGROUP  1  SOLID
ENODES
 1  1 2 3 4 5 6 7 8
LOADS  ELEMENT
 1  R 1 1
 1  T 1 1
*
FIXBOUNDARIES  1  /  1 2 5 6
FIXBOUNDARIES  2  /  2 3 6 7
FIXBOUNDARIES  3  /  5 6 7 8
*
VIEW  ID=1  XVIEW=1 YVIEW=1 ZVIEW=0.5
SET  VIEW=1 PLOTORIENTATION=PORTRAIT NSYMBOLS=YES BCODE=ALL
MESH  NNUMBERS=YES VECTOR=LOAD EAXES=RST
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B20A  CONCRETE MATERIAL CURVES IN COMPRESSION, SZZ=SYY
*
SET  DIAGRAM=GRID PLOTORIENTATION=PORTRAIT
*
DATABASE  CREATE
*
AXIS  1  VMIN=-0.004 VMAX=0.002 LABEL='STRAIN-ZZ AND STRAIN-XX'
AXIS  2  VMIN=-0.004 VMAX=0.002 LABEL='STRAIN-ZZ AND STRAIN-YY'
AXIS  3  VMIN=-400   VMAX=0.    LABEL='STRESS-ZZ'
AXIS  4  VMIN=-0.006 VMAX=0.    LABEL='VOLUMETRIC STRAIN'
*
USERCURVE  1  SORT=NO / READ  B20UNIE3.DAT
USERCURVE  2  SORT=NO / READ  B20UNIE1.DAT
USERCURVE  3  SORT=NO / READ  B20UNIVO.DAT
USERCURVE  4  SORT=NO / READ  B20EX1E1.DAT
USERCURVE  6  SORT=NO / READ  B20EX1E3.DAT
USERCURVE  7  SORT=NO / READ  B20EX1VO.DAT
*
EXYPLOT  EL=1 POINT=1 XKIND=EZZ YKIND=SZZ XAXIS=1 YAXIS=3
EXYPLOT  EL=1 POINT=1 XKIND=EXX YKIND=SZZ XAXIS=-1 YAXIS=-3
SUBFRAME=OLD
PLOT  USERCURVE  1 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD
PLOT  USERCURVE  2 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD
PLOT  USERCURVE  4 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD SYMBOL=-3
PLOT  USERCURVE  6 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD SYMBOL=-4
*
EXYPLOT  EL=1 POINT=1 XKIND=EZZ YKIND=SZZ XAXIS=2 YAXIS=3
EXYPLOT  EL=1 POINT=1 XKIND=EYY YKIND=SZZ XAXIS=-2 YAXIS=-3
SUBFRAME=OLD
PLOT  USERCURVE  1 XAXIS=-2 YAXIS=-3 SUBFRAME=OLD
PLOT  USERCURVE  2 XAXIS=-2 YAXIS=-3 SUBFRAME=OLD
PLOT  USERCURVE  4 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD SYMBOL=-3
PLOT  USERCURVE  4 XAXIS=-2 YAXIS=-3 SUBFRAME=OLD SYMBOL=-5
*
EPOINT  NAME=P1 ELEMENT=1 POINT=1
EVARIABLE  NAME=EX TYPE=SOLID KIND=EXX
EVARIABLE  NAME=EY TYPE=SOLID KIND=EYY
EVARIABLE  NAME=EZ TYPE=SOLID KIND=EZZ
EVARIABLE  NAME=SZ TYPE=SOLID KIND=SZZ
RESULTANT  NAME=VOLUME STRING='EX + EY + EZ'
RESULTANT  NAME=STRESS STRING='SZ'
*
RXYPLOT  XPOINTNAME=P1 XRESULTANTNAME=VOLUME YPOINTNAME=P1,
         YRESULTANTNAME=STRESS XAXIS=4 YAXIS=3 SUBFRAME=NEXT
PLOT  USERCURVE  3 XAXIS=-4 YAXIS=-3 SUBFRAME=OLD
PLOT  USERCURVE  7 XAXIS=-4 YAXIS=-3 SUBFRAME=OLD SYMBOL=-5
*
END
```

SOLVIA-PRE input

```
HEADING  'B20B  CONCRETE MATERIAL CURVES IN COMPRESSION,
 0.52*SZZ=SYY'
*
DATABASE  CREATE
MASTER  IDOF=000111 NSTEP=100 DT=1.
AUTOMATIC-ITERATION  NODE=1 DIRECTION=3 DISPLACEMENTS=-5.E-4,
                     DISPMAX=0.072 CONTINUATION=YES ALFA=0.8
TOLERANCES  TYPE=F RNORM=1 RTOL=0.001
*
COORDINATES
 1  0 10 20  /  2  0 0 20  /  3  5 0 20  /  4  5 10 20
 5  0 10  0  /  6  0 0  0  /  7  5 0  0  /  8  5 10  0
*
MATERIAL  1  CONCRETE E0=325000 NU=0.2 SIGMAT=28.35,
             SIGMAC=-315 EPSC=-0.0021 SIGMAU=-280 EPSU=-0.0031,
             BETA=0.5 KAPPA=15. STIFAC=0.0001 SHEFAC=0.5,
             SP311=1.0 SP312=1.24 SP313=1.15
*
EGROUP  1  SOLID
ENODES
 1  1 2 3 4 5 6 7 8
LOADS  ELEMENT
 1  R 0.52 0.52
 1  T 1 1
*
FIXBOUNDARIES  1  /  1 2 5 6
FIXBOUNDARIES  2  /  2 3 6 7
FIXBOUNDARIES  3  /  5 6 7 8
*
VIEW  ID=1  XVIEW=1 YVIEW=1 ZVIEW=0.5
SET  VIEW=1 PLOTORIENTATION=PORTRAIT NSYMBOLS=YES BCODE=ALL
MESH  NNUMBERS=YES VECTOR=LOAD EAXES=RST
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B20B  CONCRETE MATERIAL CURVES IN COMPRESSION, 0.52*SZZ=SYY
*
DATABASE  CREATE
SET  DIAGRAM=GRID PLOTORIENTATION=PORTRAIT
*
AXIS  1  VMIN=-0.004 VMAX=0.002 LABEL='STRAIN-ZZ AND STRAIN-XX'
AXIS  2  VMIN=-0.004 VMAX=0.002 LABEL='STRAIN-ZZ AND STRAIN-YY'
AXIS  3  VMIN=-400   VMAX=0.    LABEL='STRESS-ZZ'
AXIS  4  VMIN=-0.006 VMAX=0.    LABEL='VOLUMETRIC STRAIN'
*
USERCURVE  1  SORT=NO / READ  B20UNIE3.DAT
USERCURVE  2  SORT=NO / READ  B20UNIE1.DAT
USERCURVE  3  SORT=NO / READ  B20UNIVO.DAT
USERCURVE  4  SORT=NO / READ  B20EX5E1.DAT
USERCURVE  5  SORT=NO / READ  B20EX5E2.DAT
USERCURVE  6  SORT=NO / READ  B20EX5E3.DAT
USERCURVE  7  SORT=NO / READ  B20EX5VO.DAT
*
EXYPLOT  EL=1 POINT=1 XKIND=EZZ YKIND=SZZ XAXIS=1 YAXIS=3
EXYPLOT  EL=1 POINT=1 XKIND=EXX YKIND=SZZ XAXIS=-1 YAXIS=-3 SUBF=OLD
PLOT  USERCURVE  1 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD
PLOT  USERCURVE  2 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD
PLOT  USERCURVE  4 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD SYMBOL=-3
PLOT  USERCURVE  6 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD SYMBOL=-4
*
EXYPLOT  EL=1 POINT=1 XKIND=EZZ YKIND=SZZ XAXIS=2 YAXIS=3
EXYPLOT  EL=1 POINT=1 XKIND=EYY YKIND=SZZ XAXIS=-2 YAXIS=-3 SUBF=OLD
PLOT  USERCURVE  1 XAXIS=-2 YAXIS=-3 SUBFRAME=OLD
PLOT  USERCURVE  2 XAXIS=-2 YAXIS=-3 SUBFRAME=OLD
PLOT  USERCURVE  4 XAXIS=-2 YAXIS=-3 SUBFRAME=OLD SYMBOL=-3
PLOT  USERCURVE  5 XAXIS=-2 YAXIS=-3 SUBFRAME=OLD SYMBOL=-4
*
EPOINT  NAME=P1 ELEMENT=1 POINT=1
EVARIABLE  NAME=EX TYPE=SOLID KIND=EXX
EVARIABLE  NAME=EY TYPE=SOLID KIND=EYY
EVARIABLE  NAME=EZ TYPE=SOLID KIND=EZZ
EVARIABLE  NAME=SZ TYPE=SOLID KIND=SZZ
RESULTANT  NAME=VOLUME STRING='EX + EY + EZ'
RESULTANT  NAME=STRESS STRING='SZ'
*
RXYPLOT  XPOINTNAME=P1 XRESULTANTNAME=VOLUME YPOINTNAME=P1,
         YRESULTANTNAME=STRESS XAXIS=4 YAXIS=3 SUBFRAME=NEXT
PLOT  USERCURVE  3 XAXIS=-4 YAXIS=-3 SUBFRAME=OLD
PLOT  USERCURVE  7 XAXIS=-4 YAXIS=-3 SUBFRAME=OLD SYMBOL=-5
*
END
```

SOLVIA-PRE input

```
HEADING  'B20C  CONCRETE MATERIAL CURVES, COMPRESSION-TENSION,
 -0.103*SZZ=SYY'
*
DATABASE  CREATE
MASTER  IDOF=000111 NSTEP=100 DT=1.
AUTOMATIC-ITERATION NODE=1 DIRECTION=3 DISPLACEMENTS=-5.E-4,
                    DISPMAX=0.072 CONTINUATION=YES ALFA=0.8
TOLERANCES  TYPE=F RNORM=1 RTOL=0.0001
*
*
COORDINATES
 1  0 10 20  /  2  0 0 20  /  3  5 0 20  /  4  5 10 20
 5  0 10  0  /  6  0 0  0  /  7  5 0  0  /  8  5 10  0
*
MATERIAL  1  CONCRETE E0=325000 NU=0.2 SIGMAT=28.35,
             SIGMAC=-315 EPSC=-0.0021 SIGMAU=-280 EPSU=-0.0031,
             BETA=0.5 KAPPA=15. STIFAC=0.0001 SHEFAC=0.5,
             SP311=1.0 SP312=1.24 SP313=1.15
*
EGROUP  1  SOLID
ENODES
 1  1 2 3 4 5 6 7 8
LOADS  ELEMENT
 1  R -0.103 -0.103 -0.103 -0.103
 1  T  1.  1.  1.  1.
*
FIXBOUNDARIES  1  /  1 2 5 6
FIXBOUNDARIES  2  /  2 3 6 7
FIXBOUNDARIES  3  /  5 6 7 8
*
VIEW  ID=1  XVIEW=1 YVIEW=1 ZVIEW=0.5
SET PLOTORIENTATION=PORTRAIT VIEW=1 NSYMBOLS=YES BCODE=ALL
MESH NNUMBERS=YES VECTOR=LOAD EAXES=RST
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B20C  CONCRETE MATERIAL CURVES IN COMPRESSION, -0.103*SZZ=SYY
*
DATABASE  CREATE
WRITE 'B20C.LIS'
ELIST SELECT=CONCRETE
*
AXIS  1  VMIN=-0.004 VMAX=0.002 LABEL='STRAIN-ZZ AND STRAIN-XX'
AXIS  2  VMIN=-0.004 VMAX=0.002 LABEL='STRAIN-ZZ AND STRAIN-YY'
AXIS  3  VMIN=-400   VMAX=0.    LABEL='STRESS-ZZ'
*
USERCURVE  1  SORT=NO / READ  B20UNIE3.DAT
USERCURVE  2  SORT=NO / READ  B20UNIE1.DAT
USERCURVE  4  SORT=NO / READ  B20EXTE1.DAT
USERCURVE  5  SORT=NO / READ  B20EXTE2.DAT
USERCURVE  6  SORT=NO / READ  B20EXTE3.DAT
*
SET  DIAGRAM=GRID PLOTORIENTATION=PORTRAIT
EXYPLOT  EL=1 POINT=1 XKIND=EZZ YKIND=SZZ XAXIS=1 YAXIS=3
EXYPLOT  EL=1 POINT=1 XKIND=EXX YKIND=SZZ XAXIS=-1 YAXIS=-3 SUBF=OLD
PLOT  USERCURVE  1 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD
PLOT  USERCURVE  2 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD
PLOT  USERCURVE  4 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD SYMBOL=-3
PLOT  USERCURVE  6 XAXIS=-1 YAXIS=-3 SUBFRAME=OLD SYMBOL=-4
*
EXYPLOT  EL=1 POINT=1 XKIND=EZZ YKIND=SZZ XAXIS=2 YAXIS=3
EXYPLOT  EL=1 POINT=1 XKIND=EYY YKIND=SZZ XAXIS=-2 YAXIS=-3 SUBF=OLD
PLOT  USERCURVE  1 XAXIS=-2 YAXIS=-3 SUBFRAME=OLD
PLOT  USERCURVE  2 XAXIS=-2 YAXIS=-3 SUBFRAME=OLD
PLOT  USERCURVE  4 XAXIS=-2 YAXIS=-3 SUBFRAME=OLD SYMBOL=-3
PLOT  USERCURVE  5 XAXIS=-2 YAXIS=-3 SUBFRAME=OLD SYMBOL=-4
*
END
```

EXAMPLE B21

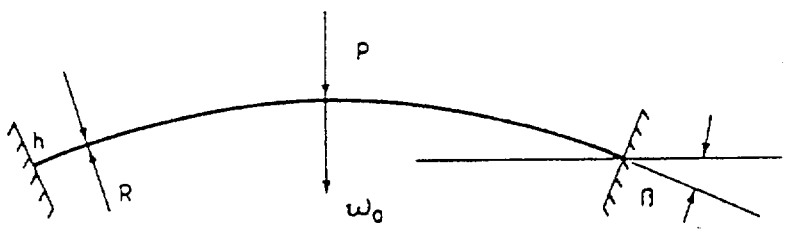
LARGE DEFLECTION ANALYSIS OF A SHALLOW ARCH

Objective

To verify the large displacement behaviour of the ISOBEAM beam element and demonstrate the restart option in static analysis.

Physical Problem

A shallow arch subjected to a concentrated apex load, as shown in the figure below, is considered.



$R = 133.144$ in
$h = 0.1875$ in
$b = 1.00$ in (width)
$\beta = 7.3397°$
$E = 10^7$ psi
$\nu = 0.2$

Finite Element Model

The finite element model is shown in the figure on page B21.2. Using symmetry conditions, four cubic ISOBEAM elements are used to model one half of the structure. The apex load is applied in seventeen equal load steps and the restart option is employed after the first eight steps. Stiffness reformations and BFGS equilibrium iterations are employed for each load step.

Solution Results

The input data on page B21.4 is used for the first eight load steps and the input data on page B21.5 is used in the restart finite element analysis.

The predicted apex displacement in the analysis is in good agreement with results reported in [1].

The top figure on page B21.3 shows the deformed mesh at load step 8 and load step 17. The left bottom figure on page B21.3 shows the mid-span deflection history and the reaction moment history at the build-in end.

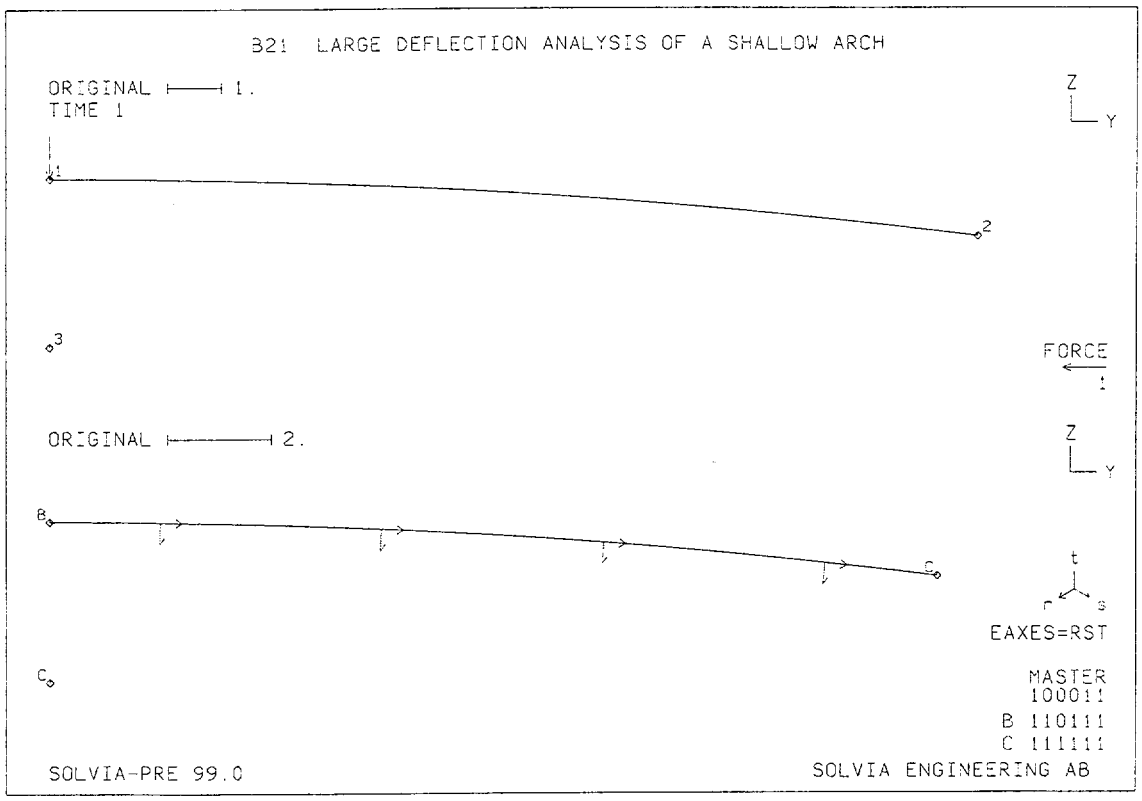
The right bottom figure shows the history of the element axial stress closest to the mid-span and the history of the element shear stress closest to the built-in end.

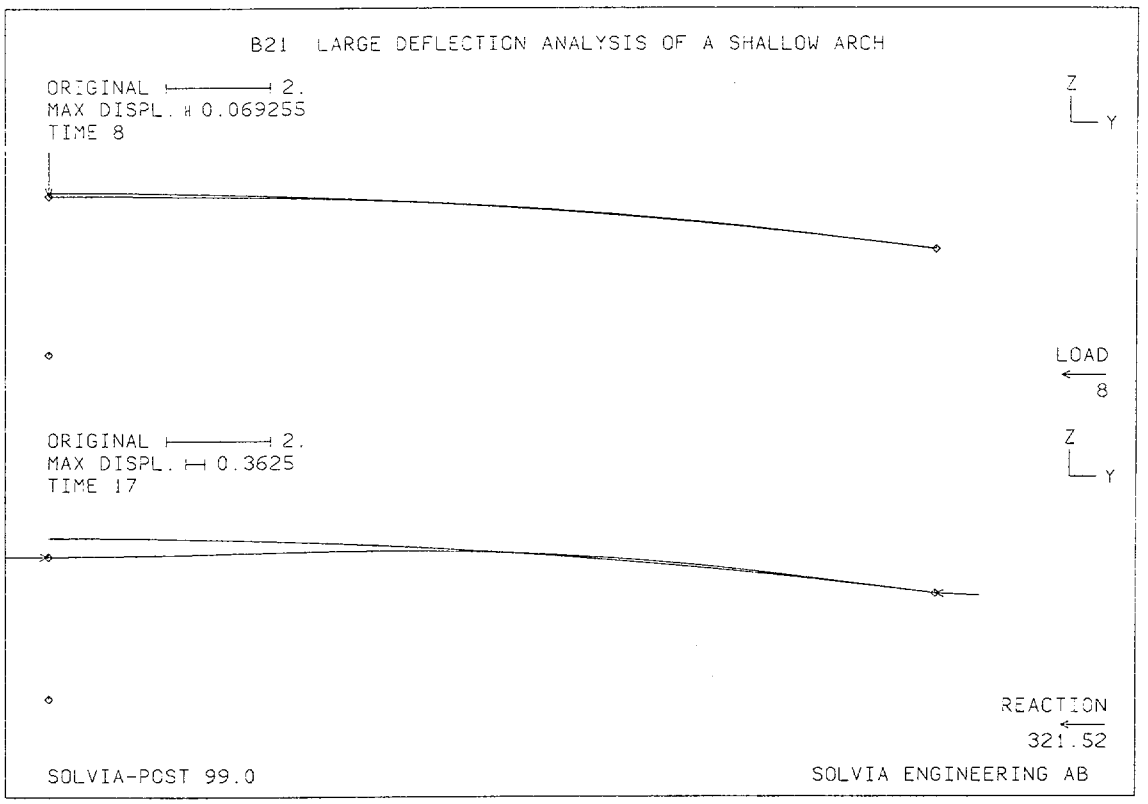
User Hints

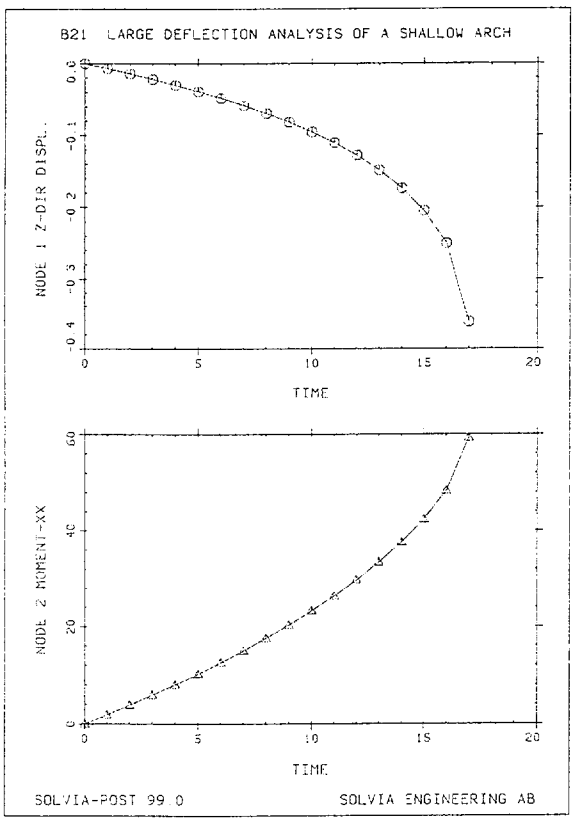
- Note that few changes are required in the input data for a restart solution and that the SOLVIA restart file, SOLVIA08.DAT, is always saved after an analysis, and available for a restart analysis.
- Results at all time steps are available in the SOLVIA-POST database since DATABASE RESTART causes the restart results to be added to the SOLVIA-POST database with results from the first run.

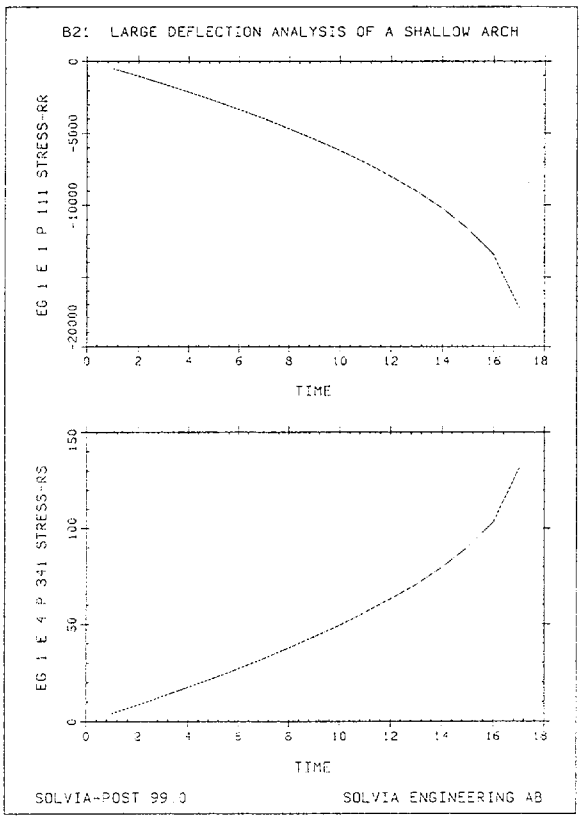
Reference

[1] Bathe, K.J., and Bolourchi, S., "Large Displacement Analysis of Three-Dimensional Beam Structures", Int. J. Num. Meth. Eng., Vol. 14, 1979, pp. 961-986.




B21 LARGE DEFLECTION ANALYSIS OF A SHALLOW ARCH
ORIGINAL 2.
MAX DISPL. 0.069255
TIME 8
Z
Y
LOAD
8
ORIGINAL 2.
MAX DISPL. 0.3625
TIME 17
Z
Y
REACTION
321.52
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B21 LARGE DEFLECTION ANALYSIS OF A SHALLOW ARCH
NODE 1 Z-DIR DISPL.
TIME
NODE 2 MOMENT-XX
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B21 LARGE DEFLECTION ANALYSIS OF A SHALLOW ARCH
EG 1 E 1 P 111 STRESS-RR
TIME
EG 1 E 4 P 341 STRESS-RS
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B21  LARGE DEFLECTION ANALYSIS OF A SHALLOW ARCH'
*
*   First analysis
*
DATABASE  CREATE
MASTER  IDOF=100011 MODEX=EXECUTE NSTEP=8 TSTART=0
KINEMATICS  DISPLACEMENT=LARGE
ITERATION  METHOD=BFGS
TOLERANCES  ETOL=1.E-6
*
TIMEFUNCTION  1
 0. 0.  /  20.  20.
*
SYSTEM  1 CYLINDRICAL
COORDINATES  /  ENTRIES  NODE  R  THETA
 1  133.114 90.  /  2  133.114 82.6603
 3  130.    90.
*
MATERIAL  1  ELASTIC E=1.E7 NU=0.2
*
EGROUP  1  ISOBEAM
SECTION  1  SDIM=0.1875 TDIM=1.
GLINE  N1=1 N2=2 AUX=3 EL=4 NODES=4 SYSTEM=1
*
FIXBOUNDARIES  24   /  1
FIXBOUNDARIES  234  /  2 3
LOADS  CONCENTRATED
 1 3 -1.
*
SET  VIEW=X NSYMBOLS=MYNODES SMOOTHNESS=YES
MESH  NNUMBERS=MYNODES VECTOR=LOAD SUBFRAME=12
MESH  EAXES=RST BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B21  LARGE DEFLECTION ANALYSIS OF A SHALLOW ARCH
*
*   First analysis
*
DATABASE  CREATE
END
```

SOLVIA-PRE input

```
HEADING  'B21A  LARGE DEFLECTION ANALYSIS OF A SHALLOW ARCH'
*
*   Restart analysis
*
DATABASE  OPEN
MASTER  MODEX=RESTART NSTEP=9 TSTART=8
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B21A  LARGE DEFLECTION ANALYSIS OF A SHALLOW ARCH
*
*   Restart analysis
*
DATABASE  RESTART
WRITE  FILENAME='b21a.lis'
*
SET  NSYMBOLS=MYNODES VIEW=X SMOOTHNESS=YES
SUBFRAME  12
MESH  VECTOR=LOAD ORIGINAL=YES TIME=8
MESH  VECTOR=REACTION ORIGINAL=YES
*
SET  PLOTORIENTATION=PORTRAIT
SUBFRAME  12
NHISTORY  NODE=1 DIRECTION=3 OUTPUT=ALL SYMBOL=1
NHISTORY  NODE=2 DIRECTION=4 KIND=REACTION OUTPUT=ALL SYMBOL=2
*
SUBFRAME  12
EHISTORY  ELEMENT=1 POINT=111 KIND=SRR
EHISTORY  ELEMENT=4 POINT=341 KIND=SRS
*
SUMMATION  KIND=REACTION
SUMMATION  KIND=LOAD
END
```

EXAMPLE B22

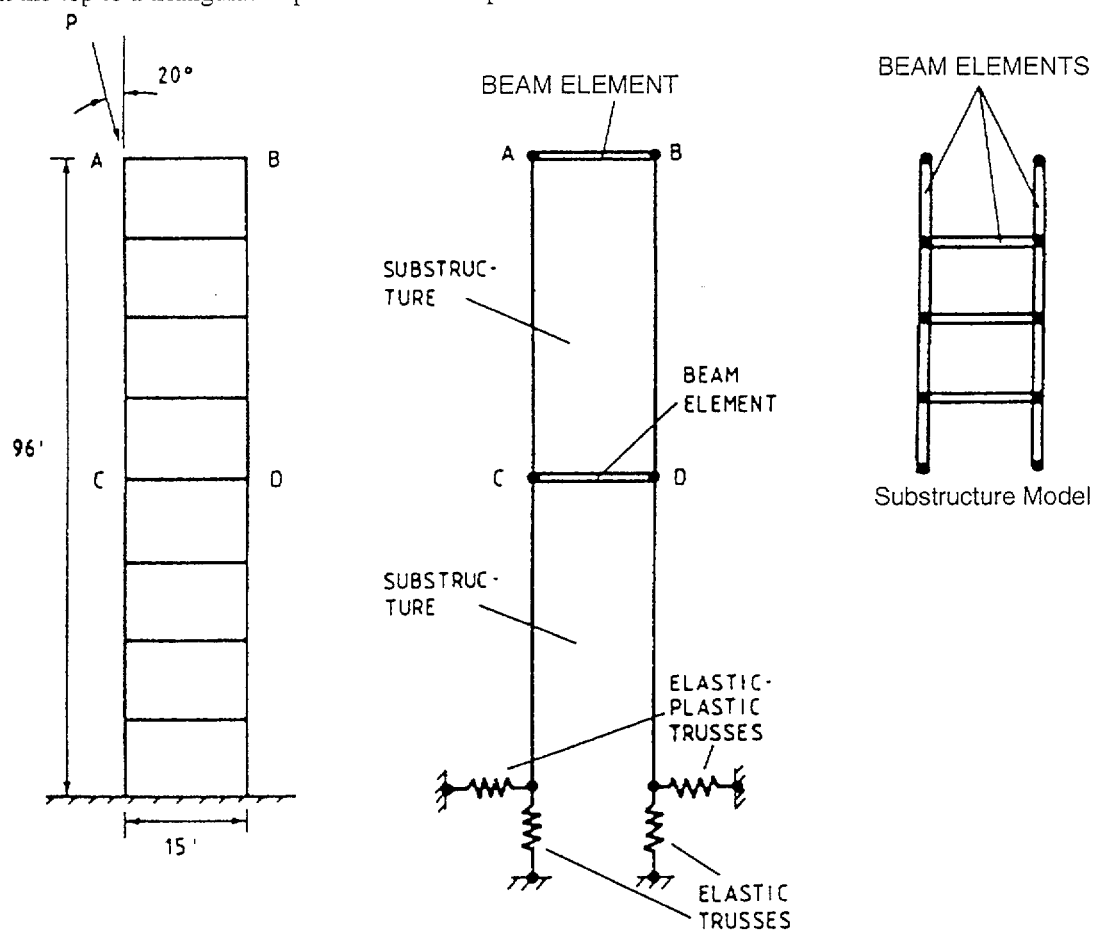
EIGHT-STORY BUILDING SUBJECTED TO IMPACT LOAD

Objective

To demonstrate the capability of performing nonlinear dynamic analysis using the substructure option.

Physical Problem

The eight-story building shown in figure below is analyzed for its dynamic response when subjected at the top to a triangular impact load of 30 kips-sec over 0.2 sec.



Main Structure Model

Beam elements:	$E = 1.09 \cdot 10^9$ psf $A = 1.00$ ft^2	$\nu = 0.2$ $\rho = 5.15$ slugs/ft^3	$I = 4/3$ ft^4	
Elastic trusses:	$E = 1.00 \cdot 10^{10}$ psf	$\rho = 0.$ slugs/ft^3	$A = 10$ ft^2	$L = 10$ ft
Elastic-plastic trusses:	$E = 1.00 \cdot 10^{10}$ psf $A = 10$ ft^2	$\sigma_y = 6000$ psf $L = 10$ ft	$ET = 0.00$ psf $\rho = 0.$ slugs/ft^3	

Finite Element Model

The finite element model is shown on page B22.3. The building is modeled with BEAM elements in the master structure and the substructures. The foundation is modeled with elastic TRUSS elements in the vertical direction and with elastic-plastic TRUSS elements in the horizontal direction. The dynamic response is evaluated using the trapezoidal rule (the Newmark method with $\alpha = 0.25$ and $\delta = 0.50$) with a time step $\Delta t = 0.02$ sec. A lumped mass matrix is used in the analysis. Full Newton equilibrium iterations are employed in every solution time step.

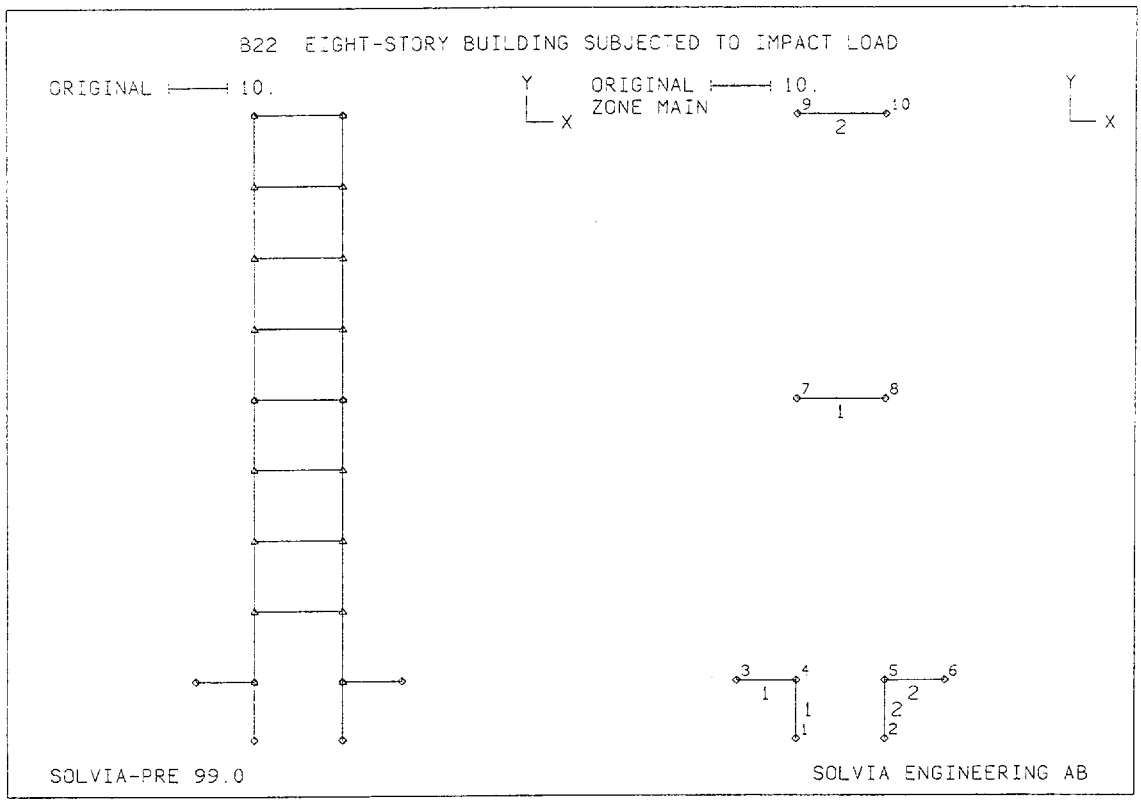
Solution Results

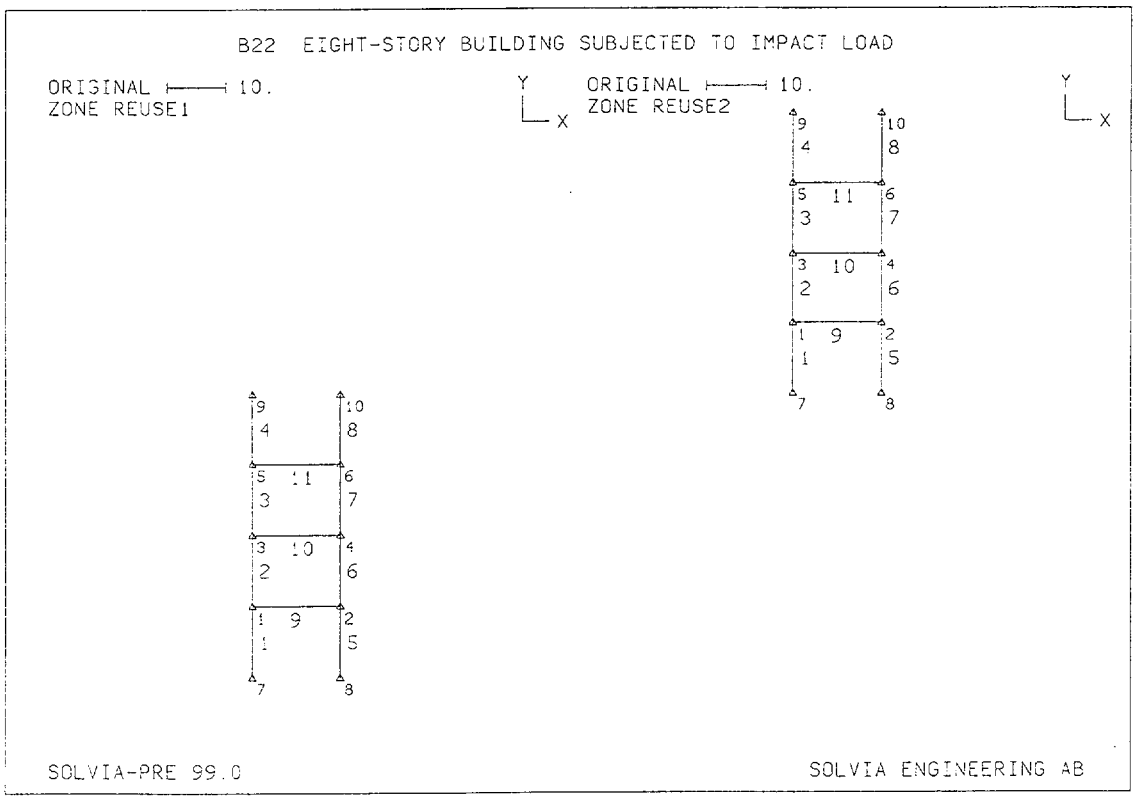
The input data on pages B22.5 and B22.6 is used in the finite element analysis. The bottom figure on page B22.4 shows the time history of the horizontal displacement and acceleration at points D and B, see figure on page B22.1, as displayed by SOLVIA-POST. The deformed finite element model at time t = 0.4 sec is shown in the top figure on page B22.4.

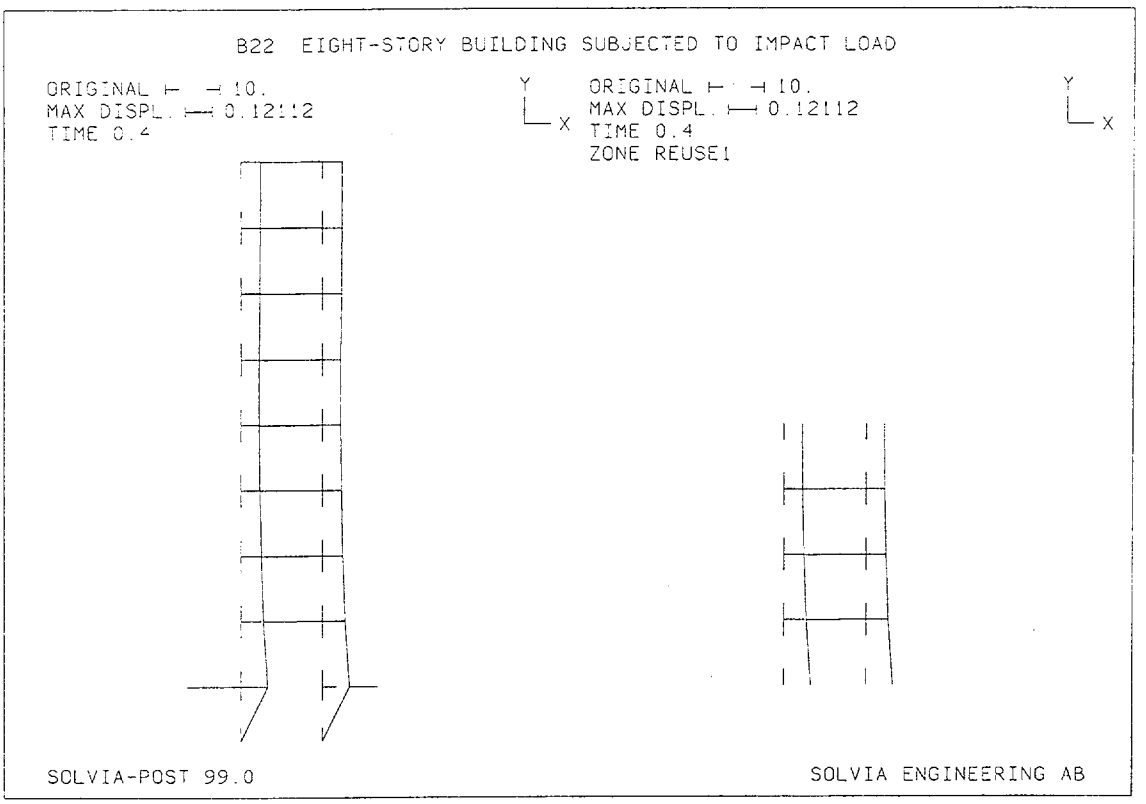
A reference solution is obtained by modeling the building as a main structure only, see figures on page B22.7 and input data on pages B22.8 and B22.9. The results of the reference solution are shown on page B22.7 and complete agreement with the substructure results can be observed.

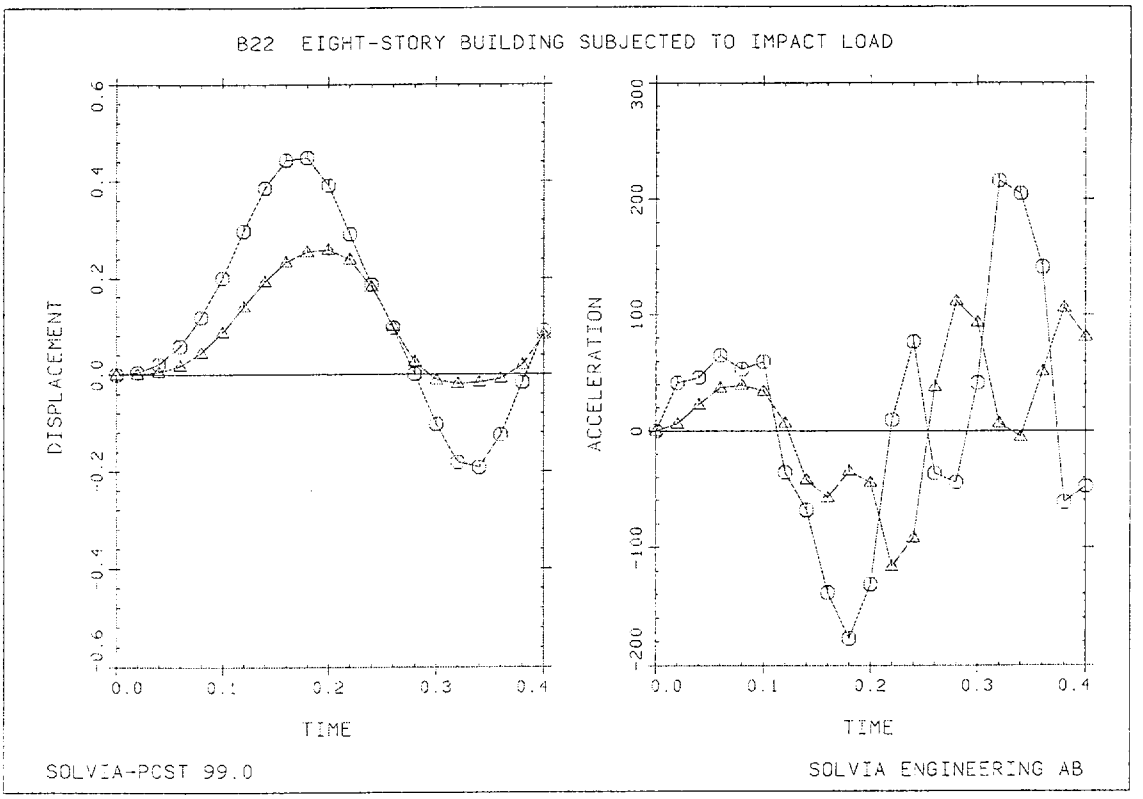
User Hints

- Note that in nonlinear dynamic analysis the convergence tolerance in the equilibrium iterations must be sufficiently tight so that the accumulation of errors in the solution is small (compare Example B2).


B22 EIGHT-STORY BUILDING SUBJECTED TO IMPACT LOAD
ORIGINAL 10.
Y
X
ORIGINAL 10.
ZONE MAIN
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


B22 EIGHT-STORY BUILDING SUBJECTED TO IMPACT LOAD
ORIGINAL 10.
ZONE REUSE1
Y
X
ORIGINAL 10.
ZONE REUSE2
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


B22 EIGHT-STORY BUILDING SUBJECTED TO IMPACT LOAD
ORIGINAL
MAX DISPL.
TIME 0.4
Y
X
ORIGINAL
MAX DISPL.
TIME 0.4
ZONE REUSE1
Y
X
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

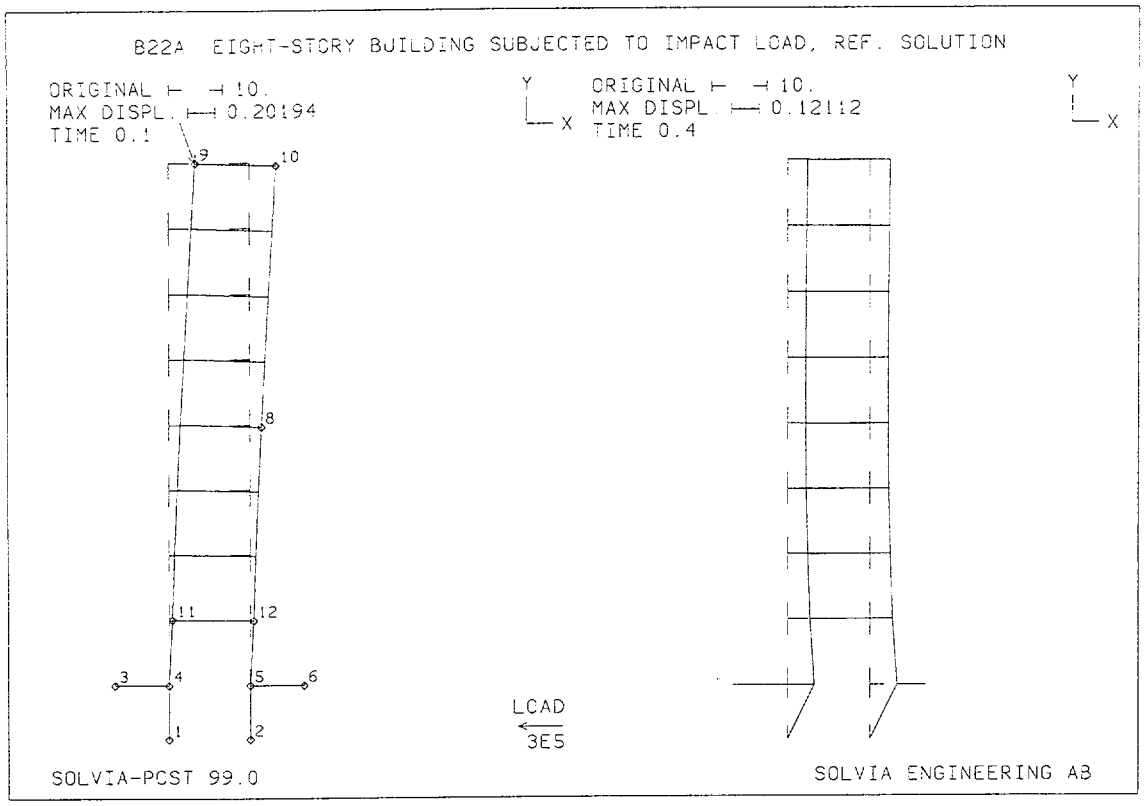

B22 EIGHT-STORY BUILDING SUBJECTED TO IMPACT LOAD
DISPLACEMENT
0.6
0.4
0.2
0.0
-0.2
-0.4
-0.6
0.0
0.1
0.2
0.3
0.4
TIME
ACCELERATION
300
200
100
0
-100
-200
0.0
0.1
0.2
0.3
0.4
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

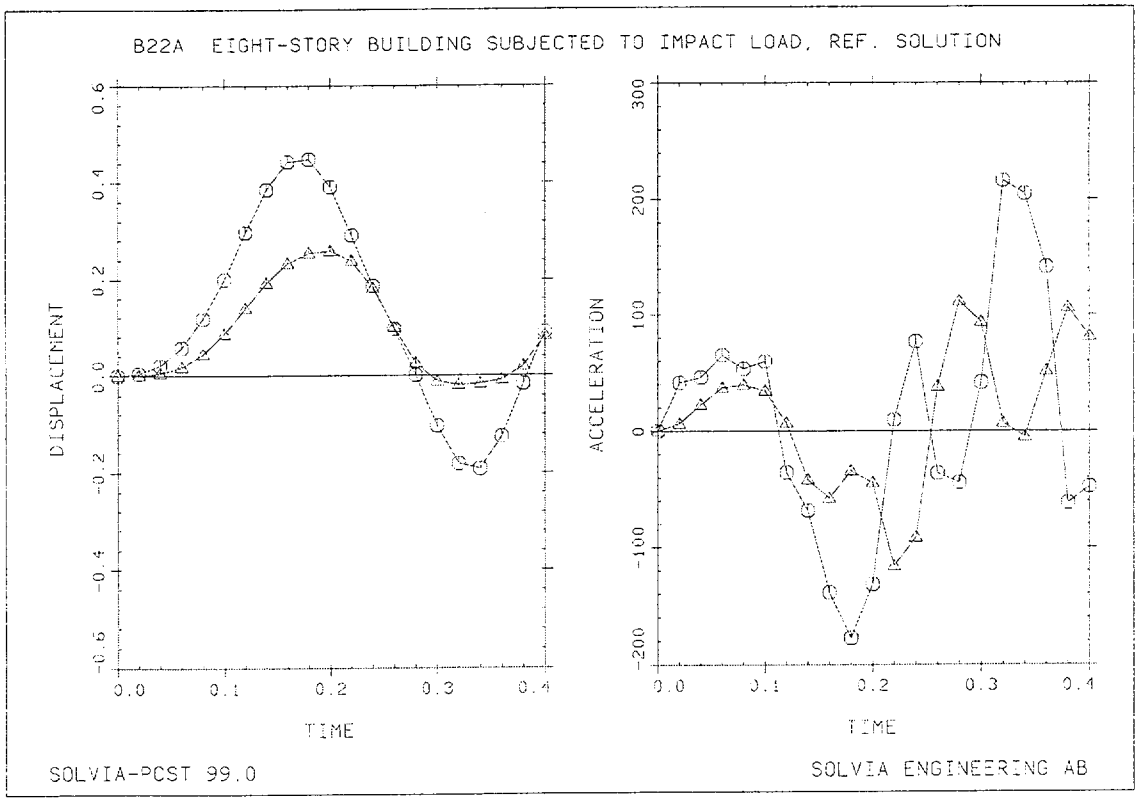
SOLVIA-PRE input

```
HEADING  'B22  EIGHT-STORY BUILDING SUBJECTED TO IMPACT LOAD'
*
DATABASE  CREATE
MASTER  IDOF=001110 NSTEP=20 DT=0.020
ANALYSIS   TYPE=DYNAMIC MASSMATRIX=LUMPED METHOD=NEWMARK
ITERATION  METHOD=FULL-NEWTON LINE-SEARCH=YES
TOLERANCES  ETOL=1.E-6
TIMEFUNCTION  1  /  0. 0.  /  0.1 3.E5  /  0.2 0.  /  1. 0.
*
COORDINATES
 1   0 -10  /  2  15 -10  /  3 -10  0  /  4  0  0  /  5  15
 6  25  0   /  7   0  48  /  8  15 48  /  9  0 96  / 10  15 96
*
MATERIAL  1  ELASTIC  E=1.E10  NU=0.
MATERIAL  2  ELASTIC  E=1.09E9 NU=0.2 DENSITY=5.15
MATERIAL  3  PLASTIC  ISOTROPIC  E=1.E10 YIELD=6000. ET=0.
*
EGROUP  1  TRUSS  MATERIAL=1
ENODES  /  1  1 4  /  2  2 5
EDATA   /  1  10.
*
EGROUP  2  TRUSS  MATERIAL=3
ENODES  /  1  3 4  /  2  5 6
EDATA   /  1  10.
*
EGROUP  3  BEAM  MATERIAL=2 RESULTS=FORCES
SECTION  1  GENERAL RINERTIA=1 SINERTIA=1 TINERTIA=1.333333 AREA=1
ENODES  /  1  1 7 8  /  2  1 9 10
FIXBOUNDARIES  /  1 2 3 6
LOADS  CONCENTRATED  /  9 1  0.342020  /  9 2 -0.939693
*
SUBSTRUCTURE  1
COORDINATES
 1   0 12  /  2  15 12  /  3   0 24  /  4  15 24  /  5   0 36
 6  15 36  /  7   0  0  /  8  15  0  /  9   0 48  / 10  15 48
*
EGROUP  1  BEAM  MATERIAL=2 RESULTS=FORCES
SECTION  1  GENERAL RINERTIA=1 SINERTIA=1 TINERTIA=1.333333 AREA=1
ENODES   / 1 8 7 1 / 2 8 1 3  / 3 8 3 5 /  4 8 5 9 /  5 7 8 2
 6 7 2 4 / 7 7 4 6 / 8 7 6 10 / 9 8 1 2 / 10 8 3 4 / 11 8 5 6
*
REUSE   1  LREUSE=0  4 5 7 8
REUSE   2  LREUSE=0  7 8 9 10
*
SET  VIEW=Z NSYMBOLS=YES
MESH  SUBFRAME=21
SET  ENUMBERS=YES NNUMBERS=YES GSCALE=OLD
MESH  ZONENAME=MAIN
SUBSTRUCTURE  1
MESH  ZONENAME=REUSE1 SUBFRAME=21
MESH  ZONENAME=REUSE2
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B22  EIGHT-STORY BUILDING SUBJECTED TO IMPACT LOAD
*
DATABASE  CREATE
WRITE  FILENAME='b22.lis'
*
SET  VIEW=Z
MESH  ORIGINAL=DASHED SUBFRAME=21
SUBSTRUCTURE  1
MESH  ZONENAME=REUSE1 GSCALE=OLD ORIGINAL=DASHED
*
AXIS  1 VMIN=-200 VMAX=300 LABEL='ACCELERATION'
AXIS  2 VMIN=-0.6 VMAX=0.6 LABEL='DISPLACEMENT'
AXIS  3 VMIN=0    VMAX=0.4 LABEL='TIME'
SUBSTRUCTURE 0
NHISTORY  NODE=10 DIRECTION=1 KIND=DISPLACEMENT YAXIS=2 XAXIS=3,
          SYMBOL=1 OUTPUT=ALL SUBFRAME=21
SUBSTRUCTURE  1
REUSE  1
NHISTORY  NODE=10 DIRECTION=1 KIND=DISPLACEMENT YAXIS=-2 XAXIS=-3,
          SYMBOL=2 OUTPUT=ALL SUBFRAME=OLD
SUBSTRUCTURE  0
NHISTORY  NODE=10 DIRECTION=1 KIND=ACCELERATION YAXIS=1 XAXIS=3,
          SYMBOL=1 OUTPUT=ALL SUBFRAME=NEXT
SUBSTRUCTURE  1
REUSE  1
NHISTORY  NODE=10 DIRECTION=1 KIND=ACCELERATION YAXIS=-1 XAXIS=-3,
          SYMBOL=2 OUTPUT=ALL SUBFRAME=OLD
*
NLIST
END
```


B22A EIGHT-STORY BUILDING SUBJECTED TO IMPACT LOAD, REF. SOLUTION
ORIGINAL 10.
MAX DISPL. 0.20194
TIME 0.1
ORIGINAL 10.
MAX DISPL. 0.12112
TIME 0.4
Y
X
LOAD
3E5
SOLVIA-PCST 99.0
SOLVIA ENGINEERING AB


B22A EIGHT-STORY BUILDING SUBJECTED TO IMPACT LOAD, REF. SOLUTION
DISPLACEMENT
TIME
ACCELERATION
TIME
SOLVIA-PCST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B22A  EIGHT-STORY BUILDING SUBJECTED TO IMPACT LOAD,
 REF. SOLUTION'
*
DATABASE  CREATE
MASTER  IDOF=001110 NSTEP=20 DT=0.020
ANALYSIS   TYPE=DYNAMIC MASSMATRIX=LUMPED METHOD=NEWMARK
ITERATION  METHOD=FULL-NEWTON LINE-SEARCH=YES
TOLERANCES  ETOL=1.E-6
TIMEFUNCTION  1  /  0. 0.  /  0.1 3.E5  /  0.2 0.  /  1. 0.
*
COORDINATES
 1   0 -10  /  2  15 -10  /  3 -10  0  /  4  0  0  /   5  15
 6  25  0   /  8  15 48   /  9   0  96 / 10  15 96  / 11  0 12
12  15 12
*
MATERIAL  1  ELASTIC  E=1.E10  NU=0.
MATERIAL  2  ELASTIC  E=1.09E9 NU=0.2 DENSITY=5.15
MATERIAL  3  PLASTIC  ISOTROPIC  E=1.E10 YIELD=6000. ET=0.
*
EGROUP  1  TRUSS  MATERIAL=1
ENODES  /  1  1 4  /  2  2 5
EDATA   /  1  10.
*
EGROUP  2  TRUSS  MATERIAL=3
ENODES  /  1  3 4  /  2  5 6
EDATA   /  1  10.
*
EGROUP  3  BEAM  MATERIAL=2 RESULTS=FORCES
SECTION  1  GENERAL RINERTIA=1 SINERTIA=1 TINERTIA=1.333333 AREA=1
ENODES  /  1  12 4 11  /  2  4 11 12 / 3 11 12 5
FIXBOUNDARIES  /  1 2 3 6
LOADS  CONCENTRATED  /  9 1  0.342020  /  9 2 -0.939693
*
TRANSLATE EG3 Y=12 COPIES=7
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B22A  EIGHT-STORY BUILDING SUBJECTED TO IMPACT LOAD,
*        REF. SOLUTION
*
DATABASE  CREATE
WRITE  FILENAME='b22a.lis'
*
SET  VIEW=Z
MESH  VECTOR=LOAD NNUMBERS=MYNODES NSYMBOL=MYNODES ,
      ORIGINAL=DASHED TIME=0.1 SUBFRAME=21
MESH  ORIGINAL=DASHED
*
AXIS  1 VMIN=-200 VMAX=300 LABEL='ACCELERATION'
AXIS  2 VMIN=-0.6 VMAX=0.6 LABEL='DISPLACEMENT'
AXIS  3 VMIN=0    VMAX=0.4 LABEL='TIME'
NHISTORY  NODE=10 DIRECTION=1 KIND=DISPLACEMENT YAXIS=2 XAXIS=3,
          SYMBOL=1 OUTPUT=ALL SUBFRAME=21
NHISTORY  NODE=8 DIRECTION=1 KIND=DISPLACEMENT YAXIS=-2 XAXIS=-3,
          SYMBOL=2 OUTPUT=ALL SUBFRAME=OLD
NHISTORY  NODE=10 DIRECTION=1 KIND=ACCELERATION YAXIS=1 XAXIS=3,
          SYMBOL=1 OUTPUT=ALL SUBFRAME=NEXT
NHISTORY  NODE=8 DIRECTION=1 KIND=ACCELERATION YAXIS=-1 XAXIS=-3,
          SYMBOL=2 OUTPUT=ALL SUBFRAME=OLD
*
END
```

EXAMPLE B23

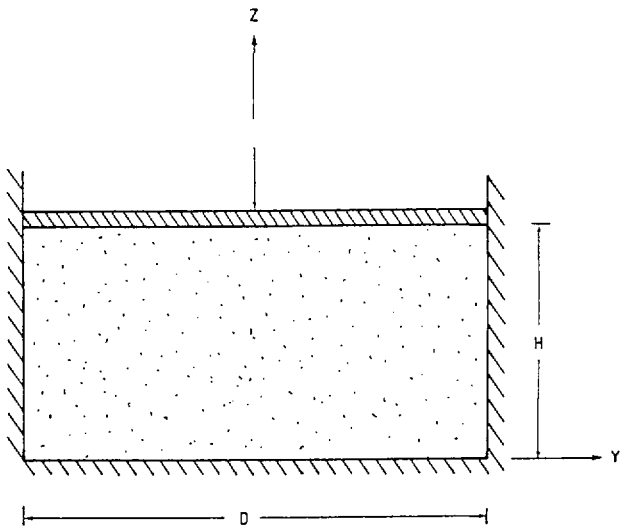
ANALYSIS OF A SAND SPECIMEN UNDER COMPRESSION

Objective

To verify the Drucker-Prager material model when used for the PLANE AXISYMMETRIC element.

Physical Problem

A uniaxial strain test for McCormick Ranch Sand is considered, see figure below.



Young's modulus, E	100.0 ksi
Poisson's ratio, ν	0.25
Yield function parameter, α	0.05
Yield function parameter, k	0.1 ksi
Cap hardening parameter, W	-0.066
Cap hardening parameter, D	-0.78 ksi^{-1}
Tension cut-off limit, T	0.01 ksi
Initial cap position, ${}^{o}I_1^a$	0.0 ksi
D = 1.0 in	
H = 0.5 in	

Finite Element Model

Four 4-node PLANE AXISYMMETRIC elements are used to model the sand specimen, see the figure on page B23.2. The top nodes in the model are subjected to prescribed displacements and the side nodes can slide in the Z-direction only. The bottom nodes are fixed.

Solution Results

The input data is shown on page B23.4. One hundred prescribed displacement steps are used with the default iteration method (BFGS) and with default tolerances.

The calculated longitudinal stress as a function of the longitudinal strain is shown in the top figure on page B23.3. The calculated solution is in good agreement with the experimental data given in [1].

The traced solution path in a Drucker-Prager yield function diagram with axes $-I_1$ and $\sqrt{J_2}$ is shown in the bottom figure on page B23.3. I_1 is the first stress invariant and J_2 is the second deviatoric stress invariant, see also page 9.27 in the SOLVIA-PRE Users Manual.

We note that upon loading the first solution point represents yielding on the cap while the subsequent yielding occurs both on the cap and on the Drucker-Prager yield surface (the vertex). The cap position is moved from $I_1 = 0$ to $I_1 = -2.26$ during yielding.

The unloading portion is first elastic but then follows yielding on the Drucker-Prager yield surface until the tensile limit $I_1 = T$, the tensile cut-off limit, is reached.

User Hints

- For the present example one 4-node element would be satisfactory, since there is no radial displacement variation and only a linear displacement variation in the vertical (Z) direction.
- In SOLVIA, compressive stresses and strains are negative. Therefore, the cap parameters must satisfy the following conditions:

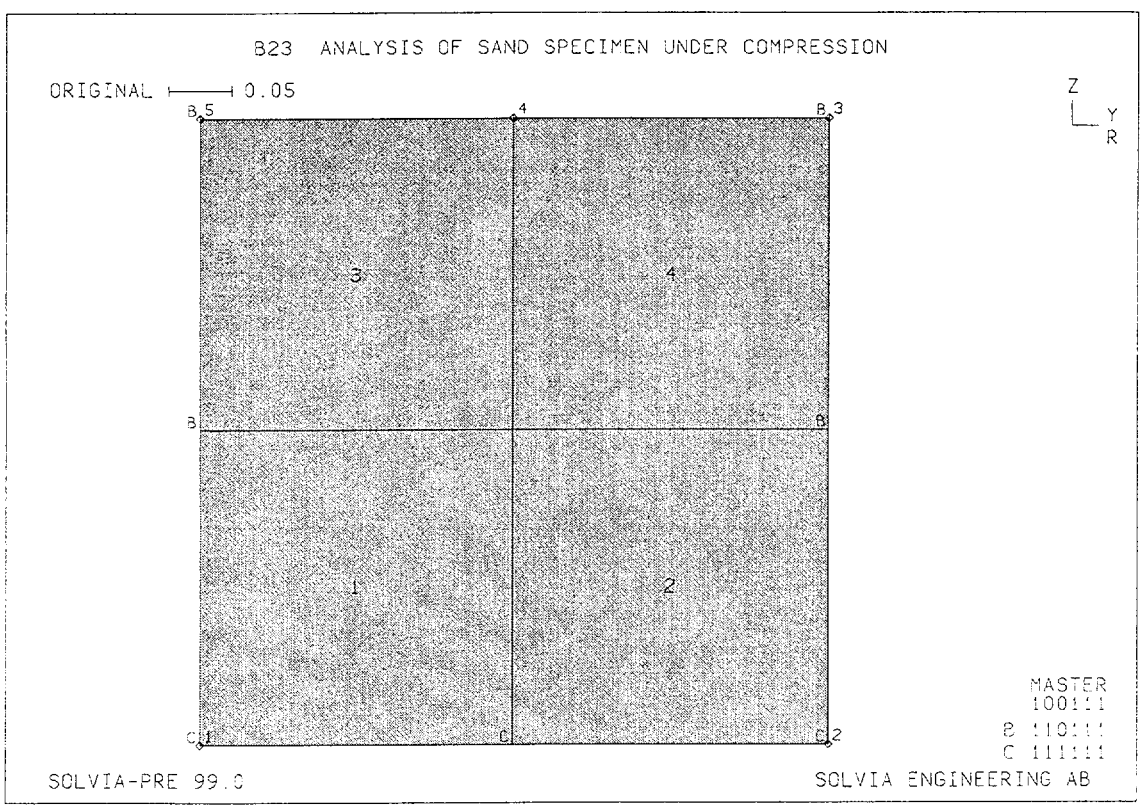
  $$W < 0 \quad D < 0 \quad {}^{o}I_1^a \le 0 \quad T \ge 0 \quad \alpha \ge 0 \quad k > 0$$

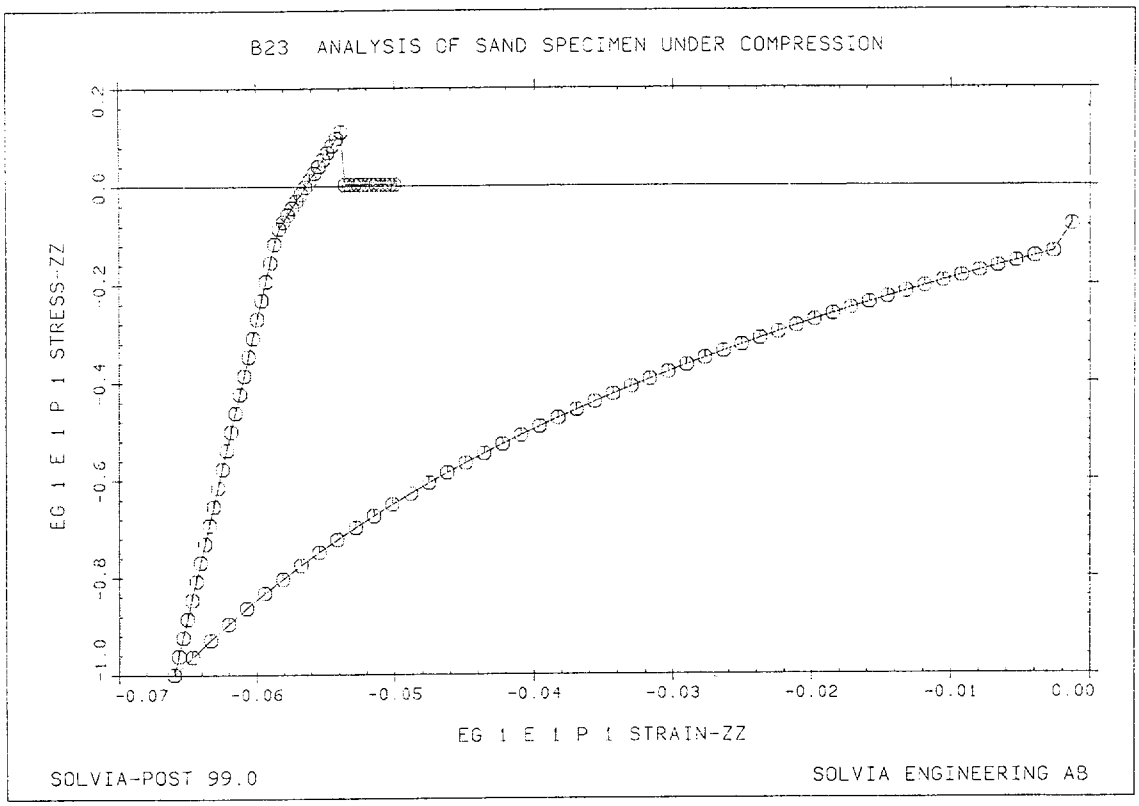
- When $\alpha = 0$ the Drucker-Prager yield criterion is the same as the von Mises yield criterion. For this case the yield function parameter k is the yield stress in shear and $k = \sigma_y / \sqrt{3}$, where σ_y is the yield stress in tension.
- When ${}^{t}I_1 \ge T$, i.e. the first stress invariant (the hydrostatic tension) reaches or exceeds the tension cut-off limit the deviatoric stresses and the normal stresses are set to

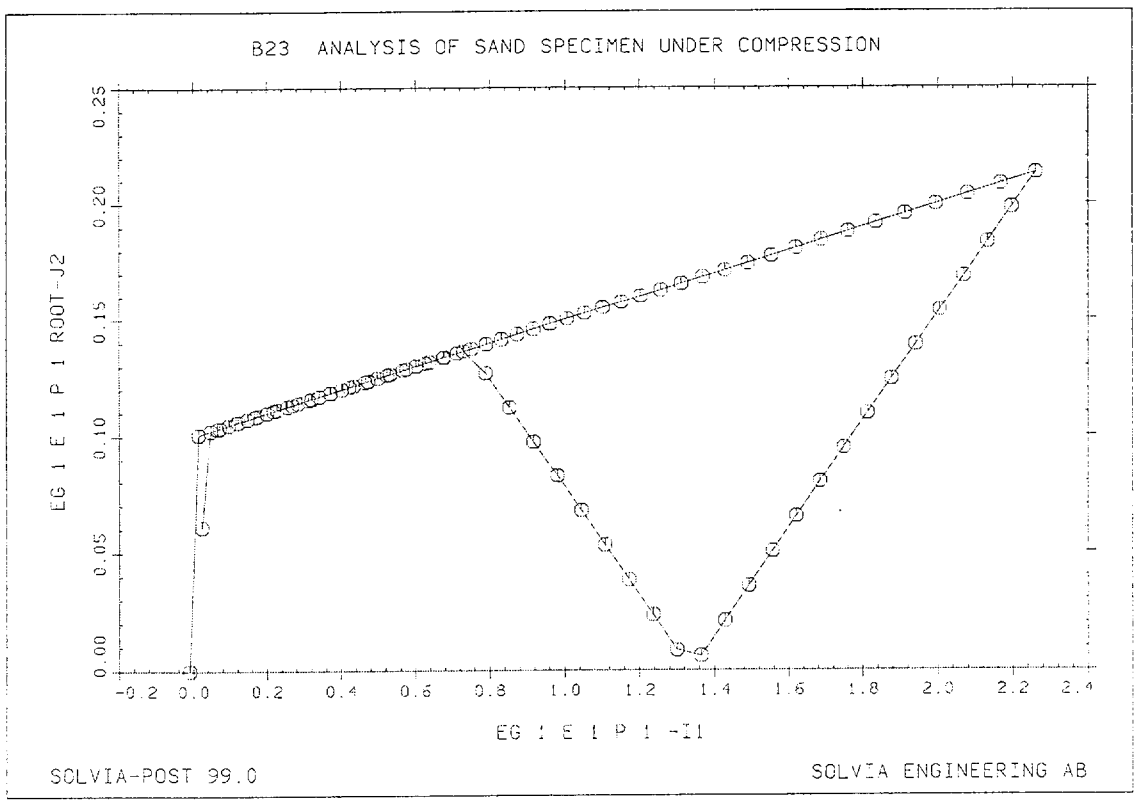
  $${}^{t}s_{ij} = 0 \qquad {}^{t}\sigma_{ij} = \delta_{ij}\, T / 3 \,, \quad \text{respectively.}$$

Reference

[1] DiMaggio, F.L., Sandler, I.S., "Material Model for Granular Soils", J. Eng. Mech. Div. ASCE 97 (EMS), pp. 935-949, 1971.




B23 ANALYSIS OF SAND SPECIMEN UNDER COMPRESSION
EG 1 E 1 P 1 STRESS-ZZ
0.2
0.0
-0.2
-0.4
-0.6
-0.8
-1.0
-0.07
-0.06
-0.05
-0.04
-0.03
-0.02
-0.01
0.00
EG 1 E 1 P 1 STRAIN-ZZ
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B23 ANALYSIS OF SAND SPECIMEN UNDER COMPRESSION
EG 1 E 1 P 1 ROOT-J2
0.25
0.20
0.15
0.10
0.05
0.00
-0.2
0.0
0.2
0.4
0.6
0.8
1.0
1.2
1.4
1.6
1.8
2.0
2.2
2.4
EG 1 E 1 P 1 -I1
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B23  ANALYSIS OF SAND SPECIMEN UNDER COMPRESSION'
*
DATABASE  CREATE
MASTER  IDOF=100111 NSTEP=100 DT=0.2
*
TIMEFUNCTION  1
 0. 0.  /  10. 0.033  /  20. 0.025
*
COORDINATES  /  ENTRIES  NODE  Y  Z
 1  /  2  0.5  /  3  0.5 0.5  TO  5  0. 0.5
*
MATERIAL  1  DRUCKER-PRAGER E=100 NU=0.25 ALFA=0.05 K=0.1,
                            W=-0.066 D=-0.78 T=0.01 I1=0.0
*
EGROUP  1  PLANE AXISYMMETRIC
GSURFACE  1 2 3 5  EL1=2 EL2=2 NODES=4
*
FIXBOUNDARIES  23 INPUT=LINES  /  1 2
FIXBOUNDARIES  2  INPUT=LINES  /  2 3  /  1 5
LOADS  DISPLACEMENT
 3 3 -1.  TO  5 3 -1.
*
MESH  NSYMBOLS=MYNODES NNUMBERS=MYNODES BCODE=ALL ENUMBERS=YES
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B23  ANALYSIS OF SAND SPECIMEN UNDER COMPRESSION
*
DATABASE  CREATE
WRITE  FILENAME='b23.lis'
*
EXYPLOT  ELEMENT=1 POINT=1 XKIND=EZZ YKIND=SZZ SYMBOL=1 OUTPUT=ALL
*
EVARIABLE SXX TYPE=PLANE KIND=SXX
EVARIABLE SYY TYPE=PLANE KIND=SYY
EVARIABLE SZZ TYPE=PLANE KIND=SZZ
EVARIABLE SYZ TYPE=PLANE KIND=SYZ
CONSTANT THREE 3.0
CONSTANT ONE   1.0
CONSTANT EPS   1.E-16
RESULTANT ROOT-J2 'SQRT((SXX*SXX+SYY*SYY+SZZ*SZZ
   -SXX*SYY-SYY*SZZ-SZZ*SXX+THREE*SYZ*SYZ)/THREE+EPS)'
RESULTANT -I1 '-(SXX+SYY+SZZ)'
EPOINT E1P1 EL=1 POINT=1
RXYPLOT XPOINT=E1P1 XRESULTANT=-I1 YPOINT=E1P1 YRESULTANT=ROOT-J2,
        SYMBOL=1 OUTPUT=ALL
*
END
```

EXAMPLE B24

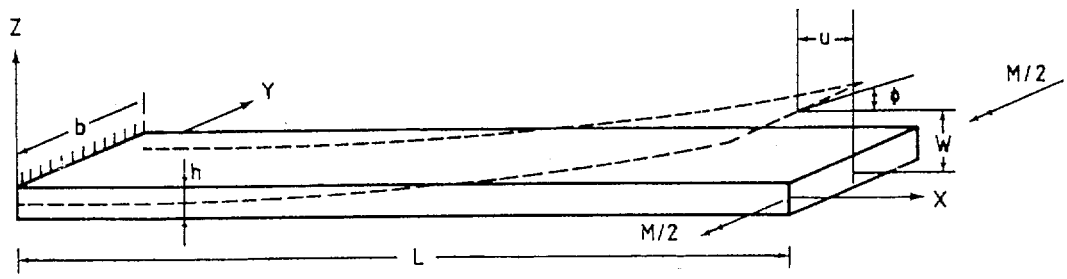
CANTILEVER UNDER LARGE DISPLACEMENTS, SHELL

Objective

To verify the large displacement/rotation behaviour of the SHELL element in pure bending.

Physical Problem

A cantilever beam subjected to a concentrated end moment, as shown in figure below, is considered.



b = 1.0 in
h = 1.0 in
L = 12.0 in

$E = 1800\ \text{lb/in}^2$
$\nu = 0.0$
M = 35 lb–in

Finite Element Model

The finite element model consists of one 8-node cubic SHELL element, as shown in figure on page B24.2. The load is applied in ten equal load steps in a large displacement analysis with full Newton iteration and line search. The iteration tolerance is $ETOL = 10^{-6}$. The element stiffness matrix is evaluated using 4×2×2 Gauss integration.

Solution Results

The input data on pages B24.4 and B24.5 is used in the finite element analysis and gives the following results for the tip rotation $\phi/2\pi$ and the deflection ratios u/L and w/L:

M	$\frac{\phi}{2\pi}$		$\frac{u}{L}$		$\frac{w}{L}$	
	Theory	SOLVIA	Theory	SOLVIA	Theory	SOLVIA
7.0	0.089	0.089	0.051	0.052	0.273	0.273
14.0	0.178	0.179	0.196	0.196	0.504	0.505
21.0	0.267	0.269	0.408	0.408	0.660	0.667
28.0	0.357	0.364	0.650	0.662	0.723	0.744
35.0	0.446	0.496	0.880	0.991	0.694	0.704

The theoretical values assuming small strains and thin shell behaviour are calculated using

$$\frac{\phi}{2\pi} = \frac{ML}{2\pi EI}$$

$$\frac{u}{L} = 1 - \frac{EI}{ML}\sin\left(\frac{ML}{EI}\right)$$

$$\frac{w}{L} = \frac{EI}{ML}\left(1 - \cos\left(\frac{ML}{EI}\right)\right)$$

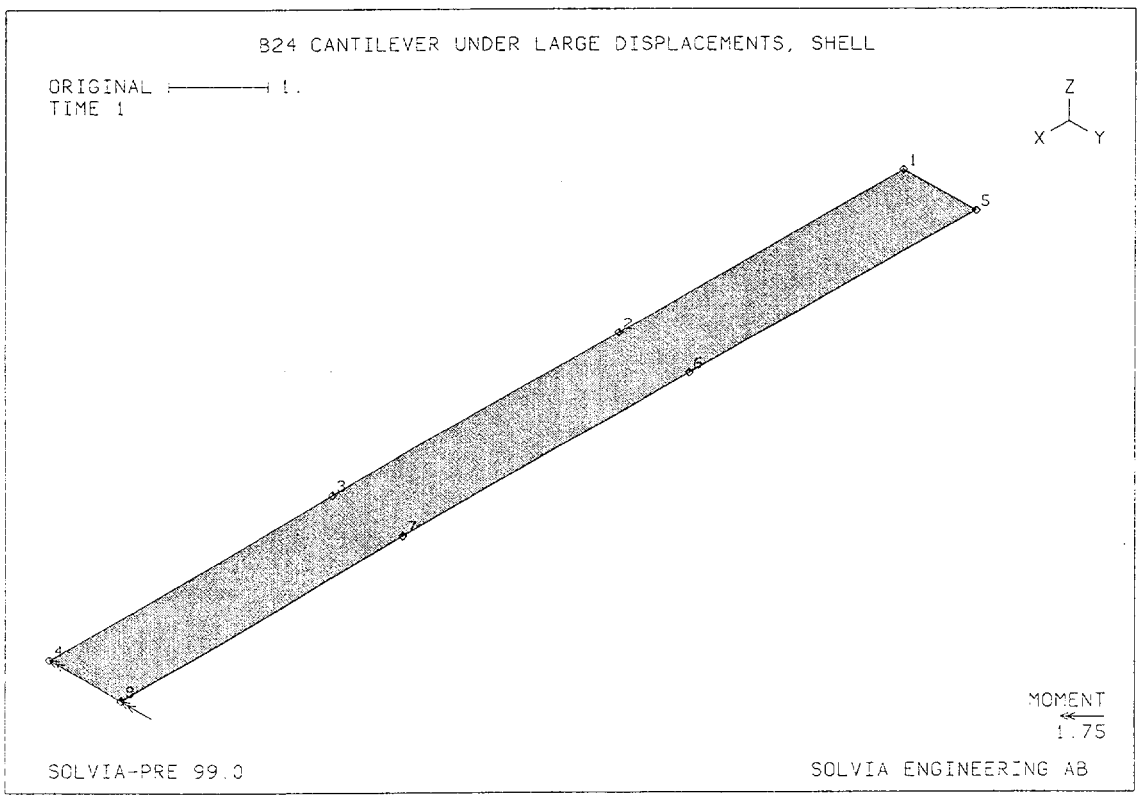
where I is the moment of inertia $\left(bh^3/12\right)$.

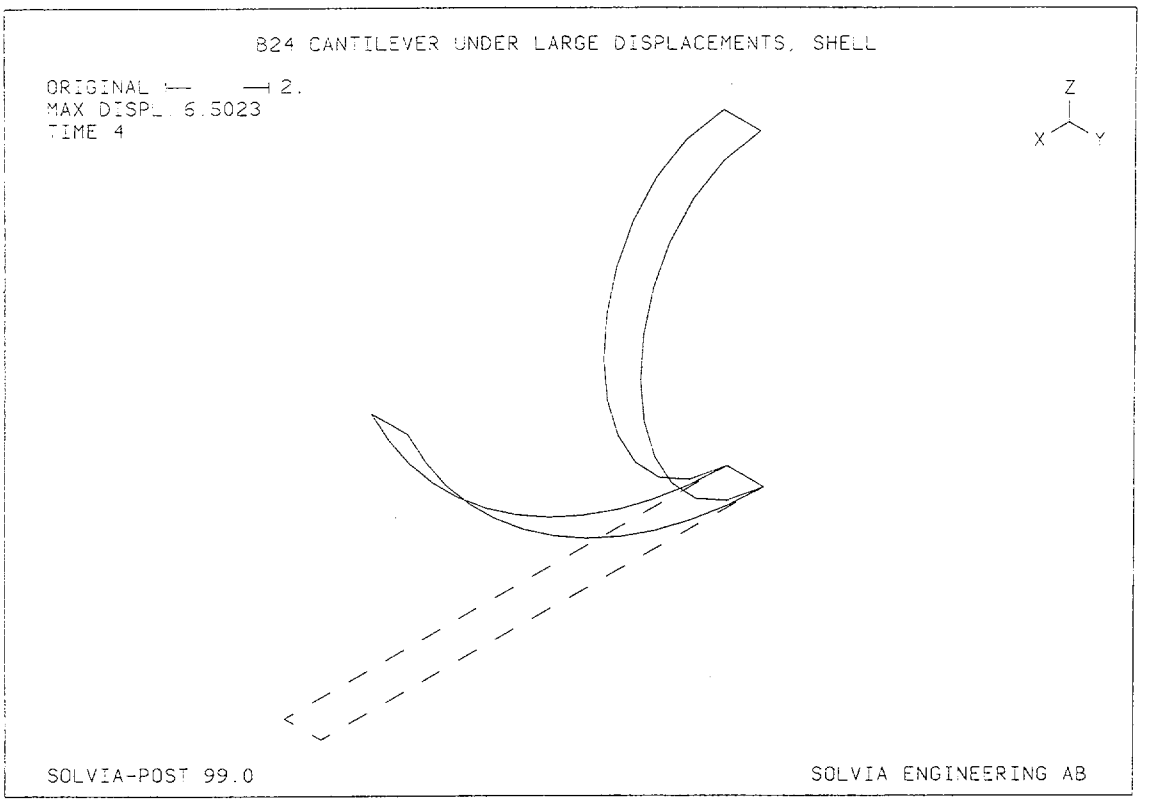
The top figure on page B24.3 shows the deformed finite element meshes at M=14 and M=35. The displacement ratios u/L, w/L and $\phi/2\pi$ as a function of the load are shown in the bottom figure on page B24.3.

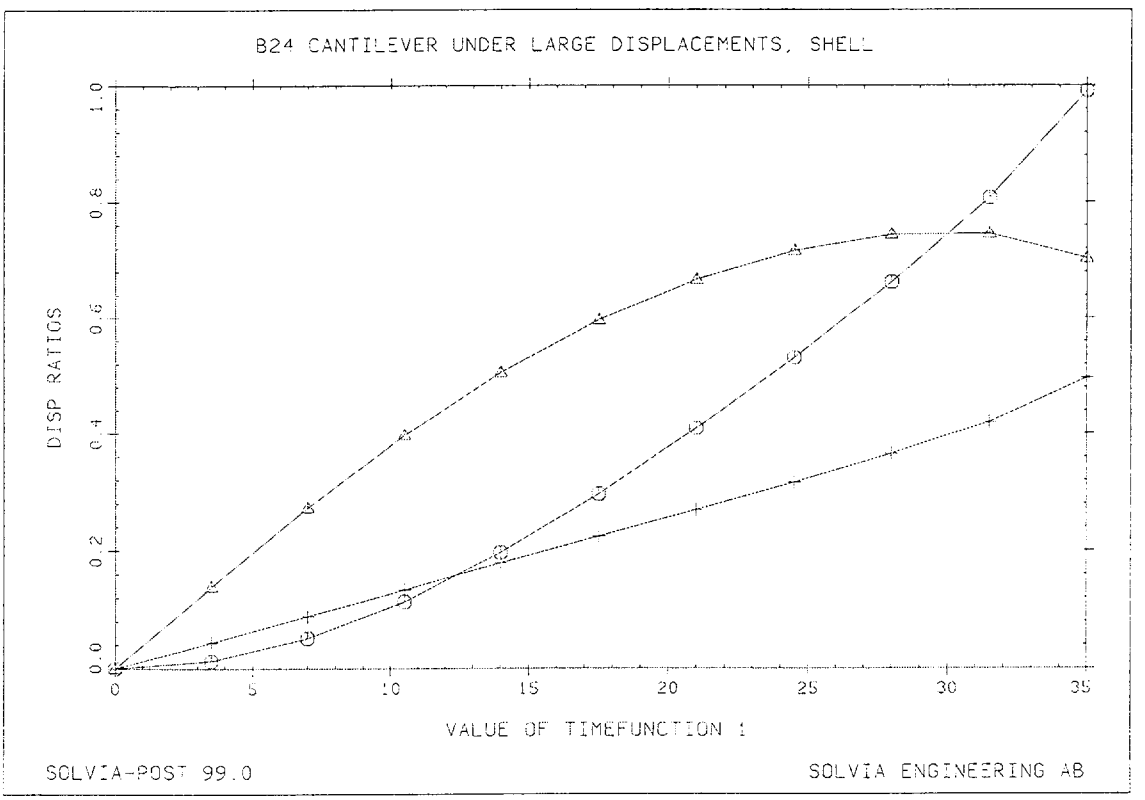
User Hints

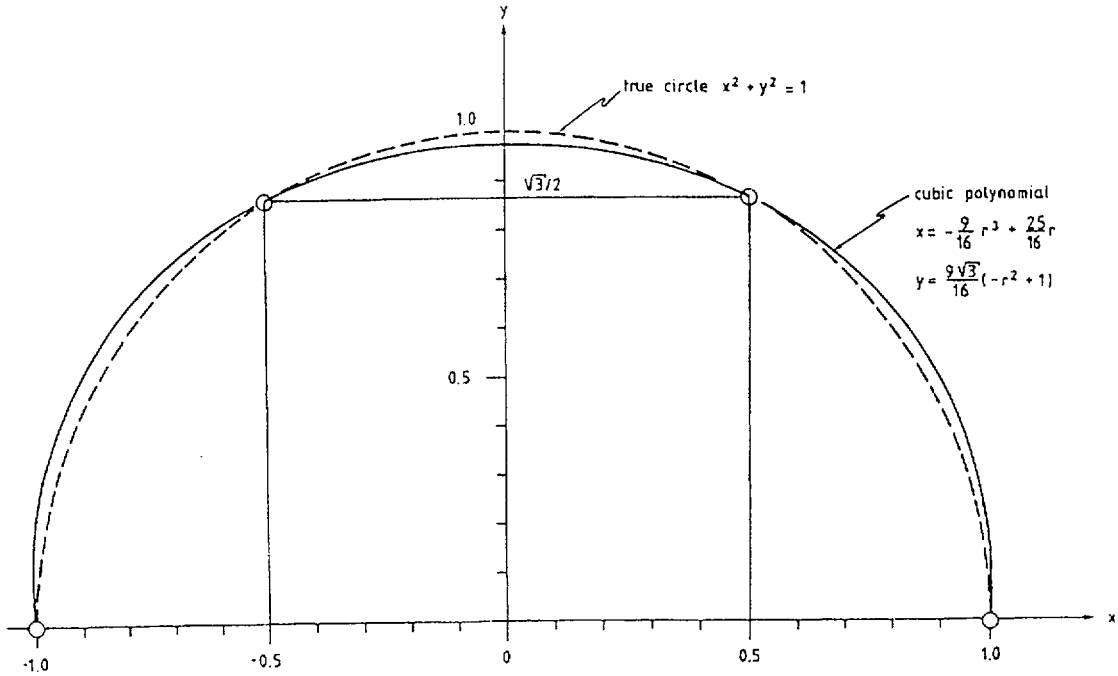
- Note that the predicted response in the finite element analysis compares well with the analytical solution up to approximately 90 degrees rotation of the cantilever tip, but for larger rotations the model is to coarse to define the geometry of the deformed cantilever accurately. As an example of the geometrical discretization error, the figure on page B24.4 shows the midsurface of a cubic shell element for which the nodes lie on a half circle ($\phi = \pi$).

- For rotations in excess of about 90 degrees more elements need, therefore, be employed.




B24 CANTILEVER UNDER LARGE DISPLACEMENTS, SHELL
ORIGINAL
MAX DISPL 6.5023
TIME 4
Z
X
Y
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B24 CANTILEVER UNDER LARGE DISPLACEMENTS, SHELL
DISP RATIOS
0.0
0.2
0.4
0.6
0.8
1.0
0
5
10
15
20
25
30
35
VALUE OF TIMEFUNCTION 1
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB



SOLVIA-PRE input

```
HEADING  'B24 CANTILEVER UNDER LARGE DISPLACEMENTS, SHELL'
*
DATABASE  CREATE
MASTER  IDOF=010101 NSTEP=10 DT=1.
KINEMATICS  DISPLACEMENT=LARGE
ITERATION  METHOD=FULL-NEWTON LINE-SEARCH=YES
TOLERANCES  ETOL=1.E-6
*
TIMEFUNCTION  1
 0.  0.  /  10.  35.
*
COORDINATES
 1  TO  4  12.  /  5  0. 1.  TO  8  12. 1.
*
MATERIAL  1  ELASTIC  E=1800. NU=0.
*
EGROUP  1  SHELL  SINT=2 RESULTS=STRESSES
ENODES
 ENTRIES  EL N1 N2 N3 N4 N5 N7 N9 N11
          1  1  4  8  5  2  7  3  6
THICKNESS  1  1.
*
FIXBOUNDARIES  /  1 5
LOADS  CONCENTRATED
 4 5 -0.5
 8 5 -0.5
*
MESH  NSYMBOLS=YES NNUMBERS=YES VECTOR=MLOAD
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B24  CANTILEVER UNDER LARGE DISPLACEMENTS, SHELL
*
DATABASE  CREATE
WRITE  FILENAME='b24.lis'
TOLERANCES
*
SET  SMOOTHNESS=YES HIDDEN-REMOVAL=NO
COLOR  MODE=LINE
MESH  TIME=4 ORIGINAL=DASHED
MESH  TIME=10 GSCALE=OLD SUBFRAME=OLD TEXT=NO
*
NPOINT  NAME=TIP NODE=4
NVARIABLE  NAME=U DIRECTION=1 KIND=DISPLACEMENT
NVARIABLE  NAME=W DIRECTION=3 KIND=DISPLACEMENT
NVARIABLE  NAME=PHI DIRECTION=5 KIND=DISPLACEMENT
CONSTANT  NAME=L VALUE=12.
CONSTANT  NAME=TWOPI VALUE=6.2832
RESULTANT  NAME=UOVERL STRING='-U/L'
RESULTANT  NAME=WOVERL STRING='W/L'
RESULTANT  NAME=ANGLE  STRING='-PHI/TWOPI'
*
AXIS  ID=1  VMIN=0 VMAX=1 LABEL='DISP RATIOS'
*
RHISTORY  POINTNAME=TIP RESULTANTNAME=UOVERL XVARIABLE=1 YAXIS=1,
          OUTPUT=ALL SYMBOL=1
RHISTORY  POINTNAME=TIP RESULTANTNAME=WOVERL XVARIABLE=1 YAXIS=-1,
          OUTPUT=ALL SYMBOL=2 SUBFRAME=OLD
RHISTORY  POINTNAME=TIP RESULTANTNAME=ANGLE  XVARIABLE=1 YAXIS=-1,
          OUTPUT=ALL SYMBOL=3 SUBFRAME=OLD
END
```

EXAMPLE B25

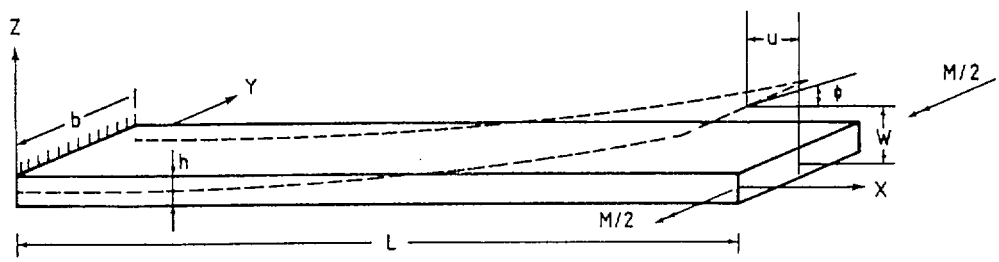
CANTILEVER UNDER LARGE DISPLACEMENTS, PLATE

Objective

To verify the large displacement/rotation behaviour of the PLATE element in pure bending.

Physical Problem

The same type of problem as in Example B24 is considered. The dimensions of the cantilever and the Young's modulus of the material are, however, different as can be seen in the figure below.



b = 10.0 in
h = 0.5 in
L = 100.0 in

$E = 12000\ lb/in^2$
$\nu = 0.0$
M = 35 lb–in

Finite Element Model

The finite element model considered is shown in the figure on page B25.2. The model consists of ten PLATE elements and the end moment is applied in ten equal load steps in a large displacement analysis. The solution is obtained using the full Newton iteration. The iteration tolerances are RTOL = 0.01, RNORM = 0.35 and RMNORM = 35.

Solution Results

The input data on pages B25.4 and B25.5 gives the following results in the finite element analysis:

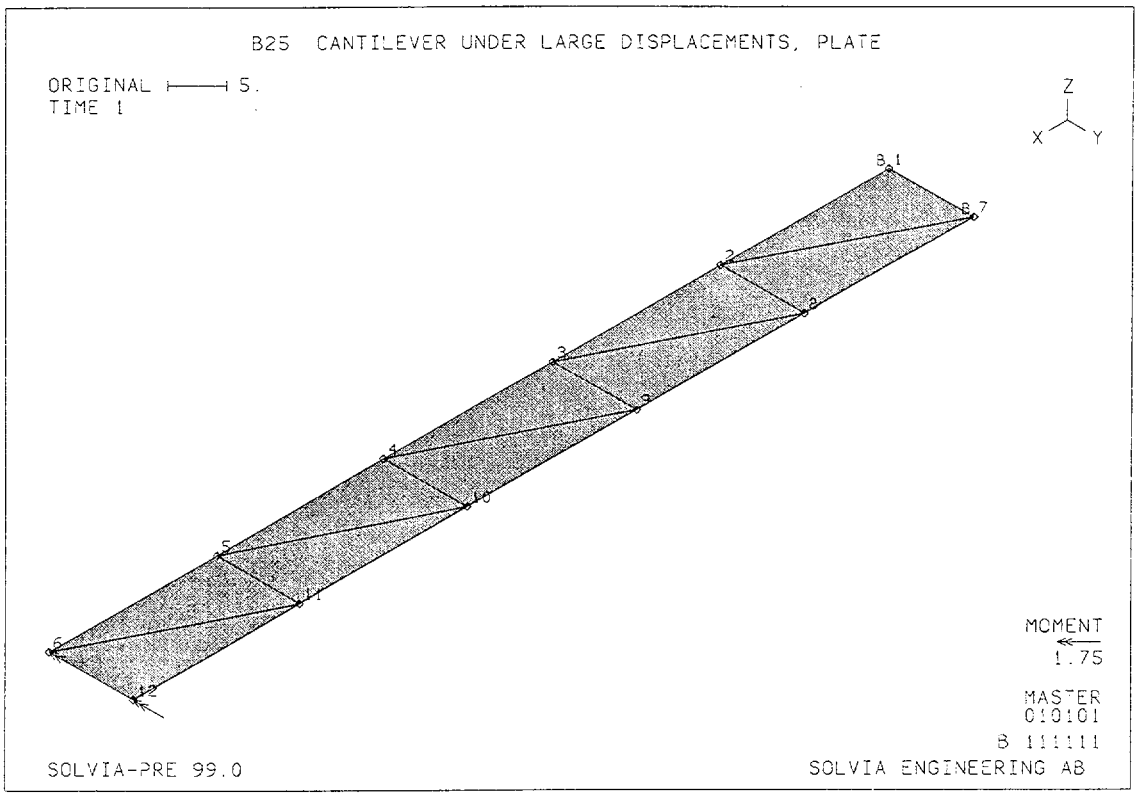
M	$\frac{\phi}{2\pi}$		$\frac{u}{L}$		$\frac{w}{L}$	
	Theory	SOLVIA	Theory	SOLVIA	Theory	SOLVIA
7.0	0.089	0.089	0.051	0.052	0.273	0.275
14.0	0.178	0.178	0.196	0.197	0.504	0.507
21.0	0.267	0.267	0.408	0.411	0.660	0.664
28.0	0.357	0.357	0.650	0.654	0.723	0.728
35.0	0.446	0.446	0.880	0.885	0.694	0.698

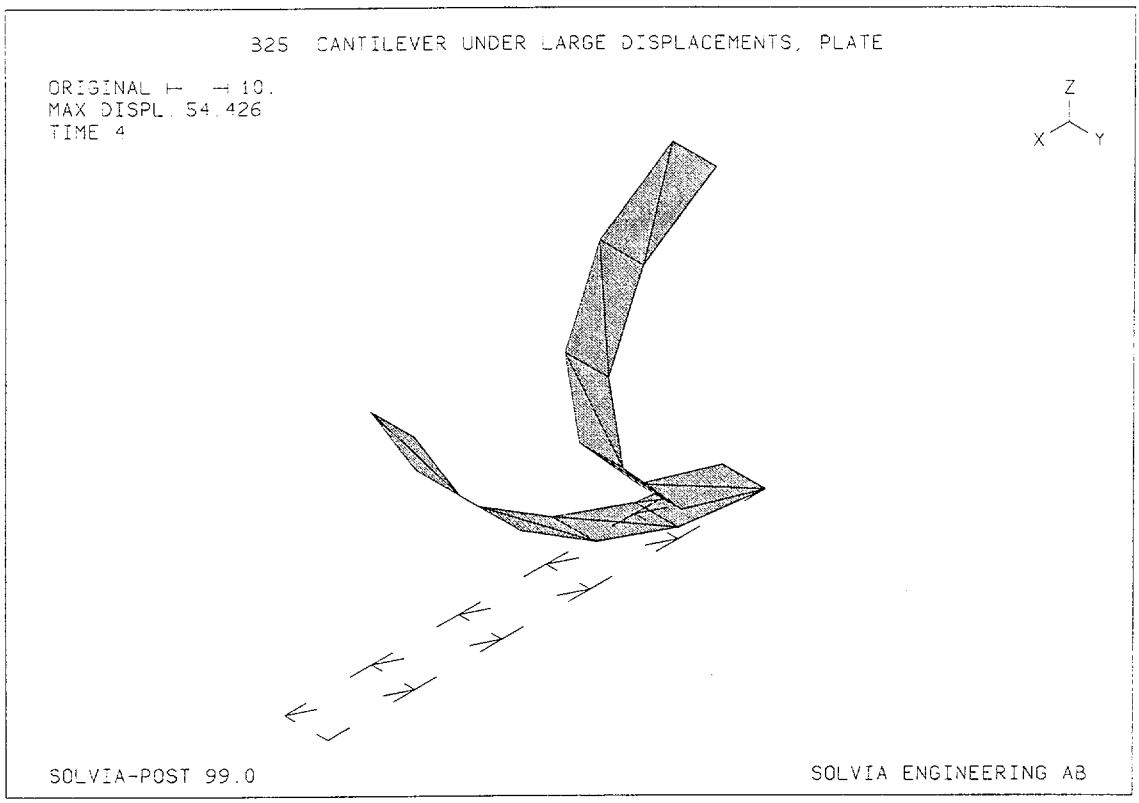
The theoretical results are calculated as in Example B24.

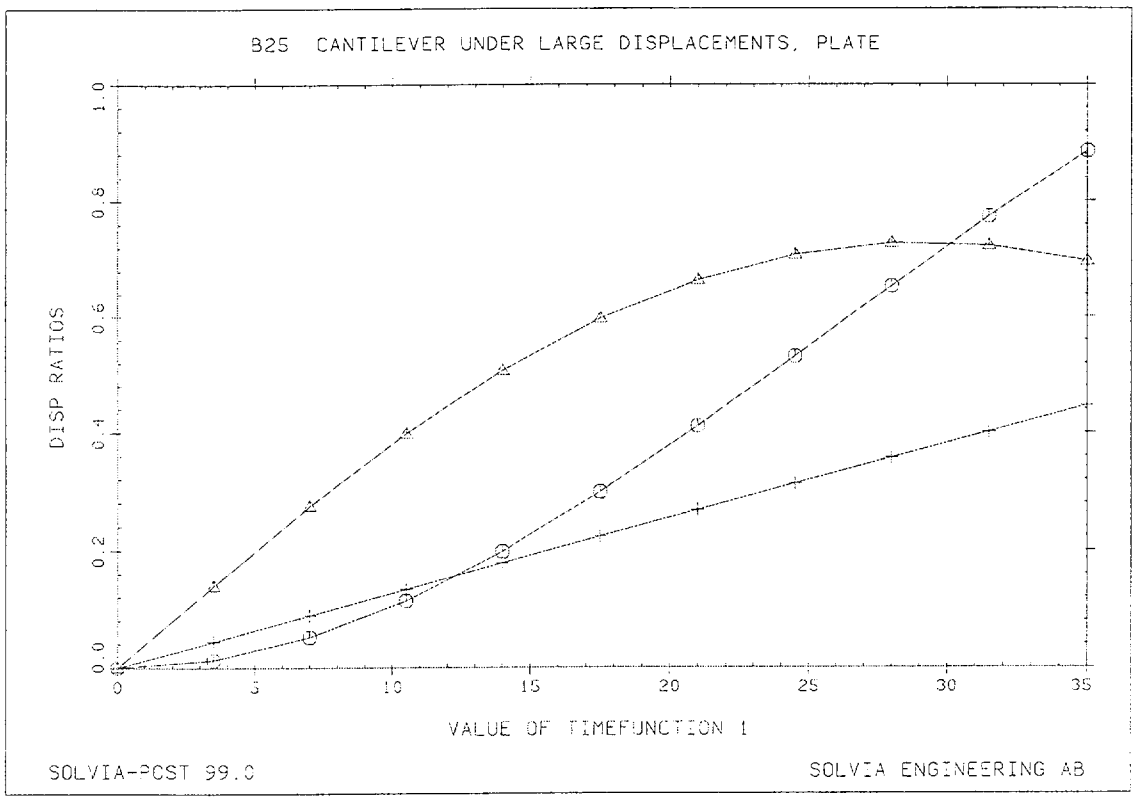
The top figure on page B25.3 shows the deformed finite element meshes at M = 14 and M = 35. The bottom figure on page B25.3 shows the displacement ratios $\phi/2\pi$, u/L and w/L as a function of the load as displayed by SOLVIA-POST.

User Hints

- Since more elements are used in this example, a better solution is obtained than in Example B24, where only one cubic SHELL element is employed.




B25 CANTILEVER UNDER LARGE DISPLACEMENTS, PLATE
ORIGINAL
MAX DISPL. 54.426
TIME 4
Z
X
Y
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B25 CANTILEVER UNDER LARGE DISPLACEMENTS, PLATE
DISP RATIOS
0.0
0.2
0.4
0.6
0.8
1.0
0
5
10
15
20
25
30
35
VALUE OF TIMEFUNCTION 1
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B25  CANTILEVER UNDER LARGE DISPLACEMENTS, PLATE'
*
DATABASE  CREATE
MASTER  IDOF=010101 NSTEP=10 DT=1
KINEMATICS  DISPLACEMENTS=LARGE
ITERATION  METHOD=FULL-NEWTON
TOLERANCES  TYPE=F RTOL=0.01 RNORM=.35 RMNORM=35
*
TIMEFUNCTION  1
 0. 0.  /  10. 35.
*
COORDINATES
 1          TO   6  100.
 7  0. 10.  TO  12  100. 10.
*
MATERIAL  1  ELASTIC  E=12000. NU=0.
*
EGROUP  1  PLATE
ENODES
 1  1 2 7  TO   5  5  6 11
 6  2 8 7  TO  10  6 12 11
EDATA  /  1  .5
*
FIXBOUNDARIES  /  1 7
LOADS  CONCENTRATED
  6 5 -0.5
 12 5 -0.5
*
MESH  NSYMBOLS=YES NNUMBERS=YES VECTOR=MLOAD BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B25  CANTILEVER UNDER LARGE DISPLACEMENTS, PLATE
*
DATABASE  CREATE
TOLERANCES
*
WRITE  FILENAME='b25.lis'
*
MESH  ORIGINAL=DASHED TIME=4
MESH  GSCALE=OLD SUBFRAME=OLD TEXT=NO
*
NPOINT  NAME=TIP NODE=6
NVARIABLE  NAME=U   DIRECTION=1 KIND=DISPLACEMENT
NVARIABLE  NAME=W   DIRECTION=3 KIND=DISPLACEMENT
NVARIABLE  NAME=PHI DIRECTION=5 KIND=DISPLACEMENT
CONSTANT  NAME=L VALUE=100.
CONSTANT  NAME=TWOPI VALUE=6.2832
RESULTANT  NAME=UOVERL STRING='-U/L'
RESULTANT  NAME=WOVERL STRING='W/L'
RESULTANT  NAME=ANGLE  STRING='-PHI/TWOPI'
*
AXIS  ID=1  VMIN=0 VMAX=1 LABEL='DISP RATIOS'
*
RHISTORY  POINTNAME=TIP RESULTANTNAME=UOVERL XVARIABLE=1 YAXIS=1,
          SYMBOL=1 OUTPUT=ALL
RHISTORY  POINTNAME=TIP RESULTANTNAME=WOVERL XVARIABLE=1 YAXIS=-1,
          SYMBOL=2 OUTPUT=ALL SUBFRAME=OLD
RHISTORY  POINTNAME=TIP RESULTANTNAME=ANGLE  XVARIABLE=1 YAXIS=-1,
          SYMBOL=3 OUTPUT=ALL SUBFRAME=OLD
END
```

EXAMPLE B26

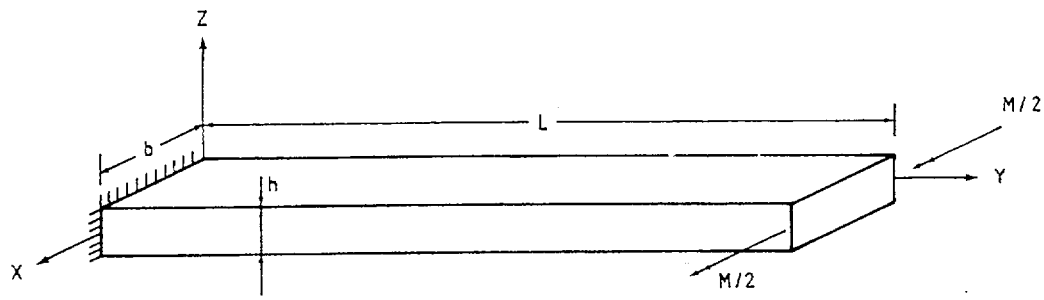
PLASTIC CANTILEVER UNDER PURE BENDING, SHELL

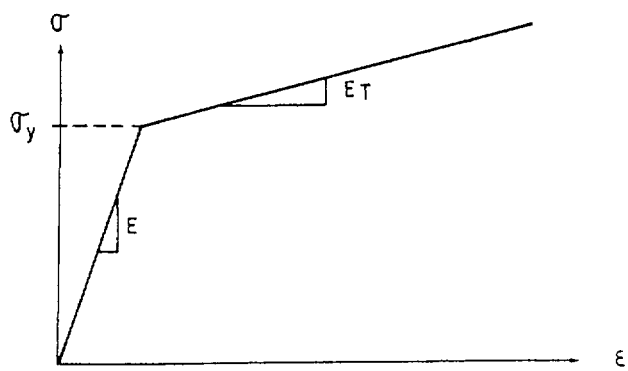
Objective

To verify the elastic-plastic behaviour of the SHELL element.

Physical Problem

The cantilever beam subjected to a concentrated end moment, as shown in the figure below, is considered. The elastic-plastic response of the cantilever is to be determined.





$b = 2.0$ in

$h = 1.0$ in

$L = 12.0$ in

$E = 3.0 \cdot 10^7$ psi

$\nu = 0.3$

$\sigma_y = 1.5 \cdot 10^3$ psi

$E_T = 3.0 \cdot 10^5$ psi

$M = 900$ lb–in

Finite Element Model

Due to symmetry conditions, only one half of the cantilever is modeled. The finite element model consists of one 16-node SHELL element as shown in the figure on page B26.2. The material of the cantilever is modeled as elastic-plastic with isotropic hardening. The element stiffness matrix is evaluated using 4×4×6 Gauss integration. The solution response is traced with the AUTOSTEP method using BFGS iteration. The maximum solution time step is set to 3 and request is made of intermediate solutions at 6 specified solution times.

Solution Results

The input data on pages B26.4 and B26.5 is used in the finite element analysis.

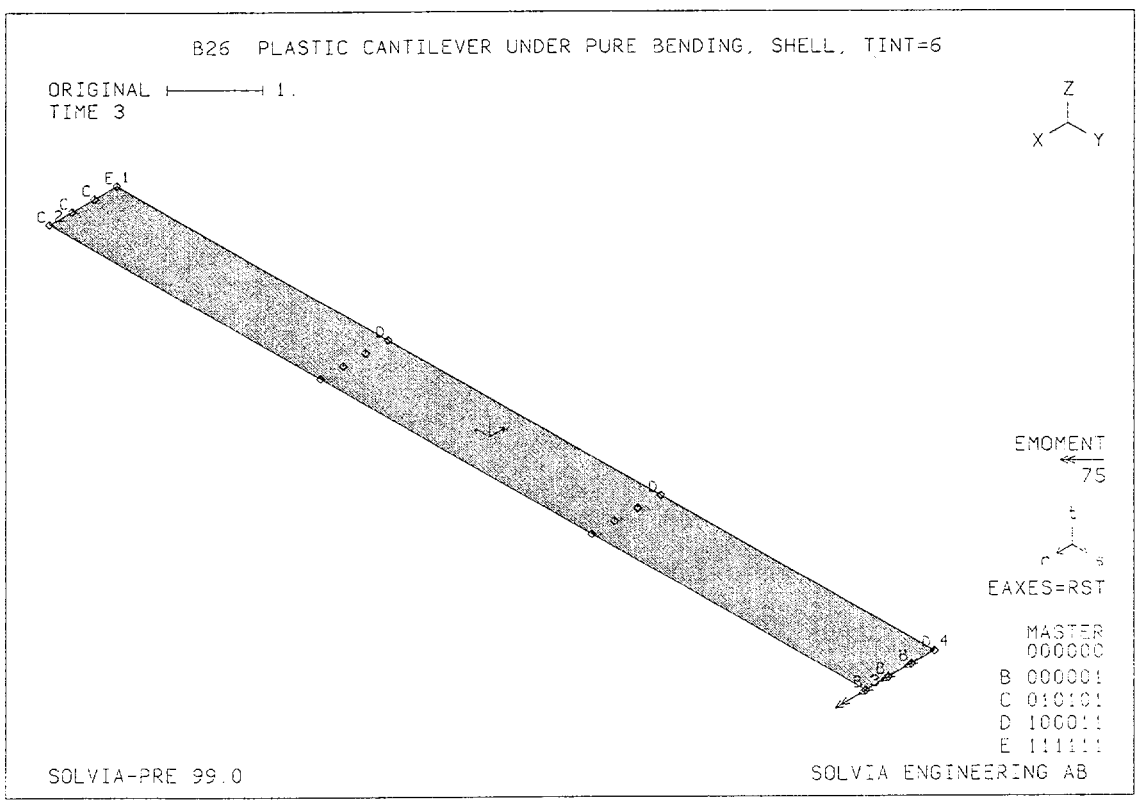
In the table on the next page the calculated tip rotations for 4×4×6 and 4×4×4 Gauss integration are compared with an analytical solution using beam theory.

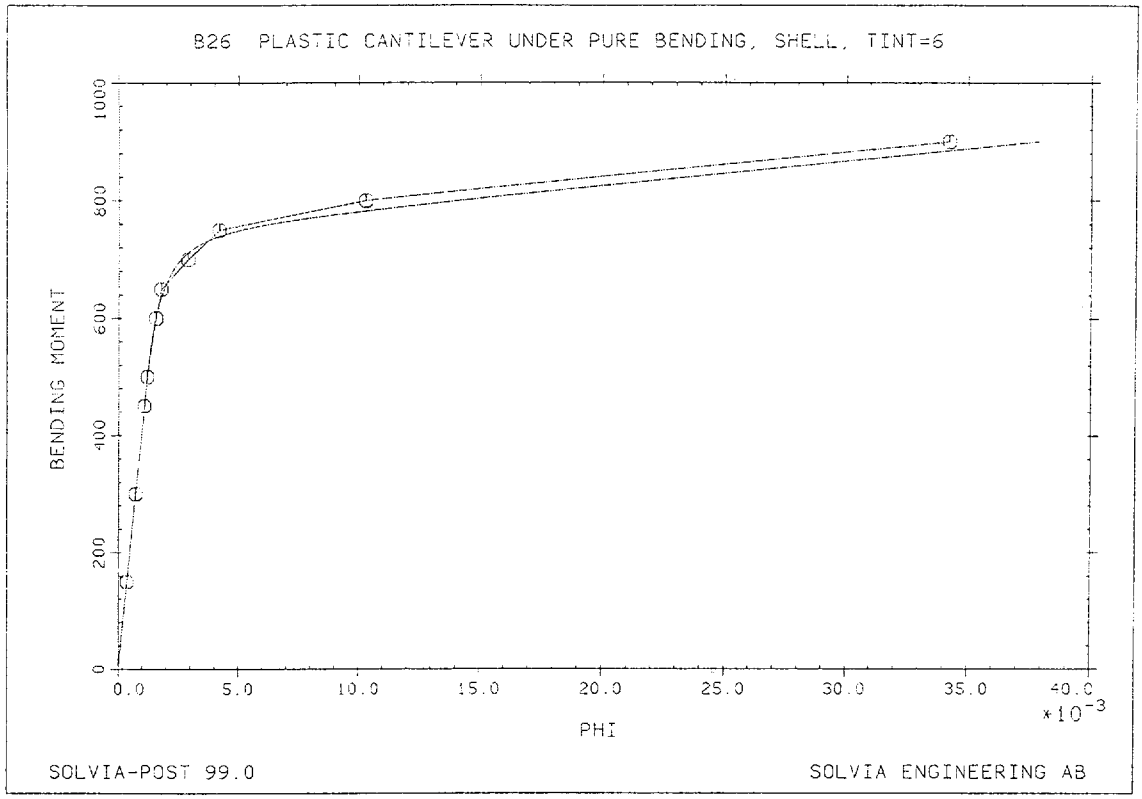
Moment	Rotation, ϕ Beam theory	SOLVIA (4×4×6)	SOLVIA (4×4×4)
500	$1.20 \cdot 10^{-3}$	$1.20 \cdot 10^{-3}$	$1.20 \cdot 10^{-3}$
600	$1.55 \cdot 10^{-3}$	$1.56 \cdot 10^{-3}$	$1.59 \cdot 10^{-3}$
700	$2.59 \cdot 10^{-3}$	$2.88 \cdot 10^{-3}$	$2.59 \cdot 10^{-3}$
800	$1.42 \cdot 10^{-2}$	$1.03 \cdot 10^{-2}$	$6.28 \cdot 10^{-3}$
900	$3.79 \cdot 10^{-2}$	$3.42 \cdot 10^{-2}$	$3.02 \cdot 10^{-2}$

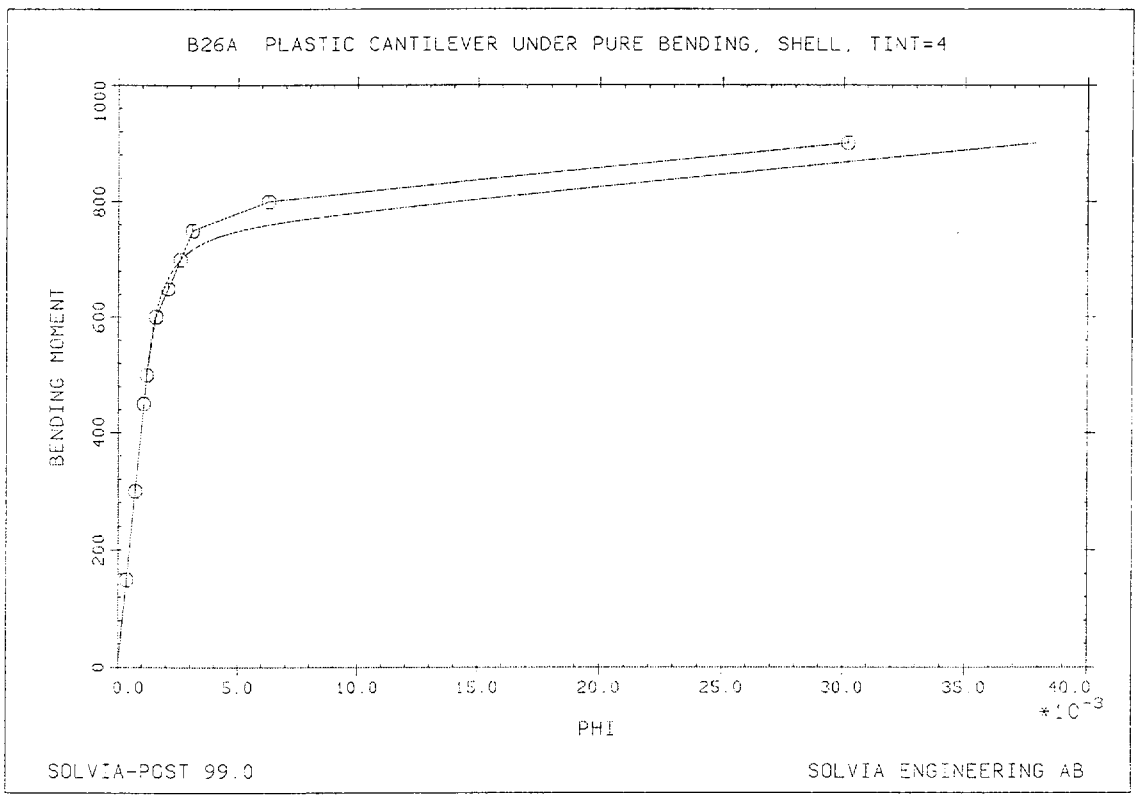
The figures on page B26.3 show the variation of the applied moment versus the tip rotation as predicted in the finite element analysis with 4×4×6 and 4×4×4 Gauss integration. For comparison, the analytical beam theory solution is also presented in the figures.

User Hints

- Note that in an elastic-plastic analysis the order of Gauss integration through the element thickness must be high enough to predict the plastic behaviour through the shell thickness accurately.
- Note that the anticlastic bending effect gives a slight variation of the transverse displacement along the width of the shell element.




B26 PLASTIC CANTILEVER UNDER PURE BENDING, SHELL, TINT=6
BENDING MOMENT
0
200
400
600
800
1000
0.0
5.0
10.0
15.0
20.0
25.0
30.0
35.0
40.0
*10^-3
PHI
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B26A PLASTIC CANTILEVER UNDER PURE BENDING, SHELL, TINT=4
BENDING MOMENT
0
200
400
600
800
1000
0.0
5.0
10.0
15.0
20.0
25.0
30.0
35.0
40.0
*10^-3
PHI
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B26  PLASTIC CANTILEVER UNDER PURE BENDING, SHELL, TINT=6'
*
DATABASE  CREATE
MASTER  NSTEP=20 DT=3.0
ITERATION  METHOD=BFGS
TOLERANCES  ETOL=1.E-8
AUTO-STEP DTMAX=3.0 TMAX=18 T1=10 12 13 14 15 16
*
TIMEFUNCTION  1
 0. 0.  /  18. 1.8
*
COORDINATES
 1  /  2  1.0  /  3  1.0 12.  /  4  0. 12.
*
MATERIAL  1  PLASTIC  E=3.E7 NU=0.3 YIELD=1.5E3 ET=3.E5
*
EGROUP  1  SHELL  TINT=6
THICKNESS  1  1.
GSURFACE  1 2 3 4  EL1=1 EL2=1 NODES=16
LOADS  ELEMENT  TYPE=MOMENT INPUT=LINE
 3 4  edge  250. 250.
*
FIXBOUNDARIES  246  INPUT=LINE  /  1 2
FIXBOUNDARIES  156  INPUT=LINE  /  1 4
FIXBOUNDARIES  3    INPUT=NODE  /  1
FIXBOUNDARIES  6    INPUT=LINE  /  3 4
*
MESH  NSYMBOLS=YES NNUMBERS=MYNODES VECTOR=LOAD BCODE=ALL,
      EAXES=RST
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B26  PLASTIC CANTILEVER UNDER PURE BENDING, SHELL
*
DATABASE  CREATE
*
WRITE  FILENAME='b26.lis'
*
AXIS 1 VMIN=0. VMAX=4.0E-2 LABELSTRING='PHI'
AXIS 2 VMIN=0. VMAX=900 LABELSTRING='BENDING MOMENT'
*
NVARIABLE NAME=XMOM  DIRECTION=4 KIND=REACTION
NVARIANLE NAME=XROT  DIRECTION=4 KIND=DISPLACEMENT
CONSTANT  NAME=TWO VALUE=-2.0
CONSTANT  NAME=EIGHT   VALUE=8.0
RESULTANT NAME=PHI 'XROT'
RESULTANT NAME=MOMENT 'XMOM*TWO*EIGHT'
NPOINT NAME=N1 NODE=1
NPOINT NAME=N4 NODE=4
RXYPLOT XPOINT=N4 XRESULTANT=PHI YPOINT=N1 YRESULTANT=MOMENT,
        SYMBOL=1 TSTART=3 XAXIS=1 YAXIS=2 OUTPUT=ALL
*
USERCURVE 1
READ B26.DAT
PLOT USERCURVE 1 XAXIS=-1 YAXIS=-2 SUBFRAME=OLD
END
```

EXAMPLE B27

PLASTIC CANTILEVER UNDER PURE BENDING, PLATE

Objective

To verify the elastic-plastic behaviour of the PLATE element.

Physical Problem

Same as in figure on page B26-1.

Finite Element Model

The finite element model considered is shown in the figure on page B27.2. Due to symmetry only one half of the cantilever beam is modeled using eight PLATE elements. The material of the beam is modeled as elastic-plastic with isotropic hardening and Ilyushin yield condition. The solution response is traced with the AUTOSTEP method using BFGS iteration. The maximum solution time step is set to 3 and request is made of intermediate solutions at 6 specified solution times.

Solution Results

The input data on pages B27.3 and B27.4 is used in the finite element analysis.

The following results for the tip rotation ϕ (rad) are obtained in the analysis:

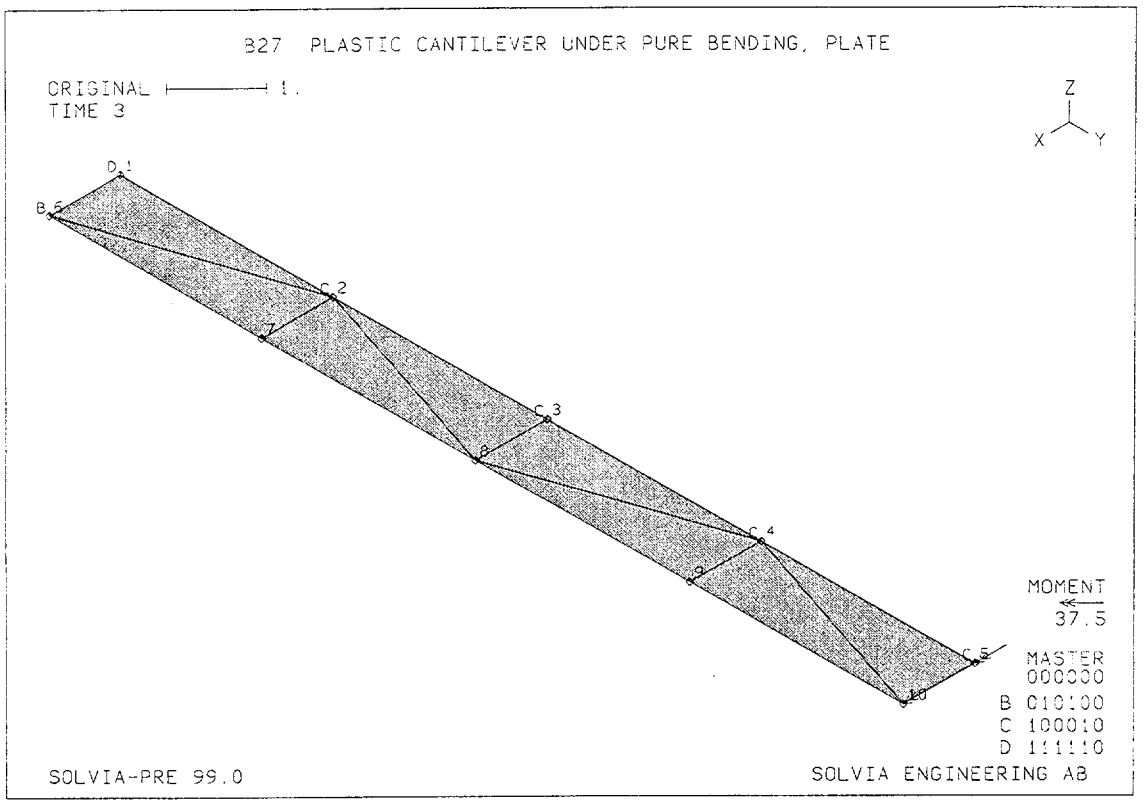
Moment	SOLVIA	Beam theory
500	$1.20 \cdot 10^{-3}$	$1.20 \cdot 10^{-3}$
750	$1.80 \cdot 10^{-3}$	$5.10 \cdot 10^{-3}$
900	$4.97 \cdot 10^{-2}$	$3.79 \cdot 10^{-2}$

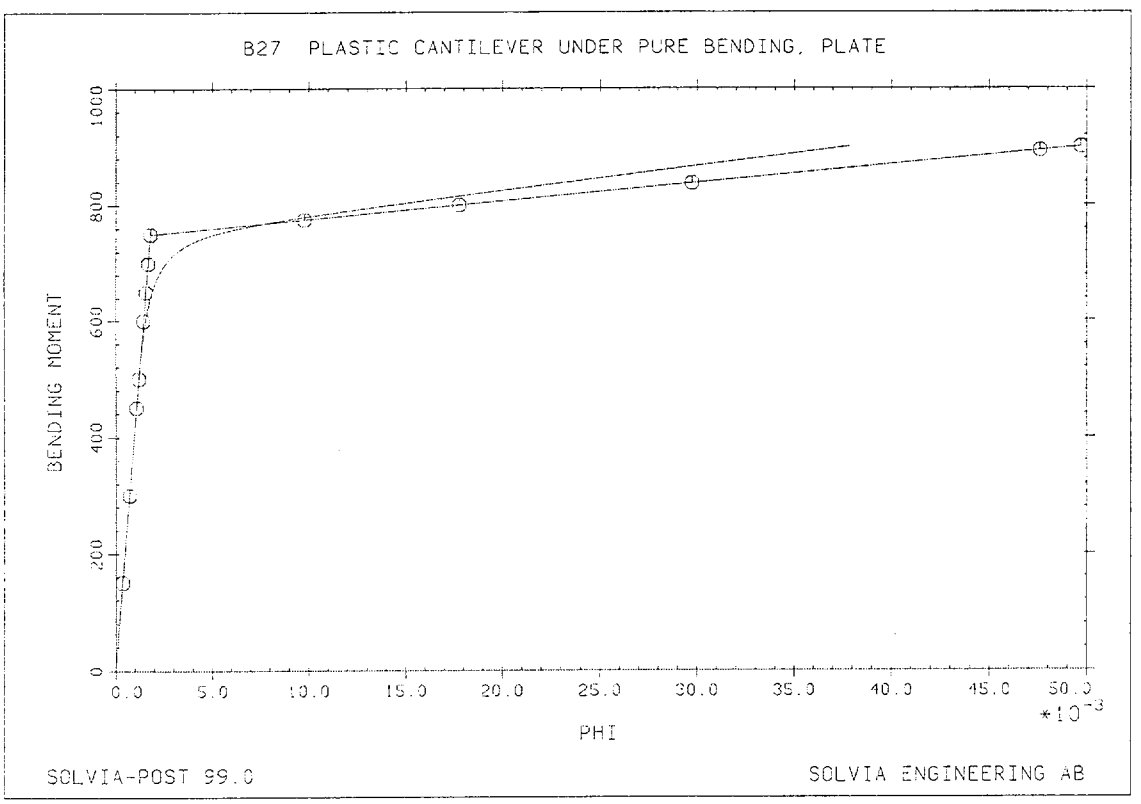
The bottom figure on page B27.2 shows the variation of the applied moment on the whole plate versus the tip rotation as predicted in the finite element analysis. An analytical beam theory solution is also presented in the same figure.

User Hints

- The PLATE element stiffness matrix is calculated using stress resultants and no numerical integration in the thickness direction is used. The yield condition employed is the one of Ilyushin [1], and the stress resultants correspond either to elastic or plastic conditions. This means that the entire cross-section at an integration point changes at the same time from elastic to plastic and vice versa.

- In this example with pure moment loading the plastic response is calculated based on the conditions at the distance h/4 from the midsurface, where h is the thickness. However, in the case of strain-hardening the incremental plastic moment can be represented by two force resultants each located h/3 from the midsurface, which is the position of the center of gravity of the corresponding triangular incremental stress distribution. Hence, the calculated plastic slope in the moment-rotation diagram on page B27.2 is 3/4 of the theoretical plastic slope asymptote.


B27 PLASTIC CANTILEVER UNDER PURE BENDING, PLATE
ORIGINAL 1.
TIME 3
Z
X
Y
MOMENT
37.5
MASTER
000000
B 010100
C 100010
D 111110
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


B27 PLASTIC CANTILEVER UNDER PURE BENDING, PLATE
BENDING MOMENT
0
200
400
600
800
1000
0.0
5.0
10.0
15.0
20.0
25.0
30.0
35.0
40.0
45.0
50.0
*10^-3
PHI
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B27  PLASTIC CANTILEVER UNDER PURE BENDING, PLATE'
*
DATABASE  CREATE
MASTER  NSTEP=20 DT=3.0
ITERATION  METHOD=BFGS
TOLERANCES  ETOL=1.E-6
AUTO-STEP DTMAX=3.0 TMAX=18. T1=10 12 13 14 15 16
*
TIMEFUNCTION  1
 0. 0.  /  18. 1.8
*
COORDINATES
 1  TO  5  0. 12.  /  6  1.0  TO  10  1.0 12.
*
MATERIAL  1  ILYUSHIN  E=3.E7 NU=0.3 YIELD=1.5E3 ET=3.E5
*
EGROUP  1  PLATE
ENODES
 1  1 2 6  /  2  2 3 8  /  3  3 4 8  /  4  4 5 10
 5  6 2 7  /  6  2 8 7  /  7  8 4 9  /  8  4 10 9
EDATA  /  1  1.
*
FIXBOUNDARIES  15     /  2 TO 5
FIXBOUNDARIES  12345  /  1
FIXBOUNDARIES  24     /  6
LOADS  CONCENTRATED
  5 4 125.
 10 4 125.
*
MESH  NSYMBOLS=YES NNUMBERS=YES VECTOR=MLOAD BCODE=ALL
*
SOLVIA
END
```

SOLVIA-POST input

```
*  B27  PLASTIC CANTILEVER UNDER PURE BENDING, PLATE
*
DATABASE  CREATE
*
WRITE  FILENAME='b27.lis'
*
AXIS 1 VMIN=0. VMAX=5.0E-2 LABELSTRING='PHI'
AXIS 2 VMIN=0. VMAX=900 LABELSTRING='BENDING MOMENT'
*
NVARIABLE NAME=XMOM  DIRECTION=4 KIND=REACION
NVARIANLE NAME=XROT  DIRECTION=4 KIND=DISPLACEMENT
CONSTANT  NAME=TWO VALUE=2.0
RESULTANT NAME=PHI 'XROT'
RESULTANT NAME=MOMENT '- XMOM * TWO * TWO'
NPOINT NAME=N1 NODE=1
NPOINT NAME=N5 NODE=5
RXYPLOT XPOINT=N5 XRESULTANT=PHI YPOINT=N1 YRESULTANT=MOMENT,
        SYMBOL=1  TSTART=3 XAXIS=1 YAXIS=2 OUTPUT=ALL
*
USERCURVE 1
READ B26.DAT
PLOT USERCURVE 1 XAXIS=-1 YAXIS=-2 SUBFRAME=OLD
END
```

EXAMPLE B28

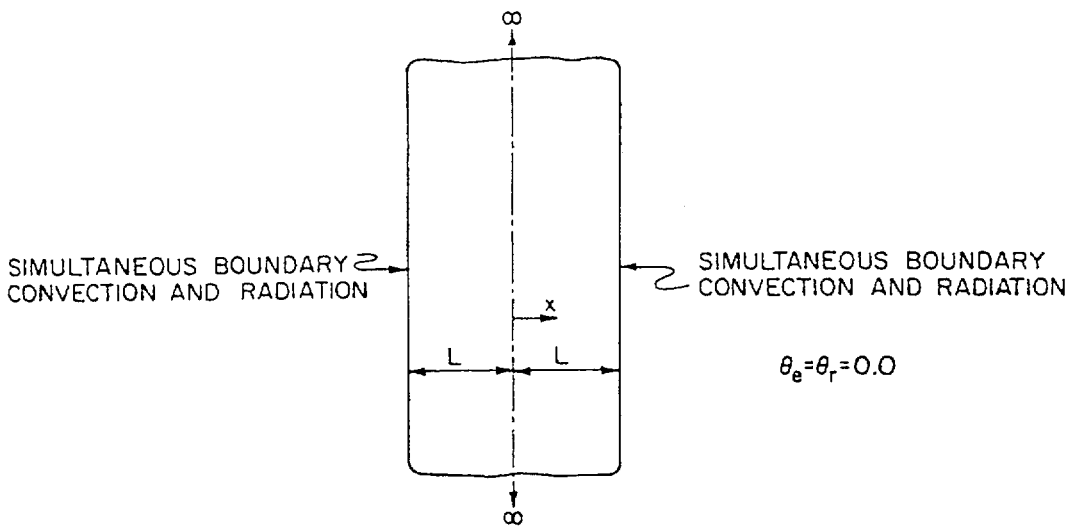
TRANSIENT TEMPERATURE ANALYSIS OF A SLAB

Objective

To verify the behaviour of the TRUSS conduction element in SOLVIA-TEMP when subjected to simultaneous boundary convection and radiation in a transient analysis.

Physical Problem

The slab shown in the figure below is considered. The slab is initially at a uniform temperature θ_i and at time $t = 0^+$ the slab surfaces are exposed to convection and radiation. The time history of the temperature at the surface and at the center of the slab is to be determined.



Thickness/2	$L = 1.0$ in	Convection coefficient	$h = 0.04\ \mathrm{Btu/°F \cdot hr \cdot in^2}$
Conductivity	$k = 0.01\ \mathrm{Btu/°F \cdot hr \cdot in}$	Shape factor	$f = 1.0$
Specific heat	$c = 0.01\ \mathrm{Btu/°F \cdot in^3}$	Emissivity constant	$\varepsilon = 1.0$ (black body)
Thermal diffusivity	$\alpha = k/c = 1.0\ \mathrm{in^2/hr}$	Bolzmann's constant	$\sigma = 0.118958 \cdot 10^{-10}\ \mathrm{Btu/in^2 \cdot hr \cdot °R^4}$

$$\Gamma = \frac{\sigma \cdot f \cdot \varepsilon \cdot \theta_i^3 \cdot L}{k} = 4.0 \text{ gives } \theta_i = 1498.15 \qquad Bi = \frac{h \cdot L}{k} = 4.0$$

Finite Element Model

Because of symmetry conditions only one half of the slab is modeled. The finite element model consists of twenty equally spaced TRUSS conduction elements as shown in the left top figure on page B28.3. The thermal conductivity and the heat capacity of the material are assumed to be constant. A diagonal heat capacity matrix is used. The transient temperature response is evaluated using the Euler backward method in the step-by-step analysis. Conductivity reformation and heat flow equilibrium iteration using the modified Newton-Raphson method is performed at each time step.

In order to obtain the required accuracy in the SOLVIA-TEMP analysis the time step size is changed during the response prediction.

	$t^* = \alpha \cdot t / L^2$	$\Delta t^* = \alpha \cdot \Delta t / L^2$	Number of time steps
First interval	$0.00<t^*\leq 0.06$	0.001	60
Second interval	$0.06<t^*\leq 0.66$	0.01	60
Third interval	$0.66<t^*\leq 3.66$	0.05	60

Solution Results

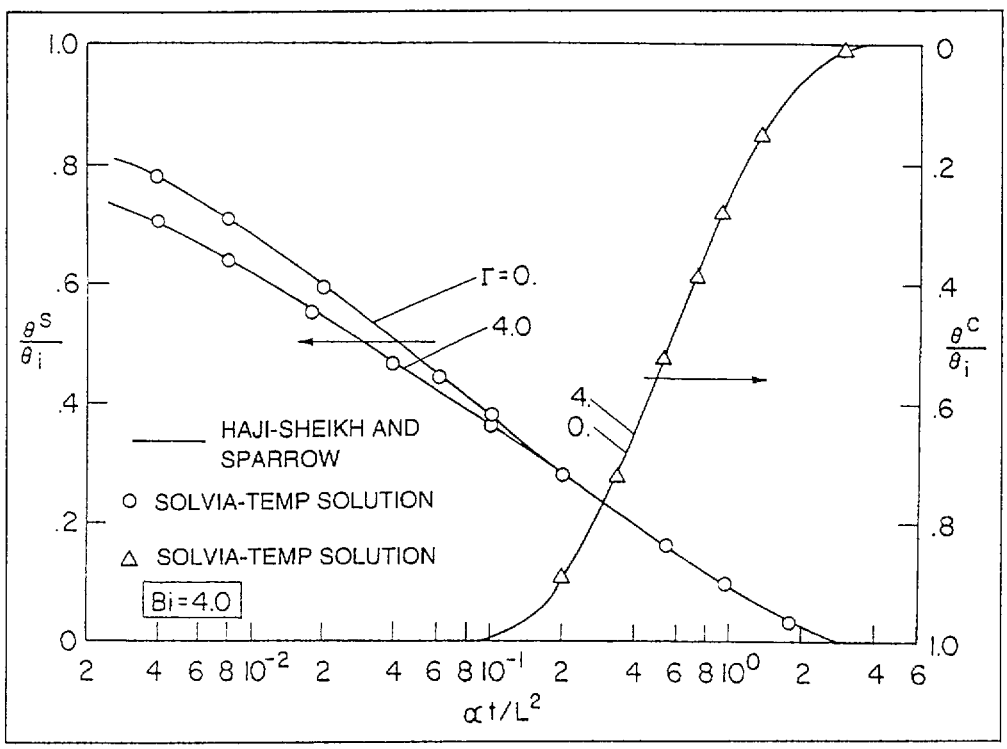
The input data on pages B28.4 to B28.5 is used in the finite element analysis. The solution of the surface temperature and the temperature at the center of the slab, θ^s and θ^c respectively, depends upon two parameters; the Biot number Bi and the radiation parameter Γ. In this analysis Bi = 4 and Γ = 4.

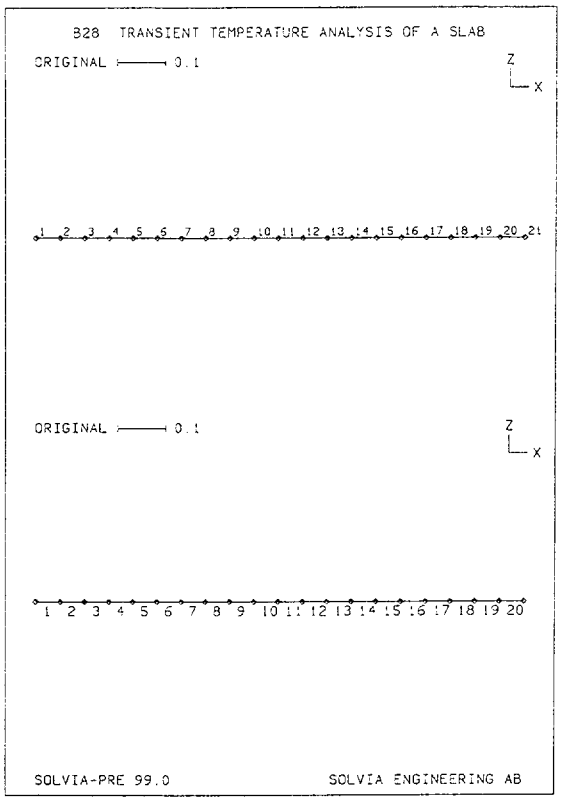
The figure below shows the temperature variation at the surface and at the center of slab for Bi = 4, and for Γ = 0 and Γ= 4. The SOLVIA-TEMP solutions are compared with solutions obtained by Haji-Sheikh and Sparrow [1] who used a probability method. The temperature variations as displayed by SOLVIA-POST are shown in the bottom figure on page B28.3.

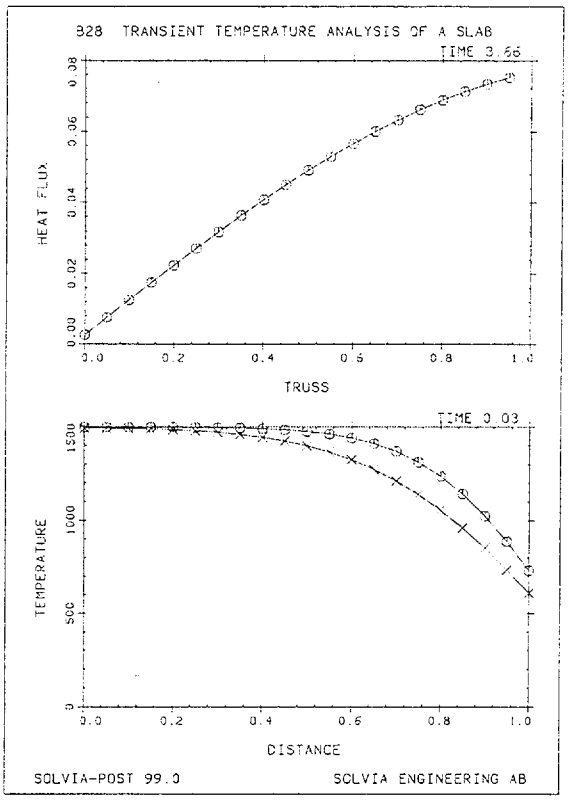
The right top figure on page B28.3 shows the distribution of heat flux along the model at time 3.66 and the temperature distributions at times 0.03 and 0.06.

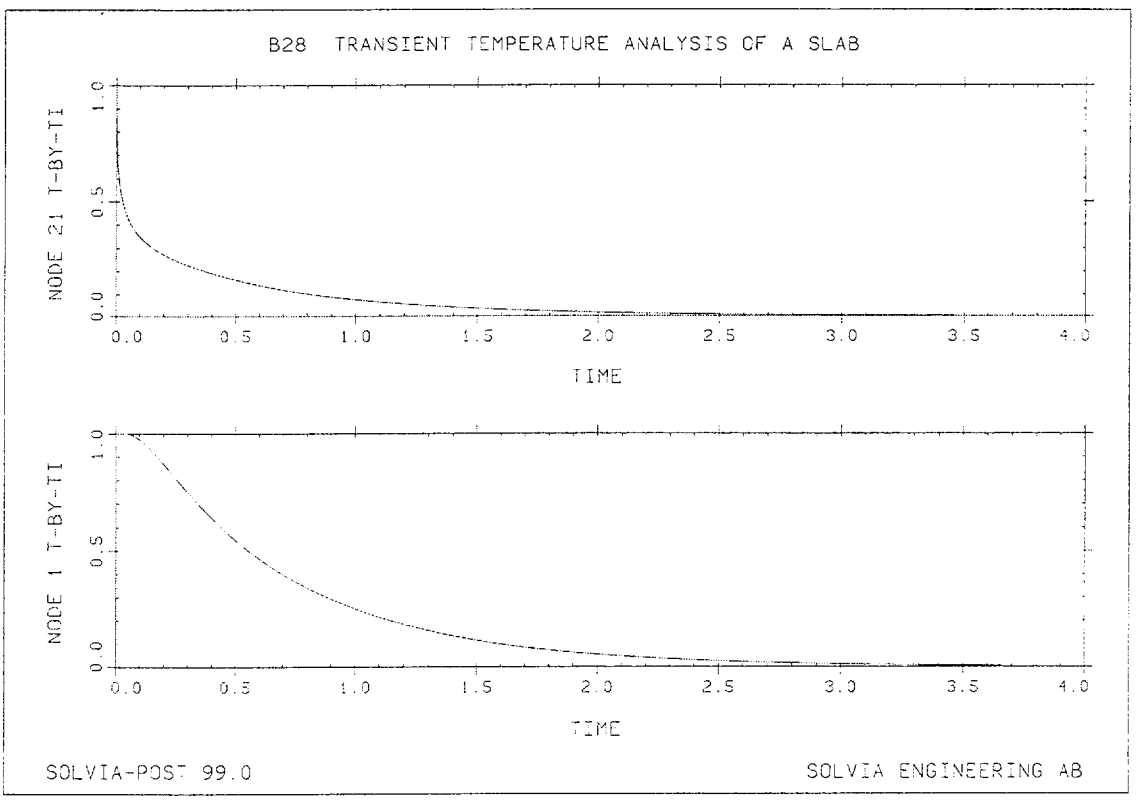
Reference

[1] Haji-Sheikh, A., and Sparrow, E.M., "The Solution of Heat Conduction Problems by Probability Methods", Trans. ASME, J. Heat Transfer, Vol. 39, pp. 121-131, 1967.




B28 TRANSIENT TEMPERATURE ANALYSIS OF A SLAB
ORIGINAL 0.1
Z
X
1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 21
ORIGINAL 0.1
Z
X
1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20
SOLVIA-PRE 99.0
SOLVIA ENGINEERING AB


B28 TRANSIENT TEMPERATURE ANALYSIS OF A SLAB
TIME 3.66
HEAT FLUX
0.00 0.02 0.04 0.06 0.08
0.0 0.2 0.4 0.6 0.8 1.0
TRUSS
TIME 0.03
TEMPERATURE
0 500 1000 1500
0.0 0.2 0.4 0.6 0.8 1.0
DISTANCE
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB


B28 TRANSIENT TEMPERATURE ANALYSIS OF A SLAB
NODE 21 T-BY-TI
0.0 0.5 1.0
0.0 0.5 1.0 1.5 2.0 2.5 3.0 3.5 4.0
TIME
NODE 1 T-BY-TI
0.0 0.5 1.0
0.0 0.5 1.0 1.5 2.0 2.5 3.0 3.5 4.0
TIME
SOLVIA-POST 99.0
SOLVIA ENGINEERING AB

SOLVIA-PRE input

```
HEADING  'B28  TRANSIENT TEMPERATURE ANALYSIS OF A SLAB'
*
DATABASE  CREATE
T-MASTER  NSTEP=60 DT=0.001 60 0.01 60 0.05
T-ANALYSIS  TRANSIENT HEATMATRIX=LUMPED METHOD=BACKWARD-EULER
T-ITERATION  TTOL=1.E-4
T-ITERATION  METHOD=MODIFIED-NEWTON
*
TIMEFUNCTION  1
 0. 0.  /  10. 0.
*
COORDINATES
 1  TO  21  1.
*
T-INITIAL  TEMPERATURES
 1  1498.1505  TO  21  1498.1505
*
T-MATERIAL  1 CONDUCTION K=0.01 SPECIFICHEAT=0.01
T-MATERIAL  2 CONVECTION H=0.04
T-MATERIAL  3 RADIATION EMISSIVITY=1. SIGMA=0.118958E-10,
              TEMPUNIT=KELVIN
*
EGROUP  1  TRUSS  MATERIAL=1
ENODES  /  1  1 2  TO  20  20 21
EDATA  /  1  1.
*
T-BOUNDARIES  SEGMENTS CON-MATERIAL=2 RAD-MATERIAL=3 INPUT=NODES
 1 21 1. 1.
T-LOADS  ENVIRONMENTAL CONVECTION=YES RADIATION=YES INPUT=NODES
 21 1.
*
SET  PLOTORIENTATION=PORTRAIT
MESH  VIEW=-Y NSYMBOLS=YES NNUMBERS=YES SUBFRAME=12
MESH  VIEW=-Y NSYMBOLS=YES ENUMBERS=YES
*
SOLVIA-TEMP
END
```

SOLVIA-POST input

```
*  B28 TRANSIENT TEMPERATURE ANALYSIS OF A SLAB
*
T-DATABASE  CREATE
WRITE  FILENAME='b28.lis'
*
EPLINE  NAME=TRUSS  /  1 1  TO  20 1
NPLINE  NAME=MEDIUM  /  1  TO  21
*
AXIS  ID=1 VMIN=0 VMAX=1 LABEL='DISTANCE'
AXIS  ID=2 VMIN=0 VMAX=1500 LABEL='TEMPERATURE'
*
SET  PLOTORIENTATION=PORTRAIT
SUBFRAME  12
ELINE  LINENAME=TRUSS KIND=TFLUX SYMBOL=1 OUTPUT=ALL
NLINE  LINENAME=MEDIUM KIND=TEMPERATURE XAXIS=1 YAXIS=2,
       TIME=0.03 OUTPUT=ALL SYMBOL=1
NLINE  LINENAME=MEDIUM KIND=TEMPERATURE XAXIS=-1 YAXIS=-2,
       TIME=0.06 OUTPUT=ALL SYMBOL=4 SUBFRAME=OLD
*
SET PLOTORIENTATION=LANDSCAPE
NVARIABLE T KIND=TEMPERATURE
CONSTANT TI 1498.1505
RESULTANT T-BY-TI 'T/TI'
NPOINT SURFACE NODE=21
NPOINT CENTER  NODE=1
SUBFRAME 12
RHISTORY POINT=SURFACE RESULTANT=T-BY-TI
RHISTORY POINT=CENTER  RESULTANT=T-BY-TI
*
NLIST
END
```

EXAMPLE B29

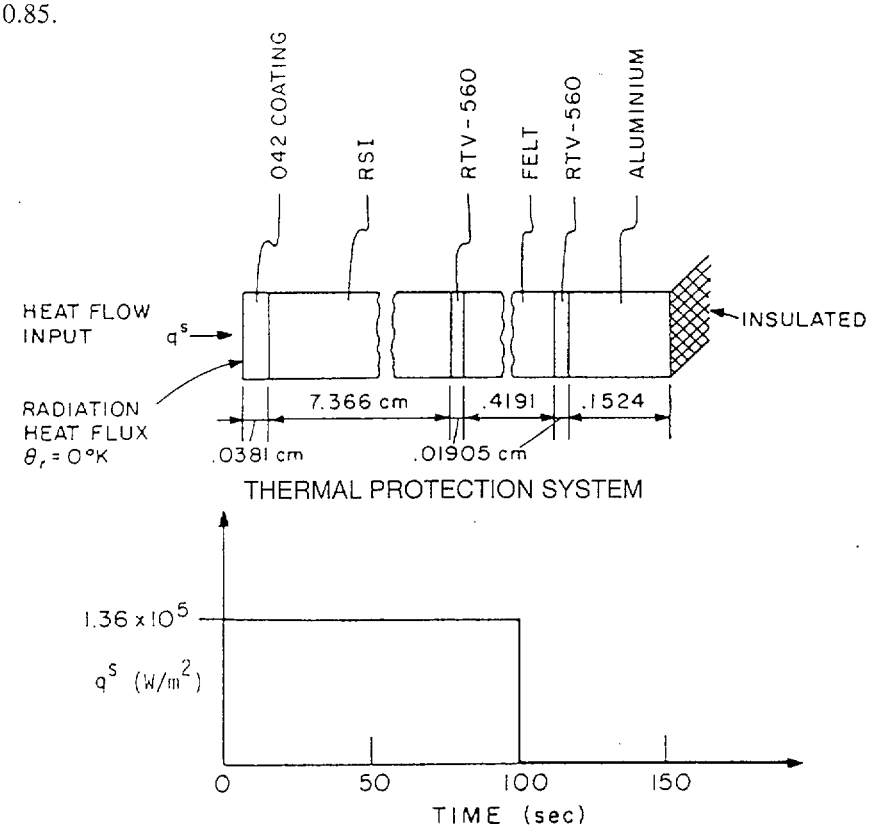
TRANSIENT TEMPERATURE ANALYSIS OF A SPACE SHUTTLE

Objective

To verify the behaviour of the TRUSS conduction element using nonlinear material models in a transient analysis.

Physical Problem

The space shuttle orbiter thermal protection system shown in the figure below is to be analyzed. The protection system is composed of different materials and the thermophysical properties of these materials are given on page B29.3. The uniform initial temperature is 322 K. At time $t = 0^+$ a step heat flow input is imposed on the surface of the thermal protection system and maintained for 100 sec. after which no heat flow input is imposed. Simultaneously, the surface of the protection system is exposed to radiation to a sink of absolute zero temperature $(\theta_r = 0\ \mathrm{K})$. It is desired to predict the transient surface temperature of the protection system. The emissivity coefficient of the radiation surface is 0.85.



Finite Element Model

The finite element model considered is shown in the bottom figures on next page. It consists of nineteen TRUSS conduction elements and one BOUNDARY SEGMENT for radiation. A lumped heat capacity matrix is used. Conductivity reformations and heat flow equilibrium iterations using the modified Newton-Raphson method are performed at each step of the analysis. The α-family time integration with $\alpha = 0.7$ and with time steps as shown in the table below are used:

Time (sec)	Time step DT (sec)	Number of time steps
$0 \le t \le 2$	0.05	40
$2 < t \le 10$	0.5	16
$10 < t \le 100$	1.0	90
$100 < t \le 110$	0.5	20
$110 < t \le 150$	1.0	40

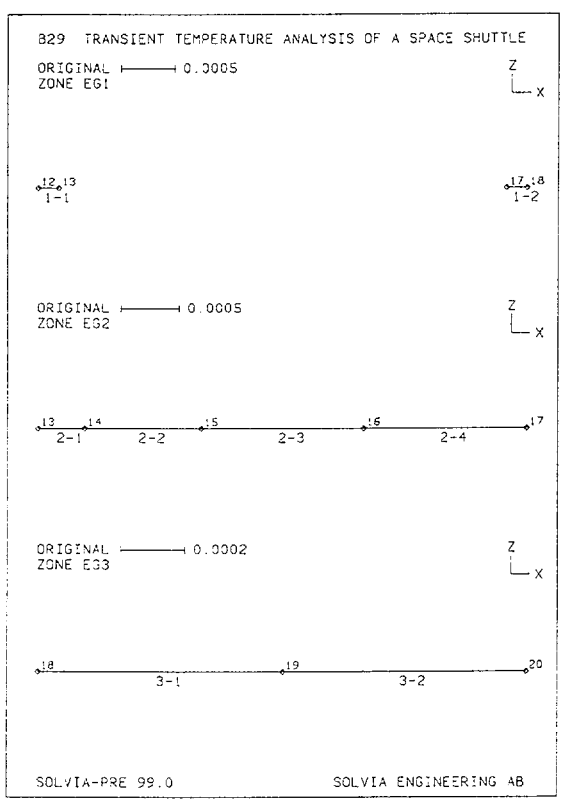
Solution Results

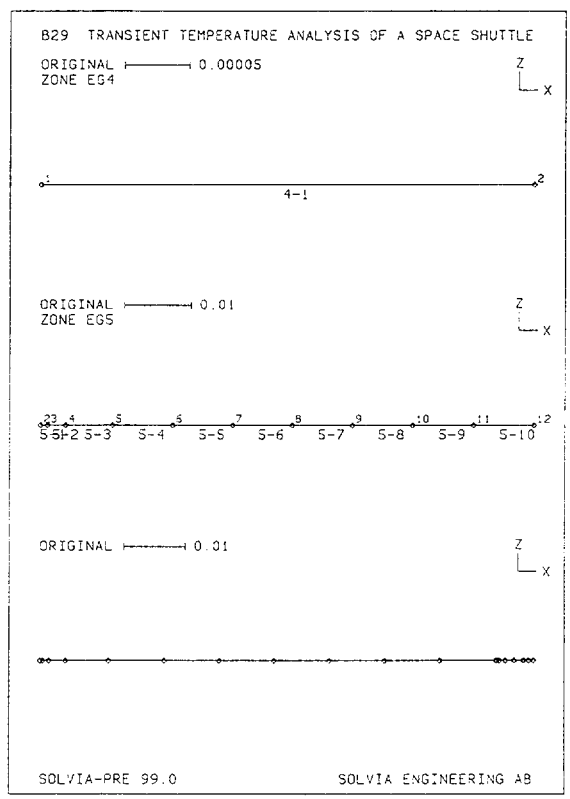
The solution for the surface temperature and the net heat flow from node 1 due to conduction is shown as function of time in the top figures on page B29.4 using the input data on pages B29.4 to B29.6. This same problem was also solved by Williams and Curry who used the finite difference method [1] and good agreement with the SOLVIA-TEMP solution presented here can be observed.

Note that the net nodal heat flow from node 1 due to conduction is not equal to the heat flow input to node 1, which is prescribed by T-LOADS HEATFLOW. The reason is that some of the input heat flow is used to heat the material lumped at node 1 and some of the input heat flow is radiated to the environment. The net remaining heat flow from node 1 is due to conduction and is shown in the right top figure of page B29.4.

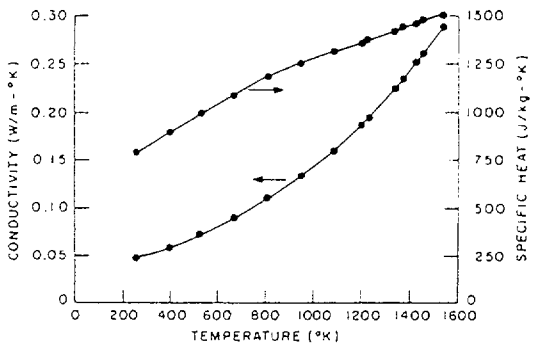
Reference

[1] Williams, S.D., and Curry, D.M., "An Implicit-Iterative Solution of the Heat Conduction Equation with a Radiation Boundary Condition", Int. J. Num. Meth. Eng., Vol. 11, pp. 1605-1620, 1977.

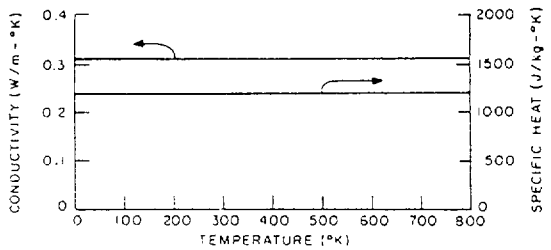




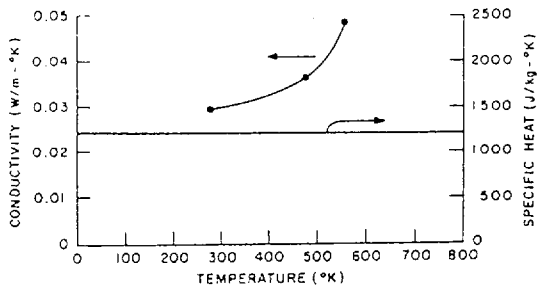
Thermophysical Properties



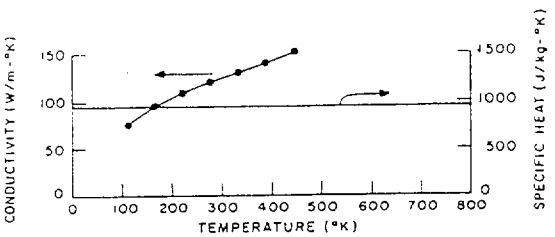
THERMOPHYSICAL PROPERTIES OF RSI
(density = 144.17 kg/m³)



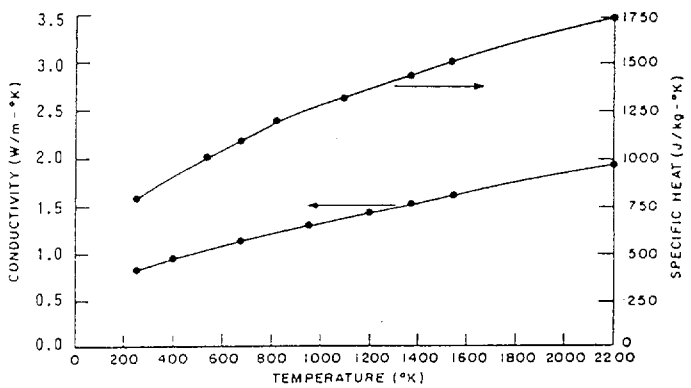
THERMOPHYSICAL PROPERTIES OF RTV-560
(density = 1409.62 kg/m³)



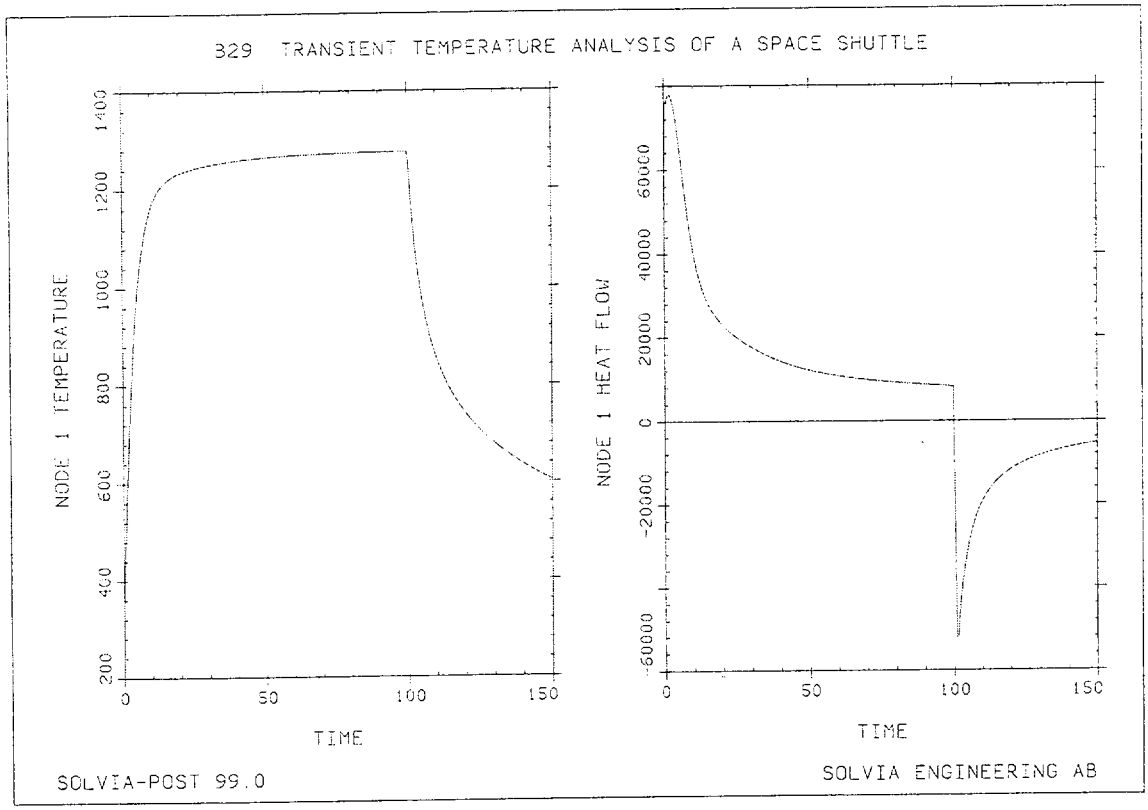
THERMOPHYSICAL PROPERTIES OF FELT
(density = 96.11 kg/m³)



THERMOPHYSICAL PROPERTIES OF ALUMINUM
(density = 2851.28 kg/m³)



THERMOPHYSICAL PROPERTIES OF 042 COATING
(density = 1665.92 kg/m³; emissivity ε = 0.8)



SOLVIA-PRE input

```
HEAD  'B29  TRANSIENT TEMPERATURE ANALYSIS OF A SPACE SHUTTLE'
*
DATABASE  CREATE
*
T-MASTER  NSTEP=40 DT=0.05 16 0.5 90 1.0 20 0.5 40 1.0
T-ANALYSIS  TYPE=TRANSIENT HEATMATRIX=LUMPED METHOD=ALPHA-FAMILY,
            ALPHA=0.7
T-ITERATION  TTOL=1.E-4 METHOD=MODIFIED-NEWTON
*
TIMEFUNCTION  1
 0. 1.  /  100. 1  /  101. 0.  /  200. 0.
*
TIMEFUNCTION  2
 0. 0.  /  200. 0.
*
COORDINATES
  1  /  2  0.000381  /  3  0.001381  /  4  0.004041
  5  0.011041   TO  12  0.074041
 13  0.0742315   /  14  0.0746285
 15  0.0756285  TO  17  0.0784225
 18  0.0786130  TO  20  0.0801370
*
T-INITIAL  TEMPERATURE  TREF=322.22
*
T-MATERIAL  1  CONDUCTION  K=0.3113 SPECIFICHEAT=1.71037652E6
*
```

SOLVIA-PRE input (cont.)

```
T-MATERIAL  2  TEMP-CONDUCTION
  277.78  0.0294  116616.03
  477.78  0.0363  116616.03
  555.56  0.0484  116616.03
*
T-MATERIAL  3  TEMP-CONDUCTION
  111.11   75.9611  2.743843E6
  166.67   95.2627  2.743843E6
  222.22  108.9606  2.743843E6
  277.78  120.7906  2.743843E6
  333.33  131.3753  2.743843E6
  388.89  141.9601  2.743843E6
  444.44  151.9222  2.743843E6
*
T-MATERIAL  4  TEMP-CONDUCTION
  255.56    0.8423  1.324340E6
  394.44    0.9512  1.672850E6
  672.22    1.1311  1.812254E6
  950.00    1.2971  1.986510E6
 1200.00    1.4338  2.195616E6
 1366.67    1.5272  2.404722E6
 1533.33    1.6137  2.509275E6
 2200.00    1.9472  2.897035E6
*
T-MATERIAL  5  TEMP-CONDUCTION
  255.56    0.0476  0.114609E6
  394.44    0.0592  0.129690E6
  533.33    0.0749  0.144770E6
  672.22    0.0922  0.156834E6
  811.11    0.1138  0.171914E6
  950.00    0.1354  0.180962E6
 1088.89    0.1629  0.190010E6
 1200.00    0.1889  0.196042E6
 1227.78    0.1960  0.199058E6
 1338.89    0.2262  0.205090E6
 1366.67    0.2349  0.208106E6
 1422.22    0.2522  0.211122E6
 1450.00    0.2608  0.212932E6
 1533.33    0.2883  0.217155E6
*
T-MATERIAL  6  RADIATION  EMISSIVITY=0.85 SIGMA=5.67E-8 TEMP=KELVIN
*
EGROUP  1  TRUSS  MATERIAL=1
ENODES  /  1 12 13  /  2 17 18
EDATA   /  1 1.
*
EGROUP  2  TRUSS  MATERIAL=2
ENODES  /  1 13 14  TO  4 16 17
EDATA   /  1 1.
*
EGROUP  3  TRUSS  MATERIAL=3
ENODES  /  1 18 19  TO  2 19 20
EDATA   /  1 1.
*
EGROUP  4  TRUSS  MATERIAL=4
ENODES  /  1 1 2
EDATA   /  1 1.
*
```

SOLVIA-PRE input cont.)

```
EGROUP  5  TRUSS  MATERIAL=5
ENODES  /  1 2 3  TO  10 11 12
EDATA   /  1 1.
*
T-BOUNDARY  SEGMENTS  RAD-MATERIAL=6 INPUT=NODES
 1 1 1. 1.
*
T-LOADS  HEATFLOW
 1 0.136E6 1
T-LOADS  ENVIRONMENT  RADIATION=YES INPUT=NODES
 1 1. 2
*
SET  VIEW=-Y PLOTORIENTATION=PORTRAIT NSYMBOL=YES,
     NNUMBERS=YES ENUMBERS=GROUP
SUBFRAME  13
MESH  ZONENAME=EG1
MESH  ZONENAME=EG2
MESH  ZONENAME=EG3
SUBFRAME  13
MESH  ZONENAME=EG4
MESH  ZONENAME=EG5
MESH  NNUMBERS=NO ENUMBERS=NO
*
SOLVIA-TEMP
END
```

SOLVIA-POST input

```
*  B29  TRANSIENT TEMPERATURE ANALYSIS OF A SPACE SHUTTLE
*
T-DATABASE  CREATE
*
WRITE  FILENAME='b29.lis'
*
SUBFRAME  21
NHISTORY  NODE=1 KIND=TEMPERATURE OUTPUT=ALL
NHISTORY  NODE=1 KIND=TFLOW
END
```