Introductory Examples Manual for LS-DYNA® Users

June, 2013
(draft)

James M. Kennedy

LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)

Table of Contents

Introduction

This document presents some LS-DYNA examples providing a basic guide in different disciplines like:

- Structural static (stress analysis, buckling analysis and modal analysis)
- Structural dynamic (vibrations and impact)
- Thermal analysis (heat transfer via conduction, convection and radiation)

This guide is mainly addressed to first-time users. The input files are always present for each problem, using the KEYWORD input format. For sake of briefness, in most problems the full node and element definitions (also, some load segments) are omitted.

Several of the problems present a closed-form solution, while others (the majority) a reference solution obtained by using an arbitrary refined mesh (NAFEMS Benchmarks). In these cases, the obtained value vs. the reference solution value is reported. Most of the problems are implicit ones. Problem-specific keywords are listed under the title of each problem.

This guide refers to LS-DYNA v.971, but most of the problems also run on the 970, 960 and 950 versions. All problems have been tested using a double precision executable.

To report inaccuracies and/or comments, please contact support at the following email address: support@lstc.com

Benchmark References

Example 1. Skew Plate with Normal Pressure (thin shell mesh)
The Standard NAFEMS Benchmarks, NAFEMS Report TNSB, Rev. 3, October, 1990, Test LE6.

Example 2. Skew Plate with Normal Pressure (thick shell mesh)
The Standard NAFEMS Benchmarks, NAFEMS Report TNSB, Rev. 3, October, 1990, Test LE6.

Example 3. Elliptical Thick Plate under Normal Pressure (coarse mesh)
Davies, G.A.O., Fenner, R.T., and Lewis, R.W., *NAFEMS Background to Benchmarks*, June, 1992, Test LE10.

Example 4. Elliptical Thick Plate under Normal Pressure (fine mesh)
Davies, G.A.O., Fenner, R.T., and Lewis, R.W., *NAFEMS Background to Benchmarks*, June, 1992, Test LE10.

Example 5. Snap-Back under Displacement Control
NAFEMS Non-Linear Benchmarks, NAFEMS Report NNB, Rev. 1, October, 1989, Test NL4.

Example 6. Straight Cantilever Beam with Axial End Point Load
NAFEMS Non-Linear Benchmarks, NAFEMS Report NNB, Rev. 1, October, 1989, Test NL6.

Example 7. Lee's Frame Buckling Problem
NAFEMS Non-Linear Benchmarks, NAFEMS Report NNB, Rev. 1, October, 1989, Test NL7.

Example 8. Pin-Ended Double Cross: In-Plane Vibration
The Standard NAFEMS Benchmarks, NAFEMS Report TNSB, Rev. 3, October, 1990, Test FV2.

Example 9. Simply Supported Thin Annular Plate (coarse mesh)
Abbassian, F., Dawswell, D.J., and Knowles, N.C., *NAFEMS Selected Benchmarks for Natural Frequency Analysis*, November, 1987, Test 14.

Example 10. Simply Supported Thin Annular Plate (fine mesh)
Abbassian, F., Dawswell, D.J., and Knowles, N.C., *NAFEMS Selected Benchmarks for Natural Frequency Analysis*, November, 1987, Test 14.

Example 11. Transient Response to a Constant Force
Biggs, J.M., *Introduction to Structural Dynamics*, McGraw-Hill Book Co., Inc., New York, New York, 1964, pg. 50, ex. E.

Example 12. Simply Supported Square Plate: Out-of Plane Vibration (solid mesh)
Abbassian, F., Dawswell, D.J., and Knowles, N.C., *NAFEMS Free Vibration Benchmarks*, October, 2001, Test FV52.

Example 13. Simply Supported Square Plate: Out-of-Plane Vibration (thick shell mesh)
Abbassian, F., Dawswell, D.J., and Knowles, N.C., *NAFEMS Free Vibration Benchmarks*, October, 2001, Test FV52.

Example 14. Simply Supported Square Plate: Transient Forced Vibration (solid mesh)
Maguire, J., Dawswell, D.J., and Gould, L.,*NAFEMS Selected Benchmarks for Forced Vibration*, February, 1989, Test 21T.

Example 15. Simply Supported Square Plate: Transient Forced Vibration (thick shell mesh)
Maguire, J., Dawswell, D.J., and Gould, L., *NAFEMS Selected Benchmarks for Forced Vibration*, February, 1989, Test 21T.

Example 16. Transient Response of a Cylindrical Disk Impacting a Deformable Surface
Thomson, W.T., *Vibration Theory and Applications*, 2nd Printing, Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 1965, pg. 110, ex. 4.6-1.

Example 17. Natural Frequency of a Linear Spring-Mass System
Timoshenko, S.P., and Young, D.H., *Vibration Problems in Engineering*, 3rd Edition, D. Van Nostrand Co., Inc., New York, New York, 1955, pg.1.

Example 18. Natural Frequency of a Nonlinear Spring-Mass System
Timoshenko, S.P., and Young, D.H., *Vibration Problems in Engineering,* 3rd Edition, D. Van Nostrand Co., Inc., New York, New York, 1955, pg. 141.

Example 19. Buckling of a Thin Walled Cylinder Under Compression
Timoshenko, S.P., and Gere, J.M., *Theory of Elastic Stability*, McGraw-Hill Book Co., Inc., New York, New York, 1961, pg. 457.

Example 20. Membrane with a Hot Spot
Davies, G.A.O., Fenner, R.T., and Lewis, R.W., *NAFEMS Background to Benchmarks*, June, 1992, Test T1.

Example 21. 1D Transient Heat Transfer with Radiation
Davies, G.A.O., Fenner, R.T., and Lewis, R.W., *NAFEMS Background to Benchmarks*, June, 1992, Test T2.

Example 22. 1D Transient Heat Transfer in a Bar
Davies, G.A.O., Fenner, R.T., and Lewis, R.W., *NAFEMS Background to Benchmarks*, June, 1992, Test T3.

Example 23. 2D Heat Transfer with Convection
Davies, G.A.O., Fenner, R.T., and Lewis, R.W., *NAFEMS Background to Benchmarks*, June, 1992, Test T4.

Example 24. 3D Thermal Load
Davies, G.A.O., Fenner, R.T., and Lewis, R.W., *NAFEMS Background to Benchmarks*, June, 1992, Test LE11.

Example 25. Cooling of a Billet via Radiation
Siegal, R., and Howell, J.R., *Thermal Radiation Heat Transfer*, 3rd Edition, Hemisphere Publishing Corporation, 1981, pg. 229, problem 21.

Example 26. Pipe Whip
Lerencz, R.M., *Element-by-Element Preconditioning Techniques for Large-Scale, Vectorized Finite Element Analysis in Nonlinear Solid and Structural Mechanics*, Ph.D. Thesis, Department of Mechanical Engineering, Stanford University, Palo Alto, California, March, 1989, pg. 142, pipe whip.

Example 27. Aluminum Bar Impacting a Rigid Wall
Lerencz, R.M., *Element-by-Element Preconditioning Techniques for Large-Scale, Vectorized Finite Element Analysis in Nonlinear Solid and Structural Mechanics*, Ph.D. Thesis, Department of Mechanical Engineering, Stanford University, Palo Alto, California, March, 1989, pg. 86, rod impact.

1. Skew Plate with Normal Pressure (thin shell mesh)

Keywords:

`*CONTROL_IMPLICIT_GENERAL`
`*CONTROL_IMPLICIT_SOLUTION`

Description:

A skew plate of equal side lengths *L* and thickness *t* is subjected to a normal pressure *P* on the top face (Figure 1.1). The plate is meshed with thin shell elements with a 4 x 4 density. The plate is simply supported on four side faces, $U_Z = 0$. Determine the maximum principal stress at plate center point E on the bottom surface.

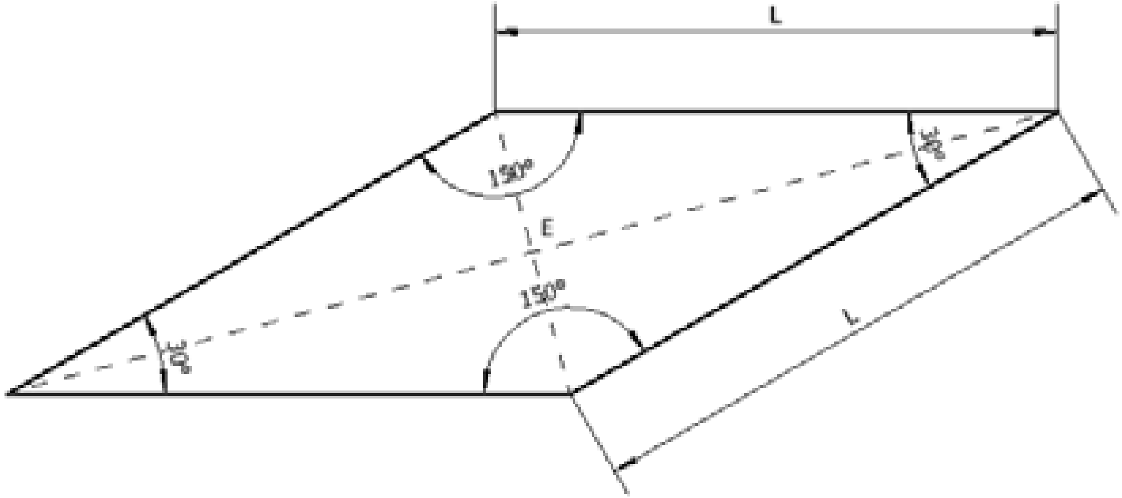

Figure 1.1 – Sketch representing the structure.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Static	Pressure	Linear	Linear	-	Implicit	1-Linear

Units:

kg, m, s, N, Pa, N-m (kilogram, meter, second, Newton, Pascal, Newton-meter)

Dimensional Data:

$L = 1.0\ m,\ t = 0.01\ m$

Material Data:

Mass Density	$\rho = 7.80 \times 10^{3}\ kg/m^{3}$
Young's Modulus	$E = 2.07 \times 10^{11}\ Pa$
Poisson's Ratio	$\nu = 0.3$

Load:

Pressure	$P = 7.0 \times 10^{2}\ Pa$

Element Types:

Belytschko-Tsay shell (elform=2)
S/R Hughes-Liu shell (elform=6)
Fully integrated shell (elform=16)

Material Models:

*MAT_001 or *MAT_ELASTIC

Results Comparison:

LS-DYNA maximum principal stress at plate center Point E (Node 13) on bottom surface plus its Z-displacement, U_Z, are compared with *Standard NAFEMS Benchmarks*, Test LE6.

Reference Condition - Point E (Node 13)	**Max Principal Stress (*Pa*)**	**U_Z (*m*)**
NAFEMS Benchmark Test LE6	0.802×10^{6}	-
Belytschko-Tsay shell (elform=2)	0.781×10^{6}	-1.616×10^{-5}
S/R Hughes-Liu shell (elform=6)	0.715×10^{6}	-1.507×10^{-5}
Fully integrated shell (elform=16)	0.696×10^{6}	-1.404×10^{-5}

These nodal displacement results were generated by *DATABASE_NODOUT keyword while the maximum principal stress results were generated by *DATABASE_ ELOUT.

LS-DYNA stress and strain output corresponds to integration point locations. Stress at a node is an artifact of the post-processor and represents an average of the surrounding integration point stresses (the value will likely be different with different post-processors).

Lobatto integration (intgrd=1 - *CONTROL_SHELL) was employed since it has an advantage in that the inner and outer integration points are on the shell surfaces. Gauss integration is the default through thickness integration rule (the default number of through thickness integration points is nip=2 - *SECTION_SHELL) in LS-DYNA, where 1-10 integration points may be specified, whereas, with Lobatto integration, 3-10 integration points may be specified (for 2 point integration, the Lobatto rule is very inaccurate).

For this coarse meshing, the one-point quadrature (low order) Belytschko-Tsay shell (elform=2) provides a good stress comparison (Figure 1.2).

The higher order, selectively reduced integration Hughes-Liu shell (elform=6) and the fully integrated Belytschko-Tsay shell (elform=16), which uses a 2x2 in-plane quadrature, provide comparatively stiffer results (Figure 1.3 and 1.4), probably due to the coarse meshing.

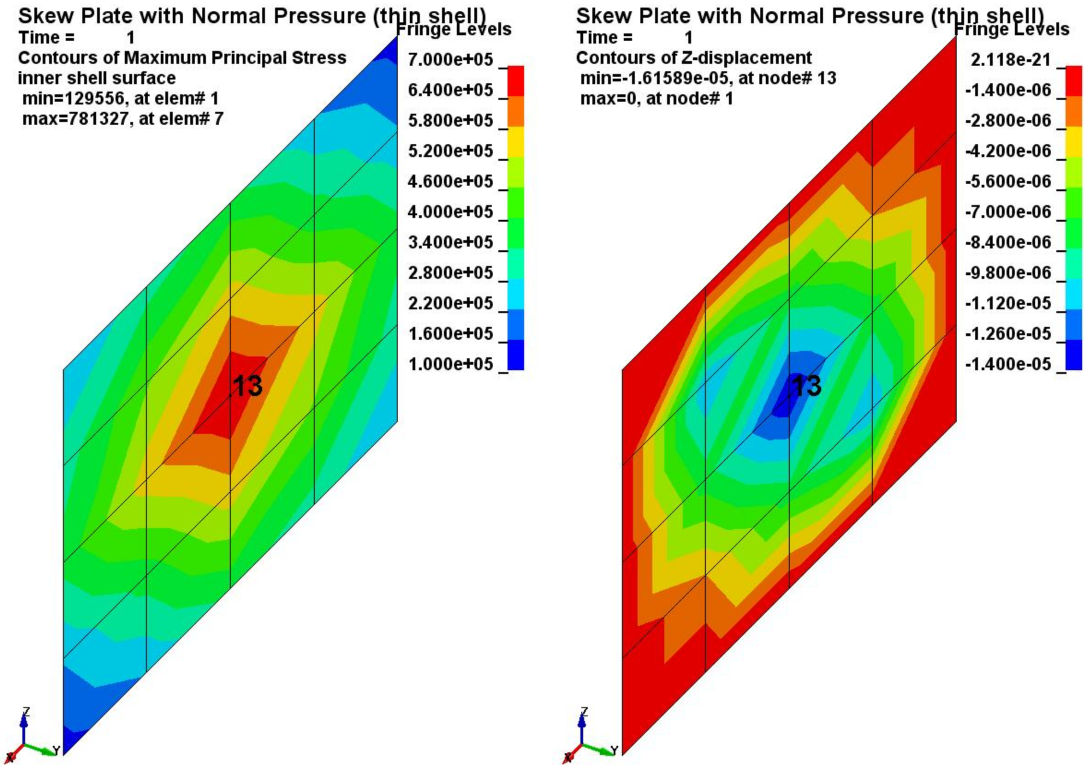

Figure 1.2 – Element formulation 2 (Belytschko-Tsay).

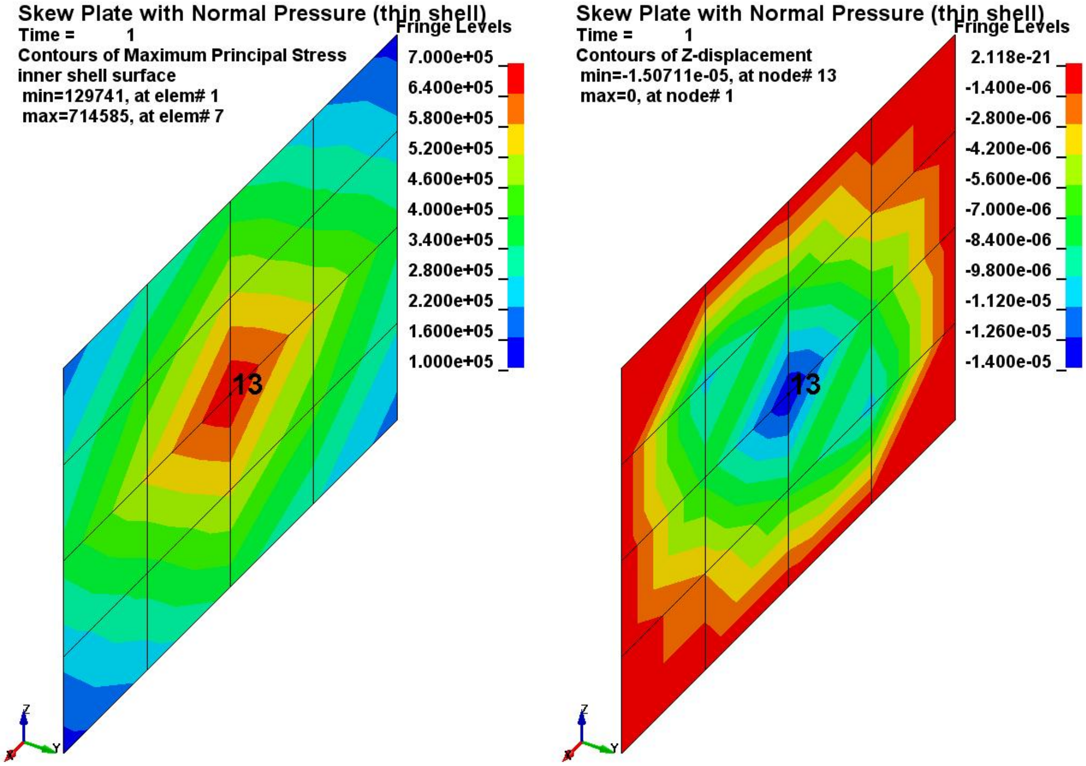

Figure 1.3 – Element formulation 6 (S/R Hughes-Liu).

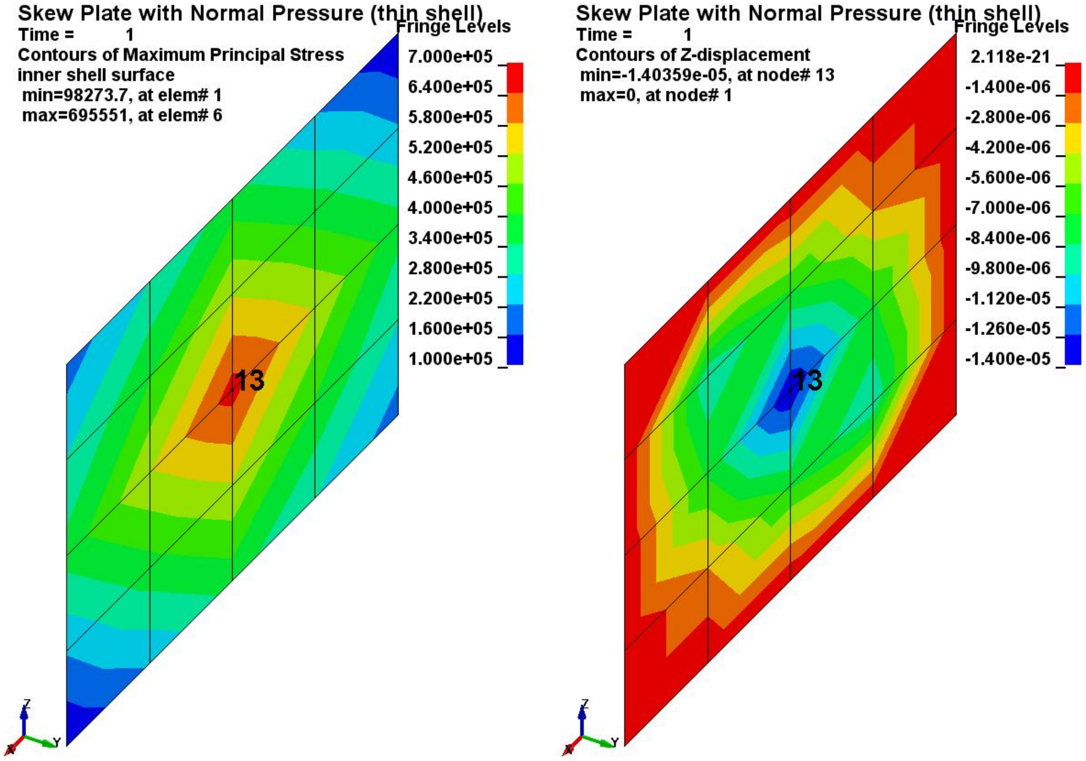

Figure 1.4 – Element formulation 16 (fully integrated).

Input Deck:

```
*KEYWORD
*TITLE
Skew Plate with Normal Pressure (thin shell mesh)
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1       0.0         2         1         2
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
         1        11        15  0.001000  0.010000       0.0  0.900000  1.000000
$#   dnorm    diverg     istif   nlprint
         2         1         1         2
$#  arcctl    arcdir    arclen    arcmth    arcdmp
         0         1       0.0         1         2
*CONTROL_SHELL
$#  wrpang     esort     irnxx    istupd    theory       bwc     miter      proj
  20.00000         0         0         0         2         2         1
$# rotascl    intgrd    lamsht    cstyp6    tshell    nfail1    nfail4
       0.0         1
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  1.000000         0       0.0       0.0       0.0
*DATABASE_ELOUT
$# .....dt
   1.0E-01
*DATABASE_BINARY_D3PLOT
$# dt/cycl
  0.100000
*DATABASE_HISTORY_SHELL
$#    eid1      eid2      eid3      eid4       ei5      eid6      eid7      eid8
         6         7        10        11
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo    dattyp
         1         0       0.0       0.0       0.0       0.0
$#                a1                  o1
                 0.0                 0.0
          1.00000000         700.0000000
*ELEMENT_SHELL
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1       6       7       2

      16       1      19      24      25      20
*NODE
$#   nid               x               y               z      tc      rc
       1             0.0             0.0             0.0       3

      25      1.86602540      0.50000000             0.0       3
*PART
$# title
material type # 1  (Elastic)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_SHELL
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
         1         2       0.0         5         0       0.0
$         1         6       0.0         5         0       0.0
$         1        16       0.0         5         0       0.0
$#      t1        t2        t3        t4      nloc     marea
  0.010000  0.010000  0.010000  0.010000         0       0.0
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1  7800.0002.1000e+11  0.300000       0.0       0.0       0.0
*LOAD_SEGMENT
$#    lcid        sf        at        n1        n2        n3        n4
         1  1.000000       0.0         1         6         7         2
         1  1.000000       0.0         2         7         8         3
         1  1.000000       0.0         3         8         9         4
         1  1.000000       0.0         4         9        10         5
         1  1.000000       0.0         6        11        12         7
```

```
         1  1.000000       0.0          7         12         13          8
         1  1.000000       0.0          8         13         14          9
         1  1.000000       0.0          9         14         15         10
         1  1.000000       0.0         11         16         17         12
         1  1.000000       0.0         12         17         18         13
         1  1.000000       0.0         13         18         19         14
         1  1.000000       0.0         14         19         20         15
         1  1.000000       0.0         16         21         22         17
         1  1.000000       0.0         17         22         23         18
         1  1.000000       0.0         18         23         24         19
         1  1.000000       0.0         19         24         25         20
*END
```

Notes:

2. Skew Plate with Normal Pressure (thick shell mesh)

Keywords:

```
*CONTROL_IMPLICIT_GENERAL
*CONTROL_IMPLICIT_SOLUTION
```

Description:

A skew plate of equal side lengths *L* and thickness *t* is subjected to a normal pressure *P* on the top face (Figure 2.1). The plate is meshed with thick shell elements with a 4 x 4 density. The plate is simply supported on four side faces of the bottom surface, $U_Z = 0$. Determine the maximum principal stress at plate center point E on the bottom surface.

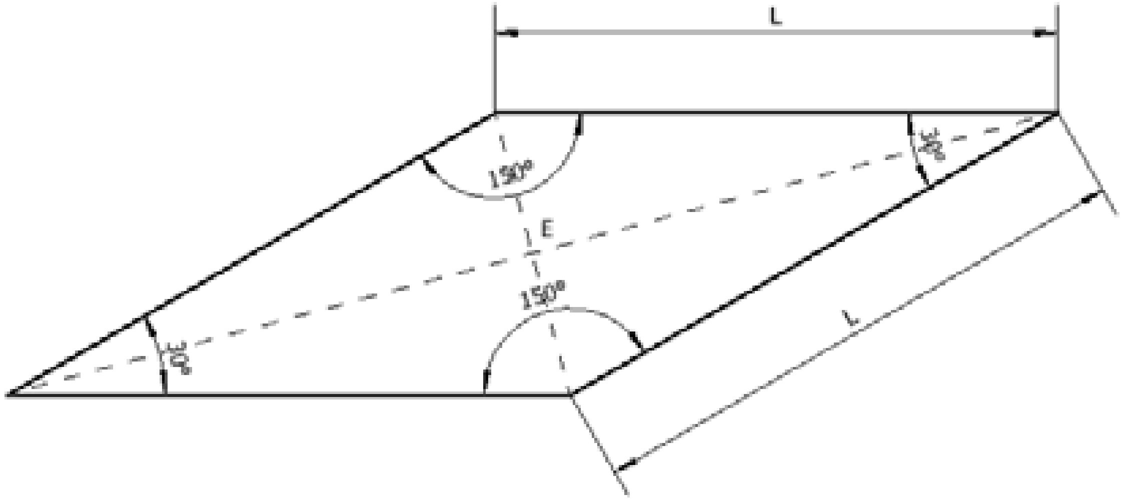

Figure 2.1 – Sketch representing the structure.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Static	Pressure	Linear	Linear	-	Implicit	1-Linear

Units:

kg, m, s, N, Pa, N-m (kilogram, meter, second, Newton, Pascal, Newton-meter)

Dimensional Data:

$L = 1.0\ m,\ t = 0.01\ m$

Material Data:

Mass Density	$\rho = 7.80\times10^{3}\ kg/m^{3}$
Young's Modulus	$E = 2.07\times10^{11}\ Pa$
Poisson's Ratio	$\nu = 0.3$

Load:

Pressure	$P = 7.0\times10^{2}\ Pa$

Element Types:

S/R 2x2 IPI thick shell (elform=2)
Assumed strain 2x2 IPI thick shell (elform=3)
Assumed strain RI thick shell (elform=5)

Material Models:

*MAT_001 or *MAT_ELASTIC

Results Comparison:

LS-DYNA maximum principal stress at plate center Point E (Node 113) on bottom surface plus its Z-displacement, U_Z, are compared with *Standard NAFEMS Benchmarks*, Test LE6.

Reference Condition - Point E (Node113)	**Max Principal Stress (*Pa*)**	U_Z **(*m*)**
NAFEMS Benchmark Test LE6	0.802×10^{6}	-
S/R 2x2 IPI thick shell (elform=2)	0.709×10^{6}	-1.496×10^{-5}
Assumed strain 2x2 IPI thick shell (elform=3)	0.021×10^{6} est.	-0.084×10^{-5}
Assumed strain RI thick shell (elform=5)	0.211×10^{6}	-0.849×10^{-5}

These nodal displacement results were generated by *DATABASE_NODOUT keyword while the maximum principal stress (nodal) results were generated by *DATABASE_ELOUT.

At least two elements through the thickness are usually recommended to capture bending response for assumed strain 2x2 IPI thick shell (elform=3) and assumed strain RI thick shell (elform=5) formulations.

LS-DYNA stress and strain output corresponds to integration point locations. Stress at a node is an artifact of the post-processor and represents an average of the surrounding integration point stresses (the value will likely be different with different post-processors).

Lobatto integration (intgrd=1 - *CONTROL_SHELL) was employed since it has an advantage in that the inner and outer integration points are on the shell surfaces. Gauss integration is the default through thickness integration rule (the default number of through thickness integration points is nip=2 - *SECTION_TSHELL) in LS-DYNA, where 1-10 integration points may be specified, whereas, with Lobatto integration, 3-10 integration points may be specified (for 2 point integration, the Lobatto rule is very inaccurate).

Only the higher order selectively reduced 2x2 IPI thick shell (elform=2) provides a reasonable stress comparison (Figure 2.2). As with other higher order options, this formulation provides a comparatively stiff result, again probably due to the coarse meshing.

The higher order assumed strain 2x2 IPI thick shell (elform=3) and assumed strain RI thick shell (elform=5) formulations do not provide acceptable solutions (Figure 2.3 and 2.4) since at least two elements through the thickness are usually recommended to capture bending response.

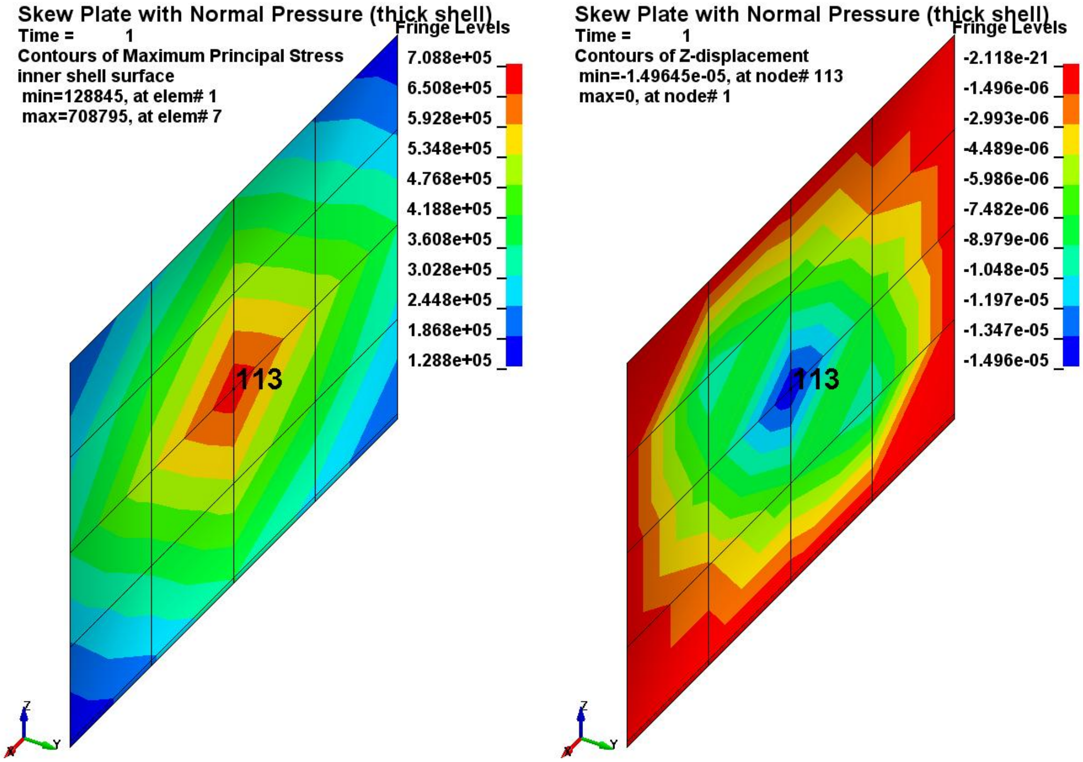

Figure 2.2 – Element formulation 2 (S/R 2x2 IPI).

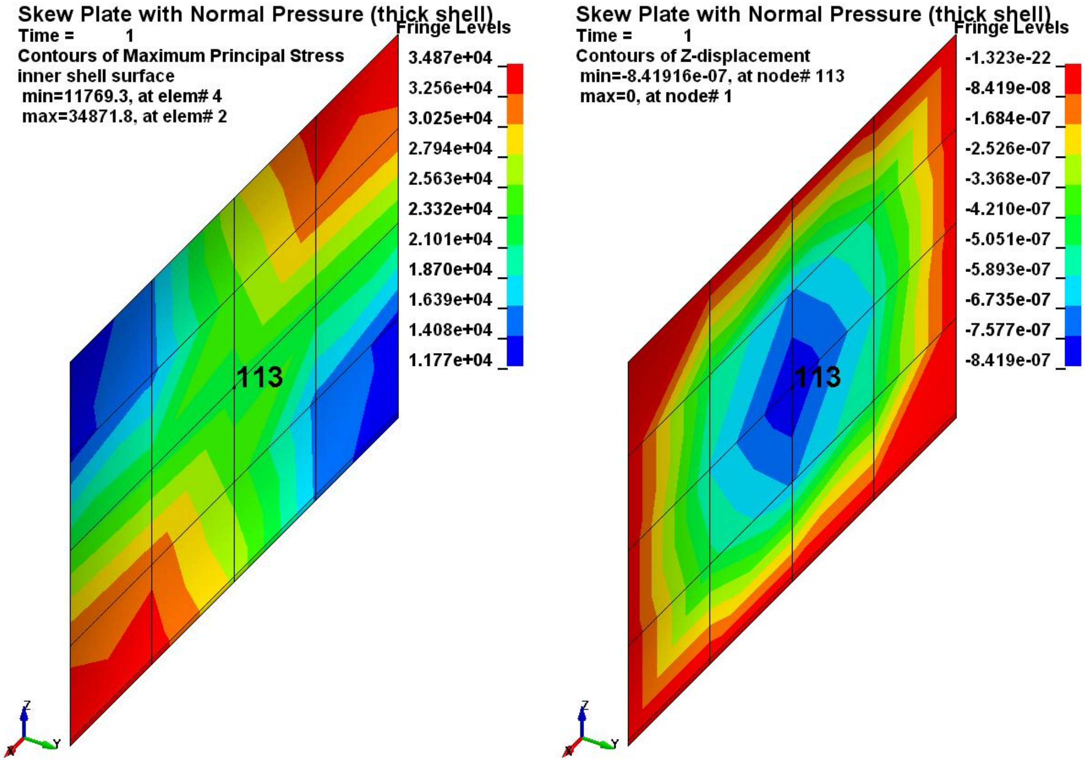

Figure 2.3 – Element formulation 3 (assumed strain 2X2 IPI).

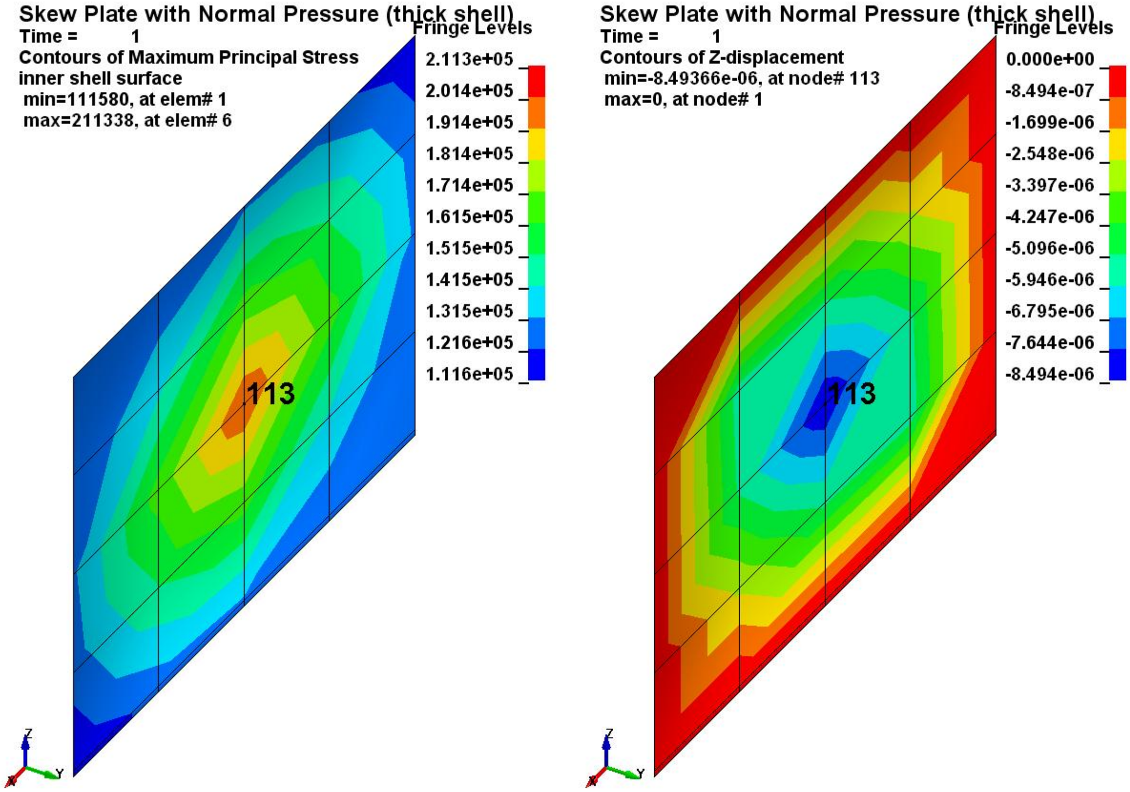

Figure 2.4 - Element formulation 5 (assumed strain RI).

Input Deck:

```
*KEYWORD
*TITLE
Skew Plate with Normal Pressure (thick shell)
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1       0.0         2         1         2
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
         1        11        15  0.001000  0.010000       0.0  0.900000  1.000000
$#   dnorm    diverg     istif   nlprint
         2         1         1         2
$#  arcctl    arcdir    arclen    arcmth    arcdmp
         0         1       0.0         1         2
*CONTROL_SHELL
$#  wrpang     esort     irnxx    istupd    theory       bwc     miter      proj
  20.00000         0         0         0         2         2         1
$# rotascl    intgrd    lamsht    cstyp6    tshell    nfail1    nfail4
       0.0         1
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  1.000000         0       0.0       0.0       0.0
*DATABASE_ELOUT
$# dt/cycl
   1.0E-01
*DATABASE_BINARY_D3PLOT
$# dt/cycl
  0.100000
*DATABASE_HISTORY_TSHELL
$#    eid1      eid2      eid3      eid4       ei5      eid6      eid7      eid8
         6         7        10        11
*DEFINE_CURVE
```

```
$#    lcid      sdir       sfa       sfo      offa      offo    dattyp
         1         0       0.0       0.0       0.0       0.0
$#                 a1                  o1
                 0.0                 0.0
          1.00000000         700.0000000
*ELEMENT_TSHELL
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1       6       7       2     101     106     107     102

      16       1      19      24      25      20     119     124     125     120
*NODE
$#   nid               x               y               z      tc      rc
       1             0.0             0.0             0.0       3

     125      1.86602540      0.50000000           0.010
*PART
$# title
material type # 1  (Elastic)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_TSHELL
$#   secid    elform      shrf       nip     propt   qr/irid     icomp    tshear
         1         2       0.0         5         0       0.0
$         1         3       0.0         5         0       0.0
$         1         5       0.0         5         0       0.0
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1  7800.0002.1000e+11  0.300000       0.0       0.0       0.0
*LOAD_SEGMENT
$#    lcid        sf        at        n1        n2        n3        n4
         1  1.000000       0.0       101       106       107       102
         1  1.000000       0.0       102       107       108       103
         1  1.000000       0.0       103       108       109       104
         1  1.000000       0.0       104       109       110       105
         1  1.000000       0.0       106       111       112       107
         1  1.000000       0.0       107       112       113       108
         1  1.000000       0.0       108       113       114       109
         1  1.000000       0.0       109       114       115       110
         1  1.000000       0.0       111       116       117       112
         1  1.000000       0.0       112       117       118       113
         1  1.000000       0.0       113       118       119       114
         1  1.000000       0.0       114       119       120       115
         1  1.000000       0.0       116       121       122       117
         1  1.000000       0.0       117       122       123       118
         1  1.000000       0.0       118       123       124       119
         1  1.000000       0.0       119       124       125       120
*END
```

Notes:

3. Elliptical Thick Plate under Normal Pressure (coarse mesh)

Keywords:

```
*CONTROL_IMPLICIT_GENERAL
*CONTROL_IMPLICIT_SOLUTION
*CONTROL_IMPLICIT_SOLVER
```

Description:

An elliptical thick plate with thickness *t* is subjected to a normal pressure *P* on its top surface (Figure 3.1). The plate is meshed with solid hexahedra element with a 4 x 6 x 4 density. Face CC′D′D has no Y-direction displacement, $U_Y = 0$; face ABB′A′ has no X-direction displacement, $U_X = 0$; the X and Y displacements of face BCC′B′ are fixed, $U_X = U_Y = 0$; and the mid-plane (face BCC'B') has no X-, Y-, and Z-direction displacement, $U_X = U_Y = U_Z = 0$. Determine the direct stress along Y-direction at point D.

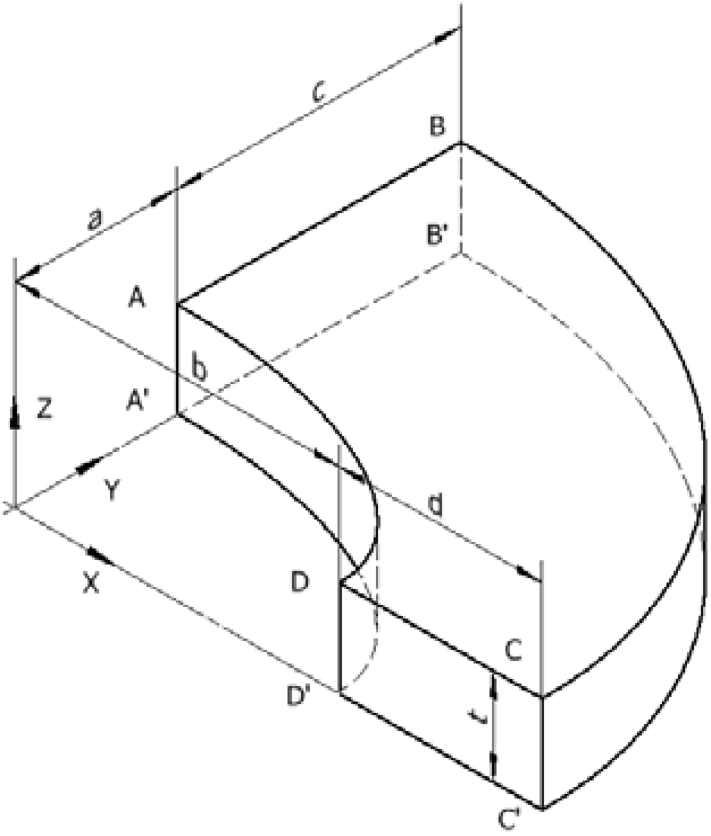

Figure 3.1 – Sketch representing the structure.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Static	Pressure	Linear	Linear	-	Implicit	1-Linear

Units:

kg, m, s, N, Pa, N-m (kilogram, meter, second, Newton, Pascal, Newton-meter)

Dimensional Data:

$a = 1.0\ m,\ b = 2.0\ m,\ c = 1.75\ m,\ d = 1.25\ m,\ t = 0.60\ m$

Material Data:

Mass Density	$\rho = 7.80 \times 10^{3}\ kg / m^{3}$
Young's Modulus	$E = 2.07 \times 10^{11}\ Pa$
Poisson's Ratio	$\nu\ = 0.3$

Load:

Pressure	$P = 1.0 \times 10^{6}\ Pa$

Element Types:

Constant stress solid (elform=1)
Fully integrated S/R solid (elform=2)
Fully integrated S/R solid - for poor aspect ratio (eff) - (elform=-1)
8 point enhanced strain solid (elform=18)

Material Models:

*MAT_001 or *MAT_ELASTIC

Results Comparison:

LS-DYNA Y-direction stress at plate edge Point D (Node 29) on top surface plus its Z-displacement, U_Z , are compared with *NAFEMS Background to Benchmarks*, Test LE10.

Reference Condition - Point D (Node 29)	**Axial Stress** σ_{yy} **(*Pa*)**	U_Z **(*m*)**
NAFEMS Benchmark Test LE10	-5.38×10^{6}	-
Constant stress solid (elform=1)	-4.78×10^{6} est	-1.022×10^{-4}
Fully integrated S/R solid (elform=2)	-4.13×10^{6}	-0.802×10^{-4}
Fully integrated S/R solid (elform=-1)	-5.35×10^{6}	-1.005×10^{-4}
8 point enhanced strain solid (elform=18)	-6.40×10^{6}	-0.973×10^{-4}

Estimated/extrapolated result calculated from $-3.67\times10^{6}\ Pa$ centroid value.

These nodal displacement results were generated by *DATABASE_NODOUT keyword while the axial stress (nodal) results were generated by *DATABASE_ELOUT (*elout* file) and *DATABASE_EXTENT_BINARY (*eloutdet* file provides detailed element output at integration points and connectivity nodes) keyword entries.

You can set intout=stress or intout= all (*DATABASE_EXTENT_BINARY) and have stresses output for all the integration points to a file called *eloutdet* (*DATABASE_ELOUT governs the output interval and *DATABASE_HISTORY_ SOLID governs which elements are output). Setting nodout=stress or nodout=all in *DATABASE_EXTENT_BINARY will write the extrapolated nodal stresses to *eloutdet*.

LS-DYNA stress and strain output corresponds to integration point locations. Stress at a node is an artifact of the post-processor and represents an average of the surrounding integration point stresses (the value will likely be different with different post-processors).

For this coarse mesh, the one-point quadrature (low order) constant stress solid (elform=1) element formulation (the LS-DYNA default) provides a fair stress comparison (Figure 3.2). Refinement of the mesh should provide a better comparison.

The higher order, fully integrated selectively reduced solid (elform=2), provides a comparatively stiff result (Figure 3.3), probably due to the coarse meshing.

The aspect ratio of the elements varies throughout the coarse meshing. An available option is the higher order, fully integrated S/R solid (the so-called efficient formulation choice) intended to address poor aspect ratios (elform=-1). This formulation provides a good comparison for this coarse meshing (Figure 3.4).

The 8 point enhanced strain solid (elform=18), developed for linear statics only, over predicts the stress (Figure 3.5); no explanation is currently available.

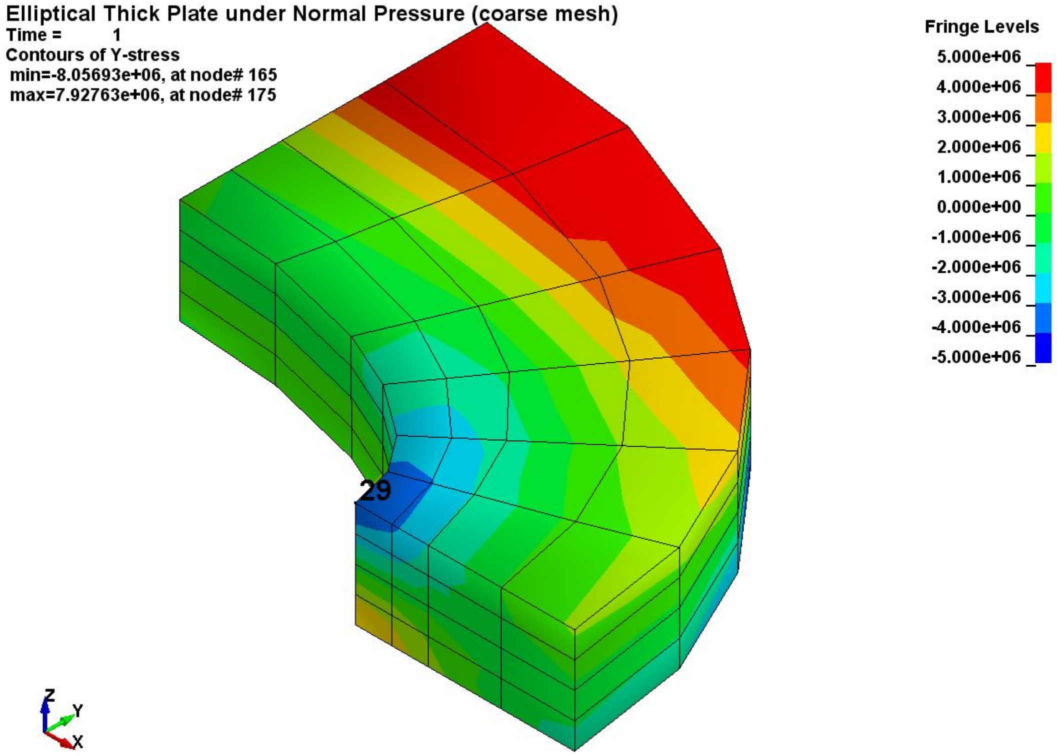

Figure 3.2 – Element formulation 1 (constant stress).

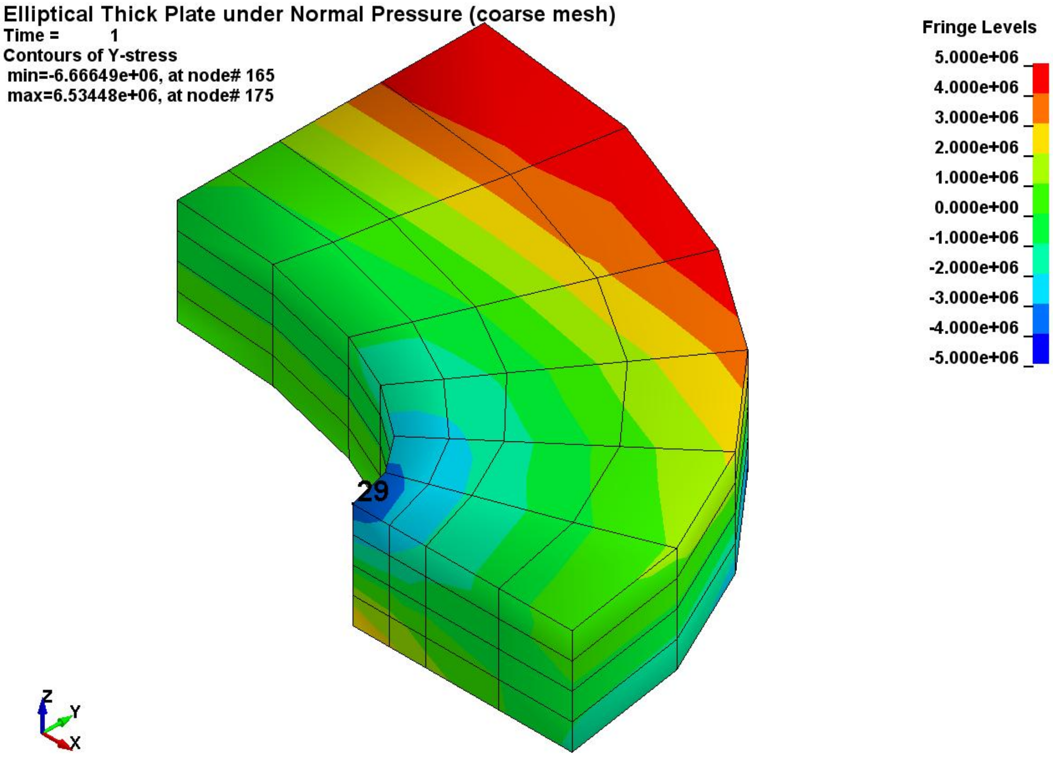

Figure 3.3 – Element formulation 2 (fully integrated S/R).

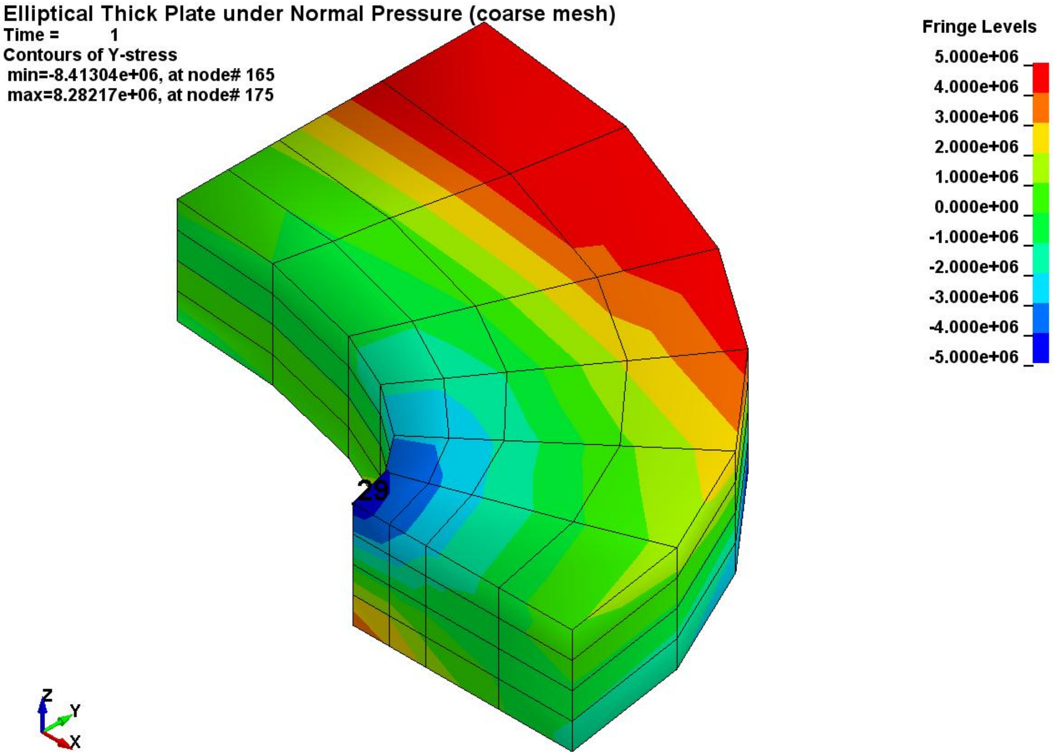

Figure 3.4 – Element formulation -1 (fully integrated S/R).

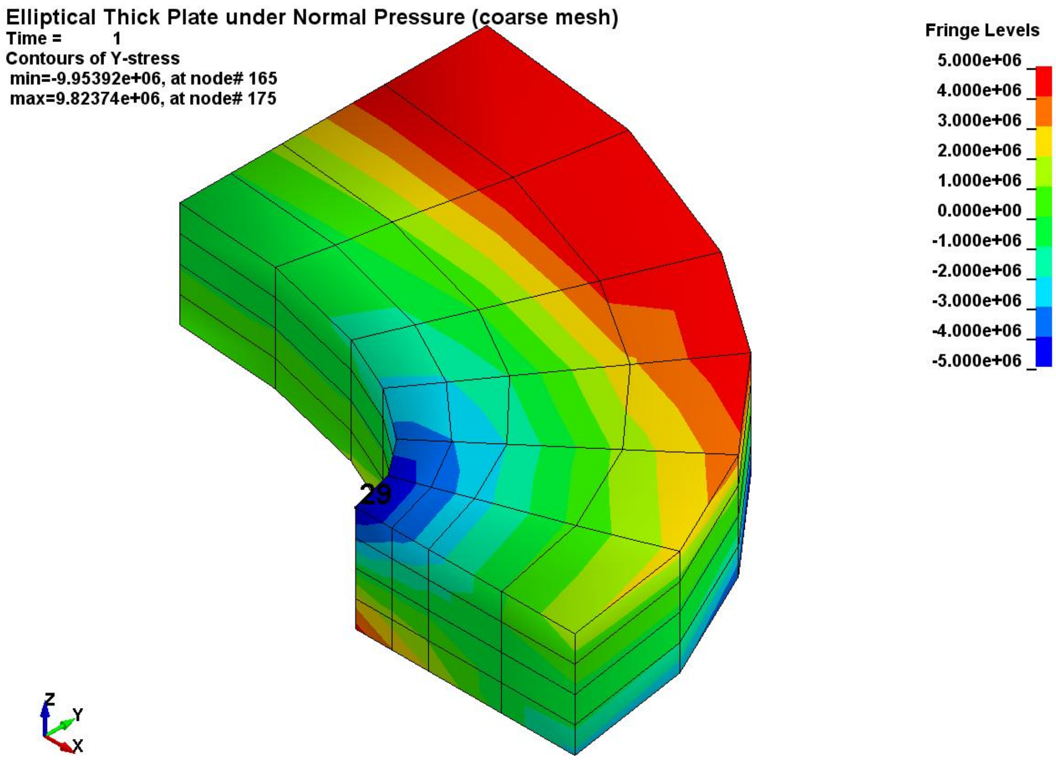

Figure 3.5 – Element formulation 18 (8 point enhanced strain).

Input Deck:

```
*KEYWORD
*TITLE
Elliptical Thick Plate under Normal Pressure (coarse mesh)
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1  0.100000         2         1         2
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol      rctol    lstol    abstol
         1        11        15  0.001000  0.010000   1.00e+10 0.900000  1.00e-10
*CONTROL_IMPLICIT_SOLVER
$#  lsolvr    lprint     negev     order      drcm    drcprm   autospc   autotol
         4         2         2         0         1       0.0         1       0.0
$#  lcpack
         2
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  1.000000         0       0.0       0.0       0.0
*DATABASE_ELOUT
$#      dt    binary      lcur     ioopt
 1.0000E-9         0         0
*DATABASE_BINARY_D3PLOT
$# dt/cycl
  1.000000
*DATABASE_EXTENT_BINARY
$#   neiph     neips    maxint    strflg    sigflg    epsflg     rtflg    engflg

$#  cmpflg    ieverp    beamip     dcomp      shge     stssz    n3thdt   ialemat

$# nintsld   pkp_sen      sclp     hydro     msscl     therm    intout    nodout
         8                 1.0                                  stress    stress
*DATABASE_HISTORY_SOLID
$#     id1       id2       id3       id4       id5       id6       id7       id8
```

```
         1         5         9        13
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo    dattyp
         1         0       0.0       0.0       0.0       0.0
$#                a1                  o1
                 0.0                 0.0
          1.00000000       1.0000000e+06
*ELEMENT_SOLID
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1      10      13       4       2      11      14       5

      96       1     168     172     174     170     169     173     175     171
*NODE
$#   nid               x               y               z      tc      rc
       1      2.00000000             0.0             0.0       2

     175             0.0      2.75000000      0.60000002       4
*PART
$# title
material type # 1  (Elastic)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_SOLID
$#   secid    elform       aet
         1         1         1
$         1         2         1
$         1        -1         1
$         1        18         1
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1  7800.0002.1000e+11  0.300000       0.0       0.0       0.0
*LOAD_SEGMENT
$#    lcid        sf        at        n1        n2        n3        n4
         1  1.000000       0.0        29        35        37        31
         1  1.000000       0.0        31        37        39        33
         1  1.000000       0.0        35        41        43        37
         1  1.000000       0.0        37        43        45        39
         1  1.000000       0.0        33        39        69        65
         1  1.000000       0.0        65        69        71        67
         1  1.000000       0.0        39        45        73        69
         1  1.000000       0.0        69        73        75        71
         1  1.000000       0.0        67        71        99        95
         1  1.000000       0.0        95        99       101        97
         1  1.000000       0.0        71        75       103        99
         1  1.000000       0.0        99       103       105       101
         1  1.000000       0.0        41       125       127        43
         1  1.000000       0.0        43       127       129        45
         1  1.000000       0.0       125       131       133       127
         1  1.000000       0.0       127       133       135       129
         1  1.000000       0.0        45       129       149        73
         1  1.000000       0.0        73       149       151        75
         1  1.000000       0.0       129       135       153       149
         1  1.000000       0.0       149       153       155       151
         1  1.000000       0.0        75       151       169       103
         1  1.000000       0.0       103       169       171       105
         1  1.000000       0.0       151       155       173       169
         1  1.000000       0.0       169       173       175       171
*END
```

Notes:

1. One should remember that the constant stress solid (elform=1), the fully integrated S/R solid (elform=2), and the fully integrated S/R solid (the so-called efficient formulation choice) intended to address poor aspect ratios (elform=-1) were originally developed for performing highly nonlinear, dynamic deformation simulations.

4. Elliptic Thick Plate under Normal Pressure (fine mesh)

Keywords:

```
*CONTROL_IMPLICIT_GENERAL
*CONTROL_IMPLICIT_SOLUTION
*CONTROL_IMPLICIT_SOLVER
```

Description:

An elliptical thick plate with thickness t is subjected to a normal pressure P on its top surface (Figure 4.1). The plate is meshed with solid hexahedra element with an 8 x 12 x 4 density. Face CC′D′D has no Y-direction displacement, $U_Y = 0$; face ABB′A′ has no X-direction displacement, $U_X = 0$**;** the X and Y displacements of face BCC′B′ are fixed, $U_X = U_Y = 0$**;** and the mid-plane (face BCC'B') has no X-, Y-, and Z-direction displacement, $U_X = U_Y = U_Z = 0$**.** Determine the direct stress along Y-direction at point D.

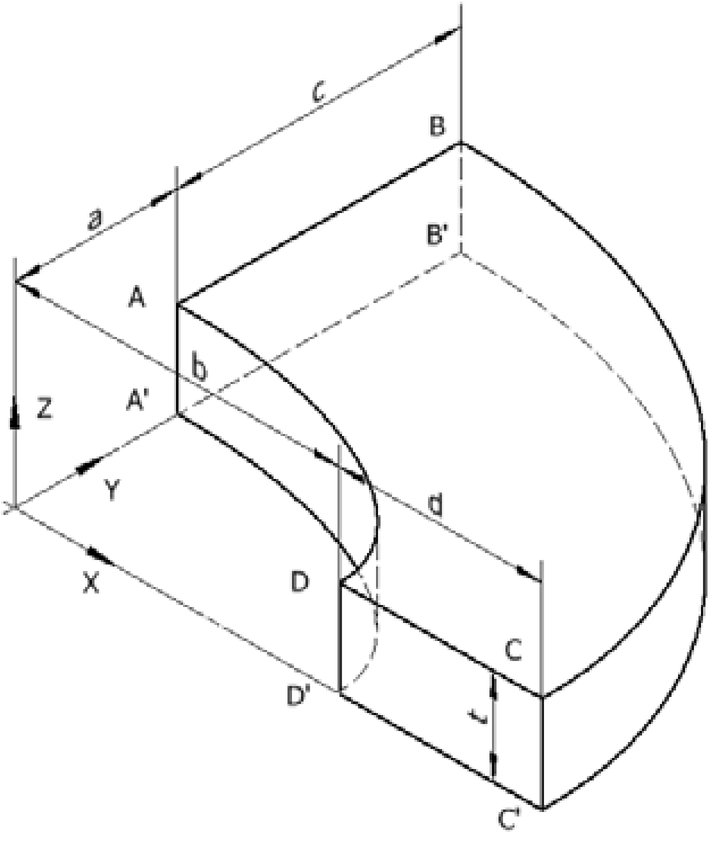

Figure 4.1 – Sketch representing the structure.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Static	Pressure	Linear	Linear	-	Implicit	1-Linear

Units:

kg, m, s, N, Pa, N-m (kilogram, meter, second, Newton, Pascal, Newton-meter)

Dimensional Data:

$a = 1.0\ m,\ b = 2.0\ m,\ c = 1.75\ m,\ d = 1.25\ m,\ t = 0.60\ m$

Material Data:

Mass Density	$\rho = 7.80 \times 10^3\ kg/m^3$
Young's Modulus	$E = 2.07 \times 10^{11}\ Pa$
Poisson's Ratio	$\nu = 0.3$

Load:

Pressure	$P = 1.0 \times 10^6\ Pa$

Element Types:

Constant stress solid (elform=1)
Fully integrated S/R solid (elform=2)
Fully integrated S/R solid (elform=-1)
8 point enhanced strain solid (elform=18)

Material Models:

*MAT_001 or *MAT_ELASTIC

Results Comparison:

LS-DYNA Y-direction stress at plate edge Point D (Node 77) on top surface plus its Z-displacement, U_Z, are compared with *NAFEMS Background to Benchmarks*, Test LE10.

Reference Condition - Point D (Node 77)	**Axial Stress** σ_{yy} **(*Pa*)**	U_Z **(*m*)**
NAFEMS Benchmark Test LE10	-5.38×10^{6}	-
Constant stress solid (elform=1)	-5.30×10^{6} est	-1.051×10^{-4}
Fully integrated S/R solid (elform=2)	-4.70×10^{6}	-0.947×10^{-4}
Fully integrated S/R solid (elform=-1)	-4.76×10^{6}	-0.991×10^{-4}
8 point enhanced strain solid (elform=18)	-6.28×10^{6}	-0.982×10^{-4}

Estimated/extrapolated result calculated from $-4.07\times10^{6}\ Pa$ centroid value.

These nodal displacement results were generated by *DATABASE_NODOUT keyword while the axial stress (nodal) results were generated by *DATABASE_ELOUT (*elout* file) and *DATABASE_EXTENT_BINARY (*eloutdet* file provides detailed element output at integration points and connectivity nodes) keyword entries.

You can set intout=stress or intout=all (*DATABASE_EXTENT_BINARY) and have stresses output for all the integration points to a file called *eloutdet* (*DATABASE_ELOUT governs the output interval and *DATABASE_HISTORY_ SOLID governs which elements are output). Setting nodout=stress or nodout=all in *DATABASE_EXTENT_BINARY will write the extrapolated nodal stresses to *eloutdet*.

LS-DYNA stress and strain output corresponds to integration point locations. Stress at a node is an artifact of the post-processor and represents an average of the surrounding integration point stresses (the value will likely be different with different post-processors).

For this fine mesh, the one-point quadrature (low order) constant stress solid (elform=1) element formulation (the LS-DYNA default) provides a better stress comparison (Figure 4.2), when compared to the coarse mesh.

The higher order, fully integrated selectively reduced solid (elform=2) still provides a comparatively stiff result (Figure 4.3); however, much improved over the coarse mesh.

Doubling the elements in the x-y plane (mesh refinement) appears to have minimized the aspect ratio issue seen in the coarse mesh. The higher order, fully integrated S/R solid (the so-called efficient formulation choice) intended to address poor aspect ratios (elform=-1) now provides a very similar result (Figure 4.4) to the fully integrated S/R solid (elform=2).

The 8 point enhanced strain solid (elform=18), developed for linear statics only, over predicts the stress result (Figure 4.5) by a fair amount (even with the change in mesh refinement); no explanation is presently available.

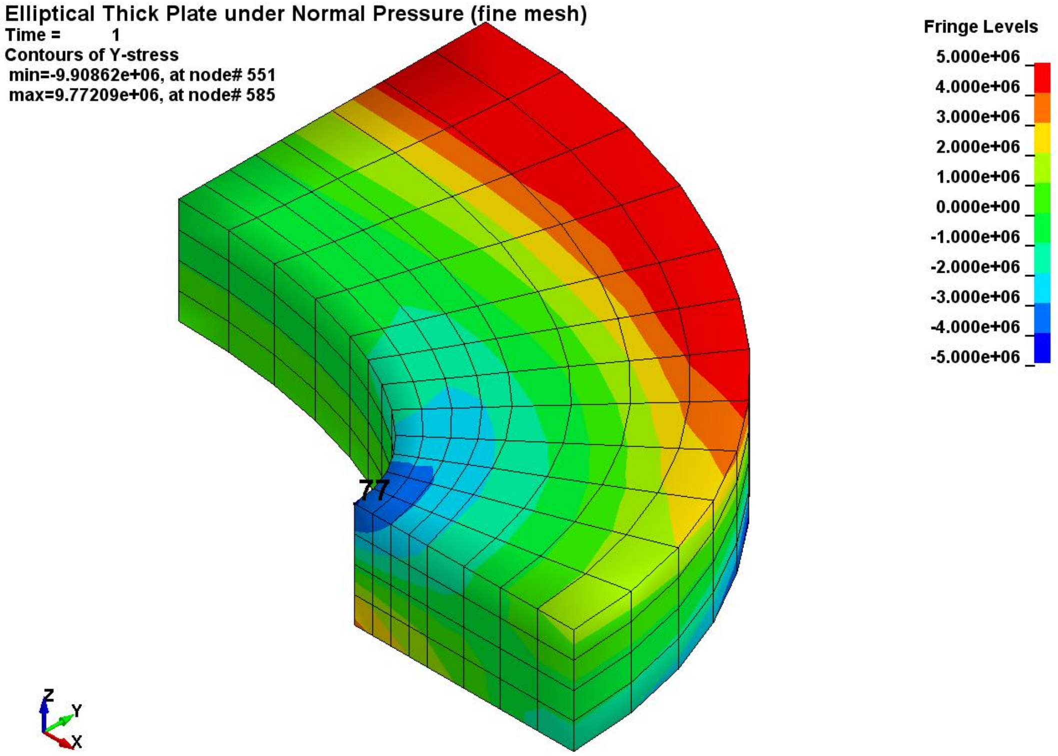

Figure 4.2 – Element formulation 1 (constant stress).

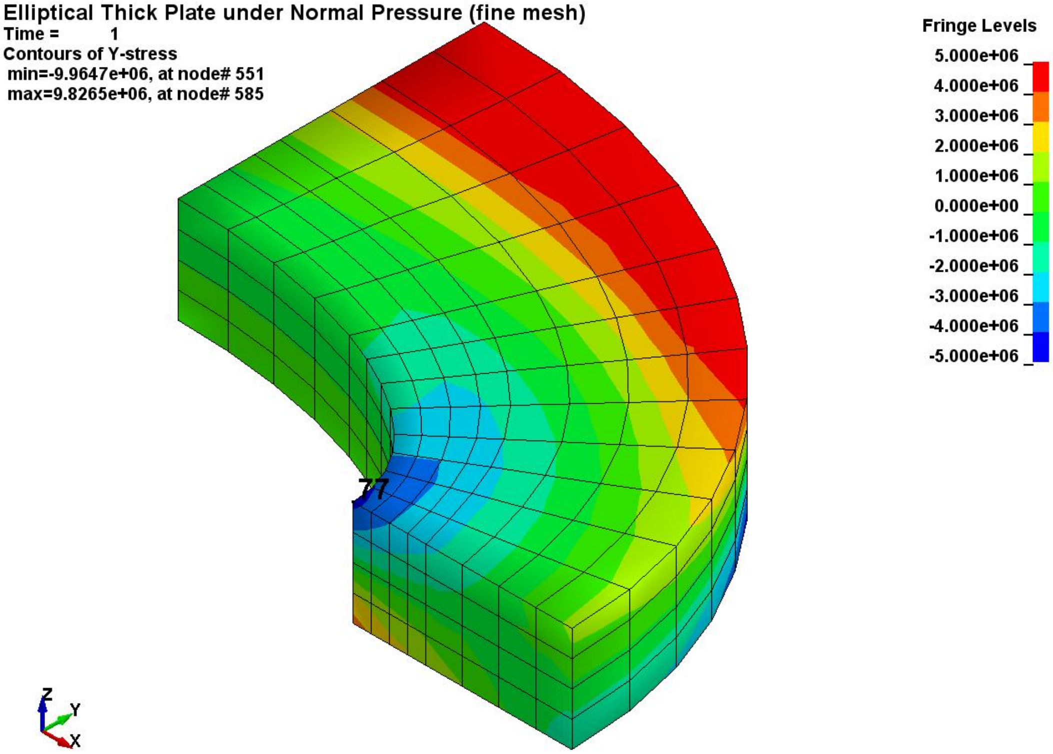

Figure 4.3 – Element formulation 2 (fully integrated S/R).

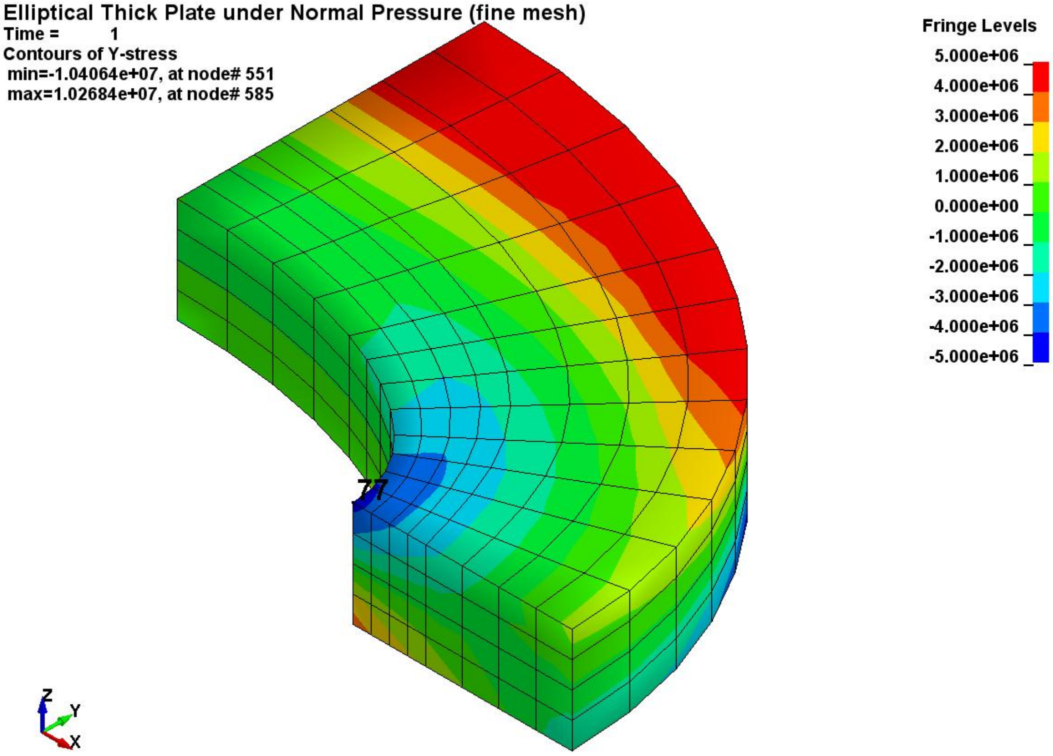

Figure 4.4 – Element formulation -1 (fully integrated S/R).

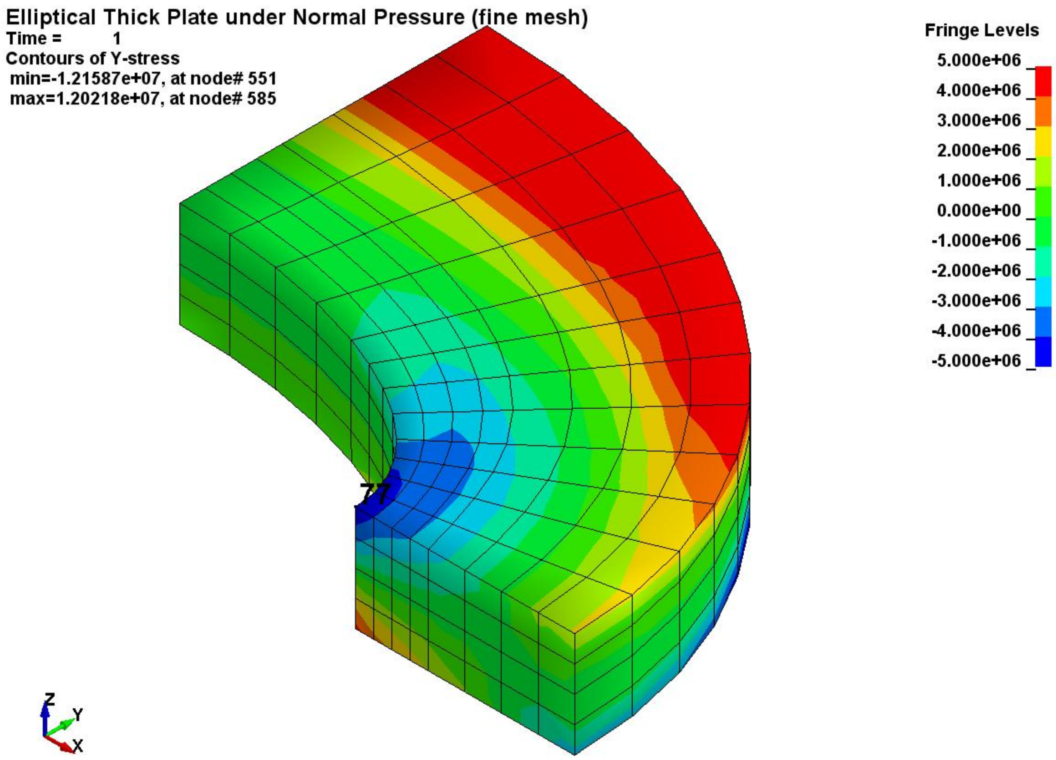

Figure 4.5 – Element formulation 18 (8 point enhanced strain).

Input Deck:

```
*KEYWORD
*TITLE
Thick Elliptic Plate under Normal Pressure (fine mesh)
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1  0.100000         2         0         2
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
         1        11        15  0.001000  0.010000  1.00e+10  0.900000  1.00e-10
*CONTROL_IMPLICIT_SOLVER
$#  lsolvr    lprint     negev     order      drcm    drcprm   autospc   autotol
         4         2         2         0         0       0.0         0       0.0
$#  lcpack
         2
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  1.000000         0       0.0       0.0       0.0
*DATABASE_ELOUT
$#      dt    binary      lcur     ioopt
 1.0000E-9         0         0
*DATABASE_BINARY_D3PLOT
$# dt/cycl
  1.000000
*DATABASE_EXTENT_BINARY
$#   neiph     neips    maxint    strflg    sigflg    epsflg     rtflg    engflg

$#  cmpflg    ieverp    beamip     dcomp      shge     stssz    n3thdt   ialemat

$# nintsld   pkp_sen      sclp     hydro     msscl     therm    intout    nodout
         8                 1.0                                  stress    stress
*DATABASE_HISTORY_SOLID
$#     id1       id2       id3       id4       id5       id6       id7       id8
        1         17        33        49
```

```
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo    dattyp
         1         0       0.0       0.0       0.0       0.0
$#                a1                  o1
                 0.0                 0.0
          1.00000000       1.0000000e+06
*ELEMENT_SOLID
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1      16      19       4       2      17      20       5

     384       1     574     582     584     576     575     583     585     577
*NODE
$#   nid               x               y               z      tc      rc
       1      2.00000000             0.0             0.0       2

     585             0.0      2.75000000      0.60000002       4
*PART
$# title
material type # 1  (Elastic)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_SOLID
$#   secid    elform       aet
         1         1         1
$         1         2         1
$         1        -1         1
$         1        18         1
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1  7800.000 2.100e+11  0.300000       0.0       0.0       0.0
*LOAD_SEGMENT
$#    lcid        sf        at        n1        n2        n3        n4
         1  1.000000       0.0        77        87        89        79
         1  1.000000       0.0        79        89        91        81

         1  1.000000       0.0       575       583       585       577
         1  1.000000       0.0       575       583       585       577
*END
```

Notes:

1. One should remember that the constant stress solid (elform=1), the fully integrated S/R solid (elform=2), and the fully integrated S/R solid (the so-called efficient formulation choice) intended to address poor aspect ratios (elform=-1) were originally developed for performing highly nonlinear, dynamic deformation simulations.

5. Snap-Back under Displacement Control

Keywords:

```
*CONTROL_IMPLICIT_AUTO
*CONTROL_IMPLICIT_GENERAL
*CONTROL_IMPLICIT_SOLUTION
```

Description:

In this problem the implicit arc length method is used in order to solve the snap-back of the system. With traditional Newton-based methods it is not possible to fully solve this problem, due to the null tangent stiffness matrix at a certain point of the analysis.

Three DOF are present.

A sketch representing the structure is shown below (Figure 5.1) along with a finite element representation (Figure 5.2).

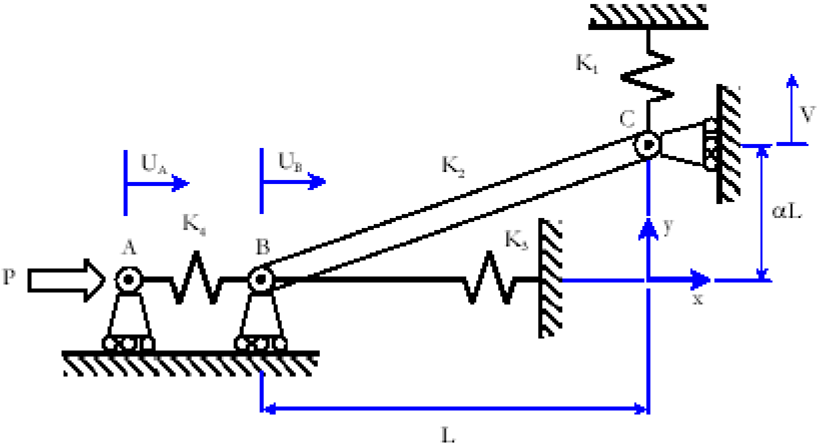

Figure 5.1 – Sketch representing the structure.

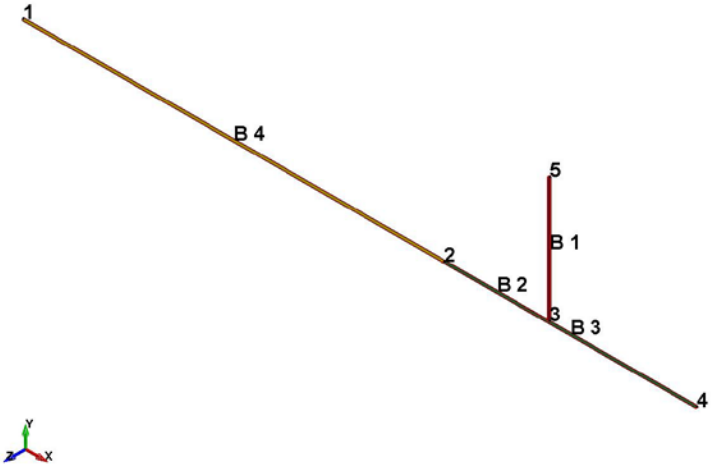

Figure 5.2 – The finite element representation of the problem. Four beams are used; the springs are modeled with discrete formulation, the truss is modeled with truss formulation. To avoid element inversion, the beams the springs are very long.

Analysis Summary:

Dim.	**Type**	**Load**	**Material**	**Geometry**	**Contact**	**Solver**	**Solution Method**
3D	Static	Force	Linear	Nonlinear	-	Implicit	6–Arc length w/BFGS

Units:

non-dimensional

Dimensional Data:

$L = 2.50\times10^{3}$, $\alpha = 1.00\times10^{-2}$, $\alpha L = 2.50\times10^{1}$

Material Data:

$AE = 5.0\times10^{7}$, $K_1 = 1.5$, $K_2 = AE / L\sqrt{(1+\alpha^2)} = 1.9999\times10^{3}$, $K_3 = 0.25$, $K_4 = 1.0$

Load:

Axial Load $P = 0.0$ varied linearly to 4.0×10^{3}
(load values of 0.6499×10^{3}, 1.300×10^{3}, 1.949×10^{3}, 2.599×10^{3}, 3.243×10^{3}, -1.099×10^{3})

Element Types:

Truss (resultant) (elform=3)
Discrete beam/cable (elform=6)

Material Models:

*MAT_001 or *MAT_ELASTIC

*MAT_074 or *MAT_ELASTIC_SPRING_DISCRETE_BEAM

Results Comparison:

LS-DYNA displacements U_A, U_B, V_C at locations A (Node 1), B (Node 2), and C (Node 3) are compared with *NAFEMS Non-Linear Benchmark*s, Test NL4 for each load value.

	***NAFEMS* NL4**	**LS-DYNA**	***NAFEMS* NL4**	**LS-DYNA**	***NAFEMS* NL4**	**LS-DYNA**
***P* (*load*)**	**U_A (*disp*)**	**U_A (*disp*)**	**U_B (*disp*)**	**U_B (*disp*)**	**V_C (*disp*)**	**V_C (*disp*)**
0.6499×10^3	650.0	650.0	0.0904	0.0903	5.241	5.242
1.300×10^3	1300.0	1300.0	0.2328	0.2329	13.260	13.266
1.949×10^3	1950.0	1949.0	0.5149	0.5150	27.080	27.079
2.599×10^3	2600.0	2600.0	1.3440	1.3338	56.500	56.500
3.243×10^3	3250.0	3250.0	7.0890	7.1053	162.600	162.850
-1.099×10^3	3900.0	2800.0	4999.0	3898.500	41.950	2047.200

Figure 5.3 shows the displaced geometry at selected load values.

These nodal displacement results (table above and Figures 5.4 and 5.5 below) were generated by *DATABASE_NODOUT keyword while the element stress results (Figures 5.6 and 5.7 below) were generated by *DATABASE_ELOUT.

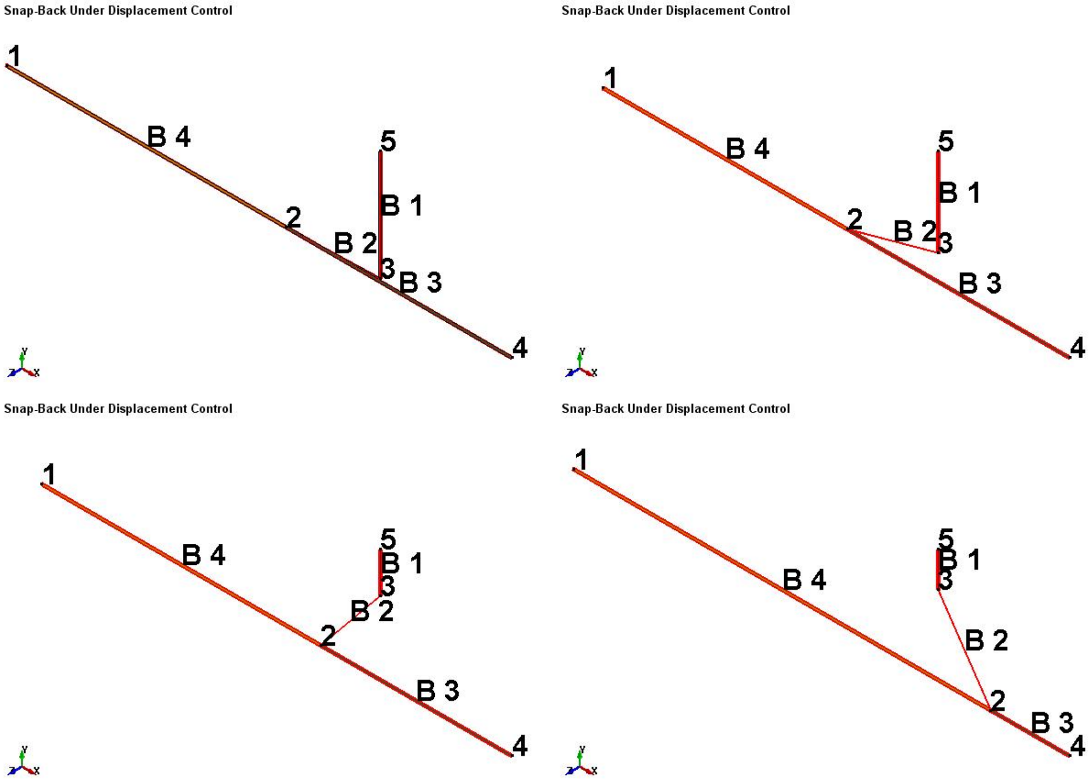

Figure 5.3 – Displaced geometry at selected loads.

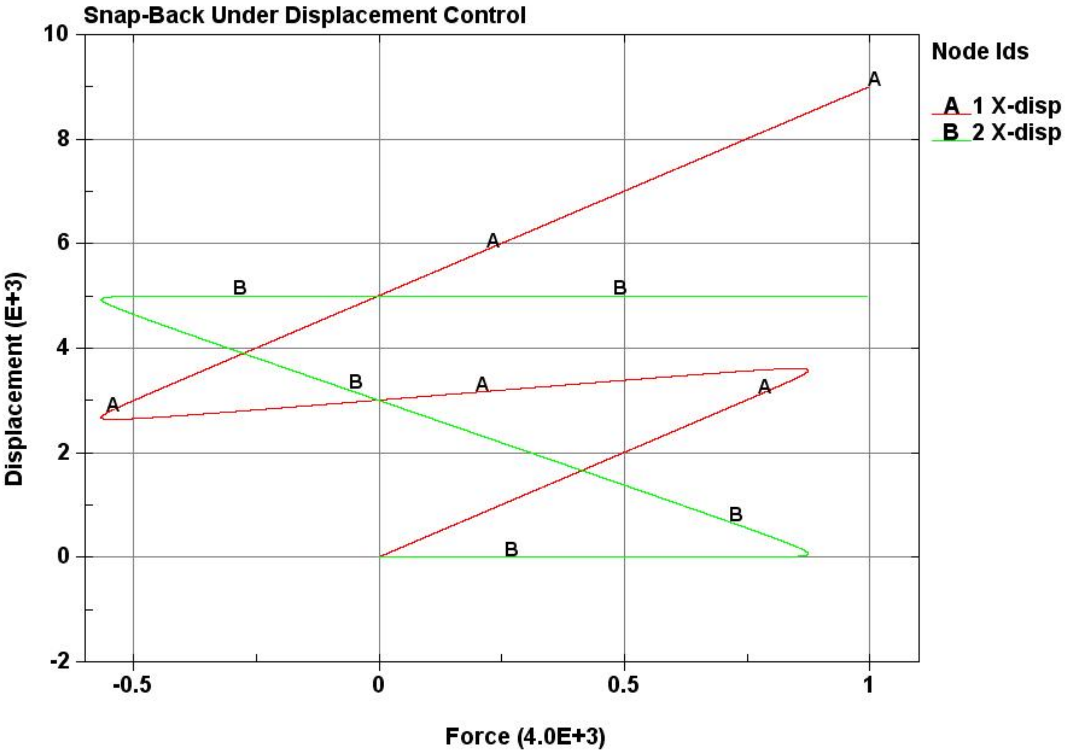

Figure 5.4 – X-displacement vs. applied load for Nodes 1 and 2.

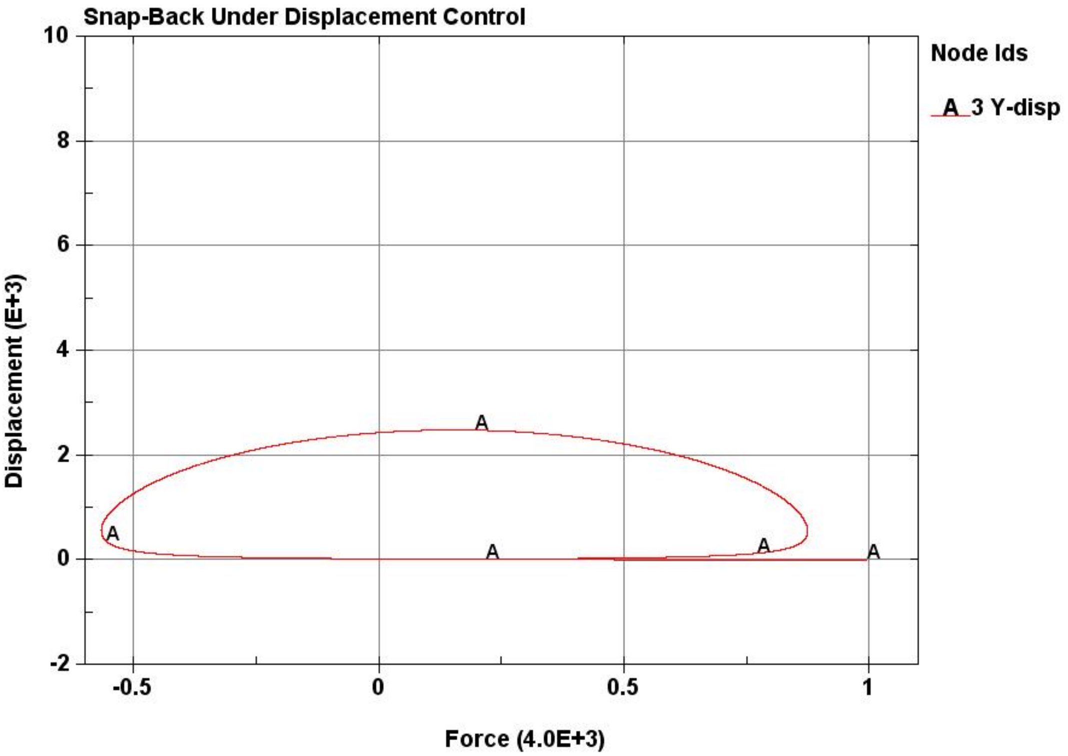

Figure 5.5 – Y-displacement vs. applied load for Node 3.

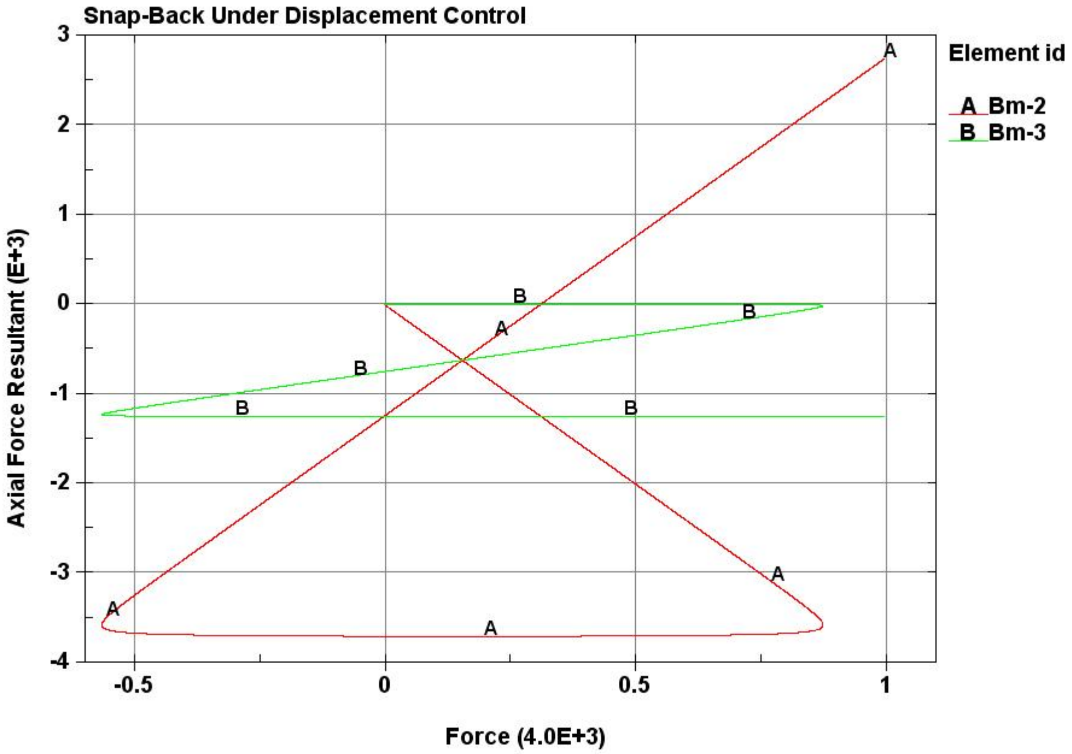

Figure 5.6 – Axial force resultant vs. applied load for Elements 2 and 3.

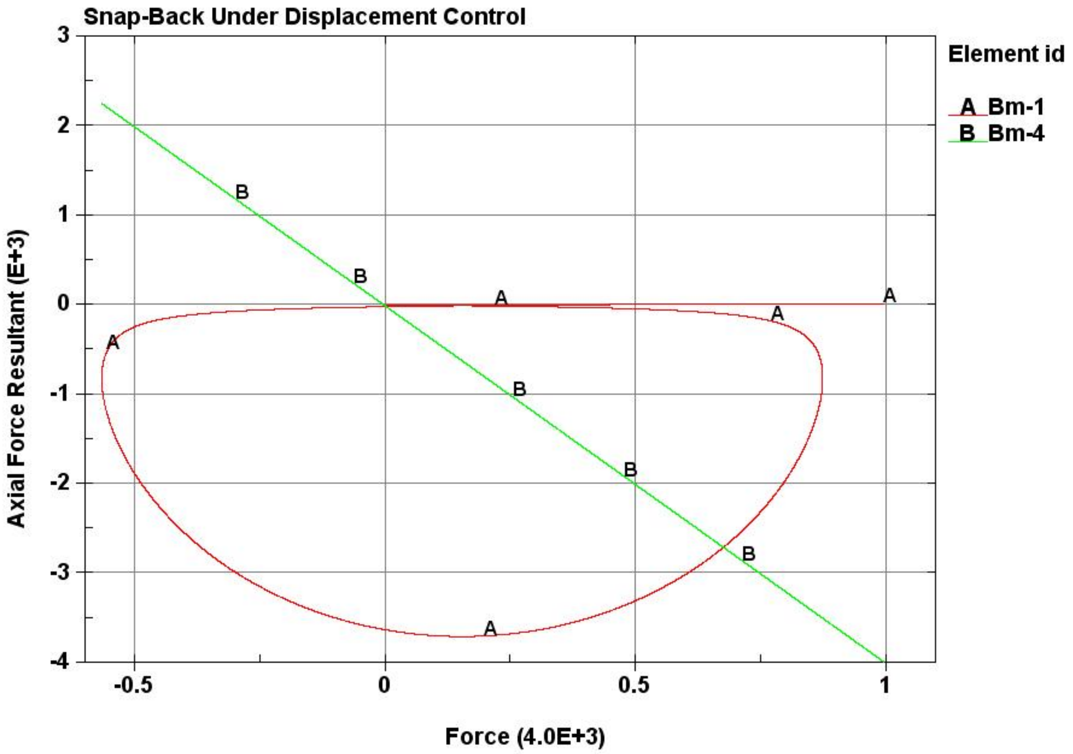

Figure 5.7 – Axial force resultant vs. applied load for Elements 1 and 4.

Input deck:

```
*KEYWORD
*TITLE
Snap-Back Under Displacement Control
*CONTROL_IMPLICIT_AUTO
$#    iauto    iteopt    itewin     dtmin     dtmax
          1        20         5 1.000e-09   0.00100
*CONTROL_IMPLICIT_GENERAL
$#   imflag       dt0    imform      nsbs       igs     cnstn      form
          1  0.001000         2         1         2         1
*CONTROL_IMPLICIT_SOLUTION
$#   nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
          6        40        15   0.00100   0.01000   0.01000  0.900000  1.000000
$#    dnorm    diverg     istif   nlprint
          2         1         1         2
$#   arcctl    arcdir    arclen    arcmth    arcdmp
          0         1       0.0         1         2
*CONTROL_TERMINATION
$#   endtim    endcyc     dtmin    endeng    endmas
  1.000000         0       0.0       0.0       0.0
*DATABASE_ELOUT
$#       dt    binary
1.0000e-04         1
*DATABASE_GLSTAT
$#       dt    binary
1.0000e-04         1
*DATABASE_MATSUM
$#       dt    binary
1.0000e-04         1
*DATABASE_NODFOR
$#       dt    binary
1.0000e-04         1
*DATABASE_NODOUT
$#       dt    binary
```

```
1.0000e-04         1
*DATABASE_SPCFORC
$#      dt    binary
1.0000e-04         1
*DATABASE_BINARY_D3PLOT
$# dt/cycl   lcdt/nr      beam     npltc    psetid
  0.010000
*DATABASE_NODAL_FORCE_GROUP
$#    nsid       cid
         1
*DATABASE_HISTORY_BEAM
$#    eid1      eid2      eid3      eid4       ei5      eid6      eid7      eid8
         1         2         3         4
*DATABASE_HISTORY_NODE
$#    nid1      nid2      nid3      nid4       ni5      nid6      nid7      nid8
         1         2         3         4         5
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo    dattyp
         1         0       0.0       0.0       0.0       0.0
$#                a1                  o1
                 0.0                 0.0
          1.00000000         4000.000000
*ELEMENT_BEAM
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       5       3       0       0       0       0       0       2
       2       2       3       2       0       0       0       0       0       2
       3       3       2       4       0       0       0       0       0       2
       4       4       2       1       0       0       0       0       0       2
*NODE
$#   nid               x               y               z      tc      rc
       1  -1.0000000e+04             0.0             0.0
       2             0.0             0.0             0.0
       3     2500.000000     25.00000000             0.0
       4     6000.000000             0.0             0.0
       5     2500.000000     3000.000000             0.0
*BOUNDARY_SPC_NODE
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         0         1         1         1         1         1
*BOUNDARY_SPC_NODE
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         2         0         0         1         1         1         1         1
         3         0         1         0         1         1         1         1
         4         0         1         1         1         1         1         1
         5         0         1         1         1         1         1         1
*PART
$# title
Spring 1
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_BEAM
$#   secid    elform      shrf   qr/irid       cst     scoor
         1         6  1.000000         0         0       0.0
$#     vol      iner       cid        ca    offset     rrcon     srcon     trcon
  1.000000  1.000000         0       0.0       0.0       0.0       0.0       0.0
*MAT_ELASTIC_SPRING_DISCRETE_BEAM
$#     mid        ro         k        f0         d       cdf       tdf
         1  1.000000  1.500000       0.0       0.0       0.0       0.0
$#   flcid     hlcid        c1        c2       dle     glcid
         0         0       0.0       0.0  1.000000
*PART
$# title
Truss 2
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         2         2         2
*SECTION_BEAM
$#   secid    elform      shrf   qr/irid       cst     scoor
         2         3       0.0         0         0       0.0
$#       a       iss       itt       irr        sa
  1.000000  1.000000  1.000000  1.000000  1.000000
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
```

```
         2  1.0000005.0000e+07       0.0       0.0       0.0       0.0
*PART
$# title
Spring 3
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         3         1         3
*MAT_ELASTIC_SPRING_DISCRETE_BEAM
$#     mid        ro         k        f0         d       cdf       tdf
         3  1.000000  0.250000       0.0       0.0       0.0       0.0
$#   flcid     hlcid        c1        c2       dle     glcid
         0         0       0.0       0.0  1.000000
*PART
$# title
Spring 4
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         4         1         4
*MAT_ELASTIC_SPRING_DISCRETE_BEAM
$#     mid        ro         k        f0         d       cdf       tdf
         4  1.000000  1.000000       0.0       0.0       0.0       0.0
$#   flcid     hlcid        c1        c2       dle     glcid
         0         0       0.0       0.0  1.000000
*LOAD_NODE_POINT
$#    node       dof      lcid        sf       cid        m1        m2        m3
         1         1         1  1.000000
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1         2         3         4         5
*END
```

Notes:

1. Using the default values (i.e., BFGS without arc length) and an automatic time stepping control, it was possible to solve the problem only up to a certain load. At this point, (a) the BFGS solution method cannot go any further, due to the tangent stiffness matrix becoming close to null, resulting in a FATAL ERROR – nonlinear solver failed to find equilibrium, or (b) the solution proceeded with an incorrect solution (no snap-back).

2. When the time step was allowed to increase up to 0.010, either by initial time step or dtmax (automatic time stepping control), a solution could be achieved, relatively quickly, but a somewhat noisy in the response.

3. Using the default tolerance (default=inactive) for the residual (force) norm appeared to result in non-convergence or inaccurate convergence (i.e. relative convergence was achieved, but the amount of out-of-balance forces became too large to guarantee the accuracy of the solution). The tolerance on force was therefore activated and set to (0.010). If this value is too large, convergence issues will result.

4. In addition to employing the BFGS solver with arc length (nsolvr=6), it was found necessary to employ the default arc length, that is, the generalized arc length method (arcctl=0), where the norm of the global displacement vector controls the solution; this includes all nodes. Attempts at employing the option whereby the arc length method was controlled based on the displacement of a single node were unsuccessful.

6. Straight Cantilever Beam with Axial End Point Load

Keywords:

```
*CONTROL_IMPLICIT_AUTO
*CONTROL_IMPLICIT_GENERAL
*CONTROL_IMPLICIT_SOLVER
*CONTROL_IMPLICIT_SOLUTION
```

Description:

The analysis involves a cantilever beam loaded at one end with a quasi-axial load (axial component=100 normal component). The material is elastic. The X-displacement, the Y-displacement and the Z-rotation of the end point of the beam are determined.

A sketch representing the structure is shown below (Figure 6.1) along with the finite element model (Figure 6.2).

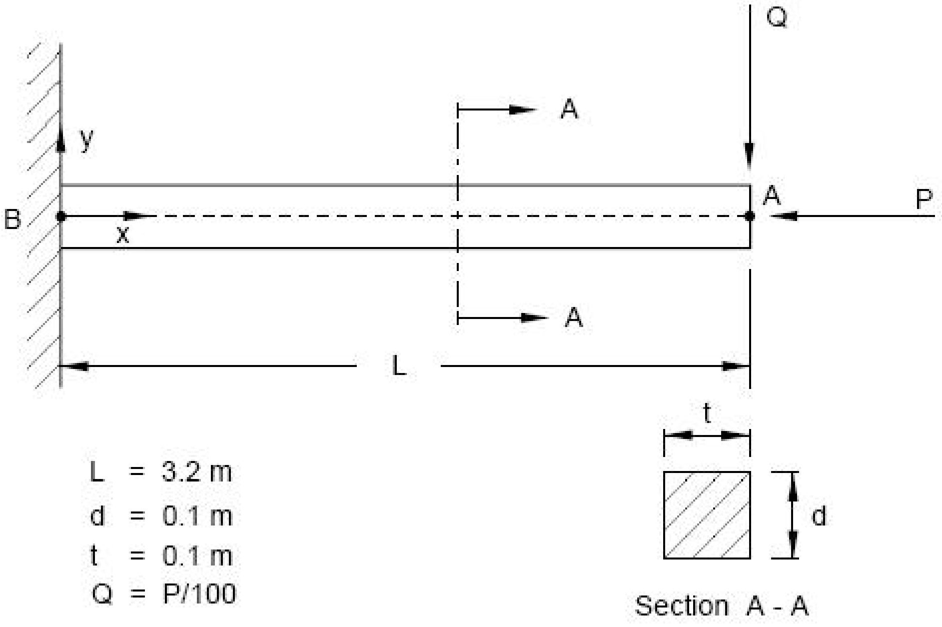

Figure 6.1 – Sketch representing the structure.

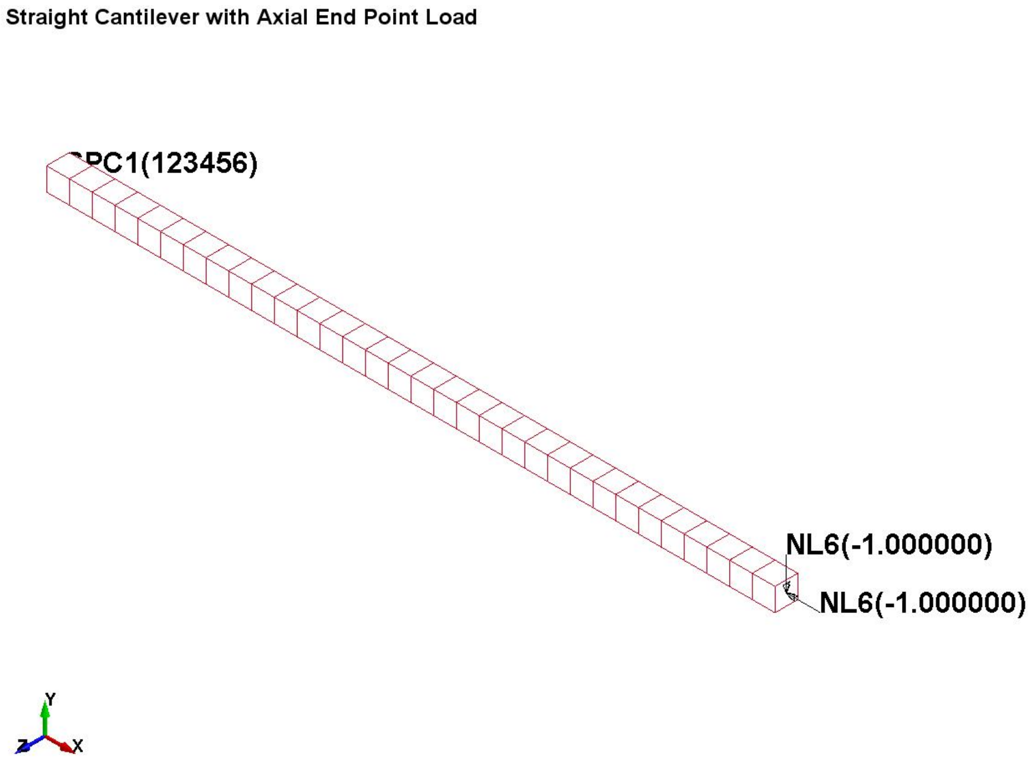

Figure 6.2 – Finite element model with applied loads and boundary conditions.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Static	Force	Linear	Nonlinear	-	Implicit	2-Nonlinear w/BFGS

Units:

kg, m, s, N, Pa, N-m (kilogram, meter, second, Newton, Pascal, Newton-meter)

Dimensional Data:

The beam has a constant square section (0.1m x 0.1m) and a total length of 3.2 m and is meshed with 32 beams of equal length.

Material Data:

Mass Density $\rho = 7.85 \times 10^{3}\ kg / m^{3}$
Young's Modulus $E = 2.10 \times 10^{11}\ Pa$
Poisson's Ratio $\nu = 0.0$

Load:

Axial Load $P = 3.844 \times 10^6\ N$

Pressure $Q = 3.844 \times 10^4\ N$

Element Types:

Hughes-Liu beam with cross section integration (elform=1)

Material Models:

*MAT_001 or *MAT_ELASTIC

Results Comparison:

LS-DYNA displacements U_X, U_Y, R_Z at the end of the beam (Node 6) are compared with *NAFEMS Non-Linear Benchmarks,* Test NL6.

	$U_X\ (m)$	$U_Y\ (m)$	$R_Z\ (rad)$
NAFEMS NL6	-5.0404	-1.3472	-3.0725
Node 6	-5.0629	-1.3607	-3.0646

These nodal displacement results were generated by *DATABASE_NODOUT keyword. The X-displacement (U_X), Y-displacement (U_Y), and Z-rotation (R_Z) histories for Node 6 are given in Figure 6.3. Figures 6.4 and 6.5 provide the contour plot of the bending moment and the axial force, respectively, at the end of the step.

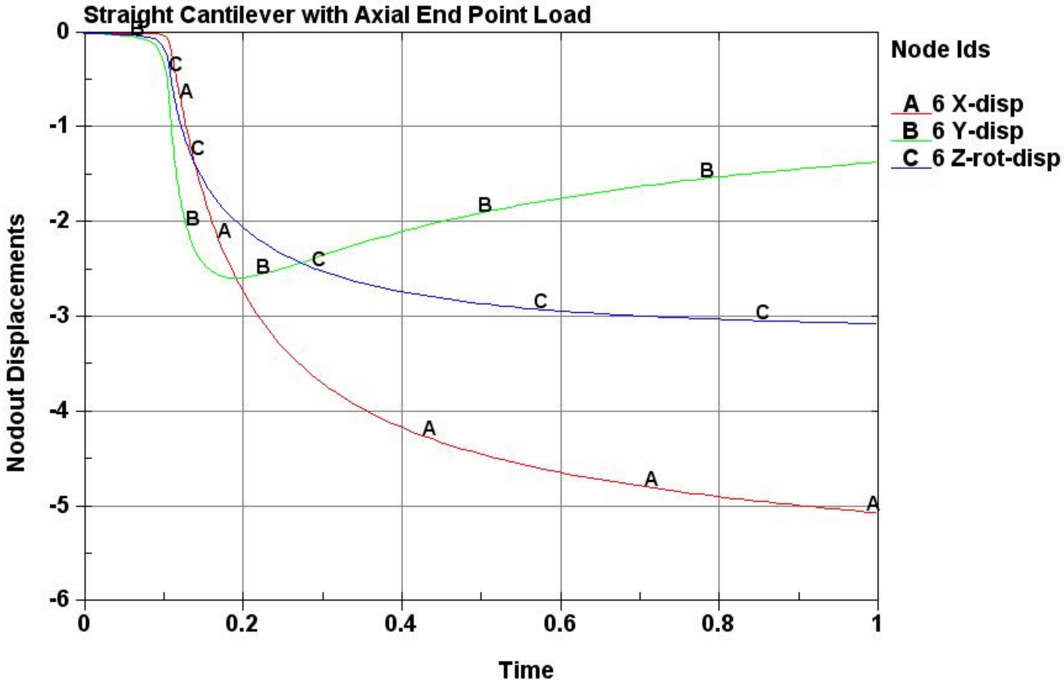

Figure 6.3 – X-displacement, Y-displacement, and Z-rotation for Node 6.

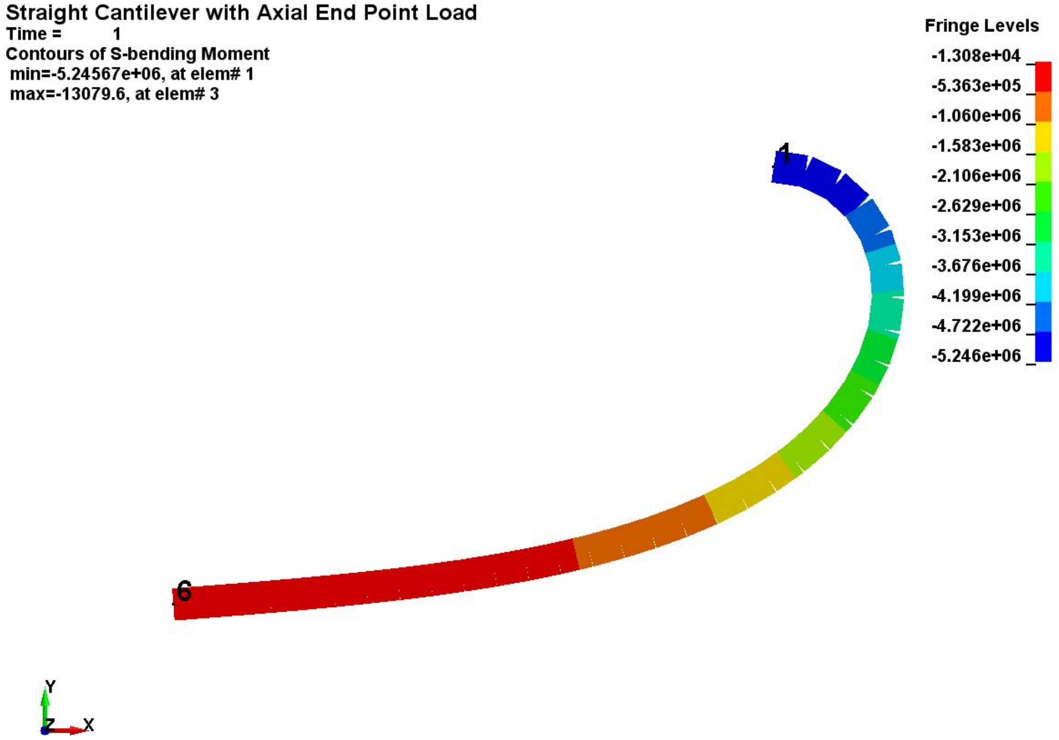

Figure 6.4 – Contour plot of the bending moment at the end of the step.

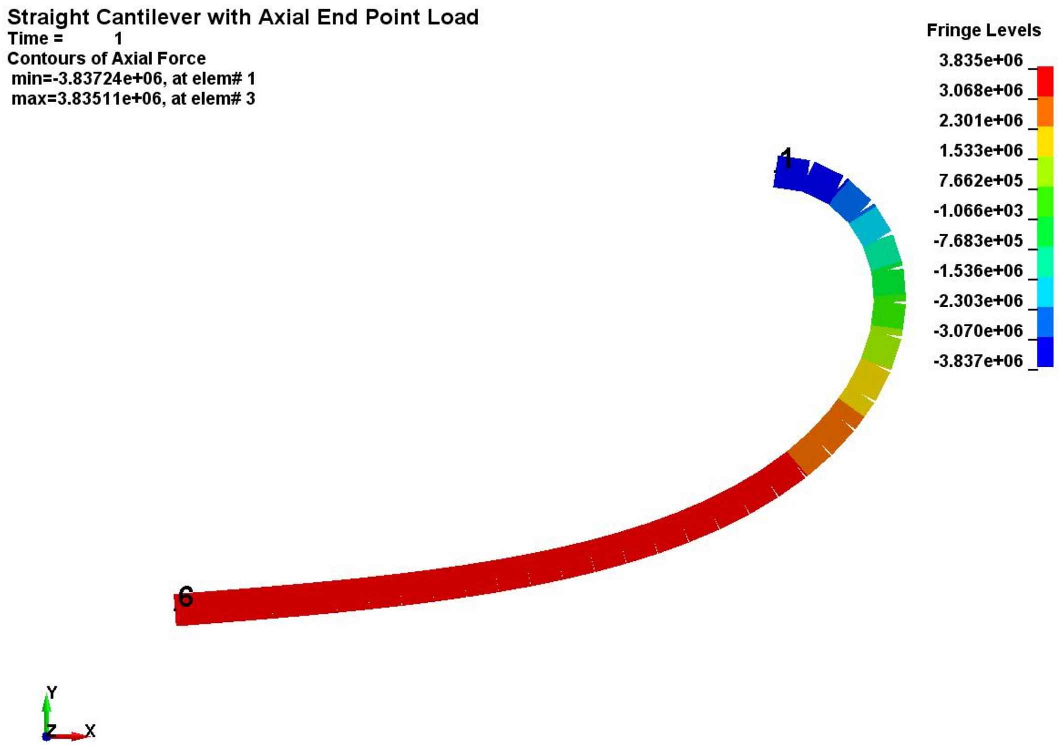

Figure 6.5 – Contour plot of the axial force at the end of the step.

Input deck:

```
*KEYWORD
*TITLE
Straight Cantilever with Axial End Point Load
*CONTROL_IMPLICIT_AUTO
$#   iauto    iteopt    itewin     dtmin     dtmax
         1        11         5              0.010000
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1  0.010000         2         0         2
*CONTROL_IMPLICIT_SOLVER
$#  lsolvr    lprint     negev     order      drcm    drcprm   autospc   autotol
         5         2         2         0         1       0.0         1       0.0
$#  lcpack
         2
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
         2        11        15    0.0010    0.0100  1.00e+10  0.900000  1.00e-10
$#   dnorm    diverg     istif   nlprint    nlnorm
         2         1         1         2         1
$#  arcctl    arcdir    arclen    arcmth    arcdmp
         6         1       0.0         1         2
*CONTROL_OUTPUT
$#   npopt    neecho    nrefup    iaccop     opifs    ipnint    ikedit    iflush
         1         3         1         0       0.0         0      1000      5000
$#   iprtf
         3
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  1.000000         0       0.0       0.0       0.0
*DATABASE_NCFORC
$#      dt    binary
  0.001000         1
```

```
*DATABASE_NODFOR
$#      dt    binary
  0.001000         1
*DATABASE_NODOUT
$#      dt    binary
  0.001000         1
*DATABASE_NODAL_FORCE_GROUP
$#    nsid       cid
         2
*DATABASE_HISTORY_NODE
$#    nid1      nid2      nid3      nid4       ni5      nid6      nid7      nid8
         6
*DATABASE_HISTORY_BEAM
$#    eid1      eid2      eid3      eid4       ei5      eid6      eid7      eid8
         3
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo    dattyp
         1         0  1.000000  1.000000       0.0       0.0
$#                a1                  o1
                 0.0                 0.0
          1.00000000       3.8440088e+06
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo    dattyp
         2         0  1.000000  1.000000       0.0       0.0
$#                a1                  o1
                 0.0                 0.0
          1.00000000         38440.08545
*ELEMENT_BEAM
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1       2       3       0       0       0       0       2

      33       1       2      92      97       0       0       0       0       2
*NODE
$#   nid               x               y               z      tc      rc
       1             0.0             0.0             0.0

      97      0.15000001             0.0      0.01000000
*BOUNDARY_SPC_NODE
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         1         1         1         1         1         1
*PART
$# title

$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_BEAM
$#   secid    elform      shrf   qr/irid       cst     scoor
         1         1  0.830000         2         0       0.0
$#     ts1       ts2       tt1       tt2     nsloc     ntloc
  0.100000  0.100000  0.100000  0.100000
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1  7850.0002.1000e+11       0.0       0.0       0.0       0.0
*LOAD_NODE_POINT
$#    node       dof      lcid        sf       cid        m1        m2        m3
         6         1         1 -1.000000
*LOAD_NODE_POINT
$#    node       dof      lcid        sf       cid        m1        m2        m3
         6         2         2 -1.000000
*SET_NODE_LIST_GENERATE
$#     sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         2        92
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4    solver
         2       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         6
*END
```

Notes:

1. Using the default values, with an initial time step dt0=0.010, the problem stops at the 12th iteration due to an energy increase. The *CONTROL_IMPLICIT_GENERAL, *CONTROL_IMPLICIT_SOLUTION, and *CONTROL_IMPLICIT_SOLVER, with no automatic time stepping (*CONTROL_IMPLICIT_AUTO) are considered to be the default keywords.

2. Allowing more iterations (*CONTROL_IMPLICIT_SOLUTION) will not help to solve the problem.

3. To resolve the energy increase and termination stated above, include the automatic time stepping (*CONTROL_IMPLICIT_AUTO) entry, in particular the specification of dtmax. The following situations occur when using different values of dtmax:

   dtmax =blank (10*dt0) or 0.100 (these are actually the same); the current step size will increase right off, eventually two energy increases will occur, where time steps are then decreased, with the simulation then continuing until termination is reached. This takes the least iterations with ASCII result plots somewhat noisy.

   dtmax =0.010 (the initial time step); solves very nicely with no energy increases, takes about 50 percent more iterations than dtmax=0.100, with smoother ASCII result plots.

   dtmax =0.001 yielded the same results as dtmax =0.010. dt0 appeared to still be considered in the time step options.

4. It is also possible to achieve a successful solution specifying an initial time step of dt0=0.001 and a similar value for the maximum allowable time step (dtmax=0.001 in the *CONTROL_IMPLICIT_AUTO keyword). Using these parameters will increase the number of iterations significantly.

7. Lee's Frame Buckling Problem

Keywords:

```
*CONTROL_IMPLICIT_AUTO
*CONTROL_IMPLICIT_GENERAL
*CONTROL_IMPLICIT_SOLUTION
```

Description:

The problem involves a framed structure deforming under the action of a load applied on one node. The frame is pinned to the ground at two nodes and the load is applied on Node 56, as shown in Figure 7-1. The finite element model is shown in Figure 7.2.

The length of the two beams is 1.2 m. The height of the square cross section is 0.02 m and the thickness is 0.03 m.

When a certain load is reached, the structure undergoes buckling and the load-deflection curve shows a typical snap-back behavior, shown in Figure 7-3.

Arc-length method is required in order to capture the post-buckling behavior of the structure.

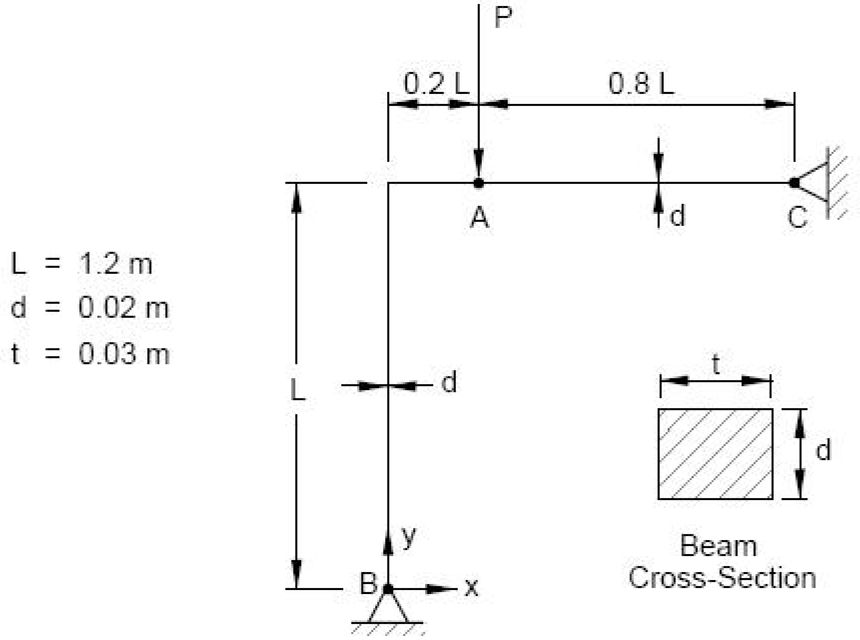

Figure 7.1 – Sketch representing the structure.

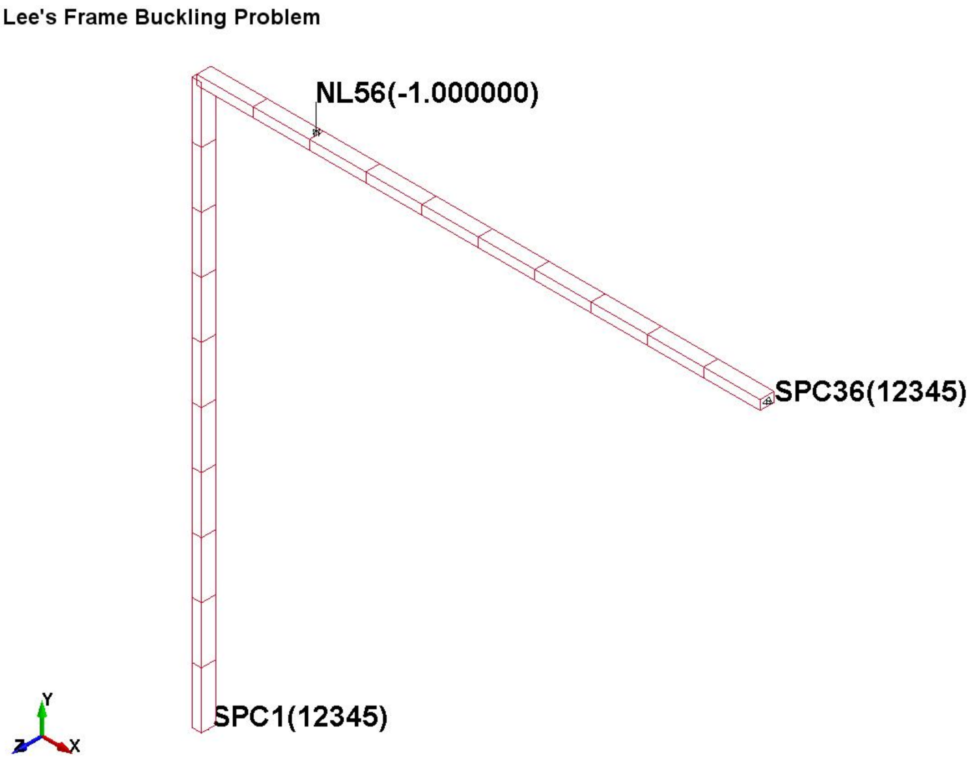

Figure 7.2 – Finite element model with applied loads and boundary conditions.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Static	Force	Linear	Nonlinear	-	Implicit	6-Arc length w/BFGS

Units:

kg, m, s, N, Pa, N-m (kilogram, meter, second, Newton, Pascal, Newton-meter)

Dimensional Data:

The beam has a constant square section (0.1m x 0.1m) and a total length of 3.2 m and is meshed with 20 beams of equal length.

Material Data:

Mass Density	$\rho = 7.85 \times 10^3 \ kg / m^3$
Young's Modulus	$E = 7.174 \times 10^{10} \ Pa$
Poisson's Ratio	$\nu = 0.0$

Load:

Load is applied incrementally to the structure until buckling occurs.

Element Types:

Hughes-Liu beam with cross section integration (elform=1)

Material Models:

*MAT_001 or *MAT_ELASTIC

Results Comparison:

LS-DYNA displacements U_X and U_Y, at the location of the applied load (Node 56), plus the critical (buckling) load P_{crit}, are compared with *NAFEMS Non-Linear Benchmarks,* Test NL7.

	$\boldsymbol{U_X \ (m)}$	$\boldsymbol{U_Y \ (m)}$	$\boldsymbol{P_{crit} \ (N)}$
NAFEMS NL7	-	0.4884	1.8485×10^4
Node 56	0.2620	0.4826	1.8228×10^4

These nodal displacement results were generated by *DATABASE_NODOUT keyword.

From *DATABASE_NODOUT results, it was also seen that the critical load increment is at 0.364568 of the total load, which would therefore correspond to a load of $P_{crit} = 0.364568 \times 5.0 \times 10^4 = 1.8228 \times 10^4 \ N$.

Figure 7.4 gives the X-displacement, Y-displacement, and resultant displacement versus load increment for Node 56.

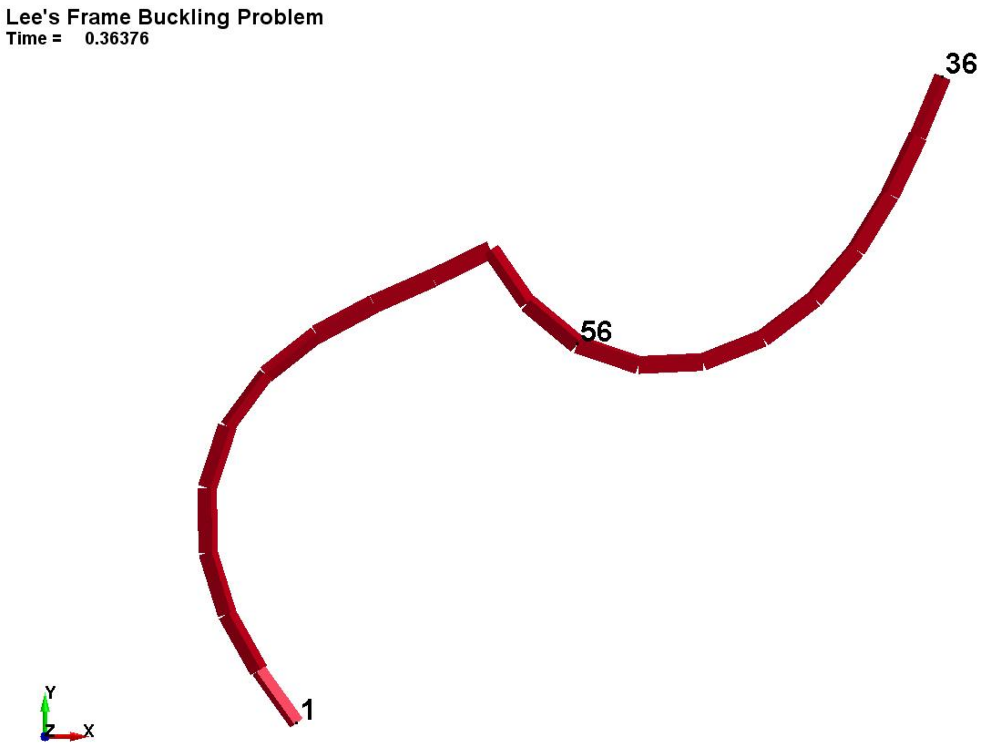

Figure 7.3 – Displaced configuration at the buckling load.

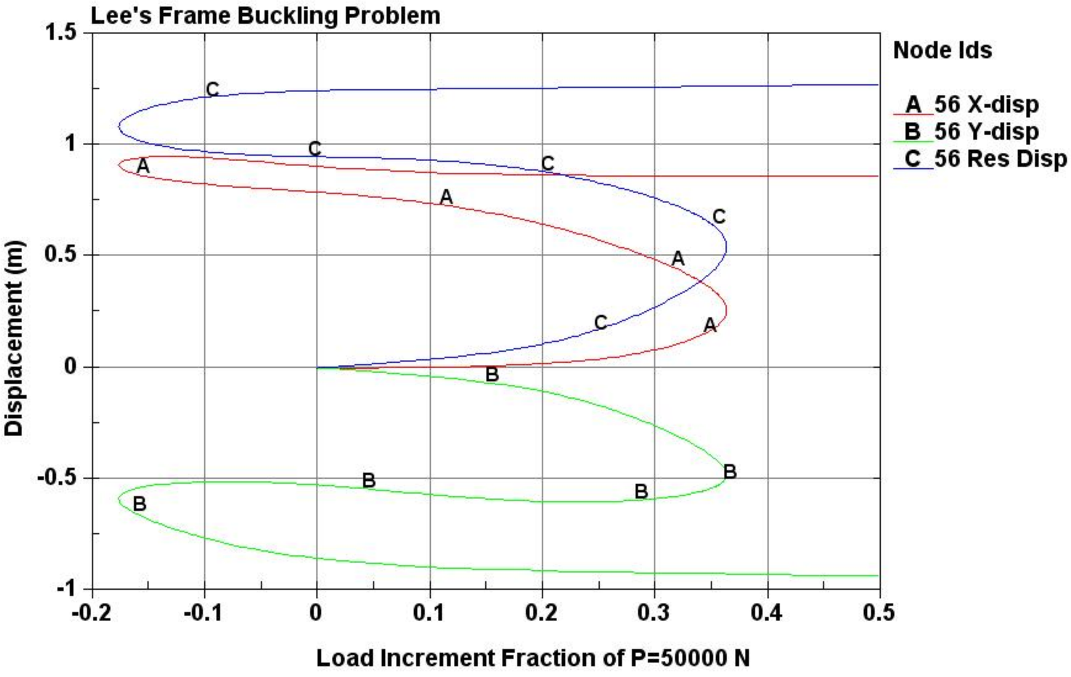

Figure 7.4 – X-displacement, Y-displacement, and resultant displacement versus load increment for Node 56.

Input deck:

```
*KEYWORD
*TITLE
Lee's Frame Buckling Problem
*CONTROL_IMPLICIT_AUTO
$#   iauto    iteopt    itewin     dtmin     dtmax
         1        20         5 1.000e-09
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1  0.003000         2       100         2         1
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
         6        30        15 1.000e-06 1.000e-05 1.000e-05  0.990000  1.000000
$#   dnorm    diverg     istif   nlprint
         2         1         1         2
$#  arcctl    arcdir    arclen    arcmth    arcdmp
         0         1       0.0         1         2
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  1.000000         0       0.0       0.0       0.0
*DATABASE_GLSTAT
$#      dt    binary
  0.001000         1
*DATABASE_MATSUM
$#      dt    binary
  0.001000         1
*DATABASE_NODFOR
$#      dt    binary
1.0000e-04         1
*DATABASE_NODOUT
$#      dt    binary
1.0000e-04         1
*DATABASE_BINARY_D3PLOT
$# dt/cycl   lcdt/nr      beam     npltc    psetid
  0.010000         0         2
*DATABASE_NODAL_FORCE_GROUP
$#    nsid       cid
         1
*DATABASE_HISTORY_NODE
$#    nid1      nid2      nid3      nid4       ni5      nid6      nid7      nid8
        56        36         1
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo    dattyp
         1         0  1.000000  1.000000       0.0       0.0
$#                a1                  o1
                 0.0                 0.0
          1.00000000       5.0000000e+04
*ELEMENT_BEAM
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1       2       3       0       0       0       0       2

      31       1      32      56      61       0       0       0       0       2
*NODE
$#   nid               x               y               z      tc      rc
       1             0.0             0.0             0.0

      61      0.17999999      1.20000005      0.10000000
*BOUNDARY_SPC_NODE
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         1         1         1         1         1
*BOUNDARY_SPC_NODE
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
        36         0         1         1         1         1         1
*PART
$# title

$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_BEAM
```

```
$#   secid    elform      shrf   qr/irid       cst     scoor
         1         1  0.830000         5         0       0.0
$#     ts1       ts2       tt1       tt2     nsloc     ntloc
  0.030000  0.030000      0.02      0.02
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1  7850.0007.1740e+10       0.0       0.0       0.0       0.0
*LOAD_NODE_POINT
$#    node       dof      lcid        sf       cid        m1        m2        m3
        56         2         1 -1.000000
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1         6        36        56
*END
```

Notes:

8. Pin-Ended Double Cross: In-Plane Vibration

Keywords:

```
*CONTROL_IMPLICIT_AUTO
*CONTROL_IMPLICIT_EIGENVALUE
*CONTROL_IMPLICIT_GENERAL
*CONTROL_IMPLICIT_SOLVER
```

Description:

This example shows the behavior of beam elements in a modal analysis. The structure is a double cross pinned to the ground as show in Figure 8.1. All inner nodes have $U_Z = R_X = R_Y = 0$. On the outer nodes $U_X = U_Y = U_Z = R_X = R_Y = 0$.

The finite element model is shown in Figure 8.2.

The problem requires the extraction of numerically close eigenvalues, making it an ideal benchmark to check the element formulation accuracy.

Each arm of the cross is modeled with 4 beams, for a total of 32 beams. The length of each arm is 5 m.

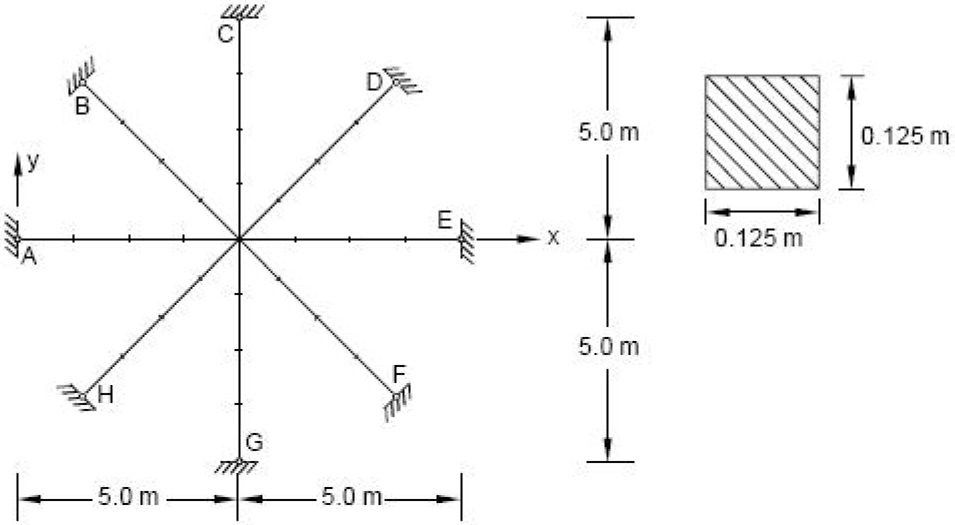

Figure 8.1 – Sketch representing the structure.

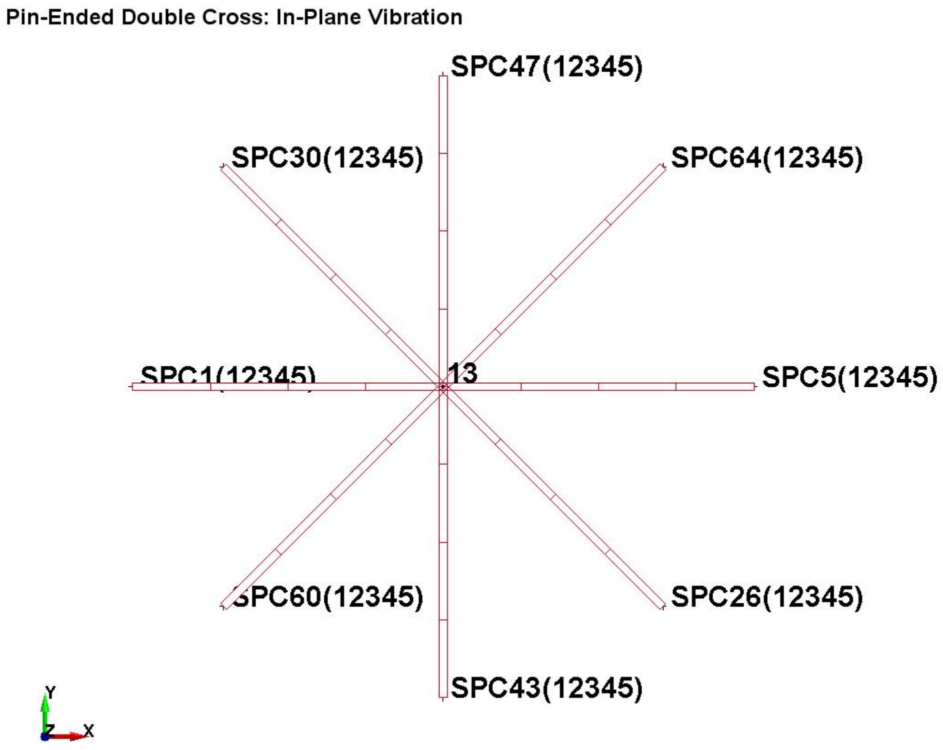

Figure 8.2 – Finite element model with end point boundary conditions.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Modal	-	Linear	Linear	-	Implicit	Block Shift and Inverted Lanczos

Units:

kg, m, s, N, Pa, N-m (kilogram, meter, second, Newton, Pascal, Newton-meter)

Dimensional Data:

Square section of the beams: 0.125 m x 0.125 m.

Material Data:

Mass Density $\rho = 8.00 \times 10^3\ kg / m^3$
Young's Modulus $E = 2.00 \times 10^{11}\ Pa$
Poisson's Ratio $\nu = 0.3$

Element Types:

Hughes-Liu beam with cross section integration (elform=1)
Belytschko-Schwer resultant beam (elform=2)
Small displacement, linear Timoshenko beam with exact stiffness (elform=13)

Material Models:

*MAT_001 or *MAT_ELASTIC

Results Comparison:

LS-DYNA natural frequencies, first 16 (frequency in *Hertz*), and mode shapes (first 6) are compared with *Standard NAFEMS Benchmarks*, Test FV2.

Mode(s)	***NAFEMS* FV2 (*Hz*)**	**Hughes-Liu Beam (*Hz*)**	**Belytschko-Schwer Beam**	**Timoshenko Beam (*Hz*)**
1	11.336	11.641	11.323	11.365
2, 3	17.709	19.080	17.621	17.803
4, 5, 6, 7, 8	17.709	19.115	17.649	17.832
9	45.345	51.691	44.833	45.620
10, 11	57.390	73.717	55.673	57.399
12, 13, 14, 15, 16	57.390	74.381	55.952	57.706

As seen in the results, using the LS-DYNA default beam (Hughes-Liu - elform=1) results in poor accuracy in the frequency calculation due to its omission of the first (no bending) and second (no rotary inertia) order terms (more elements are often needed in an attempt to overcome this limitation). The Hughes-Liu beam effectively generates a constant moment along its length, so, as with brick and shell elements, meshes need to be reasonably fine to achieve adequate accuracy.

The Belytschko-Schwer beam (elform=2) provides good frequency results throughout most of the range covered, with some minor differences at the higher frequency range. This element is often acceptable.

The Timoshenko beam (elform=13), with its inclusion of second order (rotary inertia and shear distortion) terms, provides very good results throughout the reported frequency range. This element formulation is generally recommended for this type of frequency analysis.

Eigenvalue Results:

From the *eigout* file, generated by the *CONTROL_IMPLICIT_EIGENVALUE keyword:

Hughes-Liu beam (elform=1):

```
Pin-Ended Double Cross: In-Plane Vibration
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1   5.349990E+03   7.314363E+01   1.164117E+01   8.590202E-02
           2   1.437182E+04   1.198825E+02   1.907989E+01   5.241119E-02
           3   1.437182E+04   1.198825E+02   1.907989E+01   5.241119E-02
           4   1.442423E+04   1.201009E+02   1.911465E+01   5.231589E-02
           5   1.442423E+04   1.201009E+02   1.911465E+01   5.231589E-02
           6   1.442423E+04   1.201009E+02   1.911465E+01   5.231589E-02
           7   1.442423E+04   1.201009E+02   1.911465E+01   5.231589E-02
           8   1.442423E+04   1.201009E+02   1.911465E+01   5.231589E-02
           9   1.054860E+05   3.247862E+02   5.169132E+01   1.934561E-02
          10   2.145341E+05   4.631783E+02   7.371711E+01   1.356537E-02
          11   2.145341E+05   4.631783E+02   7.371711E+01   1.356537E-02
          12   2.184158E+05   4.673498E+02   7.438102E+01   1.344429E-02
          13   2.184158E+05   4.673498E+02   7.438102E+01   1.344429E-02
          14   2.184158E+05   4.673498E+02   7.438102E+01   1.344429E-02
          15   2.184158E+05   4.673498E+02   7.438102E+01   1.344429E-02
          16   2.184158E+05   4.673498E+02   7.438102E+01   1.344429E-02
```

Belytschko-Schwer beam (elform=2):

```
Pin-Ended Double Cross: In-Plane Vibration
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1   5.061497E+03   7.114420E+01   1.132295E+01   8.831620E-02
           2   1.225757E+04   1.107139E+02   1.762066E+01   5.675155E-02
           3   1.225757E+04   1.107139E+02   1.762066E+01   5.675155E-02
           4   1.229691E+04   1.108915E+02   1.764892E+01   5.666068E-02
           5   1.229691E+04   1.108915E+02   1.764892E+01   5.666068E-02
           6   1.229691E+04   1.108915E+02   1.764892E+01   5.666068E-02
           7   1.229691E+04   1.108915E+02   1.764892E+01   5.666068E-02
           8   1.229691E+04   1.108915E+02   1.764892E+01   5.666068E-02
           9   7.935148E+04   2.816939E+02   4.483298E+01   2.230501E-02
          10   1.223614E+05   3.498019E+02   5.567270E+01   1.796212E-02
          11   1.223614E+05   3.498019E+02   5.567270E+01   1.796212E-02
          12   1.235904E+05   3.515543E+02   5.595161E+01   1.787259E-02
          13   1.235904E+05   3.515543E+02   5.595161E+01   1.787259E-02
          14   1.235904E+05   3.515543E+02   5.595161E+01   1.787259E-02
          15   1.235904E+05   3.515543E+02   5.595161E+01   1.787259E-02
          16   1.235904E+05   3.515543E+02   5.595161E+01   1.787259E-02
```

Timoshenko beam (elform=13):

```
Pin-Ended Double Cross: In-Plane Vibration
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1   5.099519E+03   7.141092E+01   1.136540E+01   8.798634E-02
           2   1.251262E+04   1.118598E+02   1.780305E+01   5.617016E-02
           3   1.251262E+04   1.118598E+02   1.780305E+01   5.617016E-02
           4   1.255348E+04   1.120423E+02   1.783209E+01   5.607869E-02
           5   1.255348E+04   1.120423E+02   1.783209E+01   5.607869E-02
           6   1.255348E+04   1.120423E+02   1.783209E+01   5.607869E-02
           7   1.255348E+04   1.120423E+02   1.783209E+01   5.607869E-02
           8   1.255348E+04   1.120423E+02   1.783209E+01   5.607869E-02
           9   8.216358E+04   2.866419E+02   4.562048E+01   2.191998E-02
          10   1.300676E+05   3.606489E+02   5.739905E+01   1.742189E-02
          11   1.300676E+05   3.606489E+02   5.739905E+01   1.742189E-02
          12   1.314649E+05   3.625808E+02   5.770653E+01   1.732906E-02
          13   1.314649E+05   3.625808E+02   5.770653E+01   1.732906E-02
          14   1.314649E+05   3.625808E+02   5.770653E+01   1.732906E-02
          15   1.314649E+05   3.625808E+02   5.770653E+01   1.732906E-02
          16   1.314649E+05   3.625808E+02   5.770653E+01   1.732906E-02
```

Mode Shapes (first six):

From the *d3plot* file, generated by the *DATABASE_BINARY_D3PLOT keyword, the user can obtain the first six mode shapes (stick view) for the Hughes-Li beam (Figure 8.3), the Belytschko-Schwer Beam (Figure 8.4), and the Timoshenko beam (Figure 8.5). Displacement contouring of the first six mode shapes are given in Figures 8.6, 8.7, and 8.8 for these three element formulations.

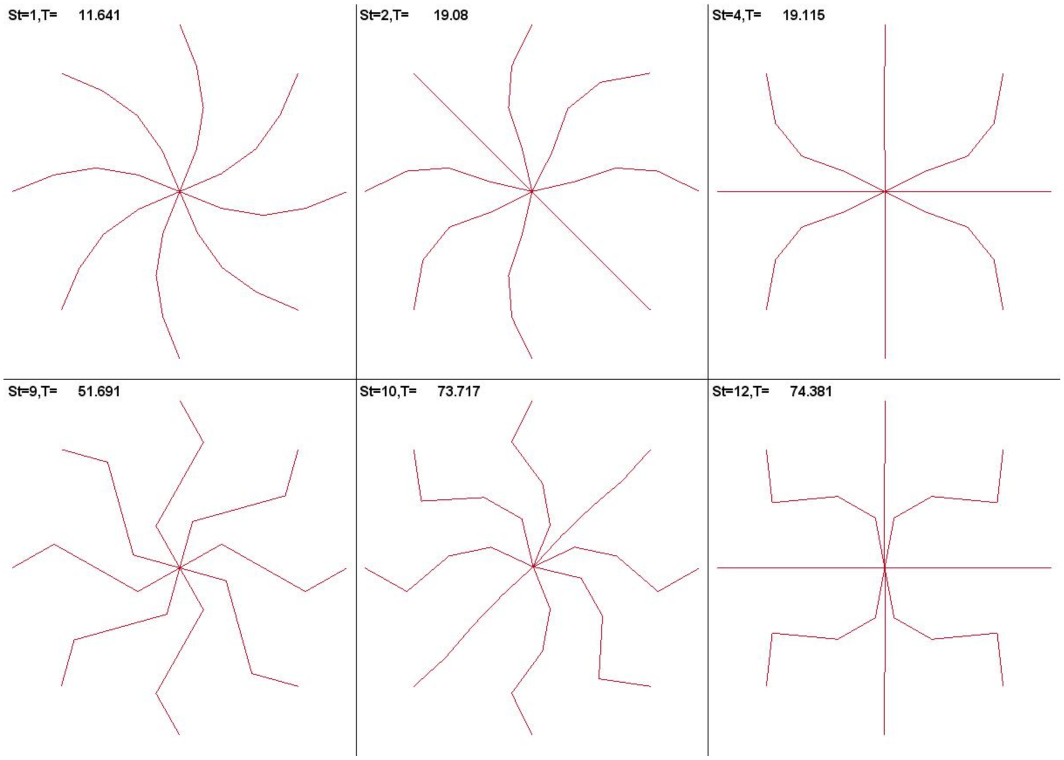

Figure 8.3 - Mode shapes for Hughes-Liu beam (stick view).

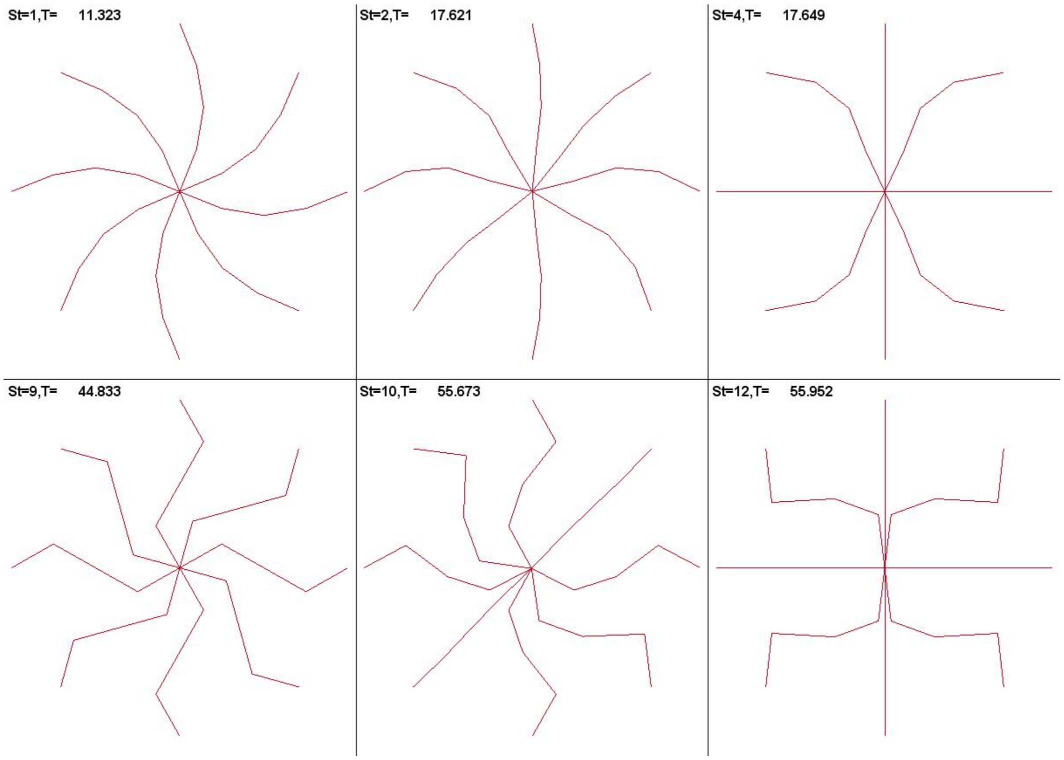

Figure 8.4 - Mode shapes for Belytschko beam (stick view).

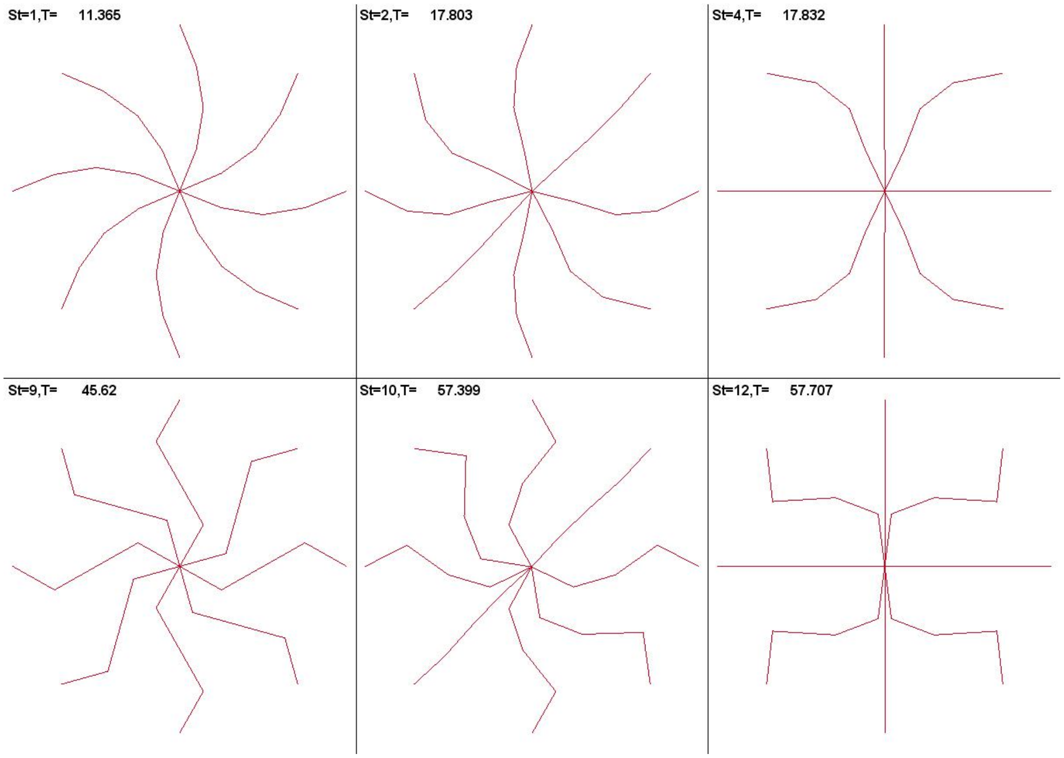

Figure 8.5 - Mode shapes for Timoshenko beam (stick view).

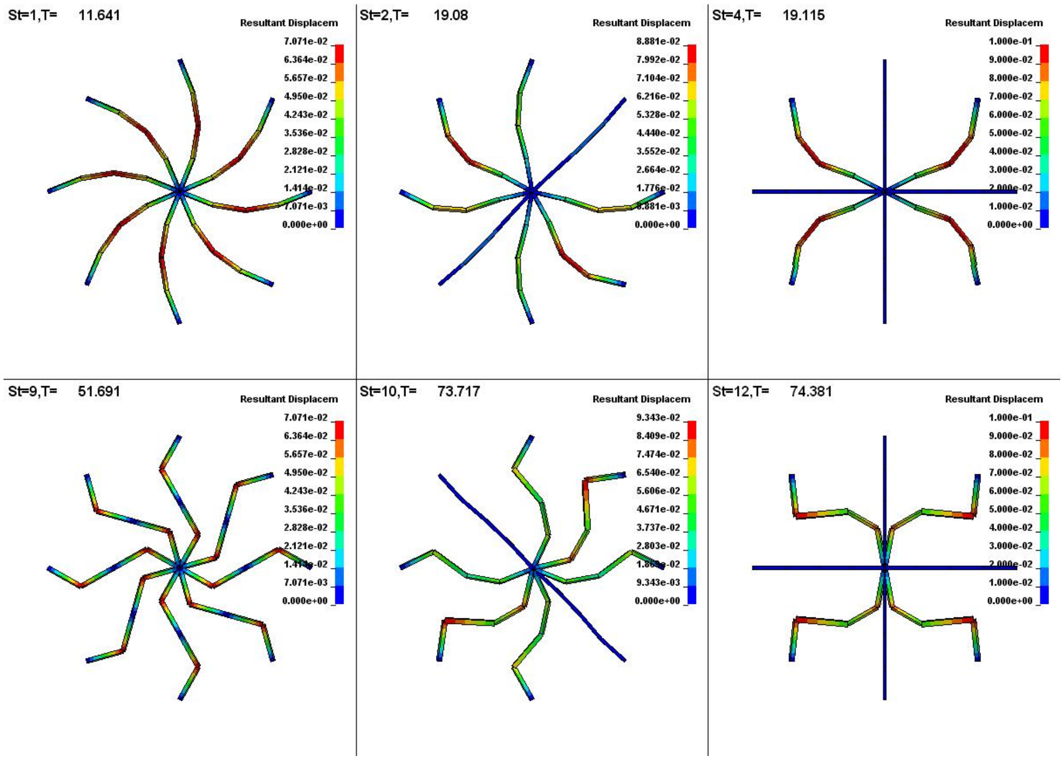

Figure 8.6 - Mode shapes for Hughes-Liu beam (displacement contouring).

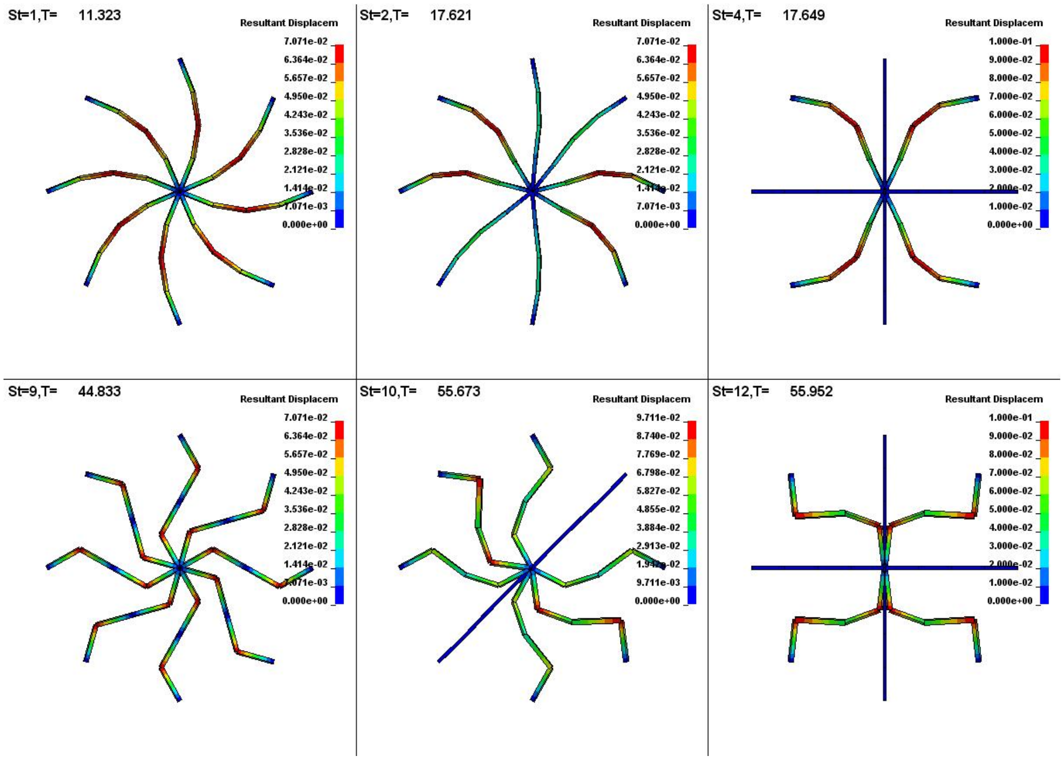

Figure 8.7 - Mode shapes for Belytschko beam (displacement contouring).

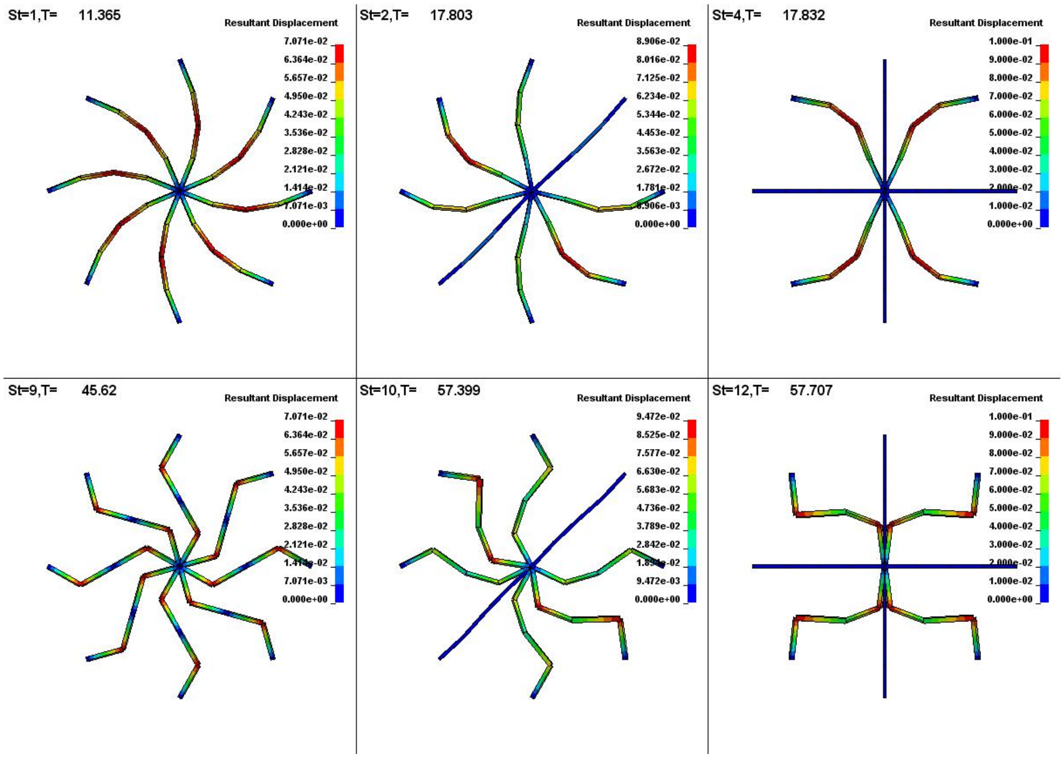

Figure 8.8 - Mode shapes for Timoshenko beam (displacement contouring).

Input deck:

```
*KEYWORD
*TITLE
Pin-Ended Double Cross: In-Plane Vibration
*CONTROL_IMPLICIT_AUTO
$#   iauto    iteopt    itewin     dtmin     dtmax
         1        11        15       0.0       0.0
*CONTROL_IMPLICIT_EIGENVALUE
$#    neig    center     lflag    lftend     rflag    rhtend    eigmth    shfscl
        16    11.000         0 -1.00e+29         0  1.00e+29         2       0.0
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1  1.00e-04         2         1         2
*CONTROL_IMPLICIT_SOLVER
$#  lsolvr    lprint     negev     order      drcm    drcprm   autospc   autotol
        16         1         1         0         1       0.0         1       0.0
*DATABASE_BINARY_D3PLOT
$# dt/cycl   lcdt/nr      beam     npltc    psetid
  0.010000         0         2
*ELEMENT_BEAM
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1       2       3       0       0       0       0       2

      32       1      61      74      76       0       0       0       0       2
*NODE
$#   nid               x               y               z      tc      rc
       1             0.0             0.0             0.0

      76      2.79029131     -2.20970869      1.00000000
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         2         0         1         1         1         1         1         0
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         0         0         1         1         1         0
*PART
$# title

$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_BEAM
$#   secid    elform      shrf   qr/irid       cst     scoor
         1         1  0.833333       2.0       0.0       0.0
$#     ts1       ts2       tt1       tt2     nsloc     ntloc
  0.125000  0.125000  0.125000  0.125000       0.0       0.0
$$#   secid    elform      shrf   qr/irid       cst     scoor
$         1         2  0.833333         2         0       0.0
$$#       a       iss       itt         j        sa       ist
$ 0.01562502.0345e-052.0345e-054.0690e-050.01302083
$$#   secid    elform      shrf   qr/irid       cst     scoor
$         1        13  0.833333         2         0       0.0
$$#       a       iss       itt         j        sa       ist
$ 0.01562502.0345e-052.0345e-054.0690e-050.01302083
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1  8000.0002.0000e+11  0.300000       0.0       0.0       0.0
*SET_NODE_LIST_GENERATE
$#     sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0
$#   b1beg     b1end     b2beg     b2end     b3beg     b3end     b4beg     b4end
         2         4         6        24        27        29        31        42
        44        46        48        59        61        63        65        76
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4    solver
         2       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1         5        30        47        64        60        43        26
*END
```

Notes:

1. The main difference among these element formulations is the inclusion of different second order terms for rotary inertia and shear distortion. The Euler (Belytschko-Schwer) beam model includes only the first order terms, lateral displacement and bending moment. The simple shear (Hughes-Liu) beam model includes only the translation first order term (no bending) plus shear distortion, while the Timoshenko beam model includes both rotary inertia and shear distortion in addition to the first order effects.

9. Simply Supported Thin Annular Plate (coarse mesh)

Keyword:

```
*CONTROL_IMPLICIT_EIGENVALUE
*CONTROL_IMPLICIT_GENERAL
```

Description:

A simply-supported annular plate of thickness t=0.06 m is to be analyzed to determine the first nine natural frequencies. The inner radius is 1.8 m and the outer radius is 6.0 m. This coarse mesh analysis has 26 shell elements (circumferential) by 3 elements (radial). All nodes have $U_X = U_Y = R_Z = 0$. On the outer nodes $U_Z = 0$.

A sketch representing the structure is shown below (Figure 9.1) along with the finite element model (Figure 9.2).

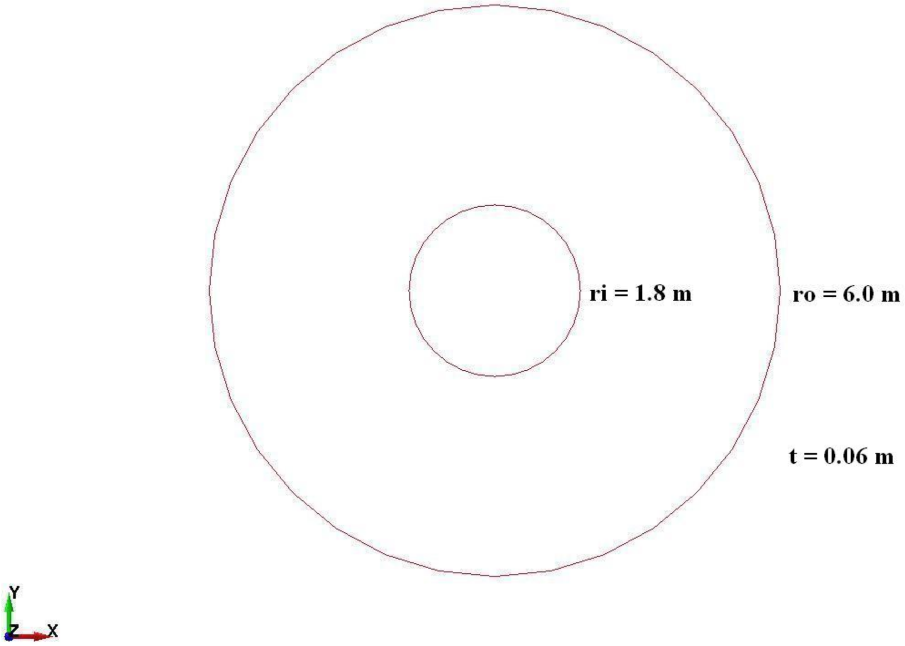

Figure 9.1 – Sketch representing the structure.

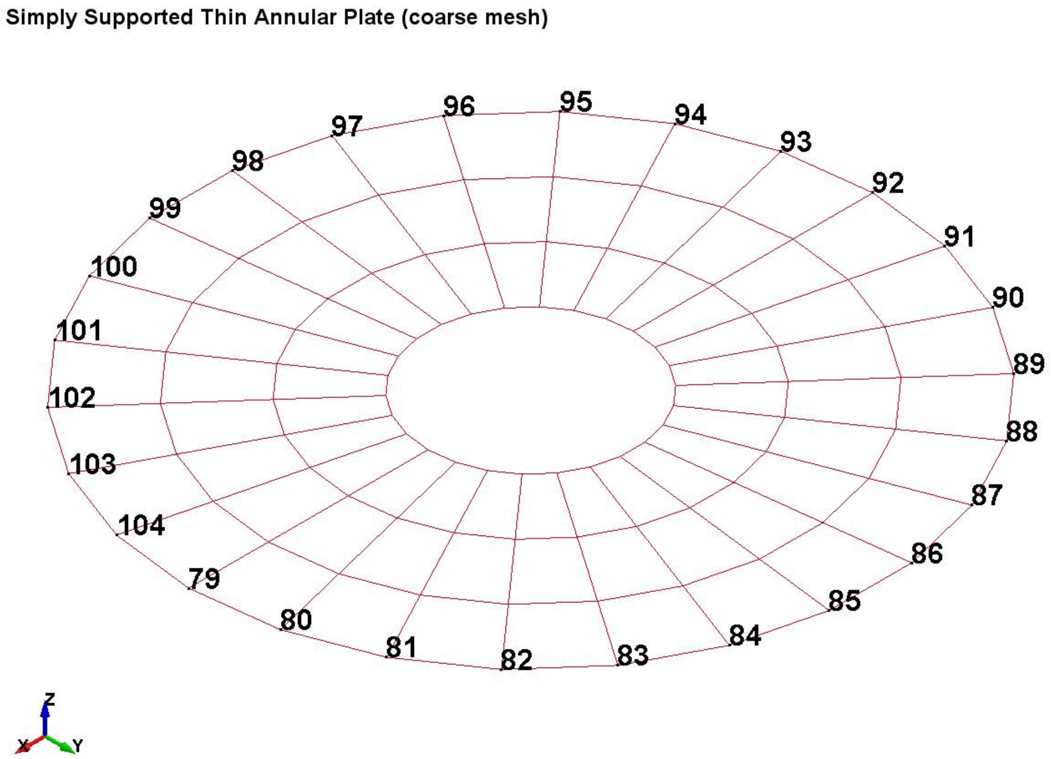

Figure 9.2 – Coarse mesh finite element model with simply supported boundary conditions on outer nodes.

Analysis Summary:

Dim.	**Type**	**Load**	**Material**	**Geometry**	**Contact**	**Solver**	**Solution Method**
3D	Modal	-	Linear	Linear	-	Implicit	Block Shift and Inverted Lanczos

Units:

kg, m, s, N, Pa, N-m (kilogram, meter, second, Newton, Pascal, Newton-meter)

Dimensional Data:

$r_o = 6\ m,\ r_i = 1.8\,m,\ t = 0.06\ m$

Material Data:

Mass Density	$\rho = 8.00\times10^{3}\ kg/m^{3}$
Young's Modulus	$E = 2.00\times10^{11}\ Pa$
Poisson's Ratio	$\nu = 0.3$

Element Types:

Fully integrated shell (elform=16)

Material Models:

*MAT_001 or *MAT_ELASTIC

Results Comparison:

LS-DYNA natural frequencies, first 10 (frequency in *Hertz*), and mode shapes (first 5) are compared with *NAFEMS Natural Frequency Benchmark* NF14.

Mode(s)	***NAFEMS* NF14 (*Hz*)**	**Coarse Mesh (*Hz*)**
1	1.870	1.806
2, 3	5.137	5.423
4, 5	9.673	10.179
6	14.850	13.217
7, 8	15.570	16.239
9, 10	18.380	16.691

It is seen that even with this rather coarse mesh refinement, the LS-DYNA natural frequency results provide a fair comparison with the *NAFEMS Selected Benchmarks for Natural Frequency Analysis*, NF14 test values.

Eigenvalue Results:

From the *eigout* file, generated by the *CONTROL_IMPLICIT_EIGENVALUE keyword:

```
Simply Supported Thin Annular Plate (coarse mesh)
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1   1.287078E+02   1.134495E+01   1.805604E+00   5.538311E-01
           2   1.161053E+03   3.407423E+01   5.423083E+00   1.843970E-01
           3   1.161053E+03   3.407423E+01   5.423083E+00   1.843970E-01
           4   4.090826E+03   6.395957E+01   1.017948E+01   9.823683E-02
           5   4.090827E+03   6.395957E+01   1.017948E+01   9.823683E-02
           6   6.896060E+03   8.304252E+01   1.321663E+01   7.566227E-02
           7   1.041122E+04   1.020354E+02   1.623944E+01   6.157849E-02
           8   1.041122E+04   1.020354E+02   1.623944E+01   6.157848E-02
           9   1.099787E+04   1.048707E+02   1.669070E+01   5.991361E-02
          10   1.099787E+04   1.048707E+02   1.669070E+01   5.991361E-02
```

Mode Shapes (first five):

Figures 9.3, 9.4, and 9.5 show the first 5 mode shapes with no contouring while Figures 9.6. 9.7, and 9.8 show the same 5 mode shapes with displacement contouring.

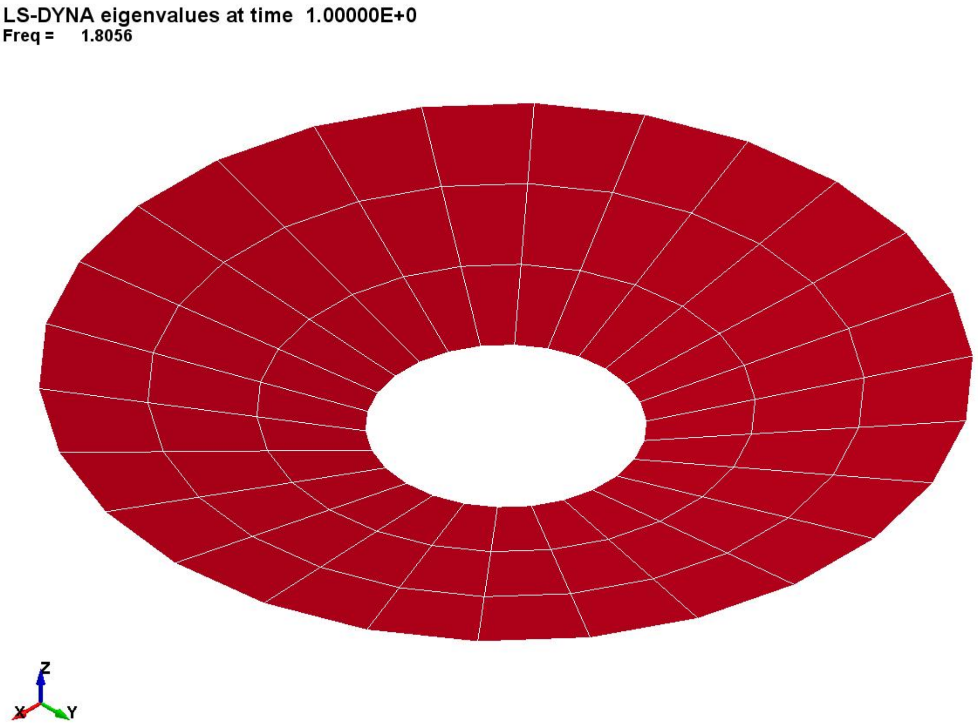

Figure 9.3 - Mode 1, 1.806 *Hz* (*NAFEMS* 1.870) - no contouring.

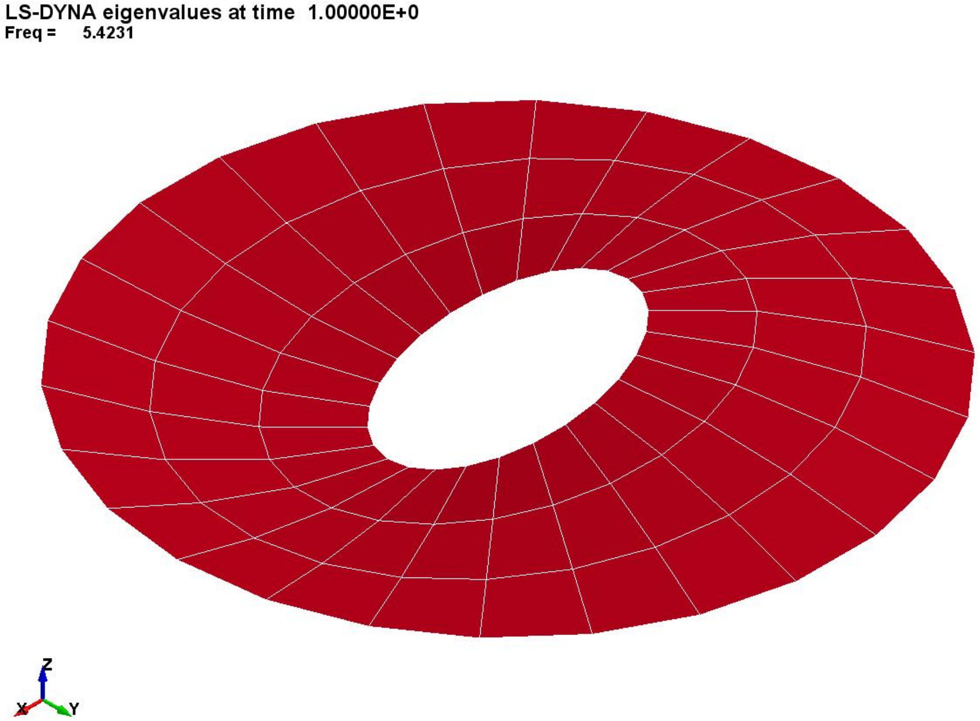

Figure 9.4 - Modes 2 and 3, 5.423 *Hz* (*NAFEMS* 5.137) - no contouring.

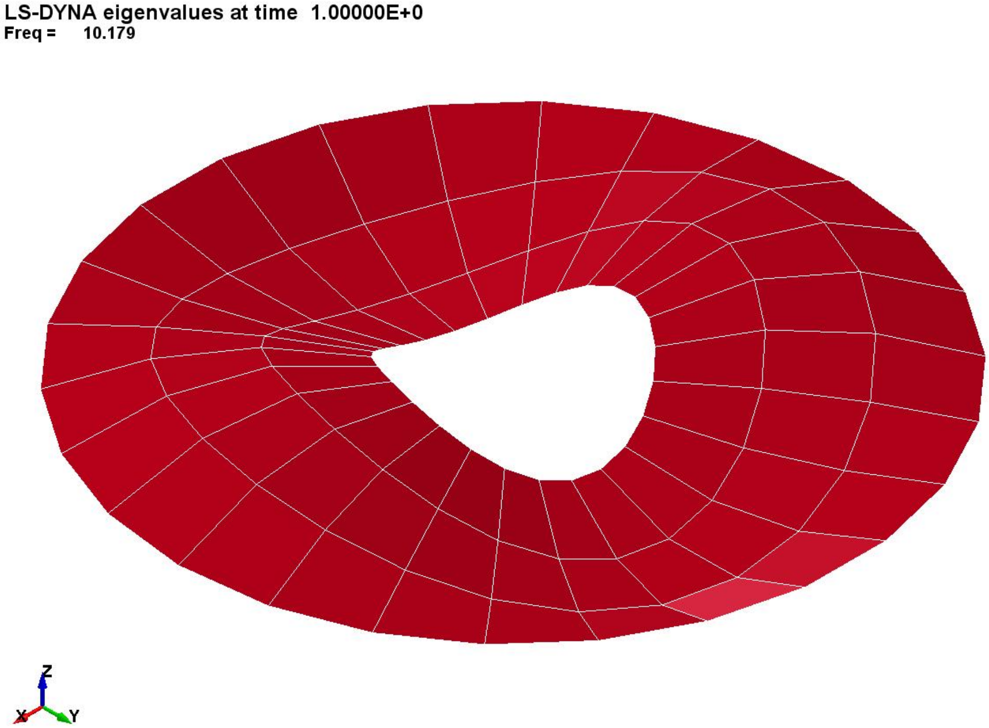

Figure 9.5 - Modes 4 and 5, 10.179 *Hz* (*NAFEMS* 9.673) - no contouring.

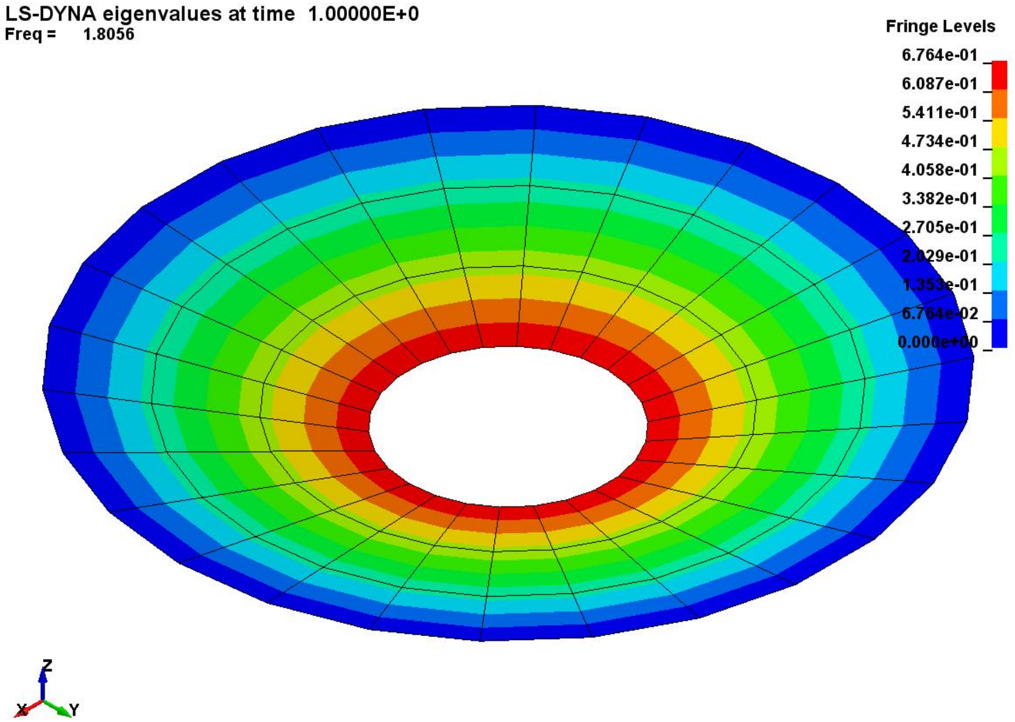

Figure 9.6 - Mode 1, 1.806 *Hz* (*NAFEMS* 1.870) - displacement contouring.

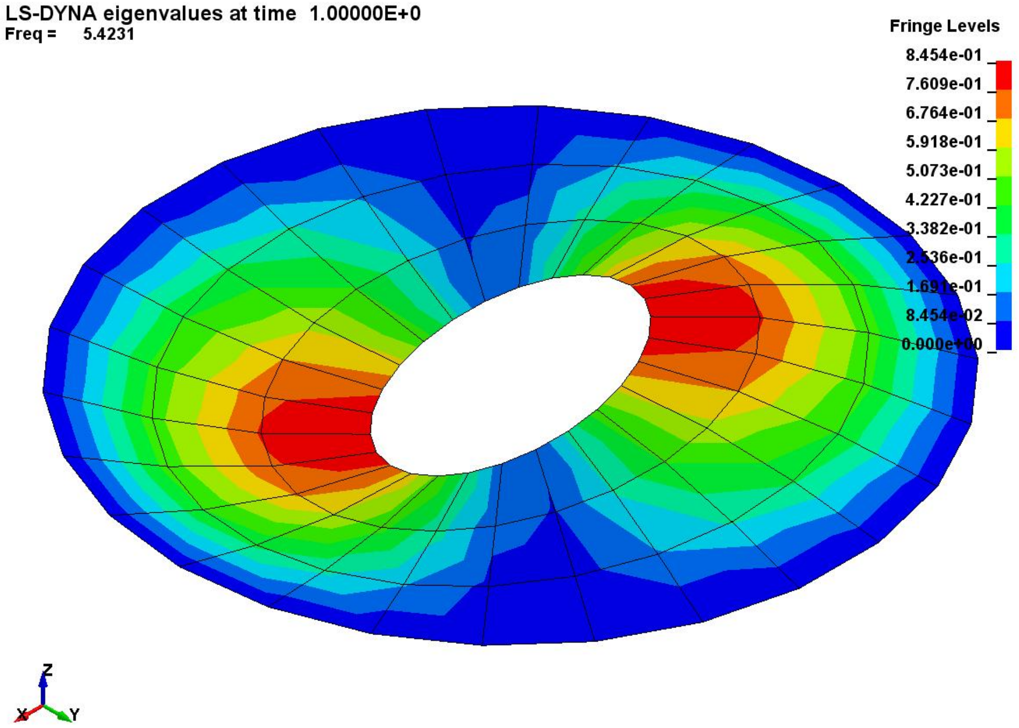

Figure 9.7 - Modes 2 and 3, 5.423 *Hz* (*NAFEMS* 5.137) - displacement contouring.

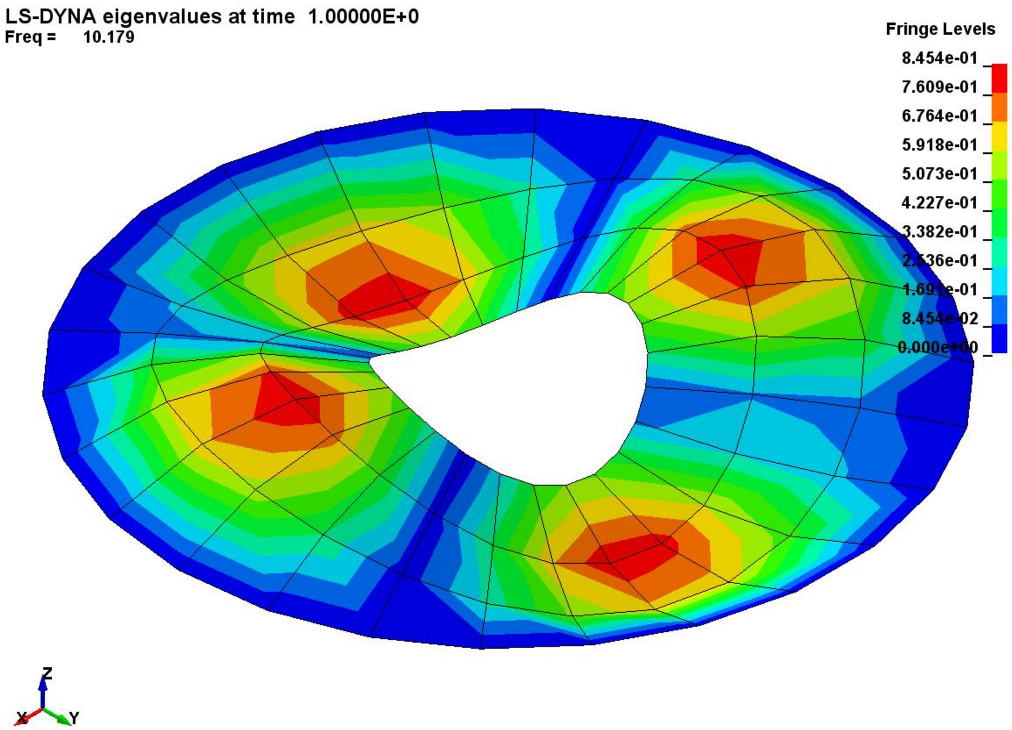

Figure 9.8 - Modes 4 and 5, 10.179 *Hz* (*NAFEMS* 9.673) - displacement contouring.

Input deck:

```
*KEYWORD
*TITLE
Simply Supported Thin Annular Plate (coarse mesh)
*CONTROL_IMPLICIT_EIGENVALUE
$#    neig    center     lflag    lftend     rflag    rhtend    eigmth    shfscl
        10       0.0         1  1.000000         1  30.00000         2       0.0
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1  1.000000
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  1.000000         0       0.0       0.0       0.0
*DATABASE_BINARY_D3PLOT
$# dt/cycl   lcdt/nr      beam     npltc    psetid
  1.000000
*ELEMENT_SHELL
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1      27      28       2

      78       1      78     104      79      53
*NODE
$#   nid               x               y               z      tc      rc
       1      1.79999995             0.0             0.0

     104      5.82565355     -1.43588233             0.0       3       1
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         1         1         1         0         0         1
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         2         0         1         1         0         0         0         1
*PART
$# title
material type # 1  (Elastic)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_SHELL
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
         1        16       0.0         0         1       0.0         0         1
$#      t1        t2        t3        t4      nloc     marea
  0.060000  0.060000  0.060000  0.060000         0       0.0
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1  8000.0002.0000e+11  0.300000       0.0       0.0       0.0
*SET_NODE_LIST_GENERATE
$#     sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0
$#   b1beg     b1end     b2beg     b2end     b3beg     b3end     b4beg     b4end
        79       104
*SET_NODE_LIST_GENERATE
$#     sid       da1       da2       da3       da4    solver
         2       0.0       0.0       0.0       0.0
$#   b1beg     b1end     b2beg     b2end     b3beg     b3end     b4beg     b4end
         1        78
*END
```

Notes:

10. Simply Supported Thin Annular Plate (fine mesh)

Keyword:

```
*CONTROL_IMPLICIT_EIGENVALUE
*CONTROL_IMPLICIT_GENERAL
```

Description:

A simply-supported annular plate of thickness t=0.06 m is to be analyzed to determine the first nine natural frequencies. The inner radius is 1.8 m and the outer radius is 6.0 m. This fine mesh analysis has 32 shell elements (circumferential) by 5 elements (radial). All nodes have $U_X = U_Y = R_Z = 0$. On the outer nodes $U_Z = 0$.

A sketch representing the structure is shown below (Figure 10.1) along with the finite element model (Figure 10.2).

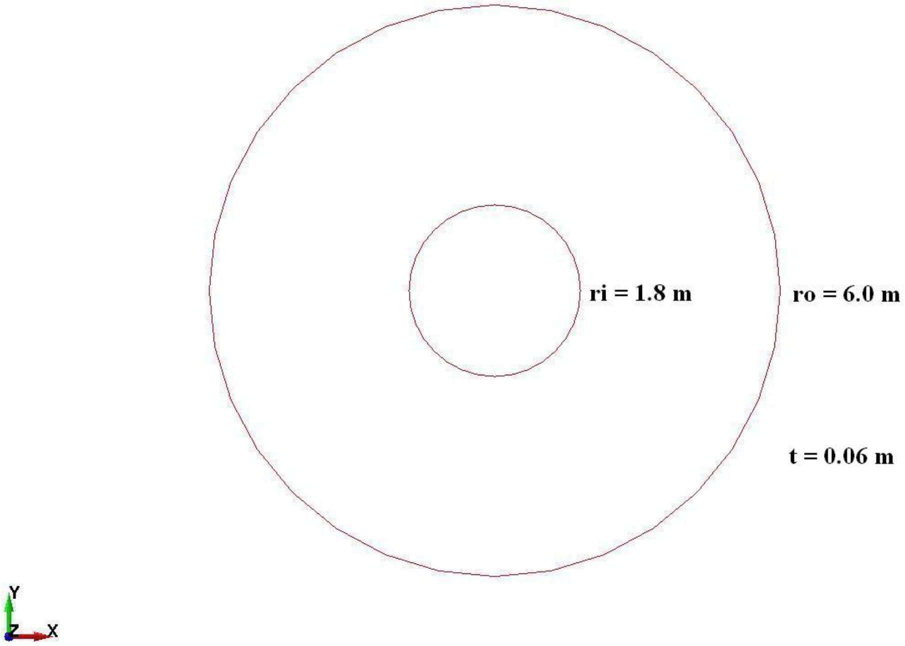

Figure 10.1 – Sketch representing the structure.

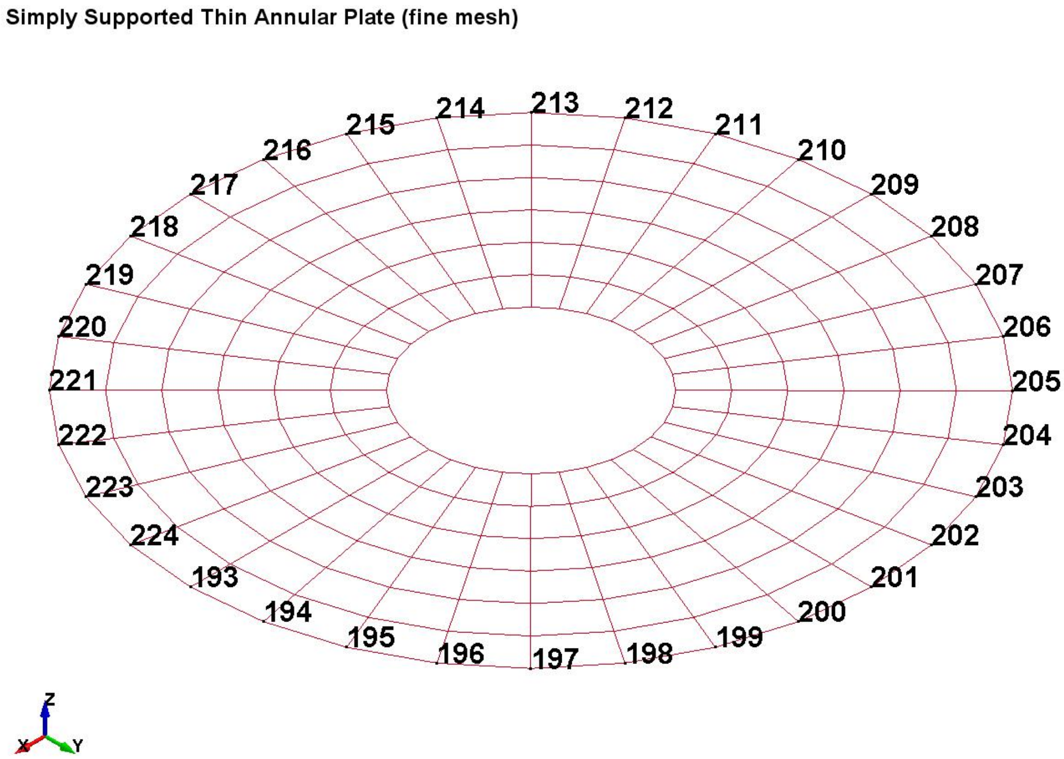

Figure 10.2 – Fine mesh finite element model with simply supported boundary conditions on outer nodes.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Modal	-	Linear	Linear	-	Implicit	Block Shift and Inverted Lanczos

Units:

kg, m, s, N, Pa, N-m (kilogram, meter, second, Newton, Pascal, Newton-meter)

Dimensional Data:

$r_o = 6\ m,\ r_i = 1.8 m,\ t = 0.06\ m$

Material Data:

Mass Density $\rho = 8.00 \times 10^3 \ kg / m^3$
Young's Modulus $E = 2.00 \times 10^{11} \ Pa$
Poisson's Ratio $\nu = 0.3$

Element Types:

Fully integrated shell (elform=16)

Material Models:

*MAT_001 or *MAT_ELASTIC

Results Comparison:

LS-DYNA natural frequencies, first 10 (frequency in *Hertz*), and mode shapes (first 5) are compared with *NAFEMS Natural Frequency Benchmark* NF14.

Mode(s)	***NAFEMS* NF14 (*Hz*)**	**Fine Mesh (*Hz*)**
1	1.870	1.867
2, 3	5.137	5.197
4, 5	9.673	9.801
6	14.850	14.471
7, 8	15.570	15.665
9, 10	18.380	17.798

It is seen that with only a slight increase in mesh refinement, the LS-DYNA natural frequency results compare nicely with the *NAFEMS Selected Benchmarks for Natural Frequency Analysis*, NF14 test values.

Eigenvalue Results:

From the *eigout* file, generated by the *CONTROL_IMPLICIT_EIGENVALUE keyword:

```
Simply Supported Thin Annular Plate (fine mesh)
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1   1.377019E+02   1.173465E+01   1.867627E+00   5.354388E-01
           2   1.066283E+03   3.265399E+01   5.197045E+00   1.924171E-01
           3   1.066283E+03   3.265399E+01   5.197045E+00   1.924171E-01
           4   3.791946E+03   6.157878E+01   9.800567E+00   1.020349E-01
           5   3.791946E+03   6.157878E+01   9.800567E+00   1.020349E-01
           6   8.267193E+03   9.092411E+01   1.447102E+01   6.910362E-02
           7   9.688143E+03   9.842836E+01   1.566536E+01   6.383511E-02
           8   9.688143E+03   9.842836E+01   1.566536E+01   6.383511E-02
           9   1.250545E+04   1.118278E+02   1.779794E+01   5.618626E-02
          10   1.250545E+04   1.118278E+02   1.779794E+01   5.618626E-02
```

Mode Shapes (first three):

Figures 10.3, 10.4, and 10.5 show the first 5 mode shapes with no contouring while Figures 10.6. 10.7, and 10.8 show the same 5 mode shapes with displacement contouring.

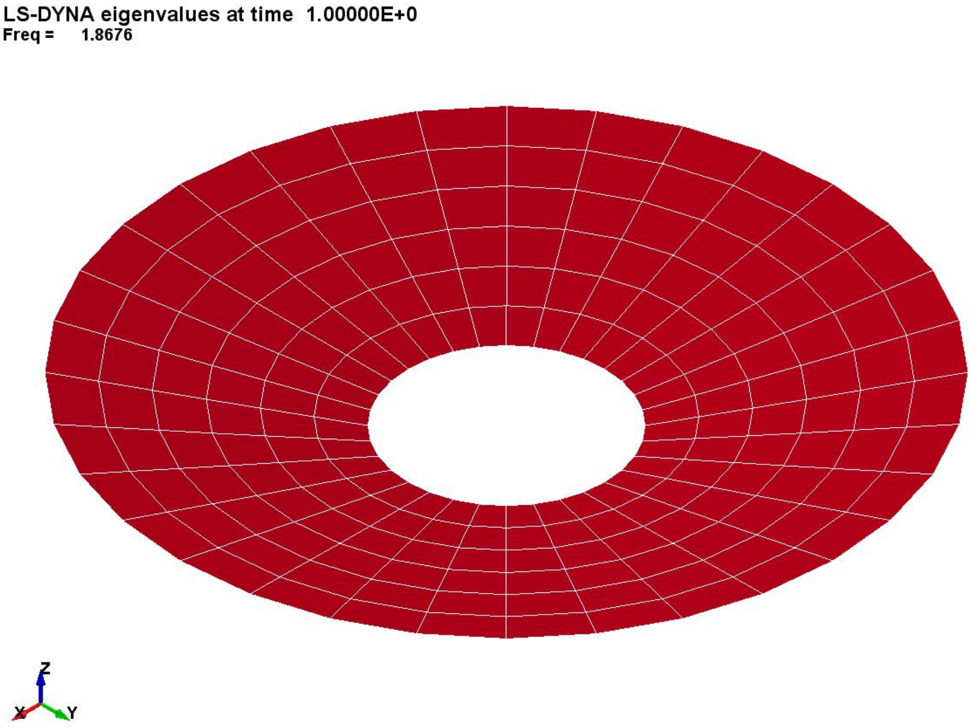

Figure 10.3 - Mode 1, 1.867 *Hz* (*NAFEMS* 1.870) - no contouring.

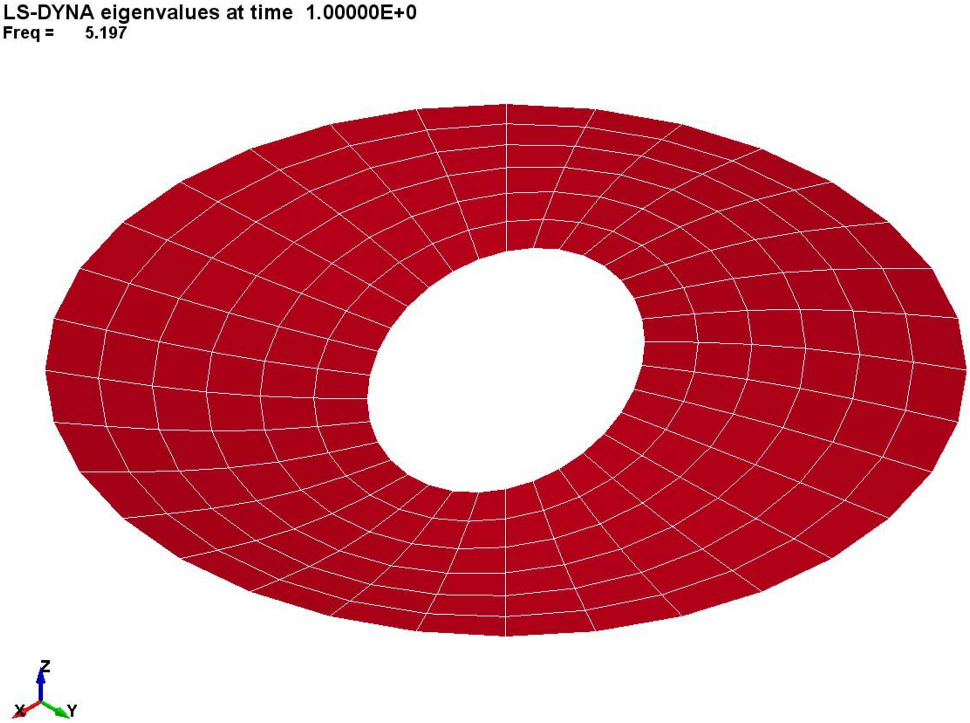

Figure 10.4 - Modes 2 and 3, 5.197 *Hz* (*NAFEMS* 5.137) - no contouring.

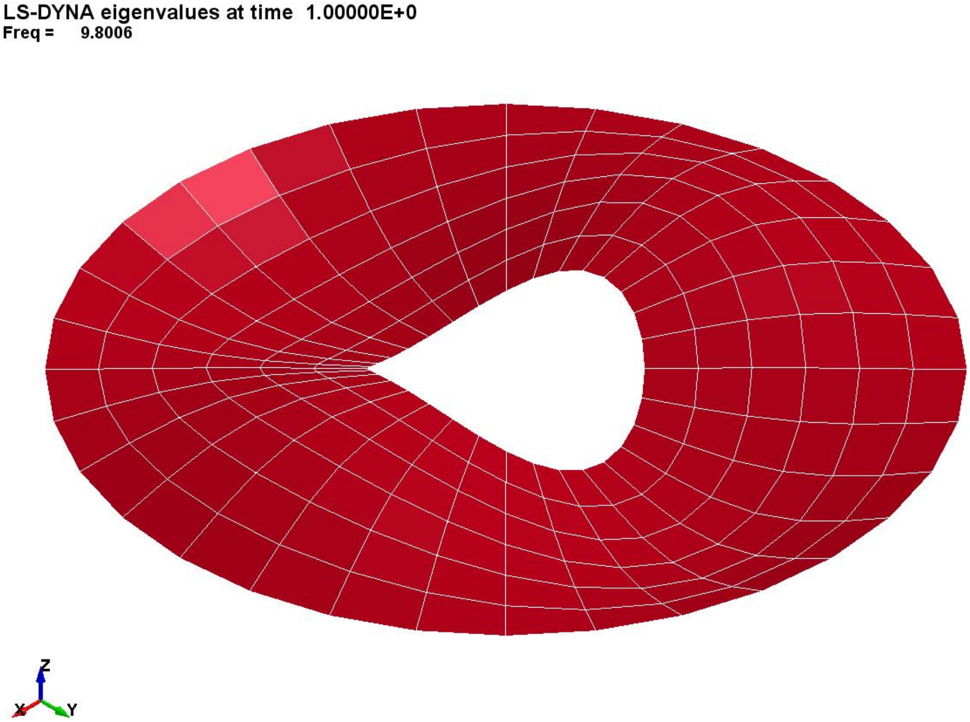

Figure 10.5 - Modes 4 and 5, 9.801 *Hz* (*NAFEMS* 9.673) - no contouring.

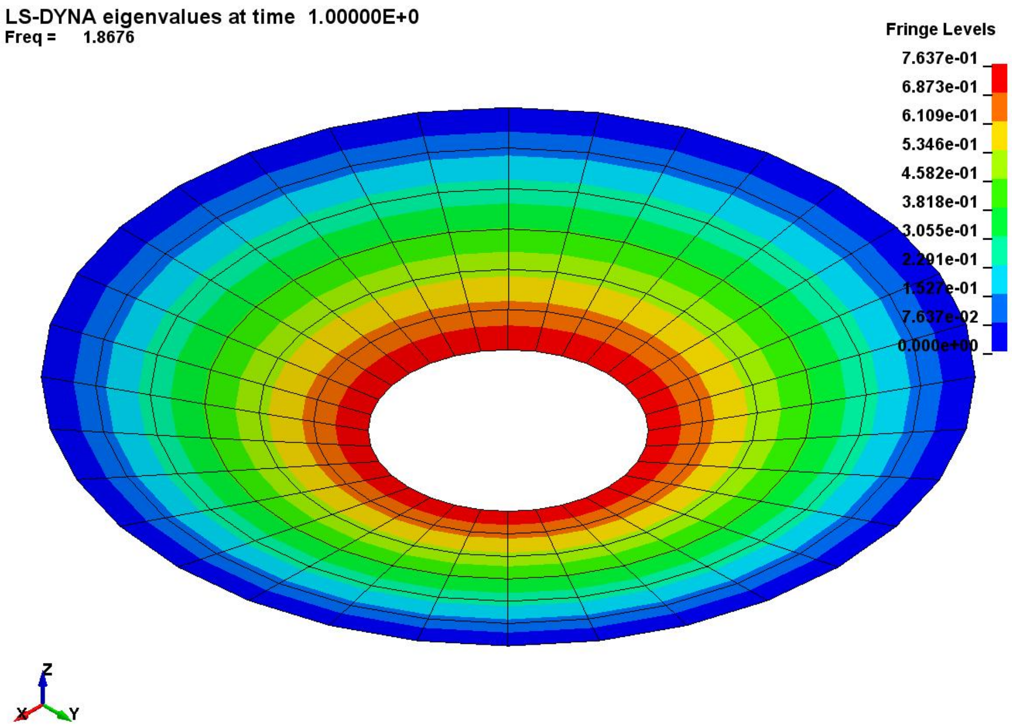

Figure 10.6 - Mode 1, 1.867 *Hz* (*NAFEMS* 1.870) - displacement contouring.

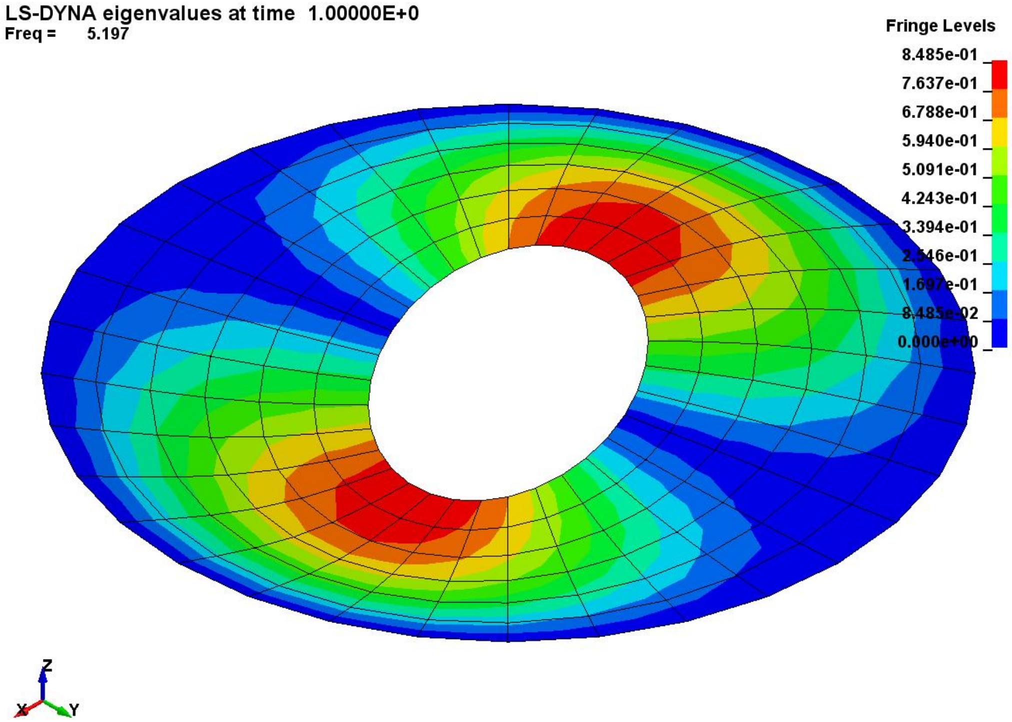

Figure 10.7 - Modes 2 and 3, 5.197 *Hz* (*NAFEMS* 5.137) - displacement contouring.

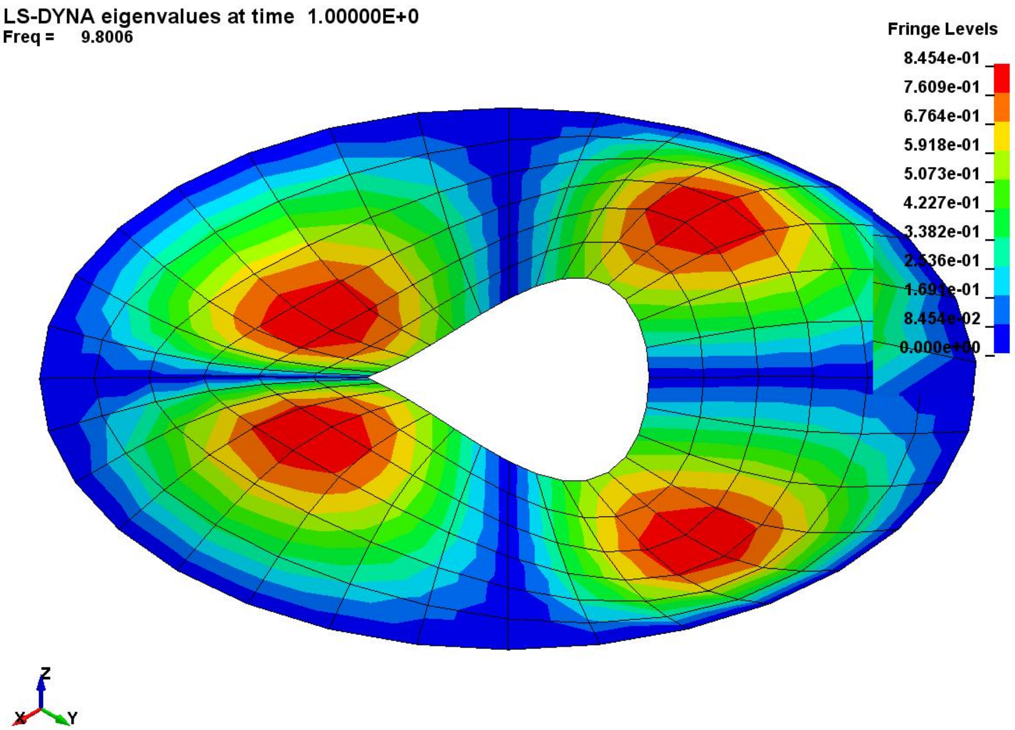

Figure 10.8 - Modes 4 and 5, 9.801 *Hz* (*NAFEMS* 9.673) - displacement contouring.

Input deck:

```
*KEYWORD
*TITLE
Simply Supported Thin Annular Plate (fine mesh)
*CONTROL_IMPLICIT_EIGENVALUE
$#    neig    center     lflag    lftend     rflag    rhtend    eigmth    shfscl
        10       0.0         1  1.000000         1  30.00000         2       0.0
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1  1.000000
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  1.000000         0       0.0       0.0       0.0
*DATABASE_BINARY_D3PLOT
$# dt/cycl   lcdt/nr      beam     npltc    psetid
  1.000000
*ELEMENT_SHELL
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1      33      34       2

     192       1     192     224     193     161
*NODE
$#   nid               x               y               z      tc      rc
       1      1.79999995             0.0             0.0

     224      5.88471174     -1.17054188             0.0       3       1
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         1         1         1         0         0         1
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         2         0         1         1         0         0         0         1
*PART
$# title
material type # 1  (Elastic)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_SHELL
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
         1         6       0.0         0         1       0.0         0         1
$#      t1        t2        t3        t4      nloc     marea
  0.060000  0.060000  0.060000  0.060000         0       0.0
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1  8000.0002.0000e+11  0.300000       0.0       0.0       0.0
*SET_NODE_LIST_GENERATE
$#     sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0
$#   b1beg     b1end     b2beg     b2end     b3beg     b3end     b4beg     b4end
       193       224
*SET_NODE_LIST_GENERATE
$#     sid       da1       da2       da3       da4    solver
         2       0.0       0.0       0.0       0.0
$#   b1beg     b1end     b2beg     b2end     b3beg     b3end     b4beg     b4end
         1       192
*END
```

Notes:

11. Transient Response to a Constant Force

Keyword:

```
*CONTROL_IMPLICIT_DYNAMICS
*CONTROL_IMPLICIT_GENERAL
*CONTROL_IMPLICIT_SOLVER
*CONTROL_IMPLICIT_SOLUTION
```

Description:

A mass m=25.9067 *lbf-s²/in* is attached in the middle of a steel beam of length l=240 inches and geometric properties shown below. The beam is subjected to a dynamic load *F*(*t*) with a rise time of 0.075 seconds and a maximum constant value of 2000 pound-force. The weight of the beam is considered negligible. Determine the time of maximum displacement response t_{max} and the maximum displacement response y_{max}. Additionally, determine the maximum bending stress σ_{bend} in the beam. The attached mass is modeled with a lumped mass element at the central node of the beam.

A sketch representing the structure is shown below (Figure 11.1) along with the finite element model (Figure 11.2).

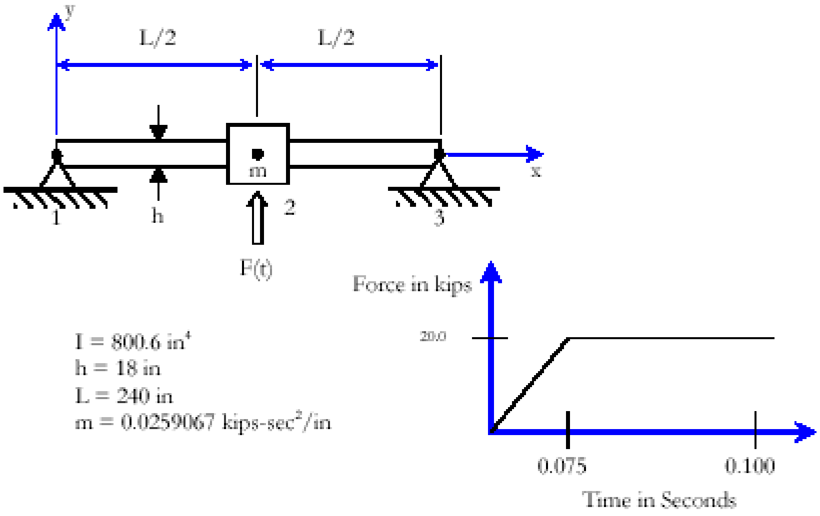

Figure 11.1 – Sketch representing the structure.

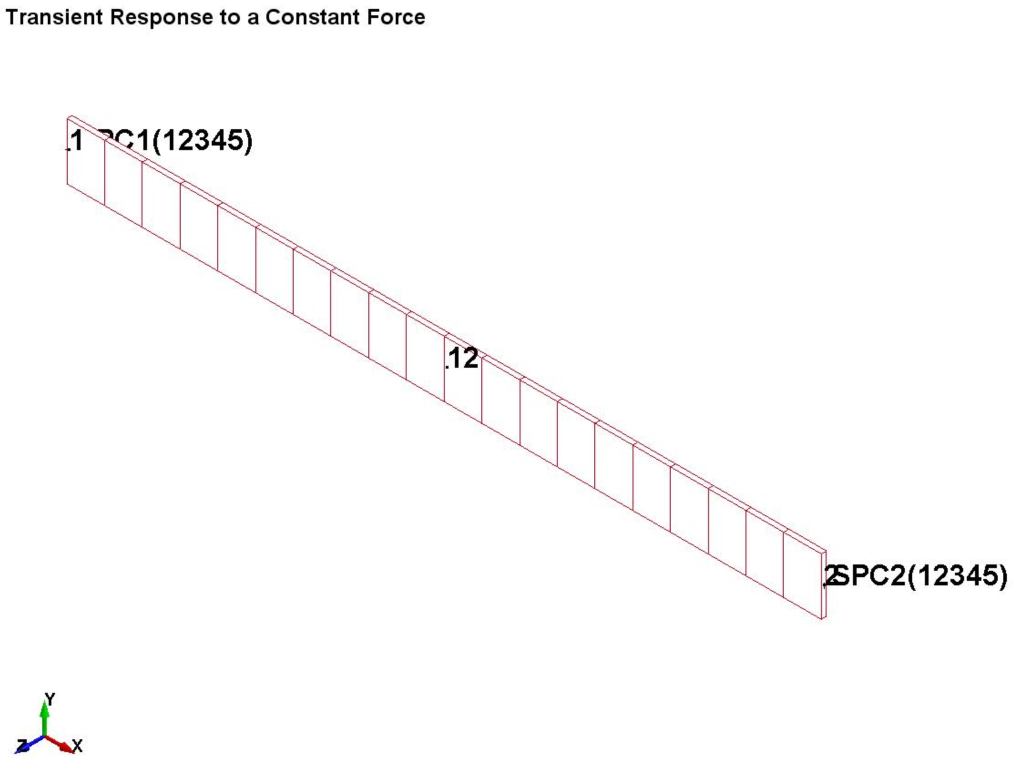

Figure 11.2 – Finite element model with applied load (Node 12) and boundary conditions. In-plane boundary conditions and lumped mass (Node 12) are not shown.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Dynamic	Force	Linear	Linear	-	Implicit	2-Nonlinear w/BFGS

Units:

lbf-s^2/in, in, s, lbf, psi, lbf-in (blob, inch, second, pound force, pound force/inch^2, pound force-inch)

Dimensional Data:

$L = 240.0\ in$, $h = 18.0\ in$, $I_z = 800.6\ in^4$

As a cross-section-integrated beam is used, the cross sectional dimension is calculated. Given $I_z = 800.6\ in^4$ and $h = 18.0\ in$, a thickness of $t = 1.647\ in$ is obtained.

Material Data:

Mass Density	$\rho = 1.0 \times 10^{-20}\ lbf - s^2 / in^4$
Young's Modulus	$E = 3.0 \times 10^7\ \ psi$
Poisson's Ratio	$\nu\ = 0.3$
Nodal Mass	$m = 25.9067\ blobs$

Load:

Lateral Load $F(t) =$ *DEFINE_CURVE

Element Types:

Hughes-Liu beam with cross section integration (elform=1)

Lumped mass (*ELEMENT_MASS entry)

Material Models:

*MAT_001 or *MAT_ELASTIC

Results Comparison:

LS-DYNA results for time of the maximum displacement response t_{max}, the maximum displacement response y_{max}, and the maximum bending stress σ_{bend} in the beam are compared with J.M. Biggs' studies in *Introduction to Structural Dynamics* (pg. 50).

	Time - t_{max} (s)	**Disp. - y_{max} (in)**	**Stress - σ_{bend} (psi)**
Biggs	0.0920	0.3310	1.8600×10^4
Node 12/Element 10	0.0930	0.3421	1.8151×10^4

These nodal time/displacement results (Figure 11.3) were generated by *DATABASE_ NODOUT keyword while the element stress results (Figure 11.4) were generated by *DATABASE_ELOUT.

LS-DYNA stress and strain output corresponds to integration point locations.

Lobatto integration (qr=4 - 3×3 quadrature - *SECTION_BEAM) was employed since it has an advantage in that the inner and outer integration points are on the beam surfaces. Gauss integration is the default quadrature rule (qr=2 - 2×2 quadrature - *SECTION_ BEAM).

The contour plots of the axial beam stresses for the upper (ip=1) and lower (ip=3) surfaces of the beam (Figure 11.5) were obtained from the *d3plot* file at t=0.093 s which were generated by the *DATABASE_BINARY_D3PLOT keyword.

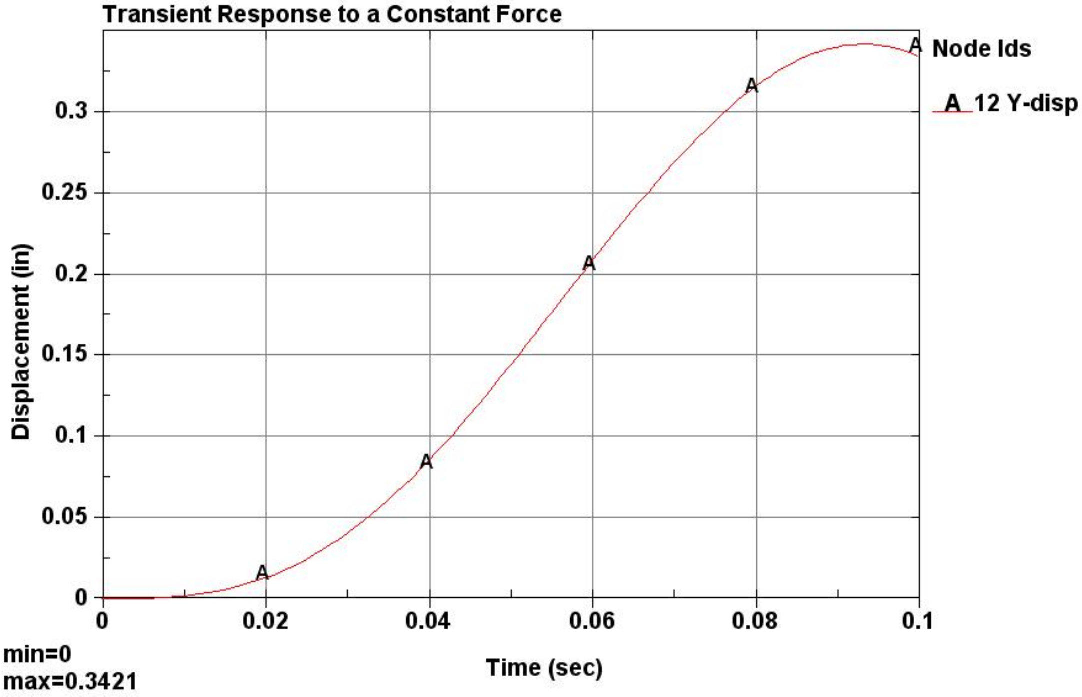

Figure 11.3 – Y-displacement vs. time for Node 12 at the center of the beam.

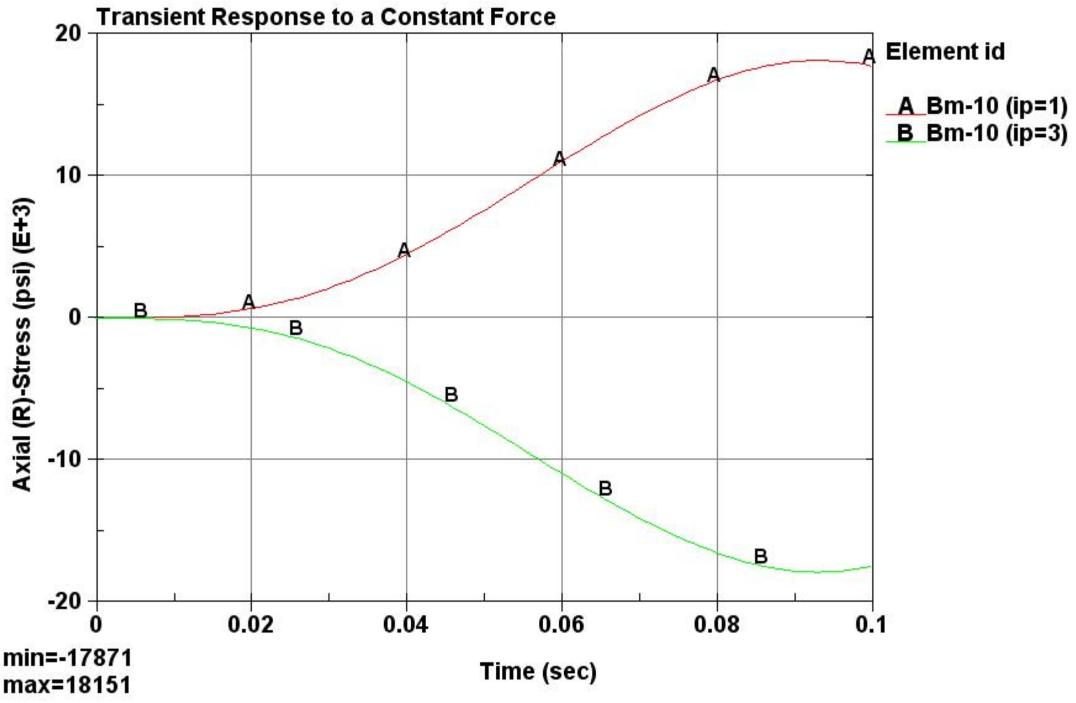

Figure 11.4 – Axial beam stress vs. time at the upper (ip=1), the point of maximum bending stress, and the lower (ip=3) beam surfaces for Element 10.

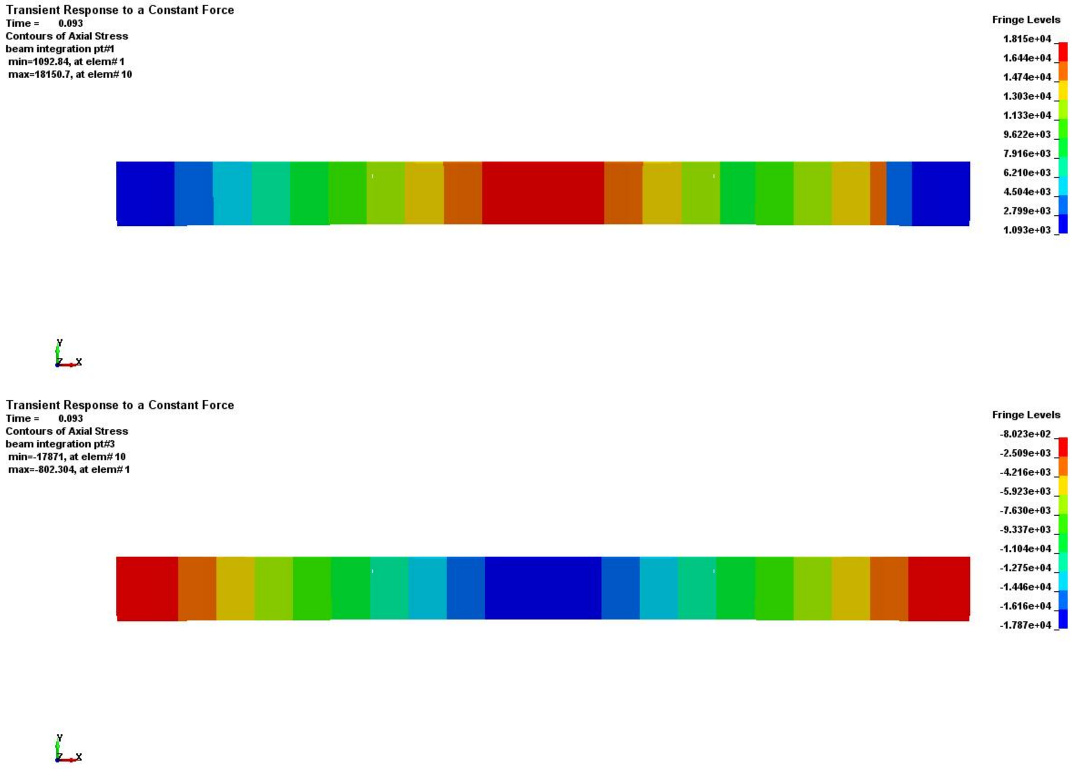

Figure 11.5 – Contour plots of the axial beam stresses for the upper (ip=1) and lower (ip=3) surfaces of the beam at t=0.093 s.

Input deck:

```
*KEYWORD
*TITLE
Transient Response to a Constant Force
*CONTROL_IMPLICIT_DYNAMICS
$#   imass     gamma      beta
         1  0.500000  0.250000
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1  0.001000         2         1         2
*CONTROL_IMPLICIT_SOLVER
$#  lsolvr    lprint     negev     order      drcm    drcprm   autospc   autotol
         4         2         2         0         1       0.0         1       0.0
$#  lcpack
         2
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
         2        11        15    0.0010    0.0100  1.00e+10  0.900000  1.00e-10
$#   dnorm    diverg     istif   nlprint
         2         1         1         2
$#  arcctl    arcdir    arclen    arcmth    arcdmp
         0         1       0.0         1         2
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  0.100000         0  0.900000       0.0       0.0
*DATABASE_ELOUT
$#      dt    binary
1.0000e-05         1
*DATABASE_MATSUM
$#      dt    binary
1.0000e-05         1
*DATABASE_NODOUT
$#      dt    binary
1.0000e-05         1
*DATABASE_SPCFORC
$#      dt    binary
1.0000e-05         1
*DATABASE_BINARY_D3PLOT
$# dt/cycl   lcdt/nr      beam     npltc    psetid
1.0000e-05
*DATABASE_BINARY_D3THDT
$# dt/cycl   lcdt/nr      beam     npltc    psetid
2.0000e-04
*DATABASE_EXTENT_BINARY
$#   neiph     neips    maxint    strflg    sigflg    epsflg    rltflg    engflg
         0         0         0         0         1         1         1         1
$#  cmpflg    ieverp    beamip     dcomp      shge     stssz    n3thdt   ialemat
         0         0        20         1         1         1         2
*DATABASE_HISTORY_NODE
$#    nid1      nid2      nid3      nid4       ni5      nid6      nid7      nid8
        12         1         2
*DATABASE_HISTORY_BEAM
$#    eid1      eid2      eid3      eid4       ei5      eid6      eid7      eid8
        10        11
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo     dattyp
         1         0  1.000000  1.000000       0.0       0.0
$#                a1                  o1
                 0.0                 0.0
          0.07500000       2.0000000e+04
          1.00000000       2.0000000e+04
*ELEMENT_BEAM
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1       3      22

      20       1      21       2      41
*ELEMENT_MASS
$#   eid      id            mass     pid
      23      12     25.90670013
```

```
*NODE
$#   nid               x               y               z      tc      rc
       1             0.0             0.0             0.0

      41     234.0000000             0.0      0.99996525
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         2         0         1         1         1         1         1
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         3         0         0         0         1         1         1
*PART
$# title
Part          1 for Mat         1 and Elem Type         1
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_BEAM
$#   secid    elform      shrf   qr/irid       cst     scoor
         1         1  0.830000         4         0       0.0
$#     ts1       ts2       tt1       tt2     nsloc     ntloc
  1.647220  1.647220      18.0      18.0
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         11.0000e-203.0000e+07  0.300000       0.0       0.0       0.0
*LOAD_NODE_SET
$#    nsid       dof      lcid        sf       cid        m1        m2        m3
         1         2         1  1.000000
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
        12
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4    solver
         2       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1         2
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4    solver
         3       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         3         4         5         6         7         8         9        10
        11        12        13        14        15        16        17        18
        19        20        21        22        23        24        25        26
        27        28        29        30        31        32        33        34
        35        36        37        38        39        40        41
*END
```

Notes:

12. Simply Supported Square Plate: Out-of-Plane Vibration (solid mesh)

Keywords:

```
*CONTROL_IMPLICIT_EIGENVALUE
*CONTROL_IMPLICIT_GENERAL
```

Description:

Determine the first 10 natural frequencies of a solid simply-supported plate of thickness t=1.0 *m*. Each side of the plate measure 10.0 *m*. The plate is meshed with solid hexahedra element with an 8 x 8 x 3 density. On the lower surface, outer boundary nodes, $U_Z = 0$.

The finite element model is shown in Figure 12.1.

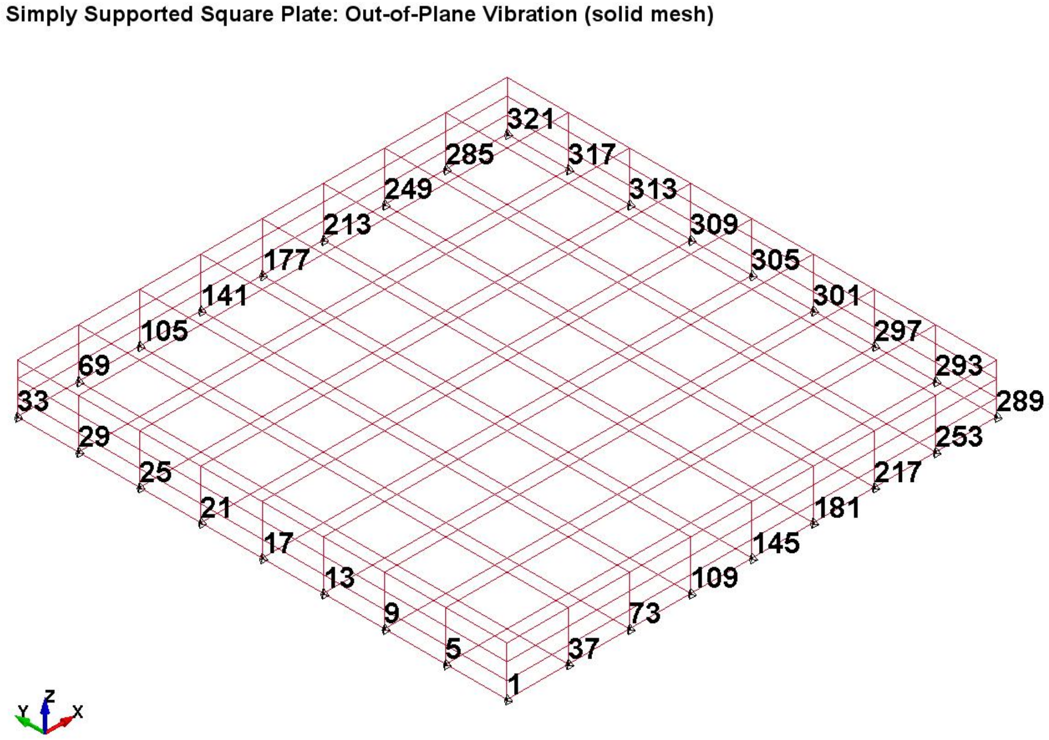

Figure 12.1 – Finite element model with simply supported boundary conditions on lower surface, outer nodes.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Modal	-	Linear	Linear	-	Implicit	Block Shift and Inverted Lanczos

Units:

kg, m, s, N, Pa, N-m (kilogram, meter, second, Newton, Pascal, Newton-meter)

Dimensional Data:

Rectangular dimensions of square plate: 10.0 *m* x 10.0 *m* x 1.00 *m*.

Material Data:

Mass Density	$\rho = 8.00 \times 10^3 \ kg / m^3$
Young's Modulus	$E = 2.00 \times 10^{11} \ Pa$
Poisson's Ratio	$\nu \ = 0.3$

Element Types:

Constant stress solid (elform=1)
Fully integrated S/R solid (elform=2)
Fully integrated S/R solid - for poor aspect ratio (eff) - (elform=-1)
Fully integrated S/R solid - for poor aspect ratio (acc) - (elform=-2)
Fully integrated quadratic 8 node element with nodal rotations (elform=3)

Material Models:

*MAT_001 or *MAT_ELASTIC

Results Comparison:

LS-DYNA natural frequencies, first 10 (frequency in *Hertz*), and mode shapes (4 through 10) are compared with *NAFEMS Free Vibration Benchmarks***,** Test FV52.

Mode(s)	***NAFEMS*** **FV52 (*Hz*)**	**elform=1 (*Hz*)**	**elform=2 (*Hz*)**	**elform=-1 (*Hz*)**	**elform=-2 (*Hz*)**	**elform=3 (*Hz*)**
1, 2, 3	rigid body	rigid body	rigid body	rigid body	rigid body	rigid body
4	45.897	44.040	48.508	45.448	46.2060	43.370
5, 6	109.440	106.468	120.388	107.114	109.265	104.703
7	167.890	155.523	169.601	159.102	163.943	153.862
8	193.590	193.582	193.526	193.518	193.526	193.227
9	206.190	200.135	200.188	198.791	200.176	196.485
10	206.190	200.135	200.188	198.873	200.176	197.280

Hourglass control (*HOURGLASS) is necessary for the constant stress solid (elform=1) element formulation (the LS-DYNA default), especially at higher frequencies. Only this element formulation (elform=1) makes use of this feature

The constant stress solid (elform=1), the fully integrated S/R solid (elform=2), and the fully integrated quadratic 8 node element with nodal rotations (elform=3) all provide similar frequency results for this analysis.

The fully integrated quadratic 8 node element with nodal rotations (elform=3) formulation provides two distinct modes and frequencies for modes 9 and 10, whereas, all the other formulations provide the same results for modes 9 and 10. This is perhaps due to the accountability of the nodal rotations.

The aspect ratio of these elements is 3.75 (ratio of side to depth length). It would, however, for this frequency analysis, appear that the element formulations available to address poor aspect ratios (elform=-1 or -2) do not offer much improvement. The constant stress solid (elform=1), the fully integrated S/R solid (elform=2), and the fully integrated quadratic 8 node element with nodal rotations (elform=3) formulation all appear to provide more than adequate results for this frequency study.

The fully integrated S/R solid (the efficient formulation choice) intended to address poor aspect ratios (elform=-1), provided somewhat different shapes for modes 9 and 10. This formulation involves a slight modification of the Jacobian matrix which can lead to a stiffness reduction for certain modes (according to Borrvall [2009]). However, modes 9 and 10 are not those modes Borrvall offered concerns for in stiffness reduction.

Eigenvalue Results:

From the *eigout* file, generated by the *CONTROL_IMPLICIT_EIGENVALUE keyword:

Constant stress solid (elform=1):

```
Simply Supported Square Plate: Out-of-Plane Vibration
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1  -3.201421E-10   1.789252E-05   2.847682E-06   3.511628E+05
           2   1.062290E-09   3.259279E-05   5.187303E-06   1.927784E+05
           3   2.881279E-09   5.367755E-05   8.543047E-06   1.170543E+05
           4   7.657028E+04   2.767133E+02   4.404030E+01   2.270648E-02
           5   4.475080E+05   6.689604E+02   1.064684E+02   9.392462E-03
           6   4.475080E+05   6.689604E+02   1.064684E+02   9.392462E-03
           7   9.548827E+05   9.771810E+02   1.555232E+02   6.429910E-03
           8   1.479411E+06   1.216310E+03   1.935818E+02   5.165775E-03
           9   1.581263E+06   1.257483E+03   2.001346E+02   4.996637E-03
          10   1.581263E+06   1.257483E+03   2.001346E+02   4.996637E-03
```

Fully integrated S/R solid (elform=2)

```
Simply Supported Square Plate: Out-of-Plane Vibration
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1  -6.009941E-09   7.752381E-05   1.233830E-05   8.104846E+04
           2   9.895302E-10   3.145680E-05   5.006505E-06   1.997401E+05
           3   5.558832E-09   7.455757E-05   1.186621E-05   8.427293E+04
           4   9.289190E+04   3.047817E+02   4.850752E+01   2.061536E-02
           5   5.721748E+05   7.564224E+02   1.203884E+02   8.306451E-03
           6   5.721748E+05   7.564224E+02   1.203884E+02   8.306451E-03
           7   1.135577E+06   1.065635E+03   1.696010E+02   5.896191E-03
           8   1.478563E+06   1.215962E+03   1.935263E+02   5.167257E-03
           9   1.582107E+06   1.257818E+03   2.001880E+02   4.995304E-03
          10   1.582107E+06   1.257818E+03   2.001880E+02   4.995304E-03
```

Fully integrated S/R solid - for poor aspect ratio (eff) - (elform=-1)

```
Simply Supported Square Plate: Out-of-Plane Vibration
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1   1.043372E-08   1.021456E-04   1.625698E-05   6.151205E+04
           2   1.335866E-08   1.155797E-04   1.839507E-05   5.436238E+04
           3   1.573062E-08   1.254218E-04   1.996149E-05   5.009645E+04
           4   8.154482E+04   2.855605E+02   4.544837E+01   2.200299E-02
           5   4.529502E+05   6.730158E+02   1.071138E+02   9.335866E-03
           6   4.529502E+05   6.730158E+02   1.071138E+02   9.335866E-03
           7   9.993359E+05   9.996679E+02   1.591021E+02   6.285273E-03
           8   1.478432E+06   1.215908E+03   1.935178E+02   5.167484E-03
           9   1.560100E+06   1.249040E+03   1.987908E+02   5.030413E-03
          10   1.561382E+06   1.249553E+03   1.988725E+02   5.028347E-03
```

Fully integrated S/R solid - for poor aspect ratio (acc) - (elform=-2)

```
Simply Supported Square Plate: Out-of-Plane Vibration
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1   2.211891E-09   4.703075E-05   7.485176E-06   1.335974E+05
           2   7.014023E-09   8.374977E-05   1.332919E-05   7.502332E+04
           3   1.121953E-08   1.059223E-04   1.685805E-05   5.931883E+04
           4   8.428602E+04   2.903205E+02   4.620595E+01   2.164223E-02
           5   4.713255E+05   6.865315E+02   1.092649E+02   9.152072E-03
           6   4.713255E+05   6.865315E+02   1.092649E+02   9.152072E-03
           7   1.061078E+06   1.030087E+03   1.639434E+02   6.099667E-03
           8   1.478558E+06   1.215960E+03   1.935260E+02   5.167264E-03
           9   1.581923E+06   1.257745E+03   2.001764E+02   4.995595E-03
          10   1.581923E+06   1.257745E+03   2.001764E+02   4.995595E-03
```

Fully integrated quadratic 8 node element with nodal rotations (elform=3)

```
Simply Supported Square Plate
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1   1.877197E-08   1.370108E-04   2.180595E-05   4.585904E+04
           2   2.239540E-08   1.496509E-04   2.381768E-05   4.198561E+04
           3   2.614979E-08   1.617090E-04   2.573678E-05   3.885490E+04
           4   7.425572E+04   2.724990E+02   4.336957E+01   2.305764E-02
           5   4.327897E+05   6.578675E+02   1.047029E+02   9.550837E-03
           6   4.327897E+05   6.578675E+02   1.047029E+02   9.550837E-03
           7   9.345968E+05   9.667455E+02   1.538623E+02   6.499317E-03
           8   1.473997E+06   1.214083E+03   1.932273E+02   5.175253E-03
           9   1.524111E+06   1.234549E+03   1.964846E+02   5.089458E-03
          10   1.536475E+06   1.239546E+03   1.972799E+02   5.068940E-03
```

Mode Shapes:

The constant stress solid (elform=1) mode shapes are shown in Figure 12.2, the fully integrated S/R solid (elform=2) in Figure 12.3, the fully integrated quadratic 8 node element with nodal rotations (elform=3) in Figure 12.4, the fully integrated S/R solid (the so-called efficient formulation choice) intended to address poor aspect ratios (elform=-1) in Figure 12.5, and the fully integrated S/R solid (the so-called accurate formulation choice) intended to address poor aspect ratios (elform=-2) in Figure 12.6.

The first three modes are not shown (rigid body translations). Modes 4 through 10 are shown for the selected results. Modes 5 and 6 are identical for all element formulations; modes 9 and 10 are also identical for all, with the exception of the fully integrated quadratic 8 node element with nodal rotations (elform=3) formulation.

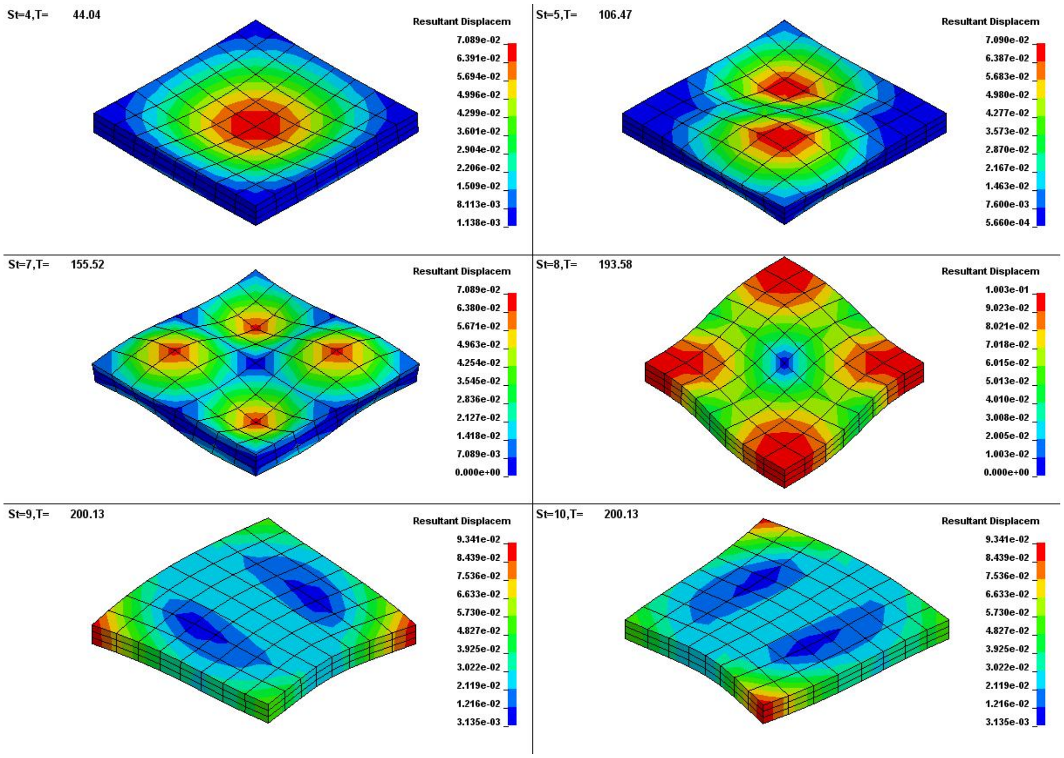

Figure 12.2 - Mode shapes for constant stress solid (elform=1).

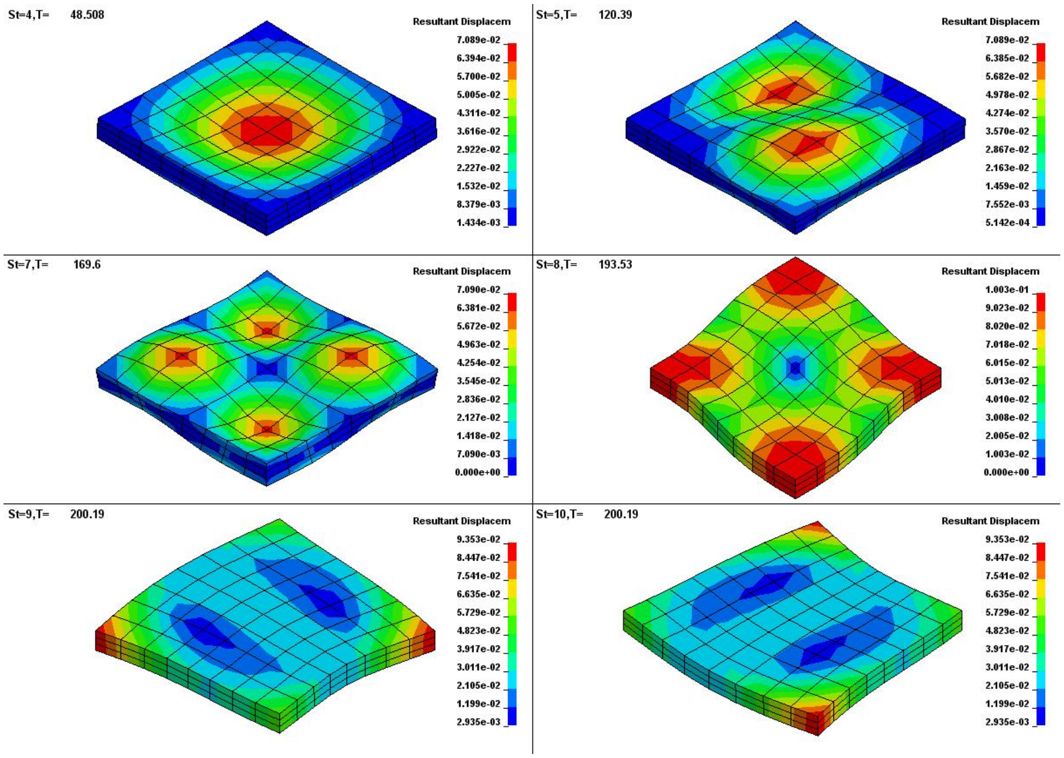

Figure 12.3 - Mode shapes for fully integrated S/R solid (elform=2).

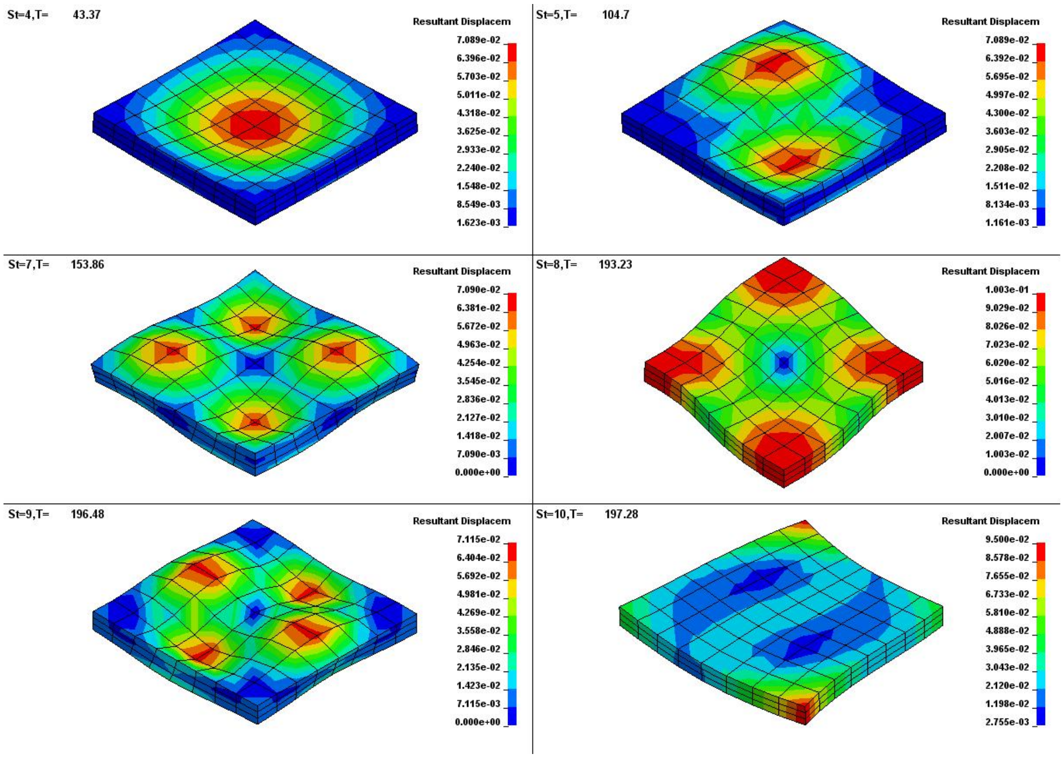

Figure 12.4 - Mode shapes for fully integrated quadratic 8 node element with nodal rotations (elform=3).

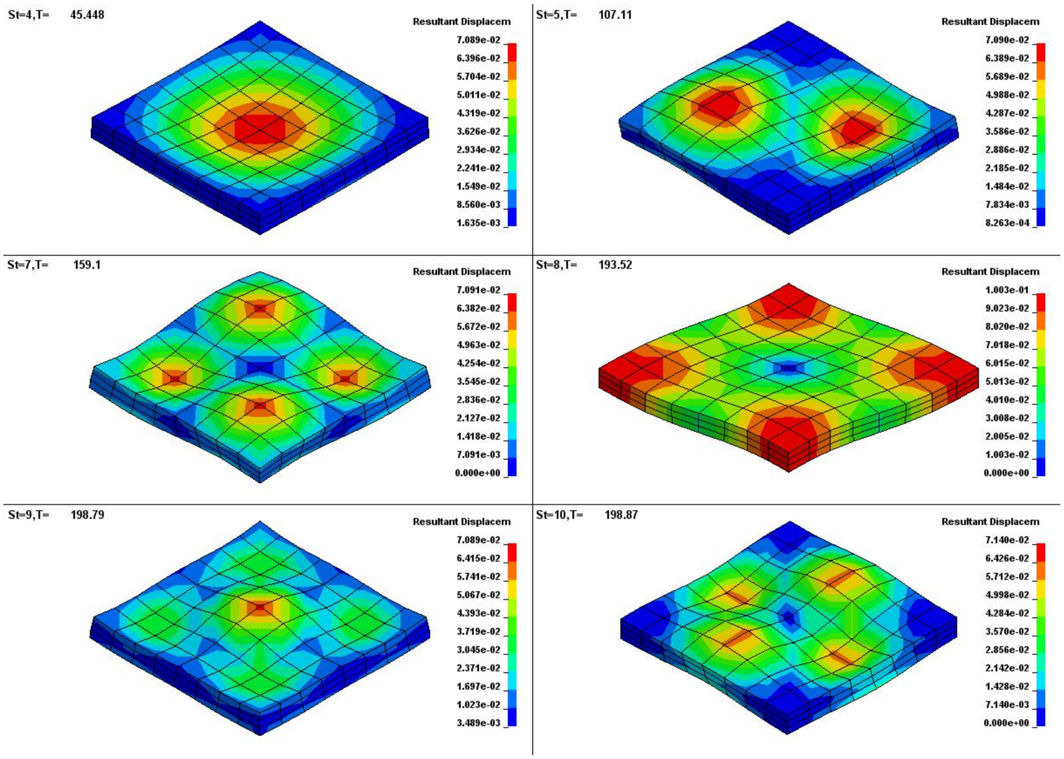

Figure 12.5 - Mode shapes for fully integrated S/R solid (elform=-1).

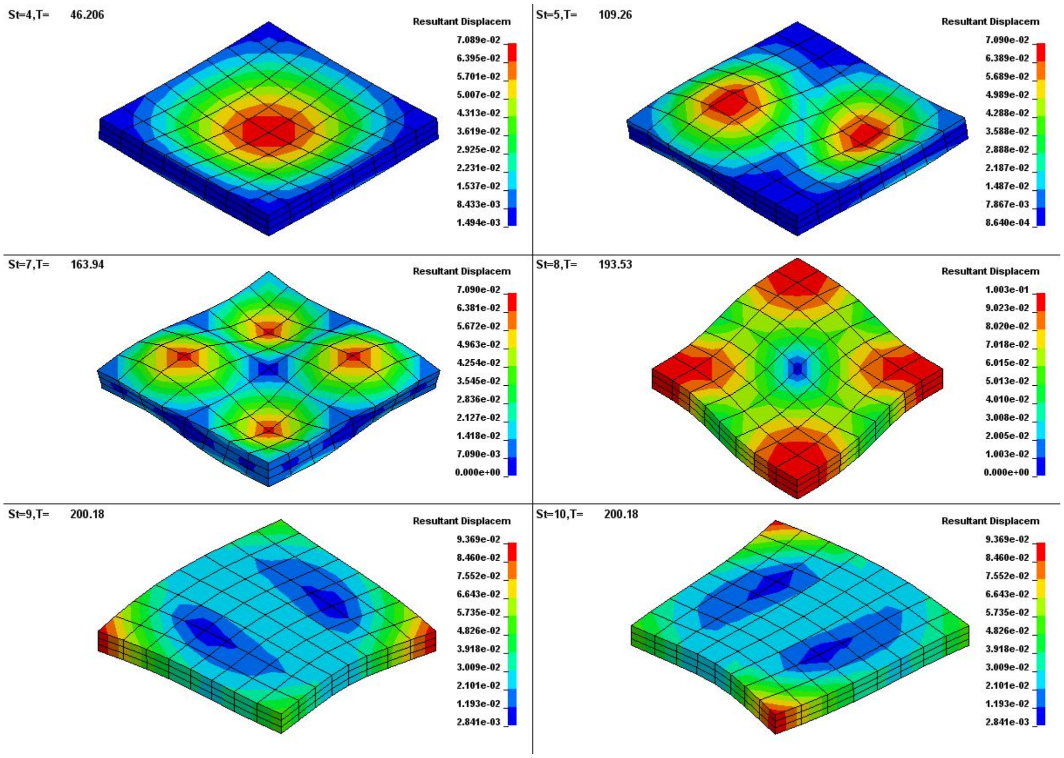

Figure 12.6 - Mode shapes fully integrated S/R solid (elform=-2).

Input deck:

```
*KEYWORD
*TITLE
Simply Supported Square Plate: Out-of-Plane Vibration (solid mesh)
*CONTROL_IMPLICIT_EIGENVALUE
$#    neig    center     lflag    lftend     rflag    rhtend    eigmth    shfscl
        10       0.0         0       0.0         0       0.0         0       0.0
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1       0.0
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  1.000000         0       0.0       0.0       0.0
*DATABASE_BINARY_D3PLOT
$# dt/cycl   lcdt/nr      beam     npltc    psetid
  1.000000
*ELEMENT_SOLID
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1      37      41       5       2      38      42       6

     192       1     283     319     323     287     284     320     324     288
*NODE
$#   nid               x               y               z      tc      rc
       1             0.0             0.0             0.0       3

     324     10.00000000     10.00000000      1.00000000
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         0         0         1
*PART
$# title
material type # 1  (Elastic)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
```

```
         1         1         1         0         1
*SECTION_SOLID
$#   secid    elform       aet
         1         1         1
$         1         2         1
$         1        -1         1
$         1        -2         1
$         1         3         1
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1  8000.0002.0000e+11  0.300000       0.0       0.0       0.0
*HOURGLASS
$#    hgid       ihq        qm       ibq        q1        q2        qb        qw
         1         6       1.0         0       0.0       0.0       0.0       0.0
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1        37        73       109       145       181       217       253
       289       293       297       301       305       309       313       317
       321       285       249       213       177       141       105        69
        33        29        25        21        17        13         9         5
*END
```

Notes:

13. Simply Supported Square Plate: Out-of-Plane Vibration (thick shell mesh)

Keywords:

```
*CONTROL_IMPLICIT_EIGENVALUE
*CONTROL_IMPLICIT_GENERAL
```

Description:

Determine the first 10 natural frequencies of a solid simply-supported plate of thickness t=1.0 *m*. Each side of the plate measure 10.0 *m*. The plate is meshed with solid hexahedra element with an 8 x 8 x 3 density. On the lower surface, outer boundary nodes, $U_Z = 0$.

The finite element model is shown in Figure 13.1.

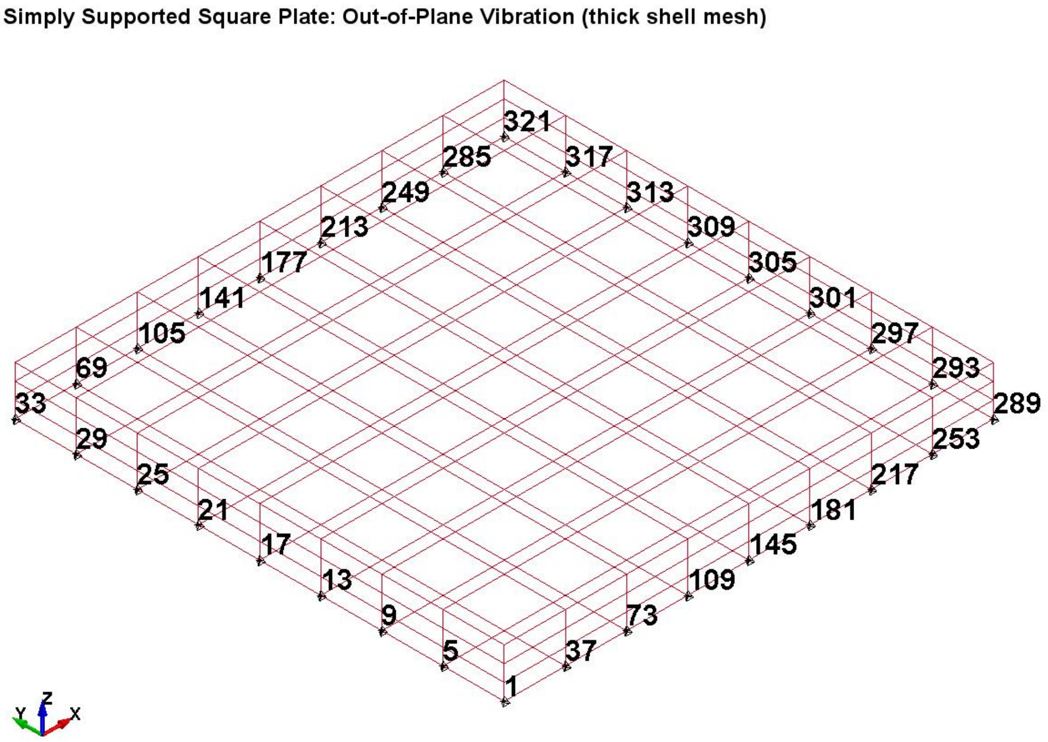

Figure 13.1 – Finite element model with simply supported boundary conditions on lower surface, outer nodes.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Modal	-	Linear	Linear	-	Implicit	Block Shift and Inverted Lanczos

Units:

kg, m, s, N, Pa, N-m (kilogram, meter, second, Newton, Pascal, Newton-meter)

Dimensional Data:

Rectangular dimensions of square plate: 10.0 *m* x 10.0 *m* x 1.00 *m*.

Material Data:

Mass Density	$\rho = 8.00 \times 10^3 \ kg / m^3$
Young's Modulus	$E = 2.00 \times 10^{11} \ Pa$
Poisson's Ratio	$\nu = 0.3$

Element Types:

S/R 2x2 IPI thick shell (elform=2)
Assumed strain 2x2 IPI thick shell (elform=3)
Assumed strain RI thick shell (elform=5)

Material Models:

*MAT_001 or *MAT_ELASTIC

Results Comparison:

LS-DYNA natural frequencies, first 10 (frequency in *Hertz*), and mode shapes (4 through 10) are compared with *NAFEMS Free Vibration Benchmarks***,** Test FV52.

Mode(s)	***NAFEMS*** **FV52 (*Hz*)**	**elform=2 (*Hz*)**	**elform=3 (*Hz*)**	**elform=5 (*Hz*)**
1, 2, 3	rigid body	rigid body	rigid body	rigid body
4	45.897	43.480	42.556	44.656
5, 6	109.440	105.363	102.983	107.290
7	167.890	152.764	150.187	157.397
8	193.590	185.301	193.378	193.583
9	206.190	193.708	197.247	200.136
10	206.190	200.997	197.295	200.136

The assumed strain 2x2 IPI thick shell (elform=3) and the assumed strain RI thick shell (elform=5) use a full three-dimensional stress update rather than the two-dimensional plane stress update of the one point reduced integration (elform=1) and the selectively reduced 2x2 IPI thick shell (elform=2).

The selectively reduced 2x2 IPI thick shell (elform=2), the assumed strain 2x2 IPI thick shell (elform=3), and the assumed strain RI thick shell (elform=5) all provide similar frequency results for this analysis.

The selectively reduced 2x2 IPI thick shell (elform=2) formulation appears to have identified (added) an unexpected result for mode 8 (an anomaly, a low energy warping mode which is believed will not cause solution troubles) due to the calculation of the out-of-plane shear stiffness terms. Code inspection indicated that the out-of-plane shear stress and stiffness is calculated at the mid-point rather than the 2x2 integration points in order to prevent shear locking in bending. Modes 9 and 10 (elform=2) are however, similar to modes 8 and 9, respectively, for the assumed strain RI thick shell (elform=5) formulation.

The assumed strain 2x2 IPI thick shell (elform=3) formulation appears to have identified two different modes and frequencies for modes 9 and 10. This is possibly due to the employment of a corotational system that rotates with the elements, which suppresses the element locking.

Eigenvalue Results:

From the *eigout* file, generated by the *CONTROL_IMPLICIT_EIGENVALUE keyword:

S/R 2x2 IPI thick shell (elform=2)

```
Simply Supported Square Plate: Out-of-Plane Vibration (thick shell mesh)
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1  -8.469215E-09   9.202834E-05   1.464676E-05   6.827446E+04
           2  -3.463356E-09   5.885028E-05   9.366313E-06   1.067656E+05
           3  -1.746230E-09   4.178791E-05   6.650753E-06   1.503589E+05
           4   7.449258E+04   2.729333E+02   4.343868E+01   2.302096E-02
           5   4.377512E+05   6.616277E+02   1.053013E+02   9.496558E-03
           6   4.377512E+05   6.616277E+02   1.053013E+02   9.496558E-03
           7   9.193747E+05   9.588403E+02   1.526042E+02   6.552901E-03
           8   1.255008E+06   1.120271E+03   1.782967E+02   5.608628E-03
           9   1.481334E+06   1.217101E+03   1.937076E+02   5.162420E-03
          10   1.594925E+06   1.262903E+03   2.009973E+02   4.975191E-03
```

Assumed strain 2x2 IPI thick shell (elform=3)

```
Simply Supported Square Plate: Out-of-Plane Vibration (thick shell mesh)
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1  -8.338247E-09   9.131400E-05   1.453308E-05   6.880856E+04
           2  -4.802132E-09   6.929742E-05   1.102903E-05   9.066983E+04
           3  -3.012246E-09   5.488394E-05   8.735050E-06   1.144813E+05
           4   7.149598E+04   2.673873E+02   4.255601E+01   2.349844E-02
           5   4.186854E+05   6.470590E+02   1.029826E+02   9.710374E-03
           6   4.186854E+05   6.470590E+02   1.029826E+02   9.710374E-03
           7   8.904817E+05   9.436534E+02   1.501871E+02   6.658361E-03
           8   1.476299E+06   1.215030E+03   1.933781E+02   5.171217E-03
           9   1.535967E+06   1.239341E+03   1.972473E+02   5.069778E-03
          10   1.536702E+06   1.239638E+03   1.972945E+02   5.068565E-03
```

Assumed strain RI thick shell (elform=5)

```
Simply Supported Square Plate: Out-of-Plane Vibration (thick shell mesh)
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1   5.966285E-10   2.442598E-05   3.887516E-06   2.572337E+05
           2   2.881279E-09   5.367755E-05   8.543047E-06   1.170543E+05
           3   5.762558E-09   7.591152E-05   1.208169E-05   8.276986E+04
           4   7.872674E+04   2.805829E+02   4.465615E+01   2.239333E-02
           5   4.544420E+05   6.741231E+02   1.072900E+02   9.320531E-03
           6   4.544420E+05   6.741231E+02   1.072900E+02   9.320531E-03
           7   9.780258E+05   9.889519E+02   1.573966E+02   6.353378E-03
           8   1.479432E+06   1.216319E+03   1.935832E+02   5.165738E-03
           9   1.581277E+06   1.257488E+03   2.001355E+02   4.996615E-03
          10   1.581277E+06   1.257488E+03   2.001355E+02   4.996615E-03
```

Mode Shapes:

The first three modes are not shown (rigid body translations). Modes 4 through 10 are shown for the selected results. Modes 5 and 6 are identical for all element formulations. Modes 9 and 10 offer three different sets of results, depending on element formulation; (1) for selectively reduced 2x2 IPI thick shell (elform=2), there is a distinct difference in natural frequencies (Figure 13.2), (2) for assumed strain 2x2 IPI thick shell (elform=3), there is a very slight difference (Figure 13.3), and (3) for assumed strain RI thick shell (elform=5), the results are identical (Figure 13.4).

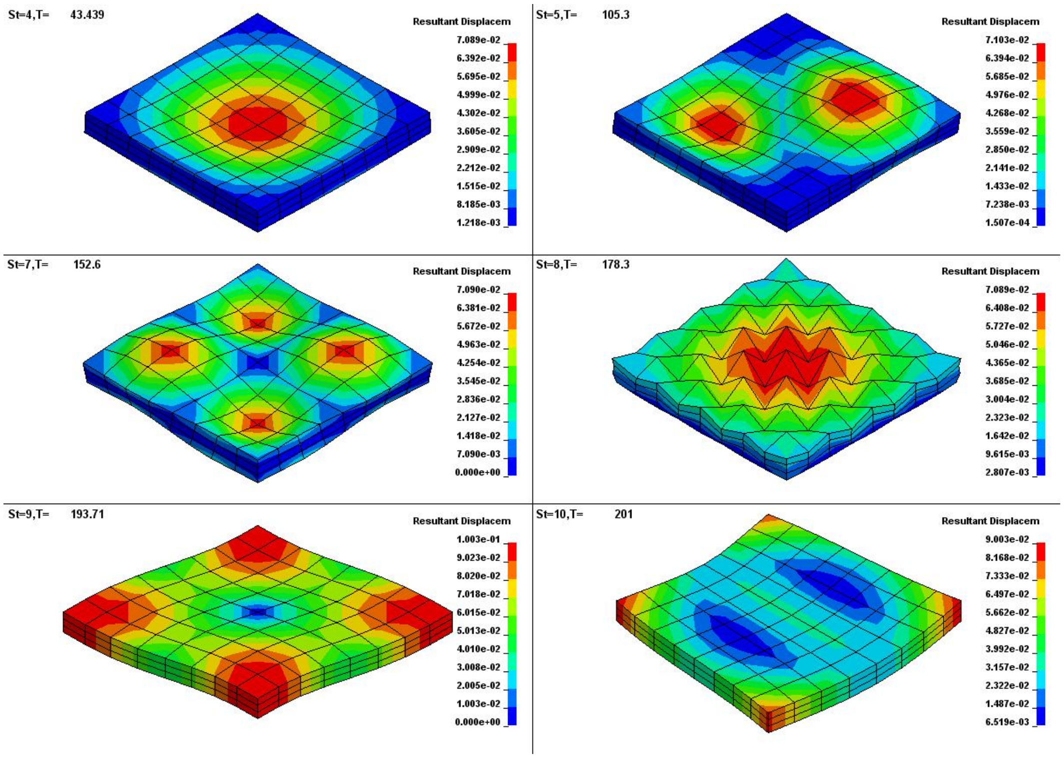

Figure 13.2 - Mode shapes for S/R 2x2 IPI thick shell (elform=2).

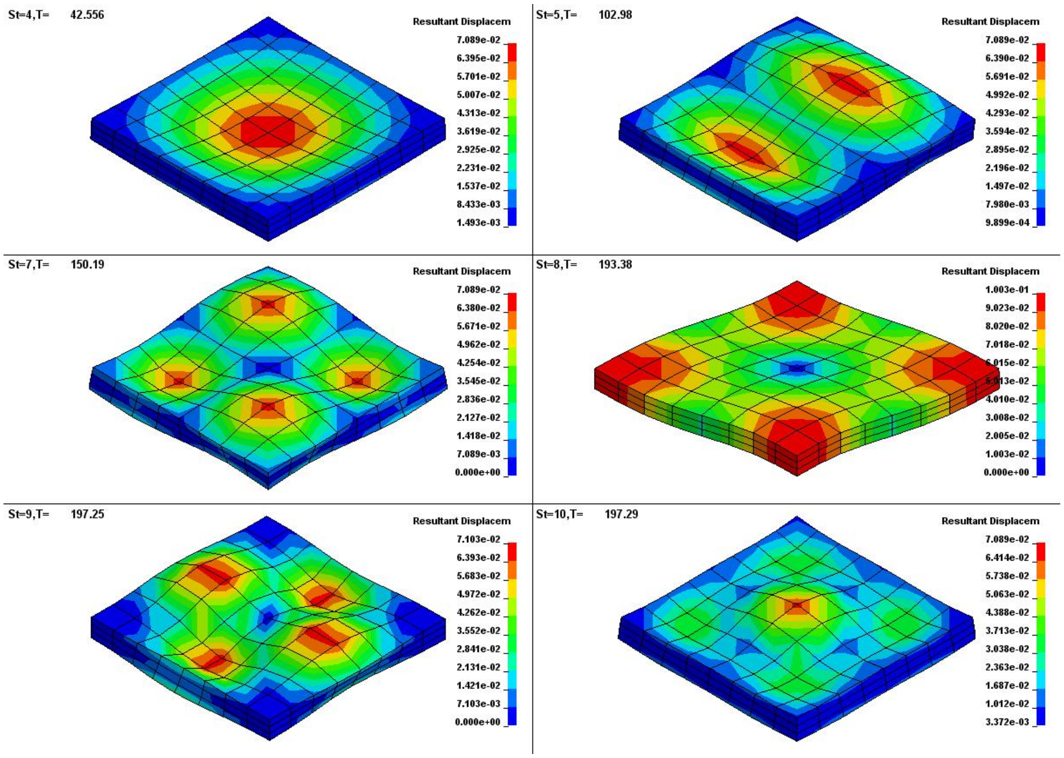

Figure 13.3 - Mode shapes for assumed strain 2x2 IPI thick shell (elform=3).

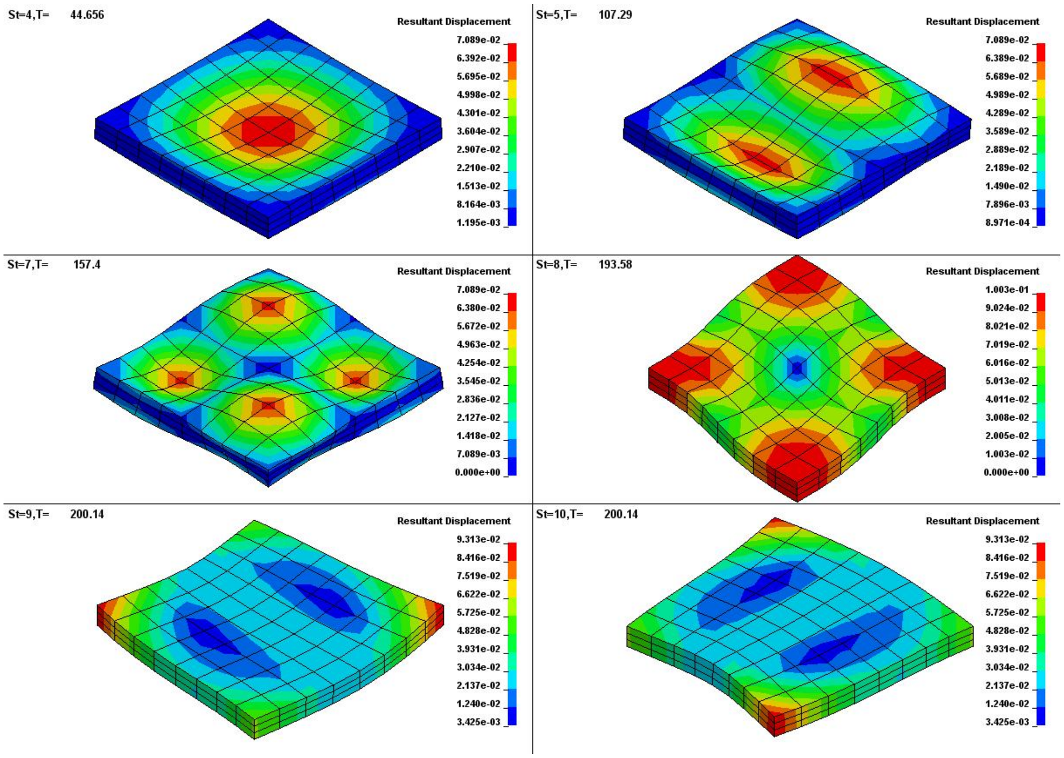

Figure 13.4 - Mode shapes for assumed strain RI thick shell (elform=5).

Input deck:

```
*KEYWORD
*TITLE
Simply Supported Square Plate: Out-of-Plane Vibration (thick shell mesh)
*CONTROL_IMPLICIT_EIGENVALUE
$#    neig    center     lflag    lftend     rflag    rhtend    eigmth    shfscl
        10       0.0         0       0.0         0       0.0         0       0.0
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1       0.0
*CONTROL_SHELL
$#  wrpang     esort     irnxx    istupd    theory       bwc     miter      proj
  20.00000         0         0         0         2         2         1
$# rotascl    intgrd    lamsht    cstyp6    tshell    nfail1    nfail4
       0.0         1
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  1.000000         0       0.0       0.0       0.0
*DATABASE_BINARY_D3PLOT
$# dt/cycl   lcdt/nr      beam     npltc    psetid
  1.000000
*ELEMENT_TSHELL
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1      37      41       5       2      38      42       6

     192       1     283     319     323     287     284     320     324     288
*NODE
$#   nid               x               y               z      tc      rc
       1             0.0             0.0             0.0       3

     324     10.00000000     10.00000000      1.00000000
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         0         0         1
*PART
$# title
material type # 1  (Elastic)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1         0         1
*SECTION_TSHELL
$#   secid    elform      shrf       nip     propt   qr/irid     icomp    tshear
         1         2       0.0         5         0       0.0
$        1         3       0.0         5         0       0.0
$        1         5       0.0         5         0       0.0
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1  8000.0002.0000e+11  0.300000       0.0       0.0       0.0
*HOURGLASS
$#    hgid       ihq        qm       ibq        q1        q2        qb        qw
         1         4       0.1         0       0.0       0.0       0.0       0.0
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1        37        73       109       145       181       217       253
       289       293       297       301       305       309       313       317
       321       285       249       213       177       141       105        69
        33        29        25        21        17        13         9         5
*END
```

Notes:

1. The assumed strain 2x2 IPI thick shell (elform=3) and the assumed strain RI thick shell (elform=5) are distortion sensitive and should not be used in situations where the elements are badly shaped.

2. With the one point reduced integration (elform=1) and the selectively reduced 2x2 IPI thick shell (elform=2), a single element through the thickness will capture bending response, but with the assumed strain 2x2 IPI thick shell (elform=3) and the assumed strain RI thick shell (elform=5), two elements are recommended to avoid excessive softness.

3. Only the selectively reduced 2x2 IPI thick shell (elform=2), the assumed strain 2x2 IPI thick shell (elform=3), and the assumed strain RI thick shell (elform=5) are available for implicit applications. If one point reduced integration (elform=1) is specified in an implicit analysis, it is internally switched to selectively reduced 2x2 IPI thick shell (elform=2).

14. Simply Supported Square Plate: Transient Forced Vibration (solid mesh)

Keywords:

```
*CONTROL_IMPLICIT_AUTO
*CONTROL_IMPLICIT_DYNAMICS
*CONTROL_IMPLICIT_GENERAL
*CONTROL_IMPLICIT_SOLVER
*CONTROL_IMPLICIT_SOLUTION
```

Description:

A plate is subjected to a suddenly applied pressure on its top. A transient analysis is performed in order to obtain the response of the plate. Damping is present. On the lower surface, outer boundary nodes, $U_Z = 0$.

The finite element model is shown in Figure 14.1.

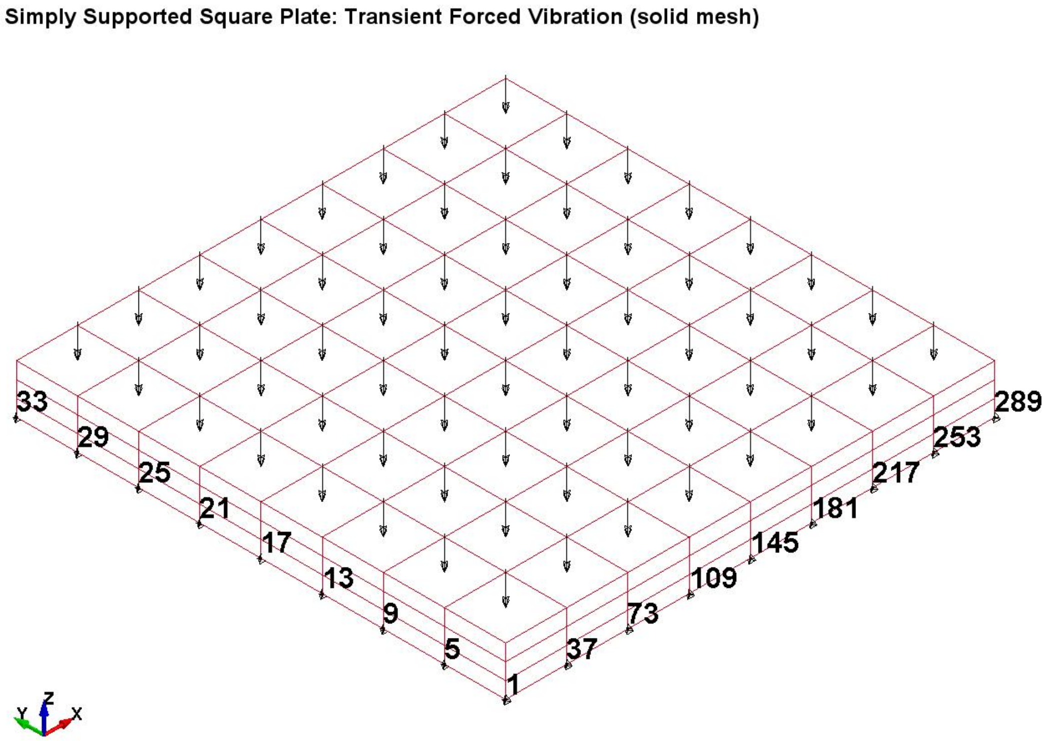

Figure 14.1 – Finite element model with applied pressure on upper surface and simply supported boundary conditions on lower surface, outer nodes.

Analysis Summary:

Dim.	**Type**	**Load**	**Material**	**Geometry**	**Contact**	**Solver**	**Solution Method**
3D	Dynamic	Pressure Damping	Linear	Linear	-	Implicit	1 - Linear

Units:

kg, m, s, N, Pa, N-m (kilogram, meter, second, Newton, Pascal, Newton-meter)

Dimensional Data:

Rectangular dimensions of square plate: 10.0 *m* x 10.0 *m* x 1.00 *m*.

Material Data:

Mass Density	$\rho = 8.00 \times 10^3 \ kg/m^3$
Young's Modulus	$E = 2.00 \times 10^{11} \ Pa$
Poisson's Ratio	$\nu = 0.3$
Damping ratio	$\zeta = 2\%$

Load:

Pressure	$P = 1.0 \times 10^6 \ Pa$

Element Types:

Constant stress solid (elform=1)
Fully integrated S/R solid (elform=2)
Fully integrated S/R solid - for poor aspect ratio (eff) - (elform=-1)
Fully integrated S/R solid - for poor aspect ratio (acc) - (elform=-2)
Fully integrated quadratic 8 node element with nodal rotations (elform=3)

Material Models:

*MAT_001 or *MAT_ELASTIC

Damping:

The damping factor d is easily found from the natural frequency of the system:

$$d_i = \zeta 2\omega_i$$

As the excited mode is the first, corresponding to $f_1 = 45.897\ Hz$ ($\omega_1 = 288.380\ rad/s$) (that from NAFEMS Benchmark Test FV52), we choose the damping factor relative to the first frequency:

$$d_1 = 11.535\ Hz$$

Results Comparison:

LS-DYNA X-direction bending stress, σ_{xx}**,** at (Node 161) on bottom surface plus its Z-displacement, U_Z, are compared with *NAFEMS Selected Benchmarks for Forced Vibration***,** Test 21T.

Reference Condition - Center (Node 161)	**Peak Bending Stress σ_{xx} (*Pa*)**	**Peak U_Z (*m*)**	**Steady-State U_Z (*m*)**
NAFEMS Benchmark Test 21T	6.211×10^{7}	-4.524×10^{-3}	-2.333×10^{-3}
Constant stress solid (elform=1)	4.638×10^{7}	-5.438×10^{-3}	-2.778×10^{-3}
Fully integrated S/R solid (elform=2)	3.732×10^{7}	-3.925×10^{-3}	-2.019×10^{-3}
Fully integrated S/R solid (elform=-1)	4.611×10^{7}	-4.435×10^{-3}	-2.242×10^{-3}
Fully integrated S/R solid (elform=-2)	4.221×10^{7}	-4.365×10^{-3}	-2.203×10^{-3}
Fully integrated quadratic element with nodal rotations (elform=3)	$x.xxx\times10^{7}$	$-x.xxx\times10^{-3}$	$-x.xxx\times10^{-3}$

The constant stress solid (elform=1) result of $4.638\times10^{7}\ Pa$ is an element centroid value.

These nodal displacement results were generated by *DATABASE_NODOUT keyword while the axial stress (nodal) results were generated by *DATABASE_ELOUT (*elout* file) and *DATABASE_EXTENT_BINARY (*eloutdet* file provides detailed element output at integration points and connectivity nodes) keyword entries.

You can set intout=stress or intout=all (*DATABASE_EXTENT_BINARY) and have stresses output for all the integration points to a file called *eloutdet*

(*DATABASE_ELOUT governs the output interval and *DATABASE_HISTORY_ SOLID governs which elements are output). Setting nodout=stress or nodout=all in *DATABASE_EXTENT_BINARY will write the extrapolated nodal stresses to *eloutdet*.

LS-DYNA stress and strain output corresponds to integration point locations. Stress at a node is an artifact of the post-processor and represents an average of the surrounding integration point stresses (the value will likely be different with different post-processors).

For this coarse mesh, the one-point quadrature (low order) constant stress solid (elform=1) element formulation (the LS-DYNA default) provides a less stiff, stress and displacement comparison. Refinement of the mesh should provide a better comparison.

The higher order, fully integrated selectively reduced solid (elform=2) provides a comparatively stiff result, both in stress and displacement, probably due to the coarse mesh.

The aspect ratio of these elements is 3.75 (ratio of side to depth length). Available options are the higher order, fully integrated S/R solid (both the so-called efficient and the so-called accurate formulation choices) intended to address poor aspect ratios (elform=-1 and -2, respectively). These formulations provide a good comparison of displacements (peak and steady-state) for this coarse mesh. Unfortunately, the stress comparison is not very good; it is not understood why at this time.

The fully integrated S/R solid (the so-called efficient formulation choice) intended to address poor aspect ratios (elform=-1), can provide a slightly less stiff solution than the so-called accurate formulation choice (elform=2). This formulation (elform=-1) involves a slight modification of the Jacobian matrix which can lead to a stiffness reduction for certain modes, in particular the out-of-plane hourglass mode (according to Borrvall [2009]).

The higher order, fully integrated quadratic 8 node element with nodal rotations (elform=3) formulation provides a (????) results. Waiting for LSTC LS-DYNA code fix to remark on this.

For the fully integrated S/R solid accurate formulation (elform=-2), the contour plot of the X-direction bending stress (Figure 14.2) and the Z-displacement (Figure 14.3) were obtained from the *d3plot* file at peak displacement time which were generated by the *DATABASE_BINARY_D3PLOT keyword.

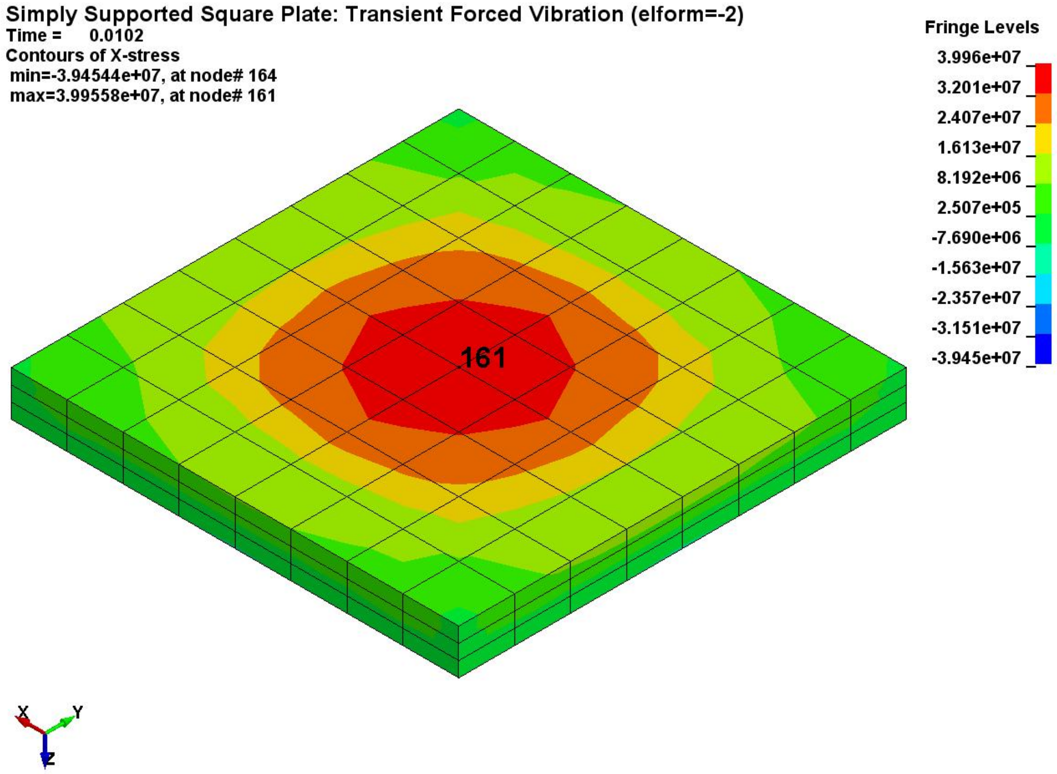

Figure 14.2 – Contour plot of the X-stress (elform=-2) at peak displacement time.

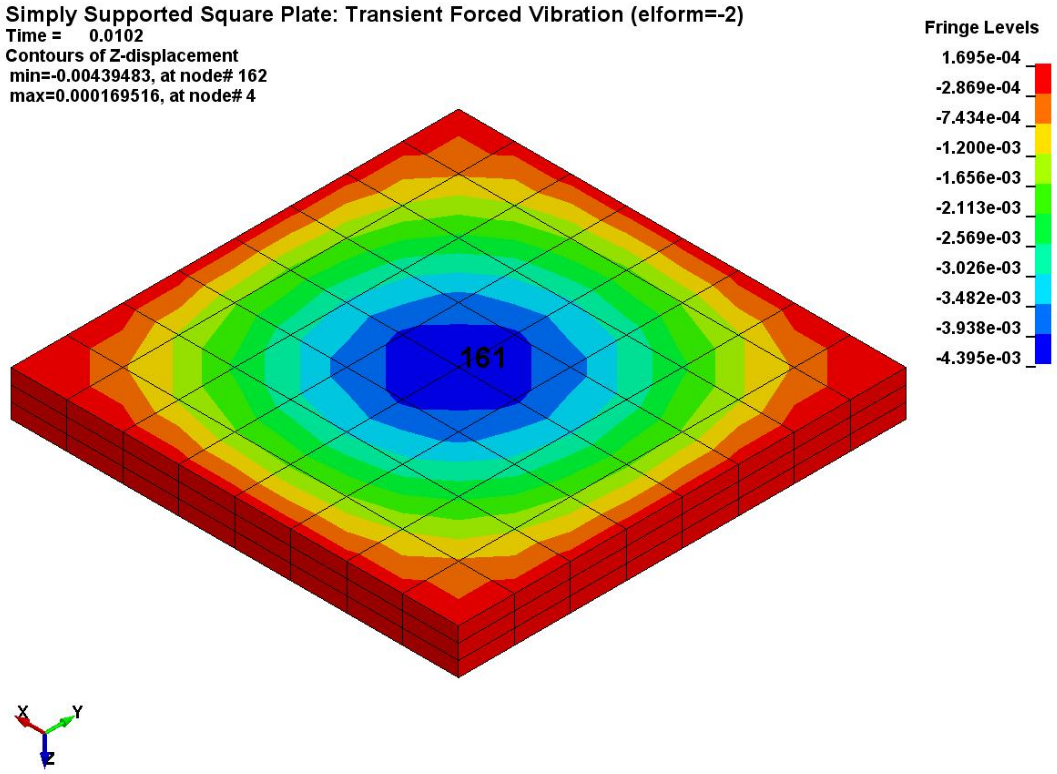

Figure 14.3 - Contour plot of Z-displacement (elform=-2) at peak displacement time.

Input deck:

```
*KEYWORD
*TITLE
Simply Supported Square Plate: Transient Forced Vibration (solid mesh)
*CONTROL_IMPLICIT_AUTO
$#   iauto    iteopt    itewin     dtmin     dtmax     dtexp     kfail    kcycle
         0        11         5       0.0       0.0
*CONTROL_IMPLICIT_DYNAMICS
$#   imass     gamma      beta
         1  0.500000  0.250000
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form    zero_v
         1  0.000100         2         1         2
*CONTROL_IMPLICIT_SOLVER
$#  lsolvr    lprint     negev     order      drcm    drcprm   autospc   autotol
         4         2         2         0         1       0.0         1       0.0
$#  lcpack
         2
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
         2        11        15    0.0010    0.0100  1.00e+10  0.900000  1.000000
$#   dnorm    diverg     istif   nlprint
         2         1         1         2
$#  arcctl    arcdir    arclen    arcmth    arcdmp
         0         1       0.0         1         2
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  0.020000         0       0.0       0.0       0.0
*DATABASE_ELOUT
$#      dt    binary      lcur     ioopt
1.0000e-06         1
*DATABASE_NODFOR
$#      dt    binary      lcur     ioopt
1.0000e-06         1
*DATABASE_NODOUT
$#      dt    binary      lcur     ioopt
1.0000e-06         1
*DATABASE_BINARY_D3PLOT
$# dt/cycl
  0.001000
*DATABASE_EXTENT_BINARY
$#   neiph     neips    maxint    strflg    sigflg    epsflg     rtflg    engflg

$#  cmpflg    ieverp    beamip     dcomp      shge     stssz    n3thdt   ialemat

$# nintsld   pkp_sen      sclp     hydro     msscl     therm    intout    nodout
         8                 1.0                                  stress    stress
*DATABASE_HISTORY_SOLID
$#     id1       id2       id3       id4       id5       id6       id7       id8
        28        29        36        37
*DATABASE_NODAL_FORCE_GROUP
$#    nsid       cid
       164
*DATABASE_HISTORY_NODE
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       161       164
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4    solver
       164       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       164
*DAMPING_GLOBAL
$#    lcid    valdmp       stx       sty       stz       srx       sry       srz
         0  11.53500       0.0       0.0       0.0       0.0       0.0       0.0
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo    dattyp
         1         0       0.0       0.0       0.0       0.0
$#                a1                  o1
                 0.0     1.0000000e+06
```

```
          0.10000000       1.0000000e+06
*ELEMENT_SOLID
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1      37      41       5       2      38      42       6

     192       1     283     319     323     287     284     320     324     288
*NODE
$#   nid               x               y               z      tc      rc
       1             0.0             0.0             0.0       3

     324     10.00000000     10.00000000      1.00000000
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         0         0         1
*PART
$# title
material type # 1  (Elastic)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_SOLID
$#   secid    elform       aet
         1         1         1
$         1         2         1
$         1        -1         1
$         1        -2         1
$         1         3         1
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1  8000.0002.0000e+11  0.300000       0.0       0.0       0.0
*LOAD_SEGMENT
$#    lcid        sf        at        n1        n2        n3        n4
         1  1.000000       0.0         4        40        44         8

         1  1.000000       0.0       284       320       324       288
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1        37        73       109       145       181       217       253
       289       293       297       301       305       309       313       317
       321       285       249       213       177       141       105        69
        33        29        25        21        17        13         9         5
*END
```

Notes:

1. One should remember that the constant stress solid (elform=1), the fully integrated S/R solid (elform=2), and the fully integrated S/R solid (both the so-called efficient and the so-called accurate formulation choices) intended to address poor aspect ratios (elform=-1 and -2, respectively) were originally developed for performing highly nonlinear, dynamic deformation simulations.

15. Simply Supported Square Plate: Transient Forced Vibration (thick shell mesh)

Keywords:

```
*CONTROL_IMPLICIT_AUTO
*CONTROL_IMPLICIT_DYNAMICS
*CONTROL_IMPLICIT_GENERAL
*CONTROL_IMPLICIT_SOLVER
*CONTROL_IMPLICIT_SOLUTION
```

Description:

A plate is subjected to a suddenly applied pressure on its top. A transient analysis is performed in order to obtain the response of the plate. Damping is present. On the lower surface, outer boundary nodes, $U_Z = 0$.

The finite element model is shown in Figure 15.1.

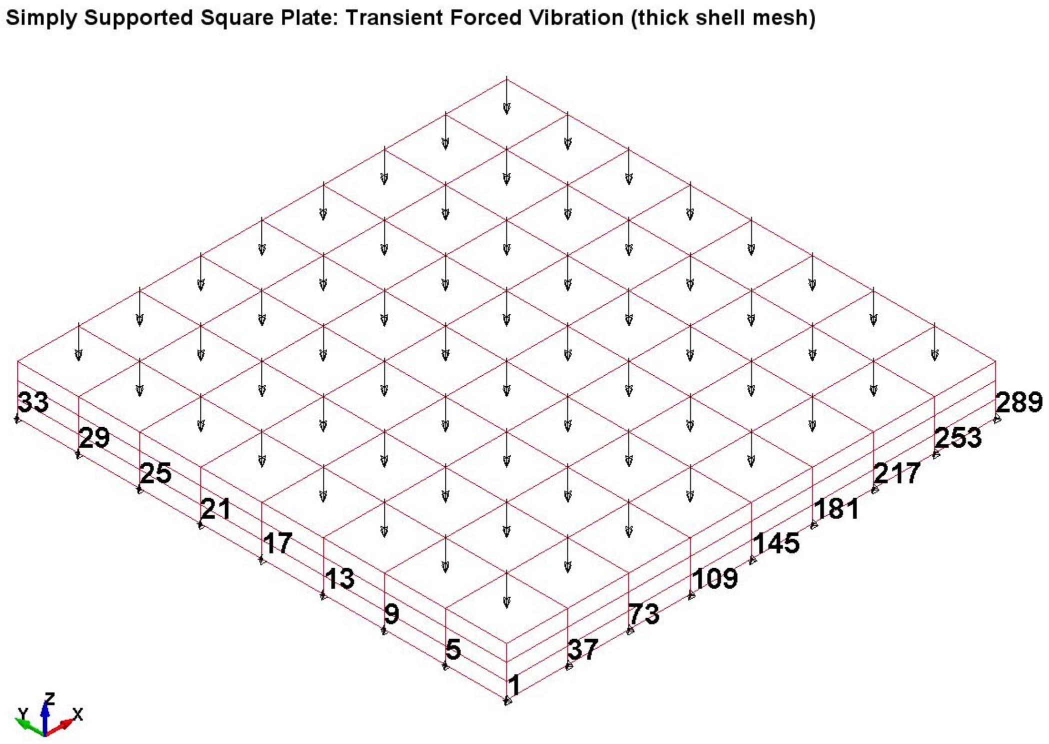

Figure 15.1 – Finite element model with applied pressure on upper surface and simply supported boundary conditions on lower surface, outer nodes.

Analysis Summary:

Dim.	**Type**	**Load**	**Material**	**Geometry**	**Contact**	**Solver**	**Solution Method**
3D	Dynamic	Pressure Damping	Linear	Linear	-	Implicit	1 - Linear

Units:

kg, m, s, N, Pa, N-m (kilogram, meter, second, Newton, Pascal, Newton-meter)

Dimensional Data:

Rectangular dimensions of square plate: 10.0 *m* x 10.0 *m* x 1.00 *m*.

Material Data:

Mass Density	$\rho = 8.00 \times 10^3 \ kg/m^3$
Young's Modulus	$E = 2.00 \times 10^{11} \ Pa$
Poisson's Ratio	$\nu = 0.3$
Damping ratio	$\zeta = 2\%$

Load:

Pressure $P = 1.0 \times 10^6 \ Pa$

Element Types:

S/R 2x2 IPI thick shell (elform=2)
Assumed strain 2x2 IPI thick shell (elform=3)
Assumed strain RI thick shell (elform=5)

Material Models:

*MAT_001 or *MAT_ELASTIC

Damping:

The damping factor d is easily found from the natural frequency of the system:

$$d_i = \zeta 2\omega_i$$

As the excited mode is the first, corresponding to $f_1 = 45.897\ Hz$ ($\omega_1 = 288.380\ rad/s$) (that from NAFEMS Benchmark Test FV52), we choose the damping factor relative to the first frequency:

$$d_1 = 11.535\ Hz$$

Results Comparison:

LS-DYNA X-direction bending stress, σ_{xx}**,** at (Node 161) on bottom surface plus its Z-displacement, U_Z, are compared with *NAFEMS Selected Benchmarks for Forced Vibration***,** Test 21T.

Reference Condition - Center (Node 161)	**Peak Bending Stress σ_{xx} (*Pa*)**	**Peak U_Z (*m*)**	**Steady-State U_Z (*m*)**
NAFEMS Benchmark Test 21T	6.211×10^{7}	-4.524×10^{-3}	-2.333×10^{-3}
S/R 2x2 IPI thick shell (elform=2)	6.398×10^{7}	-4.937×10^{-3}	-2.537×10^{-3}
Assumed strain 2x2 IPI thick shell (elform=3)	6.350×10^{7}	-5.090×10^{-3}	-2.616×10^{-3}
Assumed strain RI thick shell (elform=5)	6.319×10^{7}	-5.022×10^{-3}	-2.557×10^{-3}

These nodal displacement results were generated by *DATABASE_NODOUT keyword while the axial stress (nodal) results were generated by *DATABASE_ELOUT (*elout* file) and *DATABASE_EXTENT_BINARY (*eloutdet* file provides detailed element output at integration points and connectivity nodes) keyword entries.

You can set intout=stress or intout=all (*DATABASE_EXTENT_BINARY) and have stresses output for all the integration points to a file called *eloutdet* (*DATABASE_ELOUT governs the output interval and *DATABASE_HISTORY_ TSHELL governs which elements are output). Setting nodout=stress or nodout=all in *DATABASE_EXTENT_BINARY will write the extrapolated nodal stresses to *eloutdet*.

LS-DYNA stress and strain output corresponds to integration point locations. Stress at a node is an artifact of the post-processor and represents an average of the surrounding integration point stresses (the value will likely be different with different post-processors).

Lobatto integration (intgrd=1 - *CONTROL_SHELL) was employed since it has an advantage in that the inner and outer integration points are on the shell surfaces. Gauss integration is the default through thickness integration rule (the default number of through thickness integration points is nip=2 - *SECTION_TSHELL) in LS-DYNA, where 1-10 integration points may be specified, whereas, with Lobatto integration, 3-10 integration points may be specified (for 2 point integration, the Lobatto rule is very inaccurate).

The selectively reduced 2x2 IPI thick shell (elform=2), the assumed strain 2x2 IPI thick shell (elform=3), and the assumed strain RI thick shell (elform=5) all provide similar results for this transient forced vibration example, though slightly less stiff in comparison, both in stress and displacement.

Remember that (a) only the higher order, selectively reduced 2x2 IPI thick shell (elform=2) provides a reasonable stress comparison for a single element through the thickness, although with a comparatively stiff result, and (b) the higher order, assumed strain 2x2 IPI thick shell (elform=3) and assumed strain RI thick shell (elform=5) formulations do provide acceptable results with at least two elements through the thickness (recommended) to capture the bending response.

For this transient forced vibration example, with an element aspect ratio of 3.75, it is seen that the thick shell formulations, on the whole, compare better than the solid element formulation results. The exception to this would be the displacement comparison provided by the higher order, fully integrated S/R solid (both so-called efficient and accurate formulation choices) intended to address poor aspect ratios (elform=-1 and -2, respectively).

For the selectively reduced 2x2 IPI thick shell (elform=2), the contour plot of the X-direction bending stress (Figure 15.2) and the Z-displacement (Figure 15.3) were obtained from the *d3plot* file at peak displacement time which were generated by the *DATABASE_BINARY_D3PLOT keyword.

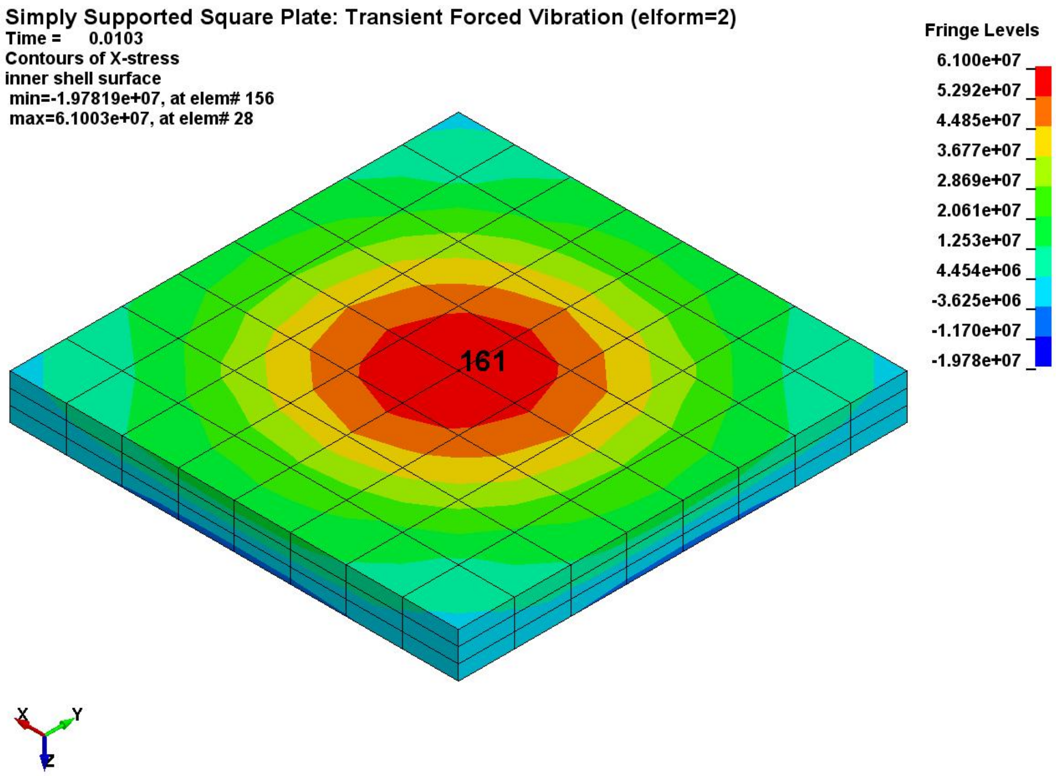

Figure 15.2 – Contour plot of the X-stress (elform=2) at peak displacement time.

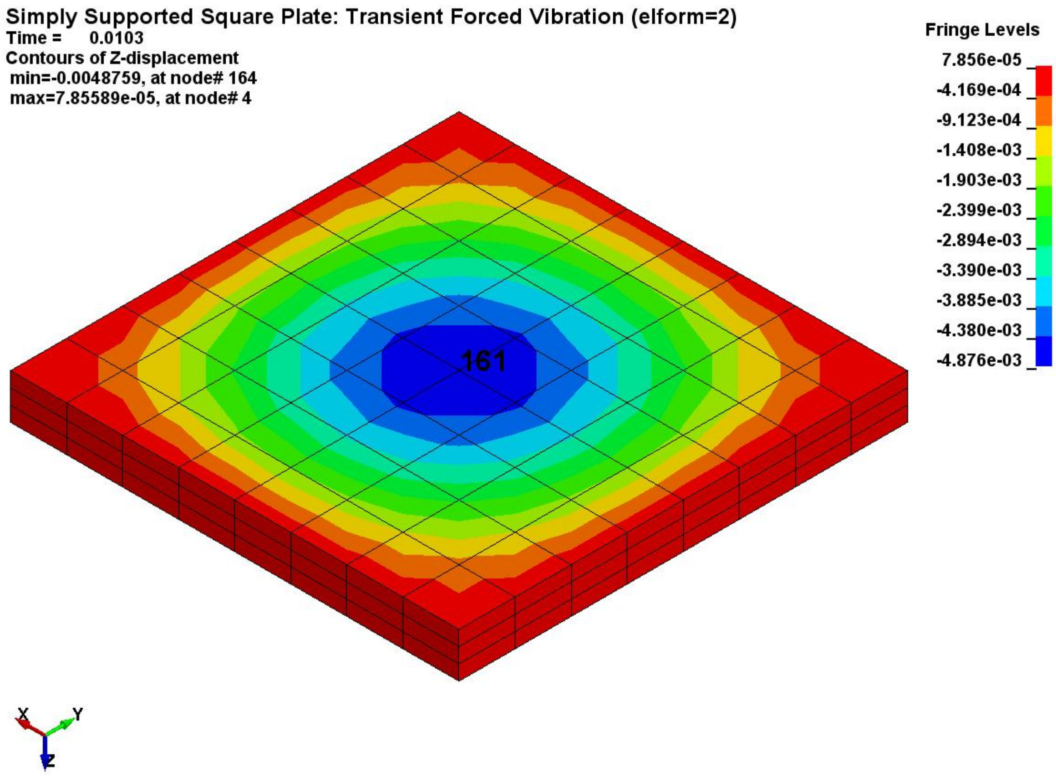

Figure 15.3 - Contour plot of Z-displacement (elform=2) at peak displacement time.

Input deck:

```
*KEYWORD
*TITLE
Simply Supported Square Plate: Transient Forced Vibration (thick shell mesh)
*CONTROL_IMPLICIT_AUTO
$#   iauto    iteopt    itewin     dtmin     dtmax     dtexp     kfail    kcycle
         0        11         5       0.0       0.0
*CONTROL_IMPLICIT_DYNAMICS
$#   imass     gamma      beta
         1  0.500000  0.250000
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form    zero_v
         1  0.000100         2         1         2
*CONTROL_IMPLICIT_SOLVER
$#  lsolvr    lprint     negev     order      drcm    drcprm   autospc   autotol
         4         2         2         0         1       0.0         1       0.0
$#  lcpack
         2
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
         2        11        15    0.0010    0.0100  1.00e+10  0.900000  1.000000
$#   dnorm    diverg     istif   nlprint
         2         1         1         2
$#  arcctl    arcdir    arclen    arcmth    arcdmp
         0         1       0.0         1         2
*CONTROL_SHELL
$#  wrpang     esort     irnxx    istupd    theory       bwc     miter      proj
  20.00000         0         0         0         2         2         1
$# rotascl    intgrd    lamsht    cstyp6    tshell    nfail1    nfail4
       0.0         1
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  0.020000         0       0.0       0.0       0.0
*DATABASE_ELOUT
$#      dt    binary      lcur     ioopt
1.0000e-06         1
*DATABASE_NODFOR
$#      dt    binary      lcur     ioopt
1.0000e-06         1
*DATABASE_NODOUT
$#      dt    binary      lcur     ioopt
1.0000e-06         1
*DATABASE_BINARY_D3PLOT
$# dt/cycl
  0.001000
*DATABASE_EXTENT_BINARY
$#   neiph     neips    maxint    strflg    sigflg    epsflg     rtflg    engflg

$#  cmpflg    ieverp    beamip     dcomp      shge     stssz    n3thdt   ialemat

$# nintsld   pkp_sen      sclp     hydro     msscl     therm    intout    nodout
         8                 1.0                                  stress    stress
*DATABASE_HISTORY_TSHELL
$#     id1       id2       id3       id4       id5       id6       id7       id8
        28        29        36        37
*DATABASE_NODAL_FORCE_GROUP
$#    nsid       cid
       164
*DATABASE_HISTORY_NODE
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       161       164
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4    solver
       164       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       164
*DAMPING_GLOBAL
$#    lcid    valdmp       stx       sty       stz       srx       sry       srz
         0  11.53500       0.0       0.0       0.0       0.0       0.0       0.0
```

```
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo     dattyp
         1         0       0.0       0.0       0.0       0.0
$#                  a1                  o1
                   0.0       1.0000000e+06
          0.10000000       1.0000000e+06
*ELEMENT_TSHELL
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1      37      41       5       2      38      42       6

     192       1     283     319     323     287     284     320     324     288
*NODE
$#   nid               x               y               z      tc      rc
       1             0.0             0.0             0.0       3

     324     10.00000000     10.00000000      1.00000000
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         0         0         1
*PART
$# title
material type # 1  (Elastic)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_TSHELL
$#   secid    elform      shrf       nip     propt   qr/irid     icomp    tshear
         1         2       0.0         5         0       0.0
$         1         3       0.0         5         0       0.0
$         1         5       0.0         5         0       0.0
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1  8000.0002.0000e+11  0.300000       0.0       0.0       0.0
*LOAD_SEGMENT
$#    lcid        sf        at        n1        n2        n3        n4
         1  1.000000       0.0         4        40        44         8

         1  1.000000       0.0       284       320       324       288
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1        37        73       109       145       181       217       253
       289       293       297       301       305       309       313       317
       321       285       249       213       177       141       105        69
        33        29        25        21        17        13         9         5
*END
```

Notes:

16. Transient Response of a Cylindrical Disk Impacting a Deformable Surface

Keywords:

```
*CONTROL_IMPLICIT_DYNAMICS
*CONTROL_IMPLICIT_AUTO
*CONTROL_IMPLICIT_GENERAL
*CONTROL_IMPLICIT_SOLVER
*CONTROL_IMPLICIT_SOLUTION
*CONTACT_2D_AUTOMATIC_NODE_TO_SURFACE
```

Description:

A rigid cylindrical disk of given mass (*m*) is released from 1.0 inch height (*h*) and, accelerated by gravity (*g*), hits a deformable surface of given stiffness (*k*). Plot the velocity, displacement, and kinetic energy of the disk, plus identify the time of impact. The maximum displacement of the cylindrical disk is also to be determined.

This simulation (Figure 16.1) is 2D plane strain (a choice of the available LS-DYNA options to address the impact - with the objects being rigid, this can be seen to negate the need for any planar type definition). The cylindrical disk is modeled with shell elements, x-y plane, which do not need additional constraints to ensure in-plane behavior. The deformable surface is modeled with rigid beam elements, x-y plane, to address contact, and a 1D translational spring element of a finite length (*l*), x-y plane, to address the deformation.

As a geometric convenience, LS-DYNA employs the *SECTION_SHELL and *ELEMENT_SHELL entries to describe 2D plane stress, plane strain, and axisymmetric solids, and the *SECTION_BEAM and *ELEMENT_BEAM entries to describe 2D axisymmetric shells, and 2D plane strain beam elements.

A suitable contact algorithm for this problem is the *CONTACT_2D_AUTOMATIC_NODE_TO_SURFACE. For this algorithm, the contact stiffness is activated when a node nears a segment at some given tolerance. The stiffness is increased as the node moves closer with the full stiffness being used when the nodal point finally makes contact. Understanding LS-DYNA contact considerations adds to the focus of this example. A plot of the contact force history is sought.

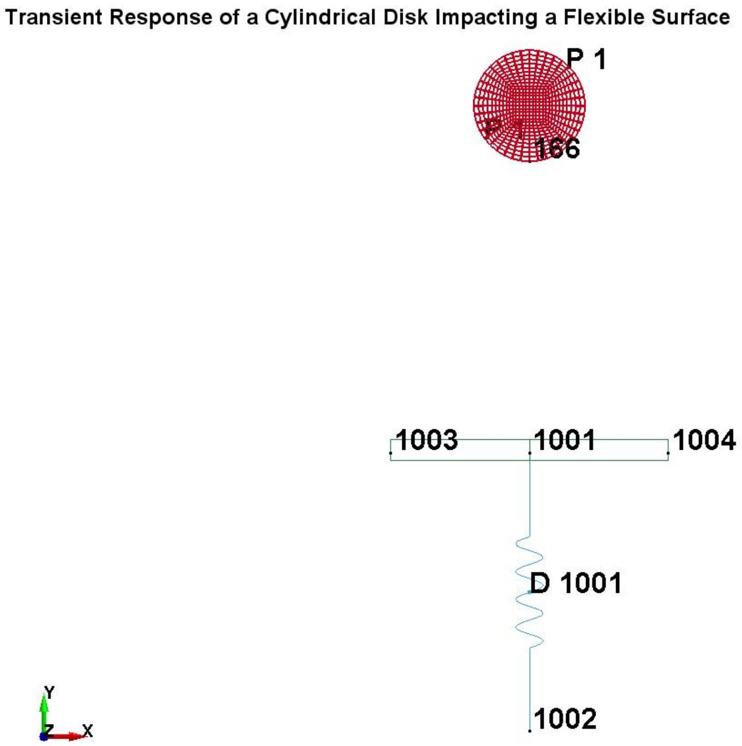

Figure 16.1 – Finite element model with selected parts, elements, and nodes identified.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
2D	Dynamic	Gravity	Linear	Linear	2D	Implicit	2-Nonlinear w/BFGS

Units:

lbf-s^2*/in, in, s, lbf, psi, lbf-in (blob, inch, second, pound force, pound force/inch*^{2}*, pound force-inch)*

Dimensional Data:

$h = 1.0\times10^{0}\ in$, $l = 1.0\times10^{0}\ in$

Material Data:

Mass Density	$\rho = 3.995281 \times 10^0 \; lbf - s^2 / in^4$
Nodal Mass	$m = 0.50 \; lbf - s^2 / in$
Spring Stiffness	$k = 1.97392 \times 10^3 \; lb / in$

Load:

Body Force $g = 0.0 \; in/s^2$ varied linearly to $3.86 \times 10^2 \; in/s^2$, then held constant

$g = 0.0 \; in/s^2, \; t = 0.0 \; s$

$g = 3.86 \times 10^2 \; in/s^2, \; t = 1.0 \times 10^{-3} \; s$

$g = 3.86 \times 10^2 \; in/s^2, \; t = 1.0 \times 10^0 \; s$

Element Types:

2D plane strain shell element (xy plane) - *SECTION_BEAM entry (elform=7)

Plane strain (x-y plane) - *SECTION_SHELL entry - (elform=13)

Translational spring - (SECTION_DISCRETE (dro=0)

Material Models:

*MAT_001 or *MAT_ELASTIC

*MAT_020 or *MAT_RIGID

Results Comparison:

LS-DYNA results for velocity $\boldsymbol{V_Y}$ and displacement $\boldsymbol{U_Y}$ of the cylindrical disk at impact (plus the time) and maximum displacement of the cylindrical disk $\boldsymbol{U_{Ymax}}$, are compared with W.T. Thomson's studies in *Vibration Theory and Applications*, 1965 (pg. 110).

	Impact Time (*s*)	**Velocity $\boldsymbol{V_Y}$ (*in/s*)**	**Displacement $\boldsymbol{U_Y}$ (*in*)**	**Max. Disp. $\boldsymbol{U_{Ymax}}$ (*in*)**
Thomson [1965]	0.07198	-27.7900	-1.0000	-1.5506
Cylindrical Disk - Part 1	0.07240	-27.7728	-0.9978	-1.5524

LS-DYNA results are reported at the closet time point for the displacement value designated as full stiffness contact (i.e. $\boldsymbol{U_Y} = -1.0000$ in).

The nodal results were generated by *DATABASE_NODOUT keyword, the kinetic energy by *DATABASE_MATSUM keyword, and the contact force by *DATABASE_ RCFORC keyword.

Figure 16.2 provides the velocity $\boldsymbol{V_Y}$ history, Figure 16.3 the vertical displacement $\boldsymbol{U_Y}$ history, and Figure 16.4 the kinetic energy history, all of the cylindrical disk.

Figure 16.5 gives the contact force between the cylindrical disk (slave) and the flexible surface (master).

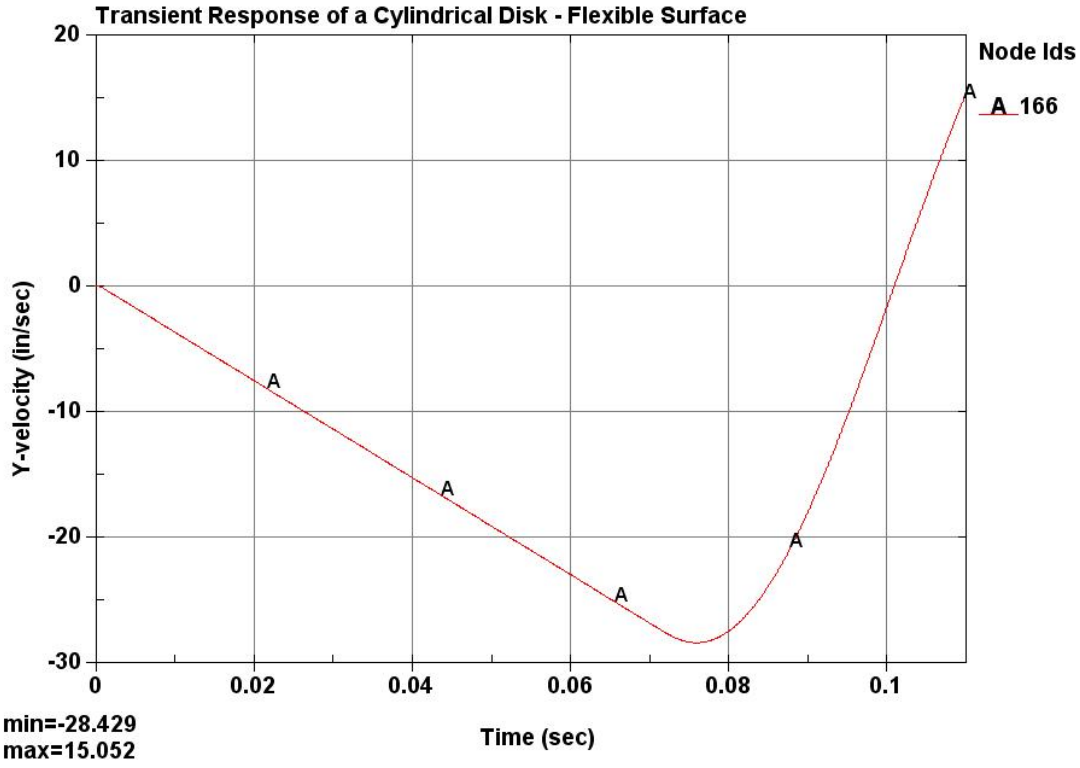

Figure 16.2 – Velocity V_Y of the cylindrical disk.

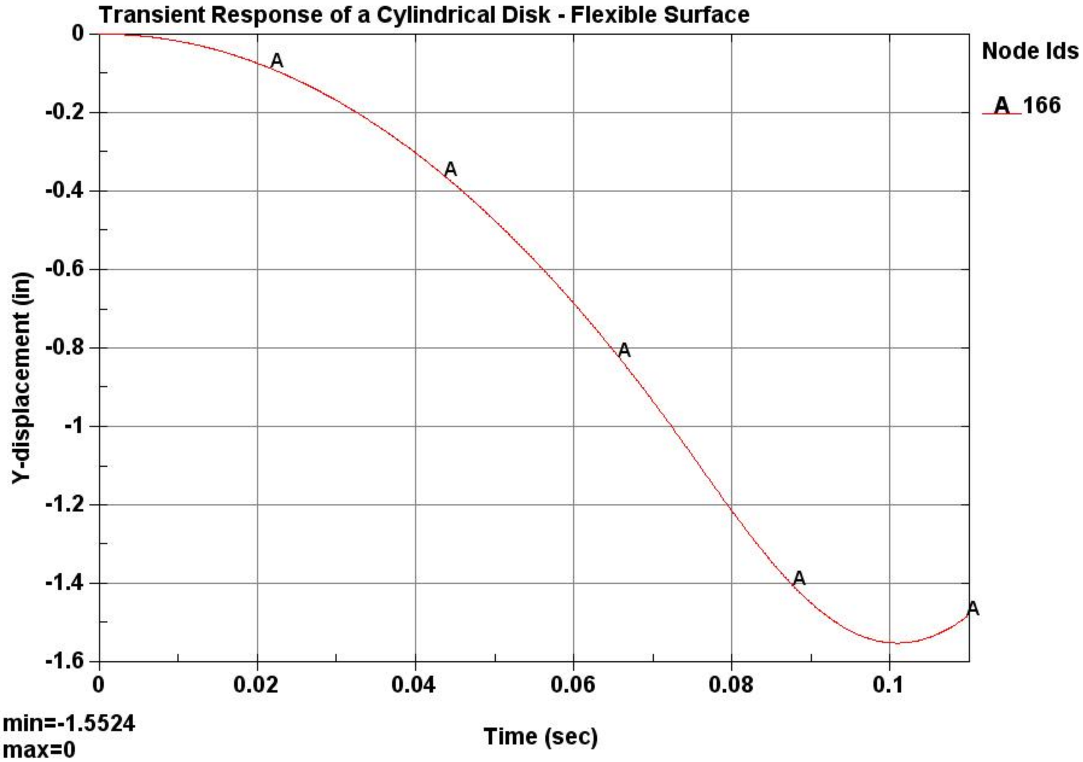

Figure 16.3 – Vertical displacement U_Y of the cylindrical disk.

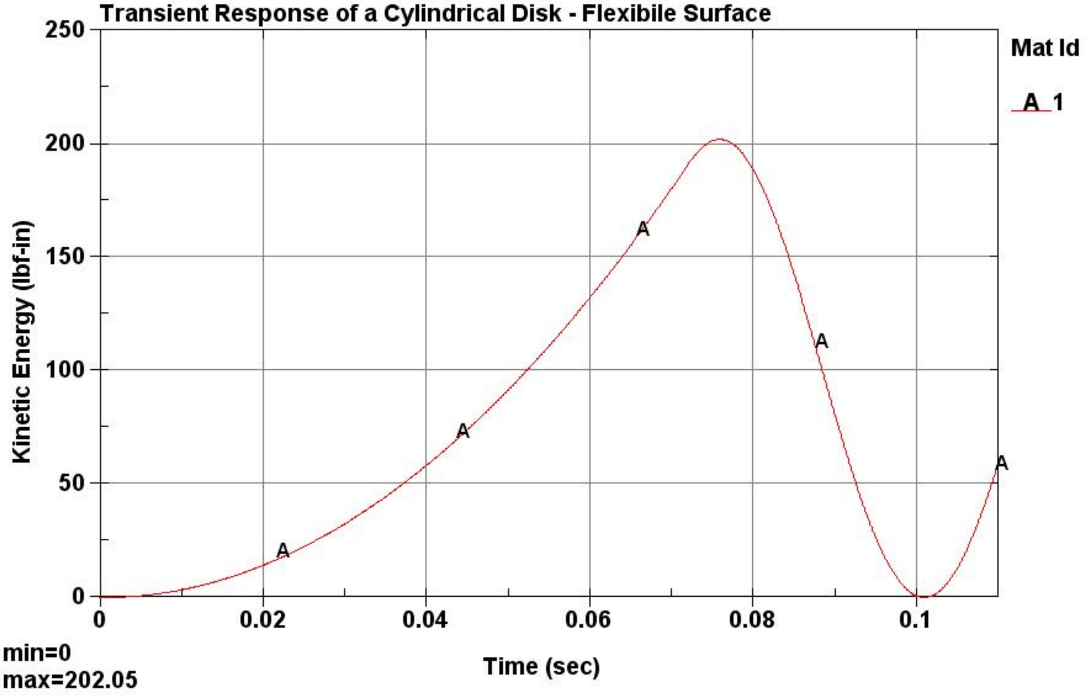

Figure 16.4 – Kinetic energy of the cylindrical disk.

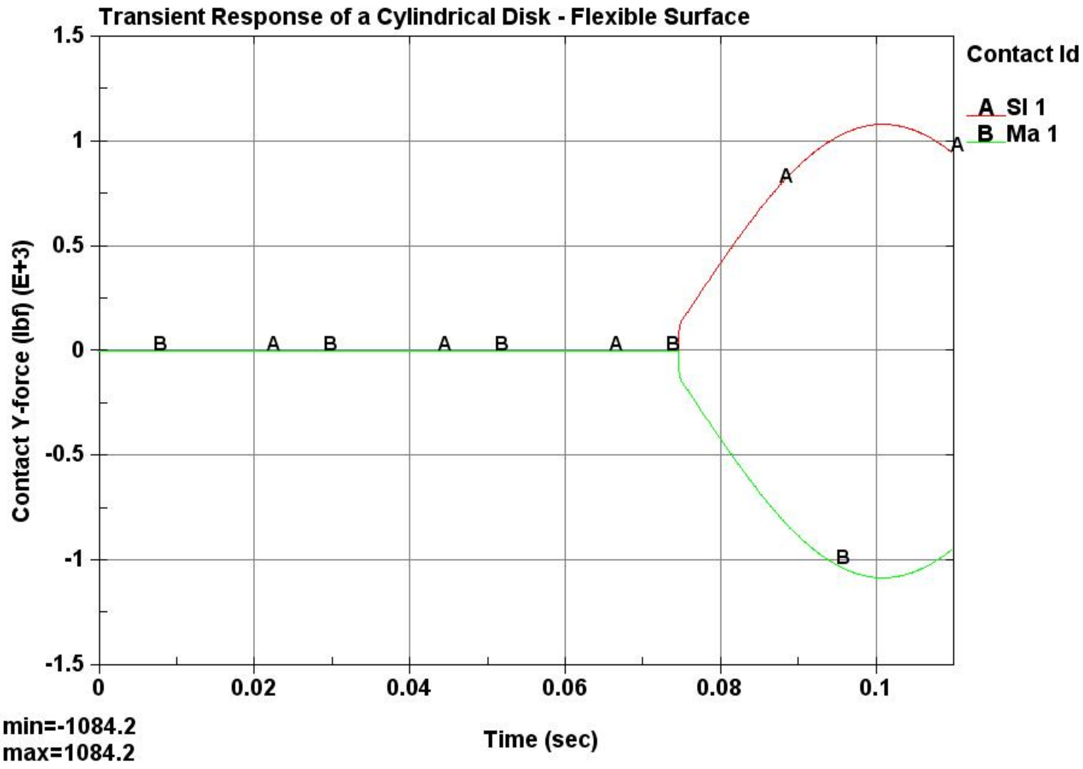

Figure 16.5 – Contact force between cylindrical disk (slave) and flexible surface (master).

Input deck:

```
*KEYWORD
*TITLE
Transient Response of a Cylindrical Disk Impacting a Flexible Surface
*CONTROL_IMPLICIT_DYNAMICS
$#   imass     gamma      beta
         1  0.500000  0.250000
*CONTROL_IMPLICIT_AUTO
$#   iauto    iteopt    itewin     dtmin     dtmax
         1        11         5  1.00e-06  1.00e-04
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1  0.000100         0         0         0
*CONTROL_IMPLICIT_SOLVER
$#  lsolvr   prntflg    negeig     order      drcm    drcprm   autospc    aspctl
         4         2         2         0         1         0         1         0
$#  lcpack
         2
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
         2        11        15    0.0010    0.0100  1.00e+10  0.900000  1.00e-10
$#   dnorm    diverg     istif   nlprint    nlnorm   d3itctl     cpchk
         2         1         1         2
$#  arcctl    arcdir    arclen    arcmth    arcdmp
         0         1       0.0         1         2
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
    0.1100
*DATABASE_GLSTAT
$#      dt    binary
```

```
1.0000e-04         1
*DATABASE_MATSUM
$#      dt    binary
1.0000e-04         1
*DATABASE_RCFORC
$#      dt    binary
1.0000e-04         1
*DATABASE_NODOUT
$#      dt    binary
1.0000e-04         1
*DATABASE_BINARY_D3PLOT
$# dt/cycl   lcdt/nr      beam     npltc    psetid
1.0000e-03
*DATABASE_HISTORY_NODE
$#    nid1      nid2      nid3      nid4       ni5      nid6      nid7      nid8
       166      1001
*PART
$# title
rigid cylindrical disk
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_SHELL
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
         1        13         0         4
$#      t1        t2        t3        t4      nloc     marea
    1.0000    1.0000    1.0000    1.0000
*MAT_RIGID
$      mid        ro         e        pr         n    couple         m     alias
         1 3.9952831  3.00e+07    0.3000       0.0       0.0       0.0       0.0
$      cmo      con1      con2
    1.0000       6.0       7.0
$lco_or_a1        a2        a3        v1        v2        v3
       0.0       0.0       0.0       0.0       0.0       0.0
*PART
discrete spring - flexible surface
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         2         2         2
*SECTION_DISCRETE
$#   secid       dro        kd        v0        cl        fd
         2         0         0         0       0.0       0.0
$#     cdl       tdl
       0.0       0.0
*MAT_SPRING_ELASTIC
$#     mid         k
         2 1973.9200
*PART
rigid plane beam (wall)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         3         3         3
*SECTION_BEAM
$#   secid    elform      shrf   qr/irid       cst     scoor      nsm
         3         7    1.0000    1.0000    0.0000
$      ts1       ts2       tt1       tt2
    0.1000    0.1000
*MAT_RIGID
$      mid        ro         e        pr         n    couple         m     alias
         3  1.00e-07  3.00e+07    0.3000       0.0       0.0       0.0       0.0
$      cmo      con1      con2
    1.0000       6.0       7.0
$lco_or_a1        a2        a3        v1        v2        v3
       0.0       0.0       0.0       0.0       0.0       0.0
*ELEMENT_BEAM
    1002       3    1003    1001
    1003       3    1001    1004
*ELEMENT_DISCRETE
$    eid     pid      n1      n2     vid               s      pf          offset
    1001       2    1001    1002       0             1.0       0             0.0
*ELEMENT_SHELL
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1      12      13       2       0       0       0       0
```

```
     500       1     512     521     431     430       0       0       0       0
*NODE
$#   nid               x               y               z      tc      rc
       1     -0.14142136      0.05857864             0.0

     521      0.16180338      0.31755710             0.0
    1001             0.0          -1.050             0.0       6       7
    1002             0.0          -2.050             0.0       7       7
    1003            -0.5          -1.050             0.0       6       7
    1004             0.5          -1.050             0.0       6       7
$    1001             0.0            -1.0             0.0       6       7
$    1002             0.0            -2.0             0.0       7       7
$    1003            -0.5            -1.0             0.0       6       7
$    1004             0.5            -1.0             0.0       6       7
*CONTACT_2D_AUTOMATIC_NODE_TO_SURFACE
$#    ssid      msid     sfact      freq        fs        fd        dc     membs
        -2        -1      0.10         0         0         0         0         0
$#  tbirth    tdeath       sos       som       nds       ndm       cof      init
         0         0         0         0         0         0         0         0
$#      vc       vdc       ipf     slide    istiff   tiedgap
                                                 2
$*CONTACT_2D_PENALTY
$#    ssid      msid    tbirth    tdeath
$         2         1
$# ext_pas    theta1    theta2    tol_ig       pen    toloff    frcscl    oneway
$                                              0.10   0.00010
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4
         1
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
      1003      1001      1004
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4
         2
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       155       166       177
*LOAD_BODY_Y
$     lcid        sf    lciddr        xc        yc        zc       cid
         1       1.0
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo    dattyp
         1         0    1.0000    1.0000       0.0       0.0
$#                a1                  o1
               0.000               0.000
               0.001              386.00
               1.000              386.00
*END
```

Notes:

1. From the LS-DYNA User's Manual: Note that the 2D and 3D element types must not be mixed, and different types of 2D elements, i.e. plane strain, plane stress, and axisymmetric, must not be used together. The discrete (spring) 1D element can be used with either 2D or 3D elements.

2. Consider the two surfaces comprising a contact. It is necessary to designate one as a slave surface and the other as a master surface. Nodal points defining the slave surface are called slave nodes, and similarly, nodes defining the master surface are called master nodes. Each slave-master surface combination is referred to as a contact surface. If one surface is more finely zoned, it should be defined as the slave surface.

3. By default, the true thickness of 2D shell elements is taken into account for *CONTACT_2D _AUTOMATIC_SURFACE_TO_SURFACE and _AUTOMATIC_ NODE_TO_SURFACE contacts. The user can override the true thickness by using the sos and som parameters on the contact entry.

   input example:

```
*CONTACT_2D_AUTOMATIC_NODE_TO_SURFACE
$#    ssid      msid     sfact      freq        fs        fd        dc     membs
        -2        -1      0.10         0         0         0         0         0
$#  tbirth    tdeath       sos       som       nds       ndm       cof      init
         0         0         0         0         0         0         0         0
$#      vc       vdc       ipf     slide    istiff   tiedgap
                                                 2
$
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4
         1
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
      1003      1001      1004
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4
         2
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       155       166       177
$
*NODE
$#   nid               x               y               z      tc       rc
    1001             0.0          -1.050             0.0       6        7
    1002             0.0          -2.050             0.0       7        7
    1003            -0.5          -1.050             0.0       6        7
    1004             0.5          -1.050             0.0       6        7
```

   There is a stiffness control variable available, cof, which allows the full stiffness to be gradually applied as a node approaches a segment. The tolerance for the stiffness appears to be hardwired internally in the LS-DYNA software. cof offers only two options: on (cof=0 - LS-DYNA default) or off (cof=1); no tolerance adjusting.

   Using cof=0 activates the contact stiffness as a node approaches a segment at some unknown value; the stiffness is increased as the node moves closer with the full stiffness being used when the nodal point finally makes contact. Using cof=1 does not turn on any contact stiffness until the nodal point makes full stiffness contact.

   It is observed that the contact output force calculation (*DATABASE_RCFORC) is not made (delayed) until full stiffness contact is made for either cof option.

   For cof=1, the contact force is calculated without a delay (due to full stiffness being applied without the gradual increase), but is nosier (oscillatory) than the other solution cof=0, at least for this example problem.

4. If the older penalty contact algorithms are used, *CONTACT_2D _PENALTY and _PENALTY_ FRICTION, the slave-master distinction is irrelevant. These contacts use the mid-surface of the 2D shell elements; thus, the shell thickness is not taken into account. This sometimes may make it necessary to modify various physical

coordinates to achieve reasonable results, e.g., the arrival time of a dropped rigid sphere onto a 2D shell plate of a moderate thickness.

input example:

```
*CONTACT_2D_PENALTY
$#    ssid      msid    tbirth    tdeath
         2         1
$# ext_pas    theta1    theta2    tol_ig       pen    toloff    frcscl    oneway
                                              0.10   0.00010
       155       166       177
$
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4
         1
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
      1003      1001      1004
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4
         2
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       155       166       177
$
*NODE
$#   nid               x               y               z      tc      rc
    1001             0.0            -1.0             0.0       6       7
    1002             0.0            -2.0             0.0       7       7
    1003            -0.5            -1.0             0.0       6       7
    1004             0.5            -1.0             0.0       6       7
```

There is an adjustable, stiffness control variable available, toloff, which allows the full stiffness to be gradually applied as a node approaches a segment.

From the LS-DYNA User's Manual: toloff - Tolerance for stiffness activation for implicit solution only. The contact stiffness is activated when a node approaches a segment at a distance equal to the segment length multiplied by toloff. The stiffness is increased as the node moves closer with the full stiffness being applied when the nodal point finally makes contact.

It is observed that the contact output force calculation (*DATABASE_RCFORC) is delayed until full stiffness contact is made.

17. Natural Frequency of a Linear Spring-Mass System

Keywords:

```
*CONTROL_TIMESTEP
*ELEMENT_DISCRETE
*ELEMENT_MASS
*MAT_SPRING_ELASTIC
```

Description:

A mass (*m*) is attached to a linear spring, as shown in Figure 17.1. The mass is initially displaced $d = -1.0\ in$ from its equilibrium position and released. Determine the period of vibration τ .

The spring is modeled by one discrete element (*ELEMENT_DISCRETE) using a linear elastic spring material (*MAT_S01/*MAT_SPRING_ELASTIC). The lumped mass is modeled by an *ELEMENT_MASS entry.

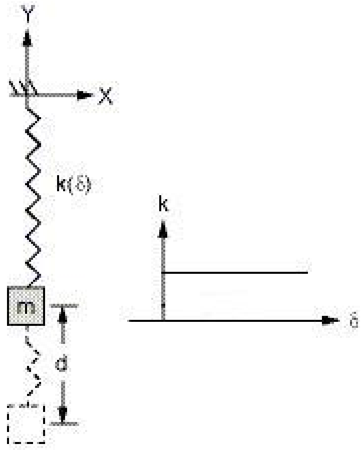

Figure 17.1 – Sketch representing the model.

Analysis Summary:

Dim.	**Type**	**Load**	**Material**	**Geometry**	**Contact**	**Solver**	**Solution Method**
3D	Dynamic	Initially Displace	Linear	Linear	-	Explicit	-

Units:

lbf-s^2/in, in, s, lbf, psi, lbf-in (blob, inch, second, pound force, pound force/inch^2, pound force-inch)

Dimensional Data:

$d = 1.0 \times 10^0 \ in$

Material Data:

Nodal Mass	$m = 2.588 \times 10^{-3} \ lbf - s^2 / in$
Spring Stiffness	$k = 5.0 \ lbf \ / in$

Element Types:

Translational spring - *SECTION_DISCRETE (dro=0)

Lumped mass (*ELEMENT_MASS entry)

Material Models:

*MAT_S01 or *MAT_SPRING_ELASTIC

Results Comparison:

LS-DYNA results for the period of vibration related to this linear spring-mass system are compared with S.P. Timoshenko and D.H. Young studies in *Vibration Problems in Engineering*, 1955 (pg. 1).

	Period of Vibration τ (s)
Timoshenko and Young [1955]	0.14295
Linear spring-mass system	0.14295

This nodal displacement result and the computer period of vibration was generated by *DATABASE_NODOUT keyword (also see Figures 17.2 and 17.3).

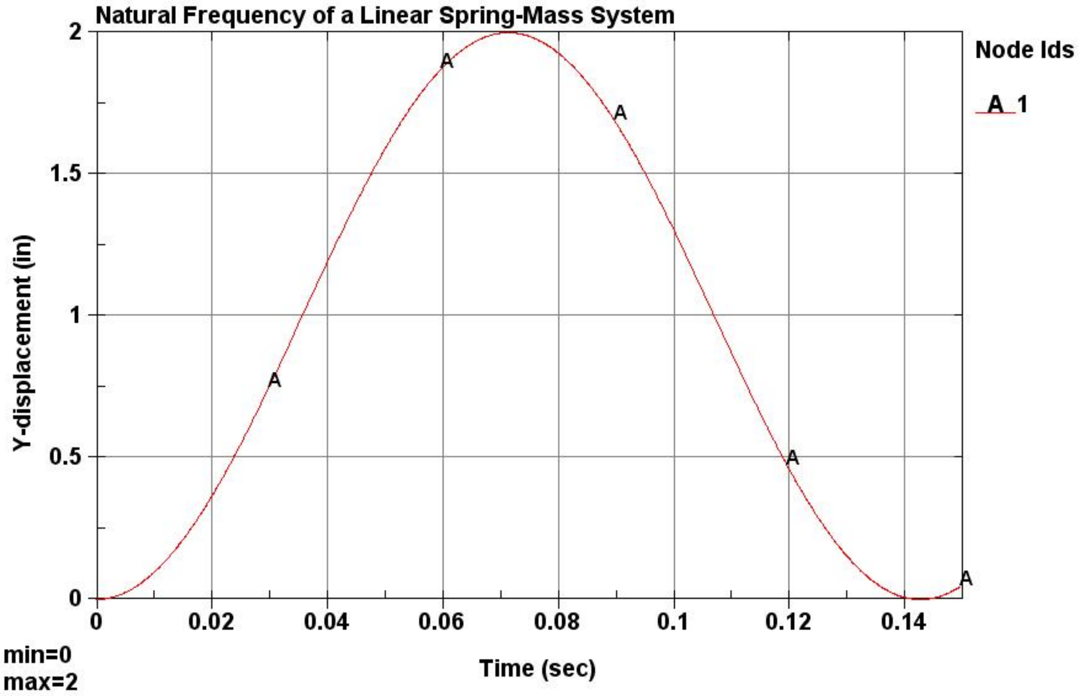

Figure 17.2 – Node 1 displacement U_Y .

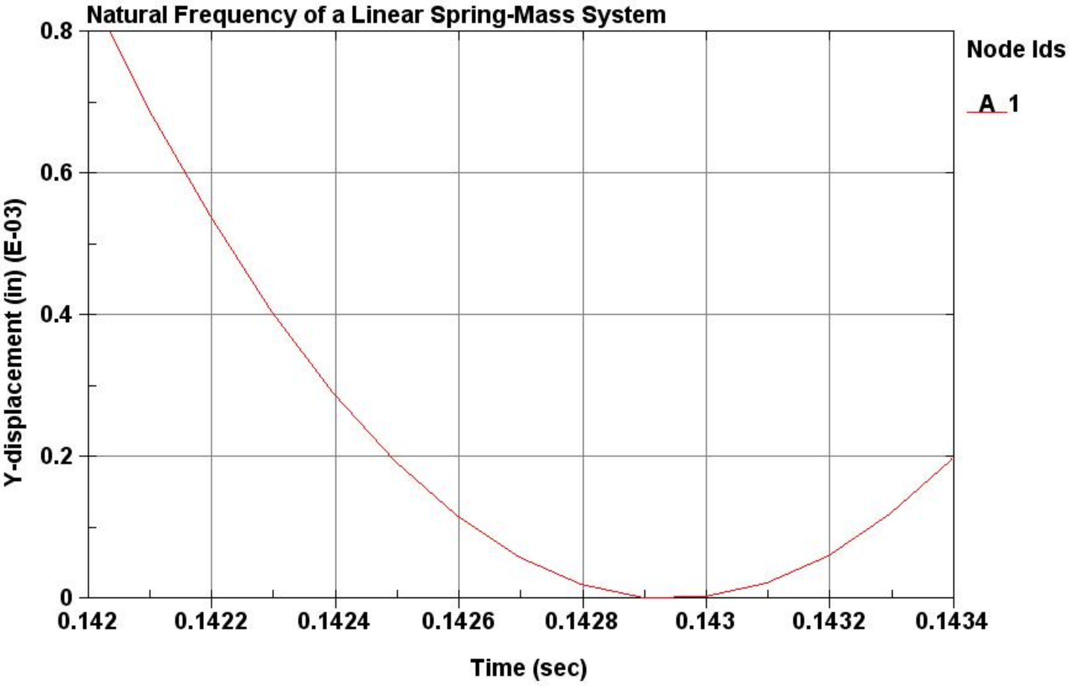

Figure 17.3 – Node 1 displacement U_Y (detailed time capture).

Input deck:

```
*KEYWORD
*TITLE
Natural Frequency of a Linear Spring-Mass System
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  0.150000         0       0.0       0.0       0.0
*CONTROL_TIMESTEP
$#  dtinit    tssfac      isdo    tslimt     dt2ms      lctm     erode     ms1st
 1.000e-04 1.000e-04         0       0.0       0.0         0         0         0
$#  dt2msf   dt2mslc     imscl
       0.0         0         0
*DATABASE_NODOUT
$#      dt    binary
  0.000100
*DATABASE_BINARY_D3PLOT
$# dt/cycl   lcdt/nr      beam     npltc    psetid
  0.001000
*DATABASE_HISTORY_NODE
$#    nid1      nid2      nid3      nid4       ni5      nid6      nid7      nid8
         1
*PART
$# title
linear elastic spring
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_DISCRETE
$#   secid       dro        kd        v0        cl        fd
         1         0       0.0       0.0       0.0       0.0
$#     cdl       tdl
       0.0       0.0
*MAT_SPRING_ELASTIC
$#     mid         k
         1      5.00
*ELEMENT_DISCRETE
$#   eid     pid      n1      n2     vid               s      pf          offset
       1       1       1       2       0         1.00000       0         1.00000
*ELEMENT_MASS
$#   eid      id            mass     pid
       2       1      0.00258800
       3       2      0.00258800
*NODE
$#   nid               x               y               z      tc      rc
       1             0.0             0.0             0.0
       2             0.0      1.00000000             0.0
*BOUNDARY_SPC_NODE
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         2         0         1         1         1         1         1         1
*END
```

Notes:

1. As an alternative, it is possible to model the linear spring with a *BEAM_ELEMENT (with the option discrete) and *MAT_066/*MAT_ LINEAR_ELASTIC_ DISCRETE_BEAM material behavior.

2. For simulations with linear stiffness, one could use the following implicit entries and perform a simple eigenvalue analysis:

```
*CONTROL_IMPLICIT_EIGENVALUE
$#    neig    center     lflag    lftend     rflag    rhtend    eigmth    shfscl
         3    11.000         0 -1.00e+29         0  1.00e+29         2       0.0
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         6  1.00e-04         2         1         2
```

The period could be obtained directly from the *eigout* results file generated by the *CONTROL_IMPLICIT_EIGENVALUE keyword as shown here:

```
Natural Frequency of a Linear Spring-Mass System
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1   4.547474E-12   2.132481E-06   3.393948E-07   2.946421E+06
           2   5.456968E-12   2.336015E-06   3.717884E-07   2.689702E+06
           3   1.931994E+03   4.395445E+01   6.995568E+00   1.429476E-01
```

18. Natural Frequency of a Nonlinear Spring-Mass System

Keywords:

```
*CONTROL_TIMESTEP
*ELEMENT_DISCRETE
*ELEMENT_MASS
*MAT_SPRING_NONLINEAR_ELASTIC
```

Description:

A mass (m) is attached to a nonlinear spring: $k = k_0 + k_1\delta^2$, as shown in Figure 18.1. The mass is initially displaced $d = -1.0\ in$ from its equilibrium position and released. Determine the period of vibration τ.

The spring is modeled by one discrete element (*ELEMENT_DISCRETE) using a nonlinear elastic spring material (*MAT_S04/*MAT_SPRING_NONLINEAR_ELASTIC). The lumped mass is modeled by an *ELEMENT_MASS entry.

To input the data for the *MAT_S04/*MAT_SPRING_NONLINEAR_ELASTIC it is necessary to convert the stiffness-deflection curve $k(\delta)$ to a force-deflection $F(\delta)$: $F = k\delta = k_0\delta + k_1\delta^3$, using a *DEFINE_CURVE entry. This curve is converted to eleven points of force-deflection points in the range $\delta = [0,1]$.

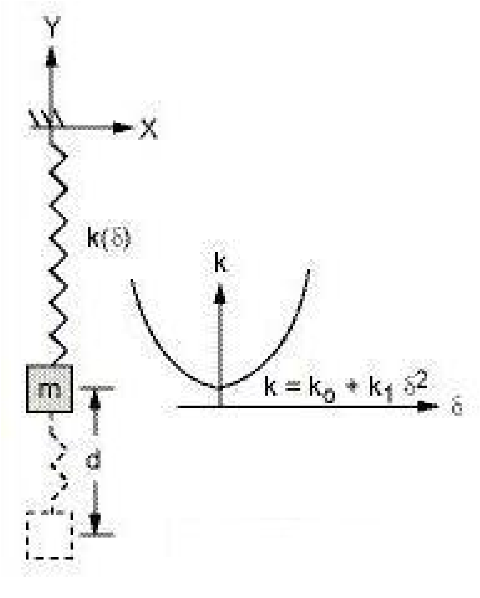

Figure 18.1 – Sketch representing the model.

Analysis Summary:

Dim.	**Type**	**Load**	**Material**	**Geometry**	**Contact**	**Solver**	**Solution Method**
3D	Dynamic	Initially Displace	Non-linear	Linear	-	Explicit	-

Units:

lbf-s^2/in, in, s, lbf, psi, lbf-in (blob, inch, second, pound force, pound force/inch^2, pound force-inch)

Dimensional Data:

$d = 1.0 \times 10^0 \ in$

Material Data:

Nodal Mass	$m = 2.588 \times 10^{-3} \ lbf - s^2 / in$
Spring Stiffness	$k = k_0 + k_1 \delta^2$, $k_0 = 2.0 \ lbf / in$, $k_1 = 4.0 \ lbf / in^3$

Element Types:

Translational spring - *SECTION_DISCRETE (dro=0)

Lumped mass (*ELEMENT_MASS entry)

Material Models:

*MAT_S04 or *MAT_SPRING_NONLINEAR_ELASTIC

Results Comparison:

LS-DYNA results for the period of vibration related to this nonlinear spring-mass system are compared with S.P. Timoshenko and D.H. Young studies in *Vibration Problems in Engineering*, 1955 (pg. 141).

	Period of Vibration τ (s)
Timoshenko and Young [1955]	0.14470
Nonlinear spring-mass system	0.14380

This nodal displacement result and the computer period of vibration was generated by *DATABASE_NODOUT keyword (also see Figures 18.2 and 18.3).

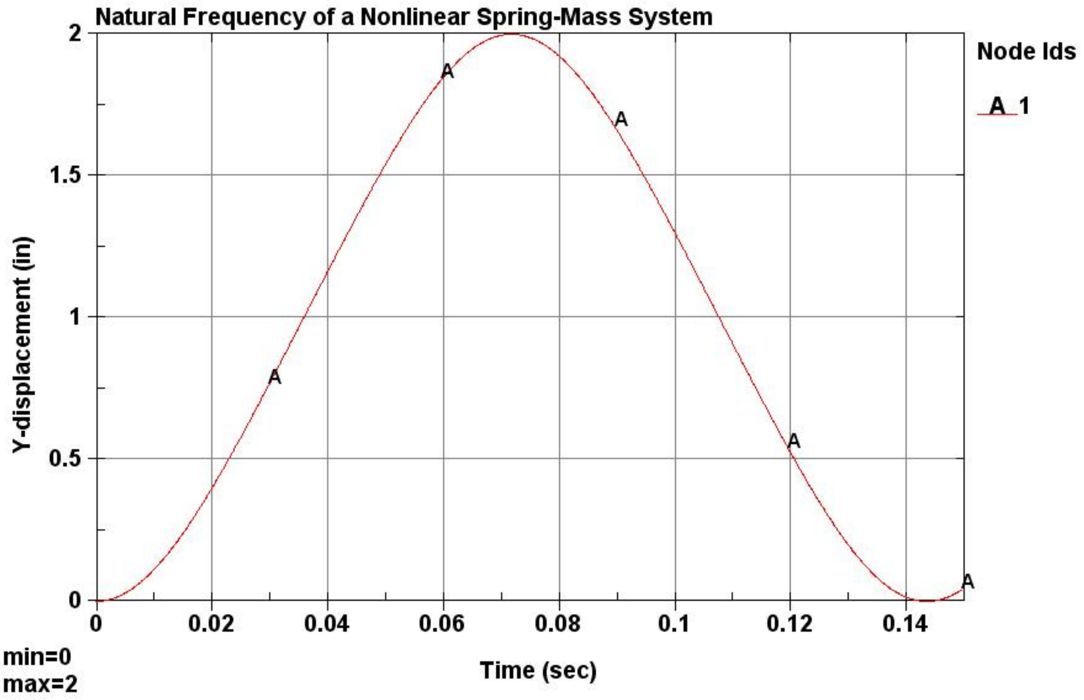

Figure 18.2 – Node 1 displacement U_Y .

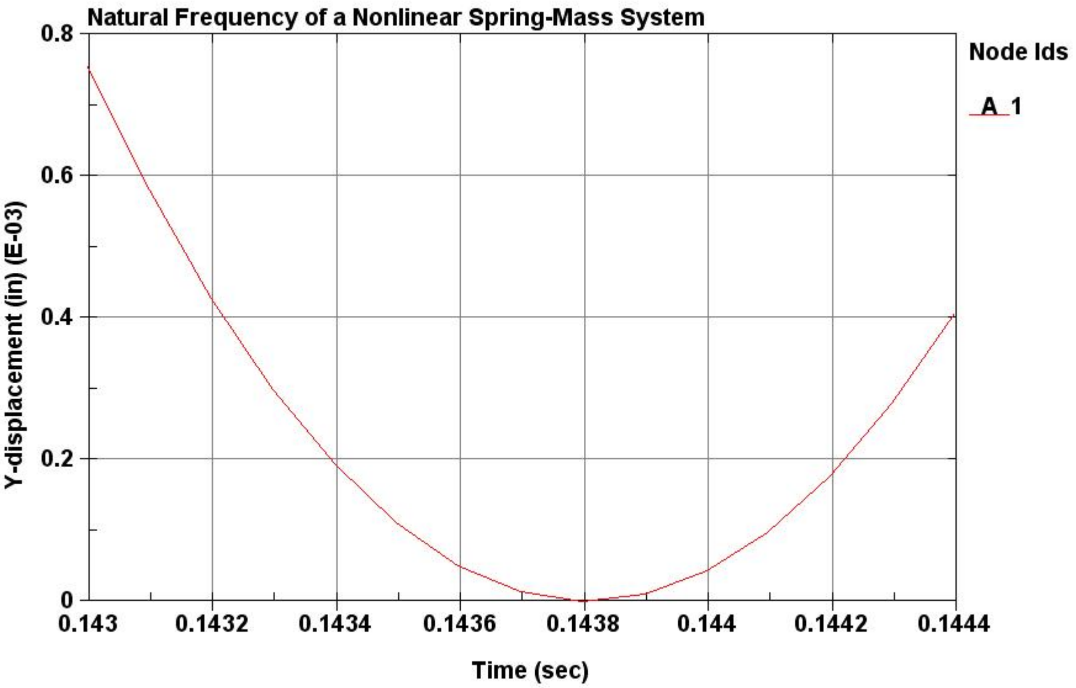

Figure 18.3 – Node 1 displacement U_Y (detailed time capture).

Input deck:

```
*KEYWORD
*TITLE
Natural Frequency of a Nonlinear Spring-Mass System
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  0.150000         0       0.0       0.0       0.0
*CONTROL_TIMESTEP
$#  dtinit    tssfac      isdo    tslimt     dt2ms      lctm     erode     ms1st
 1.000e-04 1.000e-04         0       0.0       0.0         0         0         0
$#  dt2msf   dt2mslc     imscl
       0.0         0         0
*DATABASE_NODOUT
$#      dt    binary
  0.000100
*DATABASE_BINARY_D3PLOT
$# dt/cycl   lcdt/nr      beam     npltc    psetid
  0.001000
*DATABASE_HISTORY_NODE
$#    nid1      nid2      nid3      nid4       ni5      nid6      nid7      nid8
         1
*PART
$# title
nonlinear elastic spring
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_DISCRETE
$#   secid       dro        kd        v0        cl        fd
         1         0       0.0       0.0       0.0       0.0
$#     cdl       tdl
       0.0       0.0
*MAT_SPRING_NONLINEAR_ELASTIC
$#     mid       lcd       lcr
         1         1
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo    dattyp
         1         0  1.000000  1.000000       0.0       0.0
$#                a1                  o1
                0.00              0.0000
                0.10              0.2040
                0.20              0.4320
                0.30              0.7080
                0.40              1.0240
                0.50              1.5000
                0.60              2.0640
                0.70              2.7720
                0.80              3.6480
                0.90              4.7160
                1.00              6.0000
*ELEMENT_DISCRETE
$#   eid     pid      n1      n2     vid               s      pf          offset
       1       1       1       2       0         1.00000       0         1.00000
*ELEMENT_MASS
$#   eid      id            mass     pid
       2       1      0.00258800
       3       2      0.00258800
*NODE
$#   nid               x               y               z      tc      rc
       1             0.0             0.0             0.0
       2             0.0      1.00000000             0.0
*BOUNDARY_SPC_NODE
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         2         0         1         1         1         1         1         1
*END
```

Notes:

1. A somewhat better comparison could be achieved with a more detailed representation of the nonlinear spring stiffness.

2. As an alternative, it is possible to model the nonlinear spring with a *BEAM_ELEMENT (with the option discrete) and *MAT_067/*MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM material behavior.

3. For simulations with linear stiffness, one would use the following implicit entries and perform a simple eigenvalue analysis:

```
*CONTROL_IMPLICIT_EIGENVALUE
$#    neig    center     lflag    lftend     rflag    rhtend    eigmth    shfscl
         3    11.000         0 -1.00e+29         0  1.00e+29         2       0.0
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         6  1.00e-04         2         1         2
```

The period could be obtained directly from the *eigout* results file generated by the *CONTROL_IMPLICIT_EIGENVALUE keyword as shown here:

```
Natural Frequency of a Nonlinear Spring-Mass System
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE        RADIANS        CYCLES        PERIOD
           1  -9.094947E-11   9.536743E-06   1.517820E-06   6.588397E+05
           2   4.547474E-12   2.132481E-06   3.393948E-07   2.946421E+06
           3   4.961360E+03   7.043692E+01   1.121038E+01   8.920301E-02
```

However, for this example, the spring stiffness is nonlinear, represented by a piecewise linear curve. LS-DYNA will make a stiffness, from two force-displacement pairs, to compute an eigenvalue. Which pairs used will depend on whether there is an initial offset or not (provided via *ELEMENT_DISCRETE). If there is zero initial offset, the first two force-displacement pairs are used; if there is an initial offset, the two pairs on either side of the offset would be used; if the offset and displacement value are equal, LS-DYNA uses this as the upper pair. Using this stiffness value will not yield a correct period of vibration.

It is not recommended to use the eigenvalue solver for nonlinear simulations

19. Buckling of a Axially Loaded Thin Walled Cylinder

Keywords:

```
*CONTROL_IMPLICIT_GENERAL
*CONTROL_IMPLICIT_SOLUTION
*CONTROL_IMPLICIT_BUCKLE
*CONTROL_IMPLICIT_EIGENVALUE
```

Description:

A cylinder is loaded with a uniform distributed (equal value to each node) line load of $P = 1000\,lbs$ (compressive) along the top edge. Determine the critical buckling load.

The lower end of the cylinder is clamped, i.e. fixed for all translational and rotational directions ($z = 0$: $U_X = U_Y = U_Z = R_X = R_Y = R_Z = 0$, the upper end of the cylinder is only fixed in x and y direction ($z = L$: $U_X = U_Y = 0$).

The finite element model is shown in Figure 19.1.

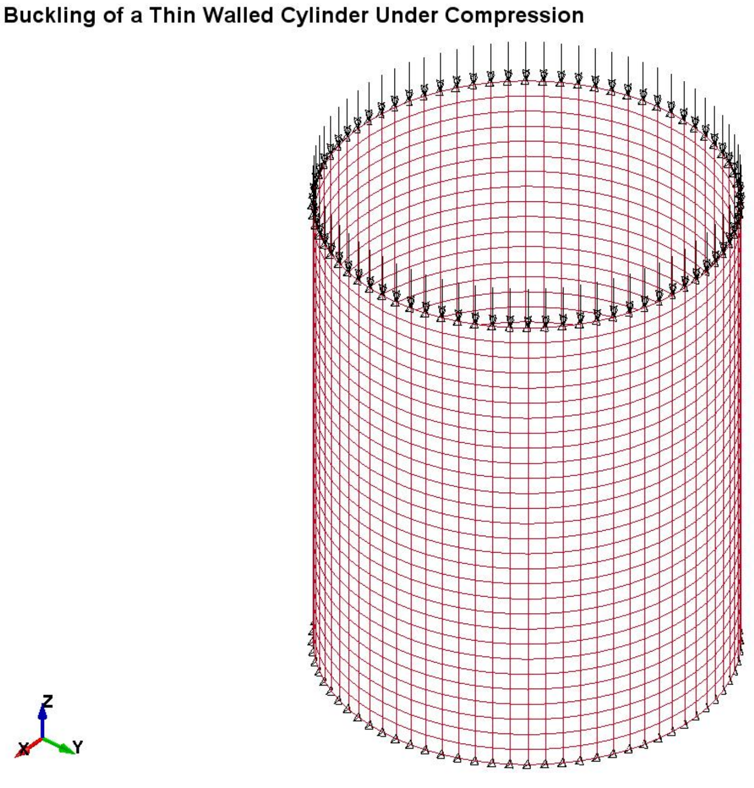

Figure 19.1 – Finite element model with applied axial load and boundary nodes (marked with□'s). There are 29 elements axially and 76 elements circumferentially.

Analysis Type:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Static	Force	Linear	Linear	-	Implicit	2-Nonlinear w/BFGS

Units:

lbf-s^2/in, in, s, lbf, psi, lbf-in (blob, inch, second, pound force, pound force/inch^2, pound force-inch)

Dimensional Data:

$L = 1.20 \times 10^2\ in$, $r_c = 4.8 \times 10^1\ in$, $t = 1.0 \times 10^{-1}\ in$

Material Data:

Mass Density	$\rho = 1.00 \times 10^{-2}\ lbf - s^2 / in^4$
Young's Modulus	$E = 1.00 \times 10^7\ lbf\ / in^2$
Poisson's Ratio	$\nu = 0.3$

Load:

Axial Load	$P = 1.00 \times 10^3\ lbs$

Element Types:

Belytschko-Tsay shell (elform=2)
S/R Hughes-Liu shell (elform=6)
Belytschko-Wong-Chiang shell (elform=10)
Fully integrated shell (elform=16)

Material Models:

*MAT_001 or *MAT_ELASTIC

Results Comparison:

LS-DYNA results for the critical buckling load of a thin walled cylinder under axial compression are compared with S.P. Timoshenko and J.M. Gere studies in *Theory of Elastic Stability*, 1961 (pg. 457).

	Critical Buckling Load P_{cr} (*lbf*)	**Critical Axial Stress σ_{cr} (*psi*)**
Timoshenko and Gere [1961]	3.8025×10^5	1.2608×10^4
Belytschko-Tsay shell (elform=2)	3.8763×10^5	1.2853×10^4
S/R Hughes-Liu shell (elform=6)	4.6226×10^5	1.5327×10^4
Belytschko-Wong-Chiang shell (elform=10)	3.8763×10^5	1.2853×10^4
Fully integrated shell (elform=16)	4.6086×10^5	1.5281×10^4

The analytical solution for this problem, from Timoshenko and Gere [1961], is:

$$\sigma_{cr}^{*} = E\frac{t}{R}\frac{1}{\sqrt{3\left(1-\nu^2\right)}} = 1.2608\times10^4\ psi$$

with a mode shape that is sinusoidal both axially and circumferentially.

The LS-DYNA critical load P_{cr} is computed from the first eigenvalue and the applied axial load:

$$P_{cr} = \lambda_1 P = 3.8762\times10^2 \times 1.0000\times10^3\ lbf = 3.8762\times10^5\ lbf$$

while the critical axial stress σ_{cr} is given by

$$\sigma_{cr} = \frac{P_{cr}}{A} = \frac{3.8763\times10^5\ lbf}{3.0159\times10^1\ in^2} = 1.2853\times10^4\ psi$$

This result, for the one point quadrature shell elements (elform=2 and elform=10), is in good agreement with the analytical solution.

The critical load and axial stress for the fully integrated shell elements (elform=6 and elform=16) is greater than the one point quadrature shell elements. The difference is not understood.

Eigenvalue Results:

From the *eigout* file, generated by the *CONTROL_IMPLICIT_BUCKLE keyword:

Belytschko-Tsay shell (elform=2):

```
Buckling of a Thin Walled Cylinder Under Compression
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1   3.876317E+02
           2   3.882852E+02
           3   3.882852E+02
```

S/R Hughes-Liu shell (elform=6)

```
Buckling of a Thin Walled Cylinder Under Compression
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1   4.622332E+02
           2   4.622523E+02
           3   4.700114E+02
```

Belytschko-Wong-Chiang shell (elform=10)

```
Buckling of a Thin Walled Cylinder Under Compression
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1   3.876317E+02
           2   3.882852E+02
           3   3.882852E+02
```

Fully integrated shell (elform=16)

```
Buckling of a Thin Walled Cylinder Under Compression
r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS        CYCLES        PERIOD
           1   4.608551E+02
           2   4.608564E+02
           3   4.681777E+02
```

The one point quadrature shell elements (elform =2 and elform=10) only provide the axial sinusoidal mode shape (10 half sine waves) as can be seen in Figure 19.2.

The fully integrated shell elements (elform=6 and elform=16) provide both the axial and circumferential sinusoidal mode shapes (2 half sine waves axially and 20 half sine waves circumferentially) as can be seen in Figure 19.3.

The eigenmodes for the one point quadrature elements and the fully integrated shell elements do not compare well. Again, this difference not understood.

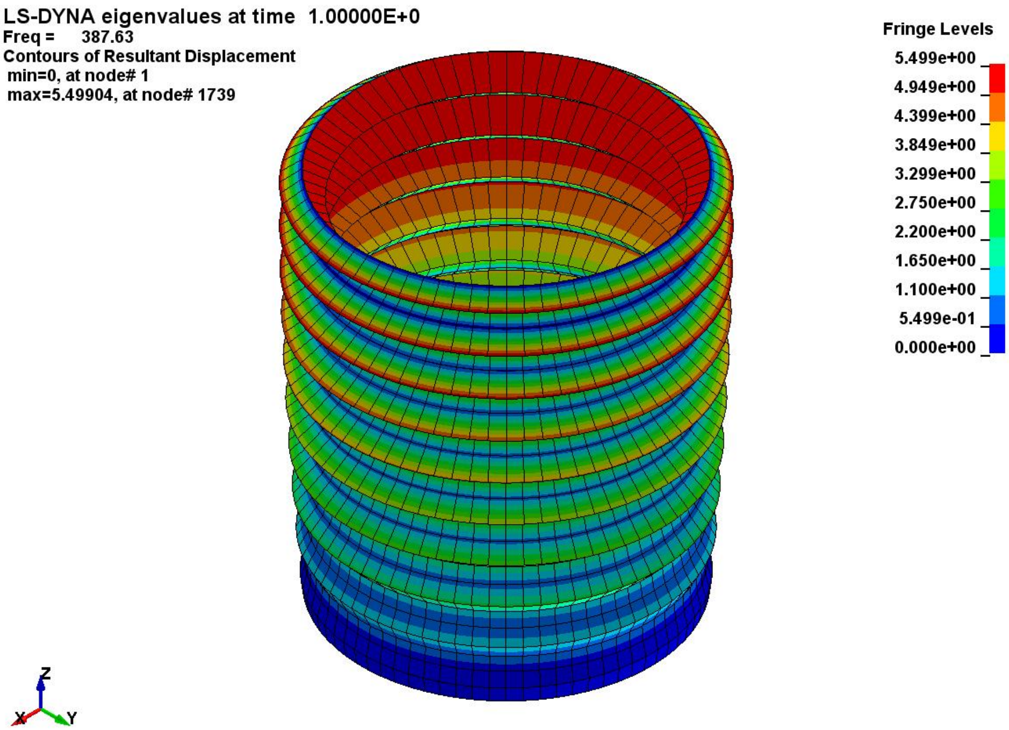

Figure 19.2 – First eigenmode with the 1000 lbf load applied (elform=10).

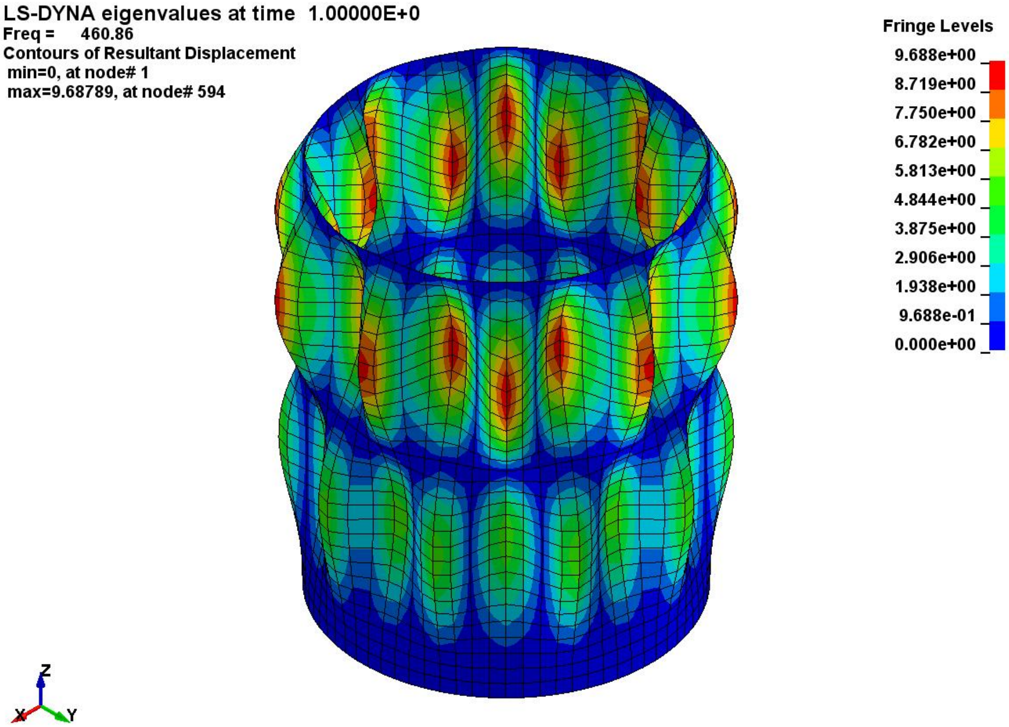

Figure 19.3 – First eigenmode with the 1000 lbf load applied (elform=16).

Input deck:

```
*KEYWORD
*TITLE
Buckling of a Thin Walled Cylinder Under Compression
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1  0.100000         2         1         2
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol  not used     lstol      rssf
         2        11        15  0.001000  0.010000       0.0  0.900000  1.000000
$#   dnorm    diverg     istif   nlprint
         2         1         1         2
$#  arcctl    arcdir    arclen    arcmth    arcdmp
         0         1       0.0         1         2
*CONTROL_IMPLICIT_BUCKLE
$#   nmode
         3
*CONTROL_IMPLICIT_EIGENVALUE
$#    neig    center     lflag    lftend     rflag    rhtend    eigmth    shfscl
                                                                           300.0
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  1.000000         0       0.0       0.0       0.0
*CONTROL_SHELL
$#  wrpang     esort     irnxx    istupd    theory       bwc     miter      proj
  20.00000         0         0         0         2         1         1         1
$# rotascl    intgrd    lamsht    cstyp6    tshell    nfail1    nfail4
       0.0         0
*DATABASE_BINARY_D3PLOT
$# dt/cycl
  0.010000
*PART
$# title
material type # 1  (Elastic)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1         0         1
*SECTION_SHELL
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
         1         2       0.0         0         1       0.0         0         1
$         1         6       0.0         0         1       0.0         0         1
$         1        10       0.0         0         1       0.0         0         1
$         1        16       0.0         0         1       0.0         0         1
$#      t1        t2        t3        t4      nloc     marea
  0.100000  0.100000  0.100000  0.100000         0       0.0
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1  0.0100001.0000e+07  0.300000       0.0       0.0       0.0
*HOURGLASS
$#    hgid       ihq        qm       ibq        q1        q2        qb        qw
         1         4       0.0         0       0.0       0.0       0.0       0.0
*SET_NODE_LIST_TITLE
bottom nodes
$#     sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nod4      nid5      nid6      nid7      nid8
         1         2         3         4         5         6         7         8
         9        10        11        12        13        14        15        16
        17        18        19        20        21        22        23        24
        25        26        27        28        29        30        31        32
        33        34        35        36        37        38        39        40
        41        42        43        44        45        46        47        48
        49        50        51        52        53        54        55        56
        57        58        59        60        61        62        63        64
        65        66        67        68        69        70        71        72
        73        74        75        76         0         0         0         0
*SET_NODE_LIST_TITLE
top nodes
$#     sid       da1       da2       da3       da4    solver
         2       0.0       0.0       0.0       0.0
```

```
$#    nid1      nid2      nid3      nod4      nid5      nid6      nid7      nid8
      2205      2206      2207      2208      2209      2210      2211      2212
      2213      2214      2215      2216      2217      2218      2219      2220
      2221      2222      2223      2224      2225      2226      2227      2228
      2229      2230      2231      2232      2233      2234      2235      2236
      2237      2238      2239      2240      2241      2242      2243      2244
      2245      2246      2247      2248      2249      2250      2251      2252
      2253      2254      2255      2256      2257      2258      2259      2260
      2261      2262      2263      2264      2265      2266      2267      2268
      2269      2270      2271      2272      2273      2274      2275      2276
      2277      2278      2279      2280         0         0         0         0
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         1         1         1         1         1         1
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         2         0         1         1         0         0         0         0
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo     dattyp
         1         0       0.0       0.0       0.0       0.0
$#                a1                  o1
                 0.0                 0.0
          1.00000000         13.15789474
*LOAD_NODE_SET
$#    nsid       dof      lcid        sf       cid        m1        m2        m3
         2         3         1 -1.000000
*ELEMENT_SHELL
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1       2      78      77       0       0       0       0

    2204       1    2204    2129    2205    2280       0       0       0       0
*NODE
$#   nid               x               y               z      tc      rc
       1           0.000       48.000000           0.000       0       0

    2280        3.963804       47.836056       120.00000       0       0
*END
```

Notes:

1. The keyword entry *CONTROL_IMPLICIT_BUCKLE allows for buckling analysis at the end of the static implicit simulation.

2. The fully integrated and one point quadrature shell elements are formulated for nonlinear analysis. Although this analysis is linear, it was solved with a nonlinear solution method to demonstrate the use of these elements.

Mesh Convergence Study:

Seven different mesh refinements were studied for this simulation:

original mesh 29 axial by 76 circumferential elements (Figure 19.4a)

- 4.1379×10^{0} *in* by 3.9671×10^{0} *in*

1st mesh refinement 58 axial by 152 circumferential elements

- 2.0690×10^{0} *in* by 1.9840×10^{0} *in*

2nd mesh refinement 87 axial by 228 circumferential elements (Figure 19.4b)

- 1.3793×10^{0} *in* by 1.3227×10^{0} *in*

3rd mesh refinement 116 axial by 304 circumferential elements

- 1.0345×10^{0} *in* by 0.9921×10^{0} *in*

4th mesh refinement 145 axial by 380 circumferential elements (Figure 19.4c)

- 0.8276×10^{0} *in* by 0.7937×10^{0} *in*

5th mesh refinement 174 axial by 456 circumferential elements

- 0.6897×10^{0} *in* by 0.6614×10^{0} *in*

6th mesh refinement 203 axial by 532 circumferential elements

- 0.5911×10^{0} *in* by 0.5669×10^{0} *in*

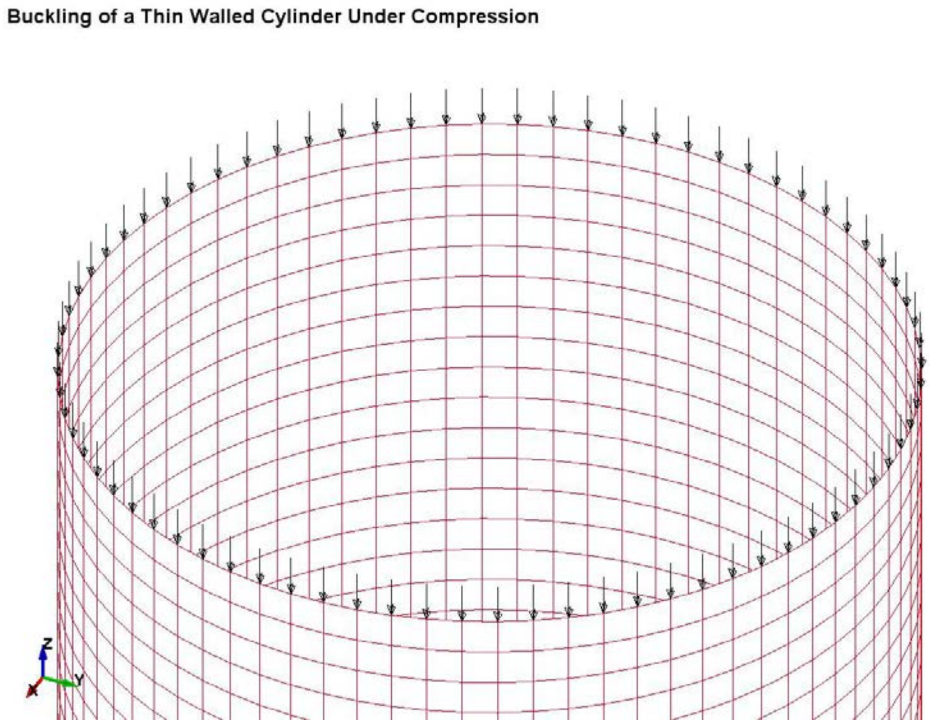

Figure 19.4a - Finite element model for the original mesh (29 axial by 76 circumferential) discretization with applied axial load.

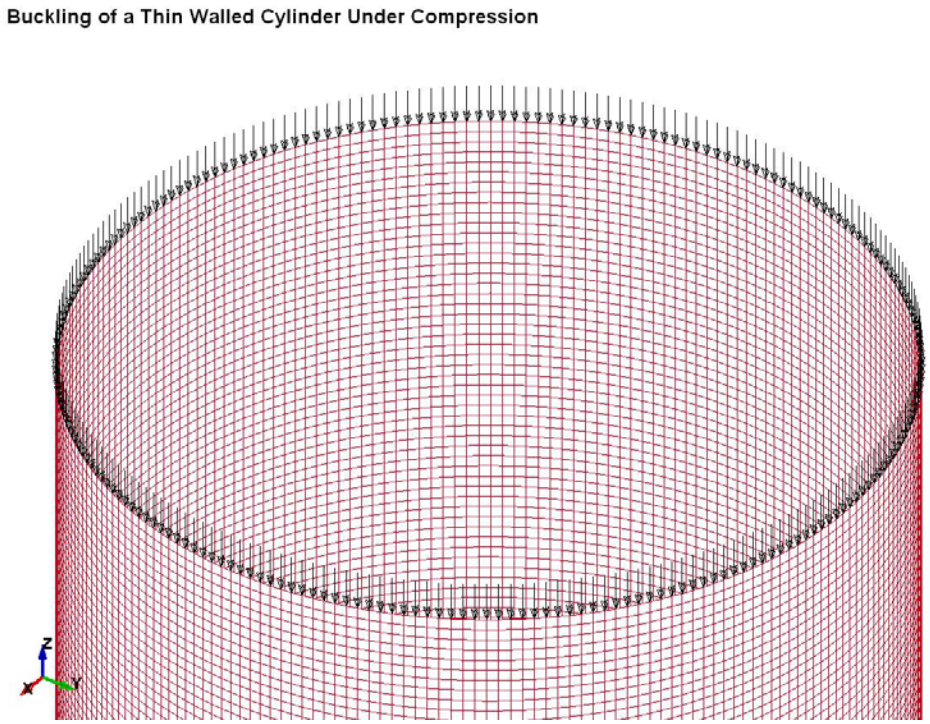

Figure 19.4b - Finite element model for the 2nd mesh refinement (87 axial by 228 circumferential) discretization with applied axial load.

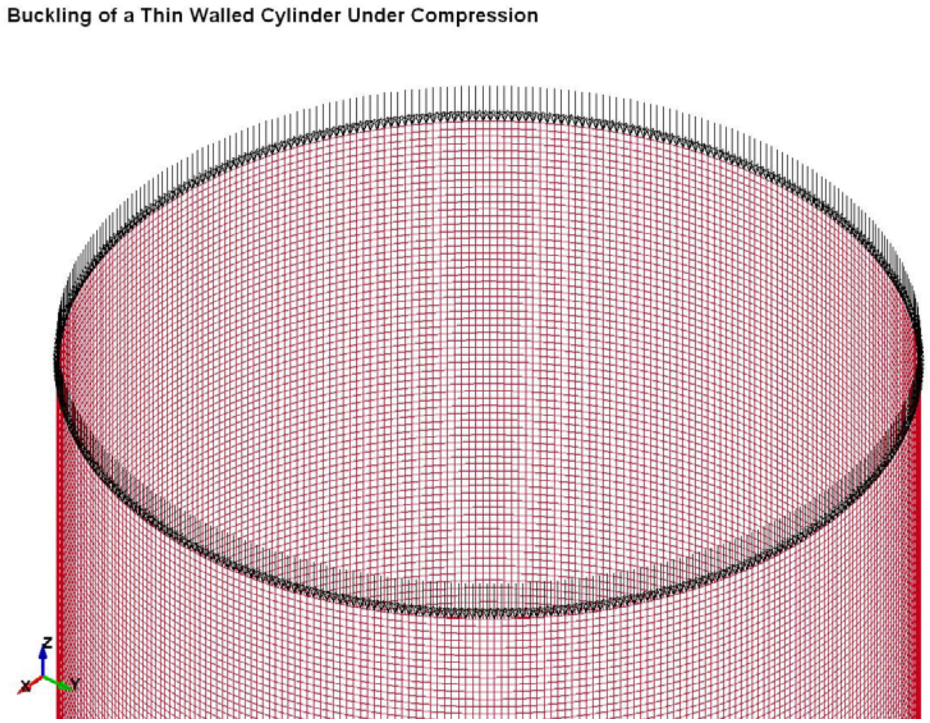

Figure 19.4c - Finite element model for the 4th mesh refinement (145 axial by 380 circumferential) discretization with applied axial load.

Mesh Convergence Results Comparison:

LS-DYNA results for the critical buckling load of a thin walled cylinder under axial compression are compared for seven different mesh discretizations.

analytical solution - 3.8025×10^{5} ***lbf***	**Belytschko-Wong-Chiang (elform=10)**	**fully integrated shell (elform=16)**
original mesh (29 axial by 76 circumferential elements)	3.8763×10^{5}	4.6086×10^{5}
1st mesh refinement (58 axial by 152 circumferential elements)	3.8115×10^{5}	4.0616×10^{5}
2nd mesh refinement (87 axial by 228 circumferential elements)	3.8052×10^{5}	3.9313×10^{5}
3rd mesh refinement (116 axial by 304 circumferential elements)	3.8033×10^{5}	3.8758×10^{5}
4th mesh refinement (145 axial by 380 circumferential elements)	3.8026×10^{5}	3.8480×10^{5}
5th mesh refinement (174 axial by 456 circumferential elements)	3.8022×10^{5}	3.8312×10^{5}
6th mesh refinement (203 axial by 532 circumferential elements)	3.8019×10^{5}	3.8207×10^{5}

For the Belytschko-Wong-Chiang (one point quadrature) shell element (elform=10), the 29 axial by 76 circumferential element mesh (original) critical buckling load result was in good agreement with the analytical solution. This element/mesh converged rapidly. The 29 axial by 76 circumferential element mesh only differed by less than 2% from the analytical solution while the 203 axial by 532 circumferential element mesh differed by less than 0.02%.

For the fully integrated shell element (elform=16), the 29 axial by 76 circumferential element mesh (original) critical buckling load result was greater (over 21%) than the analytical solution. It is not known why. Doubling the number of elements axially and circumferentially reduces the critical buckling by about 10%; however, still not in good agreement with the analytical solution, especially considering the level of mesh refinement. Two further mesh refinements (116 axial by 304 circumferential element elements) were required to reach a similar good agreement (the 2% difference) with the one point quadrature shell element (elform=10) and the original mesh discretization. The 203 axial by 532 circumferential element mesh refinement for the fully integrated shell differed by less than 0.5%.

The one point quadrature shell element (elform=10) only provides the axial sinusoidal mode shape (Figures 19.5a and 19.5b):

- 10 half sine waves in 29 axial by 76 circumferential element mesh (original),
- 13 half sine waves in 58 axial by 152 circumferential element mesh,
- 14 half sine waves in 87 axial by 228 circumferential element mesh,
- 15 half sine waves in 116 axial by 304 circumferential element mesh,
- 15 half sine waves in 145 axial by 380 circumferential element mesh,
- 16 half sine waves in 174 axial by 456 circumferential element mesh,
- 16 half sine waves in 203 axial by 532 circumferential element mesh,

estimated from the eigenmode figures. The number of half sine waves is the number of buckles. It is not known why this element formulation only provides the axial sinusoidal modes.

Figure 19.5a - First eigenmode with the 1000 lbf load applied for the six refined mesh discretizations (elform=10) - no contouring.

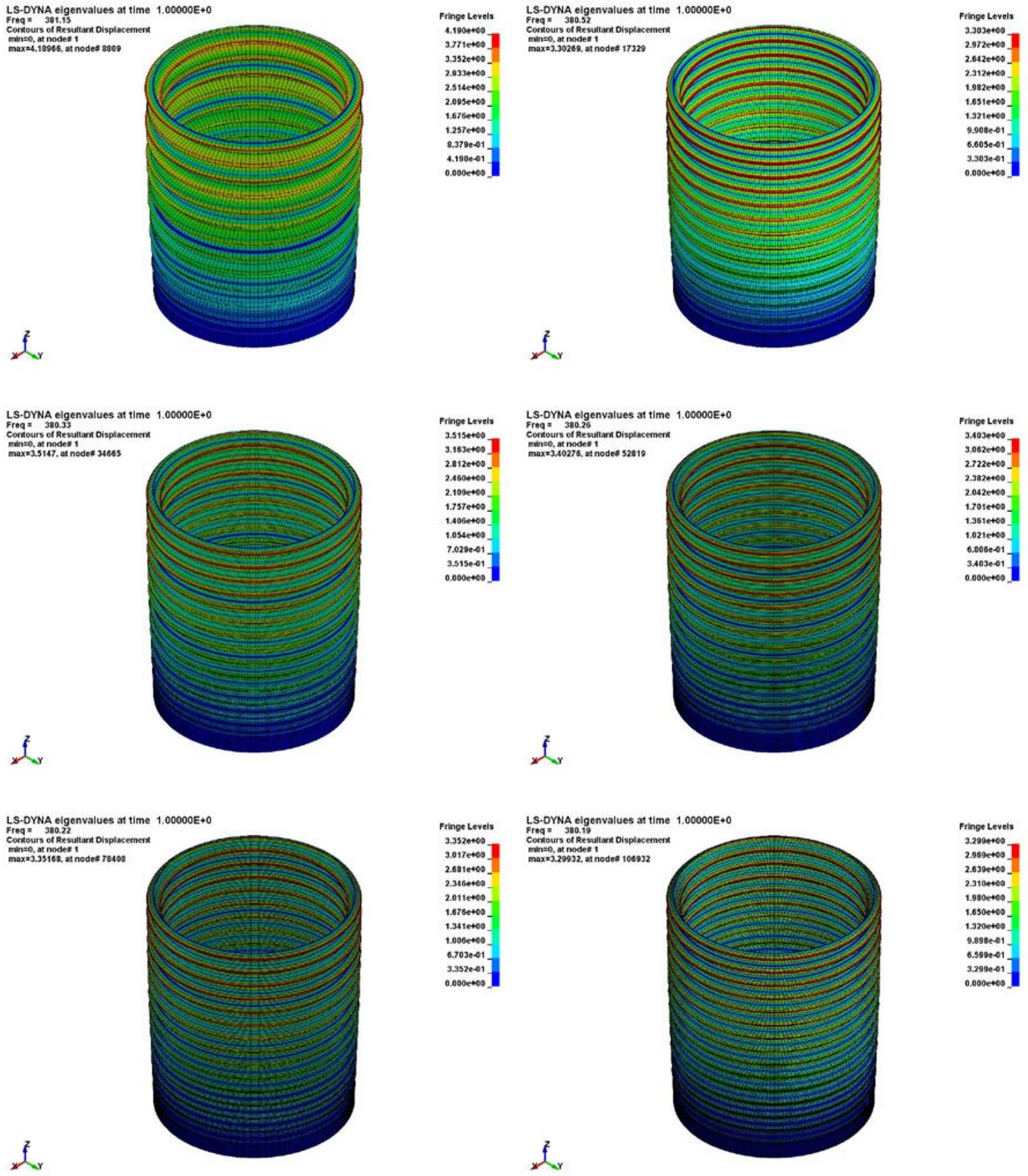

Figure 19.5b - First eigenmode with the 1000 lbf load applied for the six refined mesh discretizations (elform=10) - resultant displacement contouring.

The fully integrated shell element (elform=16) provides both the axial and circumferential sinusoidal mode shapes (Figures 19.6a and 19.6b):

- 2 half sine waves axially and 20 half sine waves circumferentially in 29 axial by 76 circumferential element mesh (original),
- 3 half sine waves axially and 24 half sine waves circumferentially in 58 axial by 152 circumferential element mesh,
- 4 half sine waves axially and 28 half sine waves circumferentially in 87 axial by 228 circumferential element mesh,
- 5 half sine waves axially and 30 half sine waves circumferentially in 116 axial by 304 circumferential element mesh,
- 6 half sine waves axially and 32 half sine waves circumferentially in 145 axial by 380 circumferential element mesh,
- 6 half sine waves axially and 32 half sine waves circumferentially in 174 axial by 456 circumferential element mesh,
- 7 half sine waves axially and 34 half sine waves circumferentially in 203 axial by 532 circumferential element mesh.

estimated from the eigenmode figures.

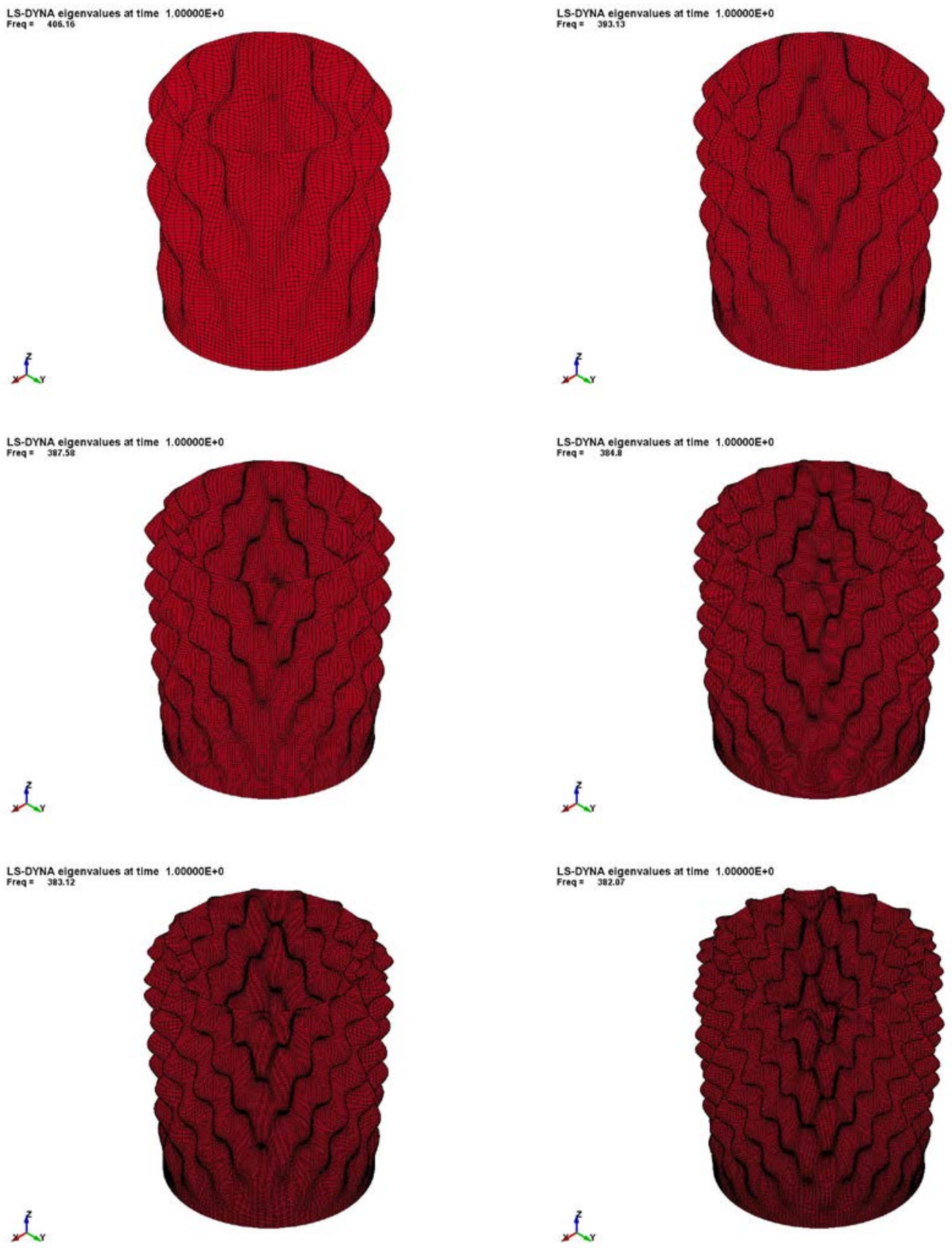

Figure 19.6a - First eigenmode with the 1000 lbf load applied for the six refined mesh discretizations (elform=16) - no contouring.

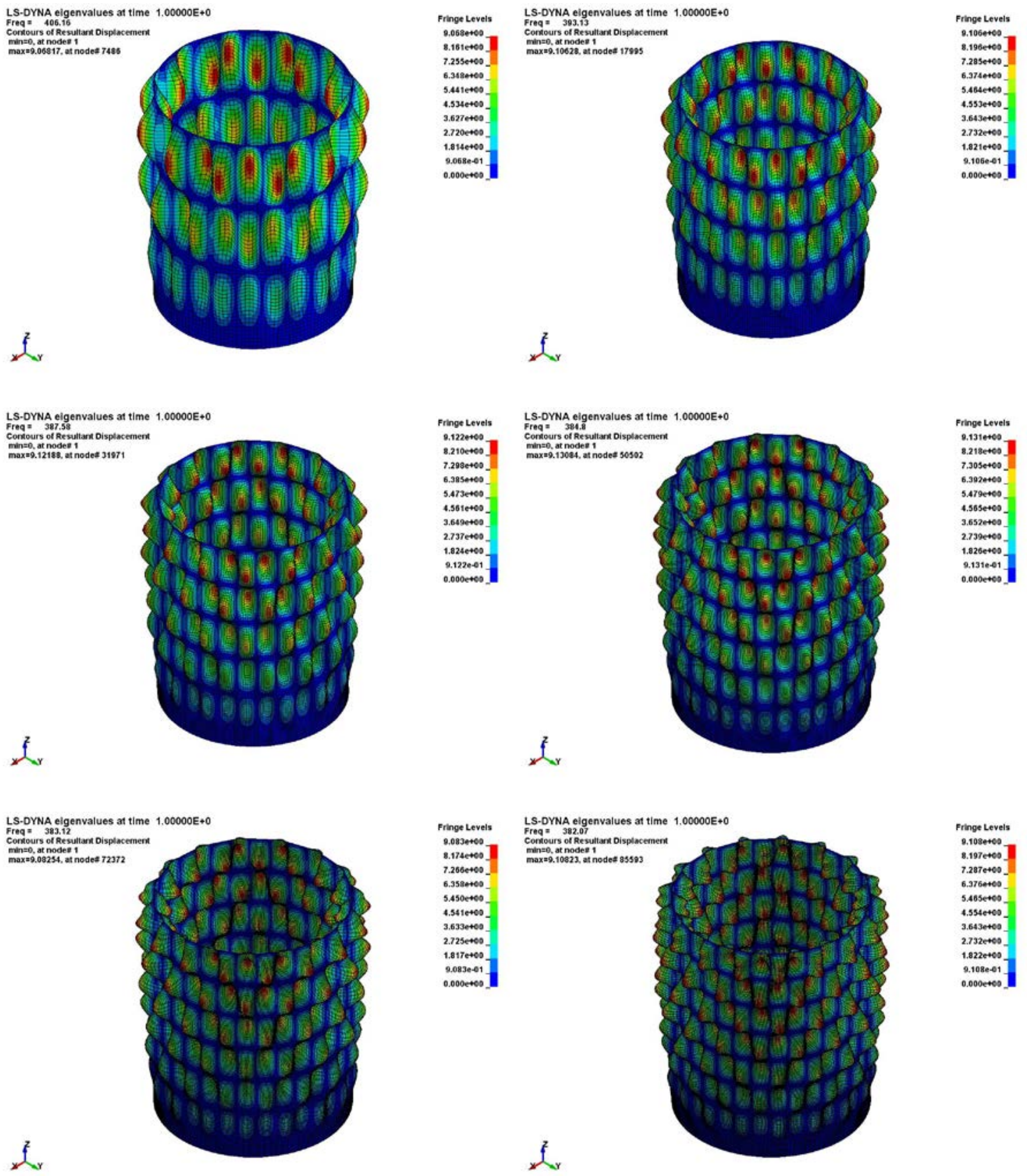

Figure 19.6b - First eigenmode with the 1000 lbf load applied for the six refined mesh discretizations (elform=16) - resultant displacement contouring.

Notes:

1. Solution problems may exist:
   - because LS-DYNA buckling solutions assume the first buckling mode will be around 1.0 and/or
   - if numerous eigenvalues are clustered around that smallest bucking frequencies.

   For the refined meshes, for this problem, it was necessary to override the internal heuristic for picking a starting point for Lanczos shift strategy, which is the initial Eigen frequency shift. In these cases, the user must specify the initial shift via the parameter shfscl. shfscl should be close to the first nonzero frequency.

20. Membrane with a Hot Spot

Keywords:

`*LOAD_THERMAL_LOAD_CURVE`
`*MAT_ELASTIC_PLASTIC_THERMAL`

Description:

This benchmark analyzes the behavior of shell elements subjected to a thermal load. Two distinct regions are modeled: the central hot-spot region (radius equal to *r*), subjected to the thermal strain $\varepsilon = \alpha T$, and the rest of the plate, which is at constant temperature with $\varepsilon = 0.0$. Due to symmetry, only ¼ of the plate (side lengths 2*L* and thickness *t*) is modeled (Figures 20.1a and 20.1b).

The material defining the hot spot is *MAT_ELASTIC_PLASTIC_THERMAL (*MAT_004), sensitive to temperature changes. The rest of the plate is defined with material *MAT_ELASTIC (*MAT_001).

The temperature is uniformly applied to the whole model by means of the *LOAD_ THERMAL_LOAD_CURVE keyword.

Determine the y-component of the stress tensor along the edge y=0, just outside the hot spot. A fine mesh is required in the region of interest.

To possibly achieve better accuracy, the value at the integration point is considered (values at nodes are interpolated from neighboring integration points).

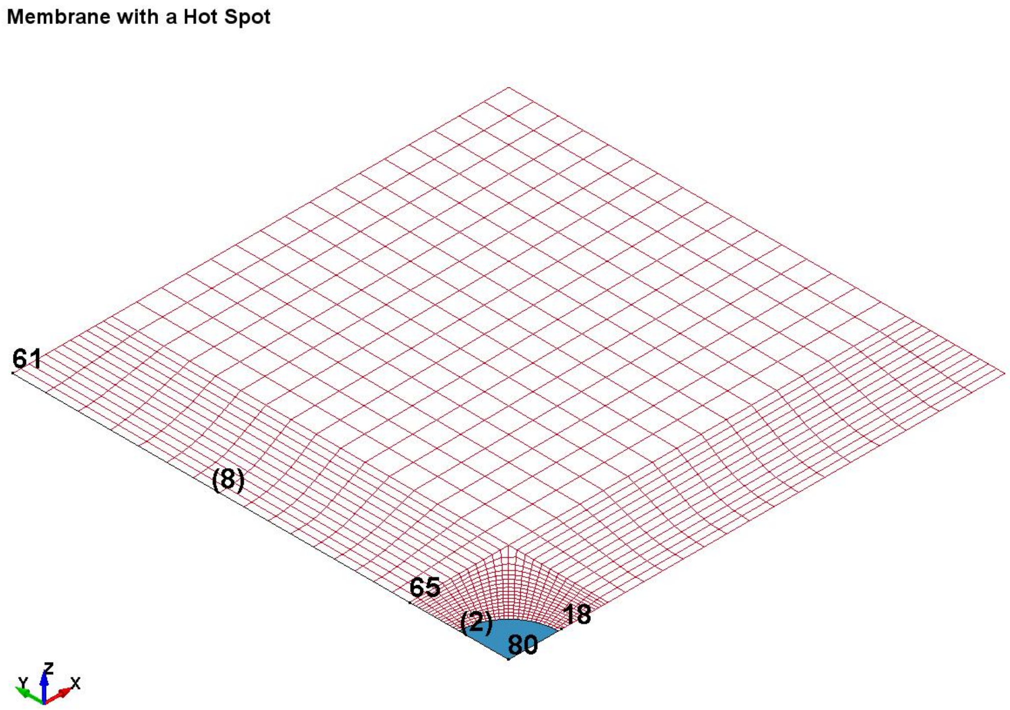

Figure 20.1a - Finite element model (¼ symmetry) with selected nodes and dimensions identified.

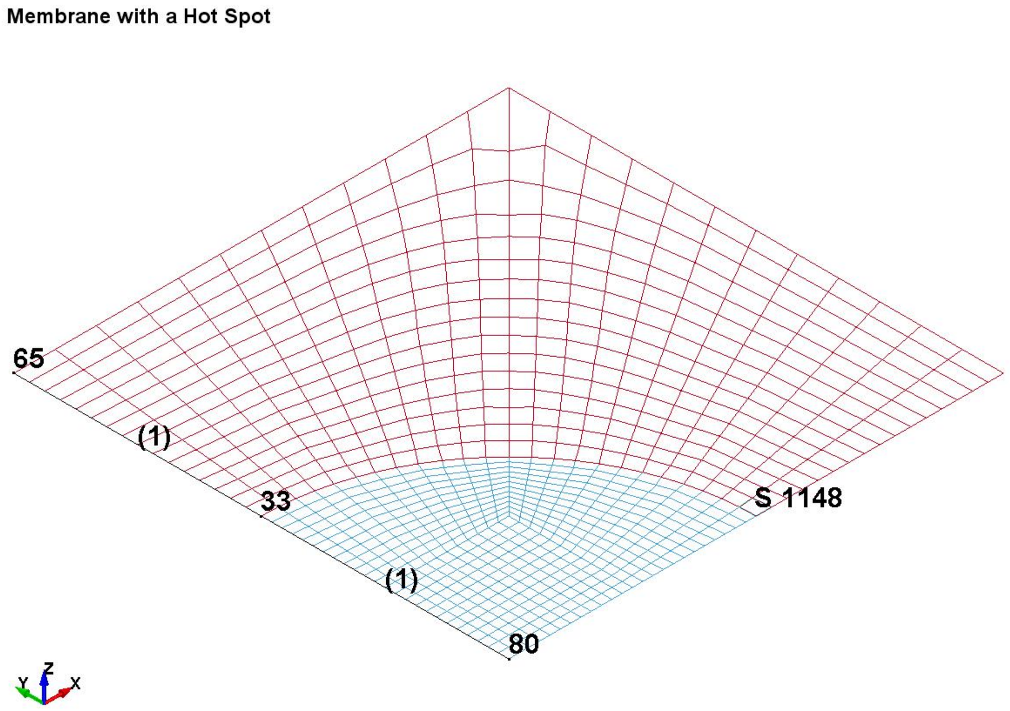

Figure 20.1b - Finite element model of hot spot (blue region) and refined surrounding mesh, with selected nodes, element, and dimensions identified.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Static	Thermal	Linear	Linear	-	Implicit	1-Linear

Units:

ton ,mm, s, N, MPa, N-m, $^{\circ}C$ *(tonne, millmeter, second, Newton, MegaPascal, Newton-millimeter, degree Centigrade)*

Dimensional Data:

$L = 10.0\ mm$, $r = 1.00\ mm$, $t = 1.00\ mm$

Material Data:

Young's Modulus	$E = 1.00 \times 10^{5}\ MPa$
Poisson's Ratio	$\nu = 0.3$
Linear Expansion	$\alpha = 1.00 \times 10^{-5}\ mm/mm/^{\circ}C$

Load:

Thermal	$T = 0.0$ varied linearly to 100 $^{\circ}C$

Element Types:

Fully integrated shell (elform=16)

Material Models:

*MAT_001 or *MAT_ELASTIC

*MAT_004 or *MAT_ELASTIC_PLASTIC_THERMAL

Results Comparison:

LS-DYNA global stress σ_{yy} at point just outside the hot spot (Node 18) is compared with *NAFEMS Background to Benchmark*, Test T1.

Reference Condition - Point Just Outside Hot Spot (Node 18)	**Global Stress - σ_{yy} (M*Pa*)**
NAFEMS Benchmark Test T1	5.0000×10^1
Element 1148 (average value)	4.7528×10^1
First in-plane integration point (2x2 quadrature) - element 1148	4.5476×10^1
Node 18	4.3974×10^1

The global stress σ_{YY} results were generated from *DATABASE_ELOUT (*elout* file) and *DATABASE_EXTENT_BINARY (*eloutdet* file provides detailed element output at integration points and connectivity nodes) keyword entries.

You can set intout=stress or intout=all (*DATABASE_EXTENT_BINARY) and have stresses output for all the integration points to a file called *eloutdet* (*DATABASE_ELOUT governs the output interval and *DATABASE_HISTORY_SHELL governs which elements are output). Setting nodout=stress or nodout=all all in *DATABASE_EXTENT_BINARY will write the extrapolated nodal stresses to *eloutdet*.

LS-DYNA stress and strain outputs correspond to integration point locations. Stress at a node is an artifact of the post-processor and represents an average of the surrounding integration point stresses (the value will likely be different with different postprocessors).

Shell element stresses are reported at through-thickness integration points. The location of those integration points depends on the number of integration points and the type of integration rule used, e. g., Gaussian, Lobatto, trapezoidal, user-defined rule (see nip and qr/irid in *SECTION_SHELL). Fully-integrated shell formulations have 4 in-plane integration points at each through-thickness location. For these formulations, the 4 values of each stress component are averaged before being written to *elout* (except for the case of linear analysis when nsolvr=1 in *CONTROL_IMPLICIT_SOLUTION, in which case all 4 stress components are written to *elout*).

Shell element stresses can be shown in the global, element, or material coordinate system. By default, shell element stresses/strains written to *d3plot* are global; shell stresses/strains written to *elout* are in the element local coordinate system (except for the case of linear analysis when nsolvr=1 in *CONTROL_IMPLICIT_SOLUTION, in which case stresses are in the global system). Shell element stresses/strains from *d3plot* are converted by LS-PrePost to the shell element coordinate system.

Even with this fine mesh in the region of interest, the large gradient temperature profile makes it difficult to capture the global stress σ_{YY} along the line of symmetry. The average global stress of the element (Figure 20.2) provides the best comparative value (~5% difference), a few percent better than the nearest element integration point (~9% difference) and the extrapolated nodal (~12% difference) results.

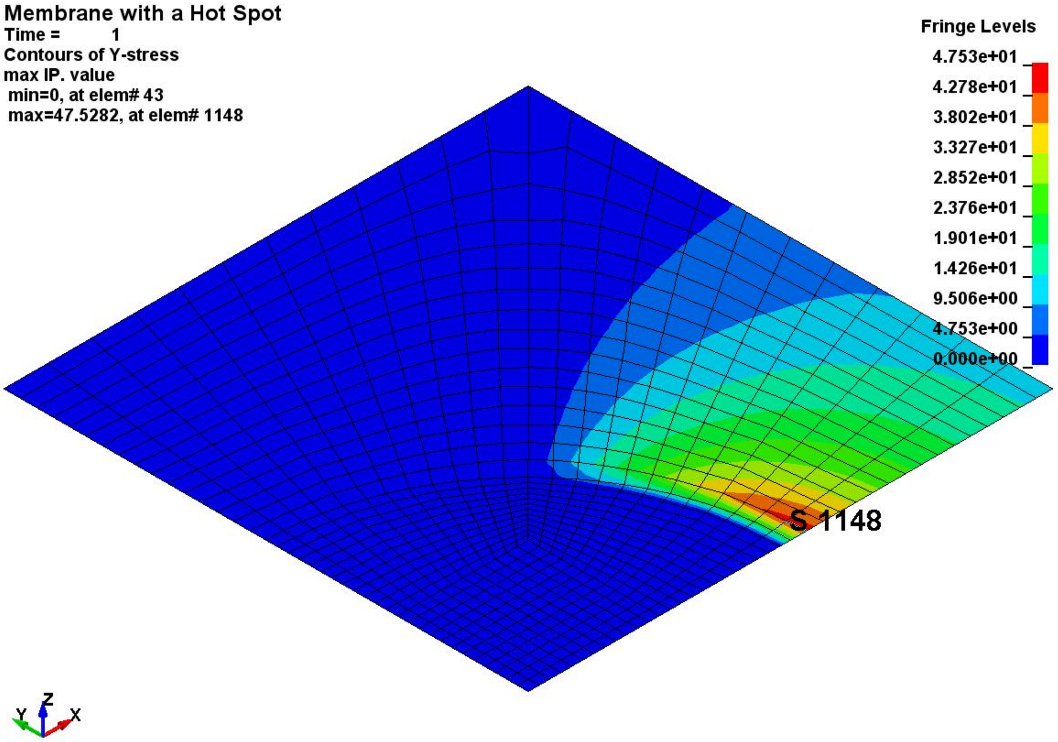

Figure 20.2 -Contour plot of global stress σ_{yy}. Maximum value at element 1148.

Input deck:

```
*KEYWORD
*TITLE
Membrane with a Hot Spot
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form    zer0_v
         1  0.100000         2         1         2         0         0         0
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
         1        11        15  0.001000  0.010000   1.0e+10  0.900000  1.000000
$#   dnorm    diverg     istif   nlprint    nlnorm    d3itcl     cpchk
         2         1         1         0         2         0         0
$#  arcctl    arcdir    arclen    arcmth    arcdmp    arcpsi    arcalf    arctim
         0         0       0.0         1         2       0.0       0.0       0.0
*CONTROL_SHELL
$#  wrpang     esort     irnxx    istupd    theory       bwc     miter      proj
  20.00000         0        -1         0        16         2         1         0
$# rotascl    intgrd    lamsht    cstyp6    tshell
  1.000000         0         0         1         0
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  1.000000         0       0.0       0.0       0.0
*DATABASE_EXTENT_BINARY
```

```
$#   neiph     neips    maxint    strflg    sigflg    epsflg     rtflg    engflg
         0         0         4         1         1         1         1         1
$#  cmpflg    ieverp    beamip     dcomp      shge     stssz    n3thdt   ialemat

$# nintsld   pkp_sen      sclp     hydro     msscl     therm    intout    nodout
         1               1.0                                  stress    stress
*DATABASE_ELOUT
$# dt/cycl
  0.100000
*DATABASE_HISTORY_SHELL
$#    eid1      eid2      eid3      eid4       ei5      eid6      eid7      eid8
      1148
*DATABASE_GLSTAT
$# dt/cycl
  0.100000
*DATABASE_MATSUM
$# dt/cycl
  0.100000
*DATABASE_BINARY_D3PLOT
$# dt/cycl
  0.100000
*PART
$# title
Part          1 for Mat         1 and Elem Type        16
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_SHELL
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
         1        16  0.830000         1         1       0.0         0         1
$#      t1        t2        t3        t4      nloc     marea
  1.000000  1.000000  1.000000  1.000000         0       0.0
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1          1.0000e+05  0.300000       0.0       0.0       0.0
*PART
$# title
Part          2 for Mat         2 and Elem Type        16
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         2         2         2
*SECTION_SHELL
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
         2        16  0.830000         1         1       0.0         0         1
$#      t1        t2        t3        t4      nloc     marea
  1.000000  1.000000  1.000000  1.000000         0       0.0
*MAT_ELASTIC_PLASTIC_THERMAL
$#     mid        ro
         2
$#      t1        t2        t3        t4        t5        t6        t7        t8
       0.0  1000.000       0.0       0.0       0.0       0.0       0.0       0.0
$#      e1        e2        e3        e4        e5        e6        e7        e8
 1.000e+05 1.000e+05       0.0       0.0       0.0       0.0       0.0       0.0
$#     pr1       pr2       pr3       pr4       pr5       pr6       pr7       pr8
  0.300000  0.300000       0.0       0.0       0.0       0.0       0.0       0.0
$#  alpha1    alpha2    alpha3    alpha4    alpha5    alpha6    alpha7    alpha8
 1.000e-05 1.000e-05       0.0       0.0       0.0       0.0       0.0       0.0
$#   sigy1     sigy2     sigy3     sigy4     sigy5     sigy6     sigy7     sigy8
       0.0       0.0       0.0       0.0       0.0       0.0       0.0       0.0
$#   etan1     etan2     etan3     etan4     etan5     etan6     etan7     etan8
       0.0       0.0       0.0       0.0       0.0       0.0       0.0       0.0
*ELEMENT_SHELL
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       2     344     368     441     345

    1456       2     434     432     433     453
*NODE
$#   nid               x               y               z      tc      rc
       1      1.00000000             0.0             0.0

    1541      2.98852444      7.92062616             0.0
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
```

```
         3         0         0         1         1         1         1         1
         4         0         1         0         1         1         1         1
         5         0         1         1         1         1         1         1
         6         0         0         0         1         1         1         0
*SET_NODE_LIST_TITLE
xsymm
$#     sid       da1       da2       da3       da4    solver
         3       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
        19        20       181       182       183       185       323       324
       325       326       327       328       330       332        17       306
       304       302       301       300       299       297       296       172
       170       169       167        48        47        46        45        18
         1        11        12       186       187       188       333       334
       335       336       337       338         8       272       270       268
       267       265       264       155       153       151        82        81
*SET_NODE_LIST_TITLE
ysymm
$#     sid       da1       da2       da3       da4    solver
         4       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
        36        83        84       147       150       154       257       258
       261       262       269       271       316       315       312       311
       308       307       177       175       173        38        37        33
        89        90        91       130       135       137       140       221
       224       231       232       235       236       239       242        65
       282       279       278       277       276       275       274       273
       160       158       157       156        68        67        66        61
*SET_NODE_LIST_TITLE
xy
$#     sid       da1       da2       da3       da4    solver
         5       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
        80
*SET_NODE_LIST_TITLE
whole
$#     sid       da1       da2       da3       da4    solver
         6       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1         2         3         4         5         6         7         8

      1537      1538      1539      1540      1541
*LOAD_THERMAL_LOAD_CURVE
$#    lcid    lciddr
         1         0
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo    dattyp
         1         0  1.000000  1.000000       0.0       0.0
$#                a1                  o1
                 0.0                 0.0
          1.00000000         100.0000000
*END
```

Notes:

1. The fully integrated and one point quadrature shell elements are formulated for non-linear analysis. Although this analysis is linear, it could have been solved with a non-linear solution method (nsolvr=2) which provides some slightly different stress output options (see Notes 2 and 3). This was done for this simulation and it was found to yield results with small differences from the linear analysis (nsolvr=1).

21. 1D Heat Transfer with Radiation

Keywords:

```
*CONTROL_SOLUTION
*CONTROL_THERMAL_SOLVER
*CONTROL_THERMAL_NONLINEAR
*CONTROL_THERMAL_TIMESTEP
*BOUNDARY_TEMPERATURE_SET
*BOUNDARY_RADIATION_SET
*MAT_THERMAL_ISOTROPIC
```

Description:

A 0.10 *m* long bar (L_z), with square 0.01 m (L_x) x 0.01 m (L_y) cross-section (Figure 21.1), radiates (steady state) to an ambient temperature of $T = 300^\circ K$ at one end (node 11). The other end (node 1) is maintained at constant temperature $T = 1000\,^\circ K$. The bar is perfectly insulated along its length. There is zero internal heat generation.

Find the temperature at node 33 (x=0.000 *m*, y=0.010 *m*, z=0.100 *m*).

The bar is meshed with 40 elements: ten elements along the length and four elements in the cross section.

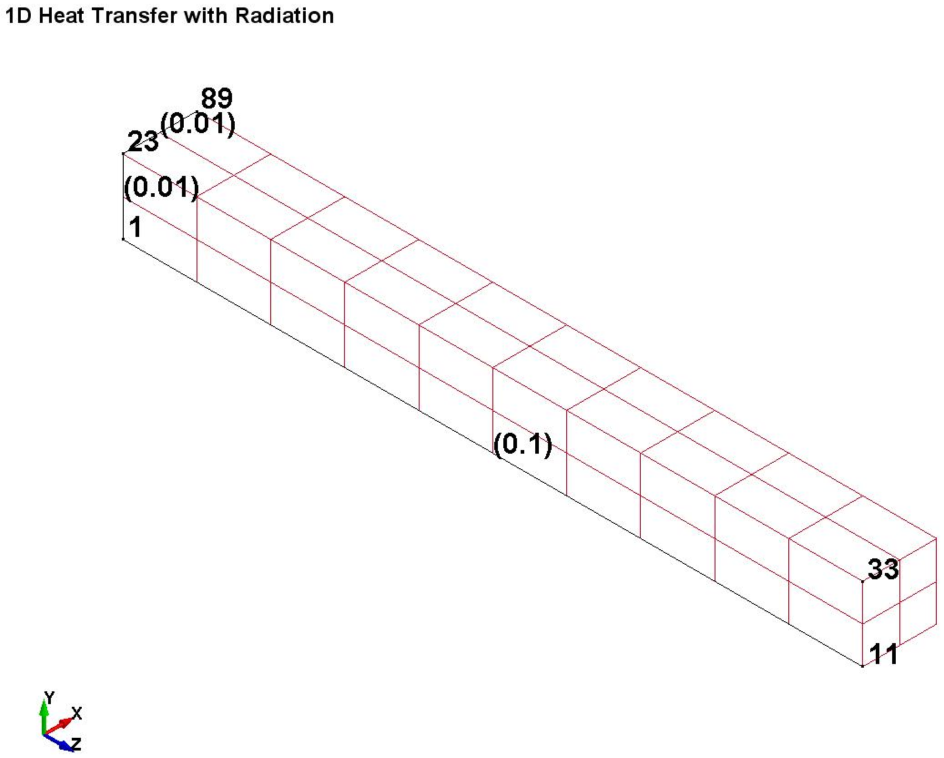

Figure 21.1 - Finite element model with selected nodes and dimensions identified.

Analysis Summary:

Dim.	**Type**	**Load**	**Material**	**Geometry**	**Contact**	**Solver**	**Solution Method**
3D	Steady State	Thermal	Linear	Linear	-	Thermal Non-Linear	3-Diagonal scaled conjugate gradient

Units:

kg, m, s, N, Pa, N-m, $^{\circ}C$ *(kilogram, meter, second, Newton, Pascal, Newton-meter, degree Centigrade) - Joule* (*J*) is a *N-m, Watt* (*W*) is a *J/s,* $1^{\circ}\,\Delta C = 1^{\circ}\,\Delta K$

Dimensional Data:

$L_x = 0.01\ m$, $L_y = 0.01\ m$, $L_z = 0.10\ m$

Material Data:

Mass Density	$\rho = 7.850 \times 10^{3}\ kg / m^{3}$
Heat Capacity	$C_p = 4.600 \times 10^{2}\ J / kg\ ^{\circ}C$
Thermal Conductivity	$k = 5.560 \times 10^{1}\ W / m\ ^{\circ}C$
Emissivity	$\varepsilon = 0.980$
Stefan-Boltzman	$\sigma = 5.670 \times 10^{-8}\, W / m^{2}\ ^{\circ}K^{4}$

Load:

Thermal	$T = 1000.0\ ^{\circ}K$ (constant)
Convection	$h = 7.500 \times 10^{2}\ W / m^{2}\ ^{\circ}C$
Initial Temperature	$T_0 = 300.0\ ^{\circ}K$ (all nodes)

Element Types:

Fully integrated S/R solid (elform=2)

Material Models:

*MAT_T01 or *MAT_THERMAL_ISOTROPIC

Results Comparison:

LS-DYNA bar temperature at x=0.000 *m*, y=0.010 *m*, z= 0.100 *m* (Node 33) is compared with *NAFEMS Background to Benchmark*, Test T2.

Reference Condition - Point Along Bar 0.1 m (Node 33) from Hot End (Node 23)	**Temperature ($^\circ K$)**
NAFEMS Benchmark Test T2	9.2700×10^2
Node 33	9.2700×10^2

The fully integrated selectively reduced solid element (elform=2) model (also see Figure 21.2) provides an exact temperature comparison for this coarse mesh.

The problem is 1D, although solved in 3D. The results are, as expected, the same for all x-y planes along the Z-direction.

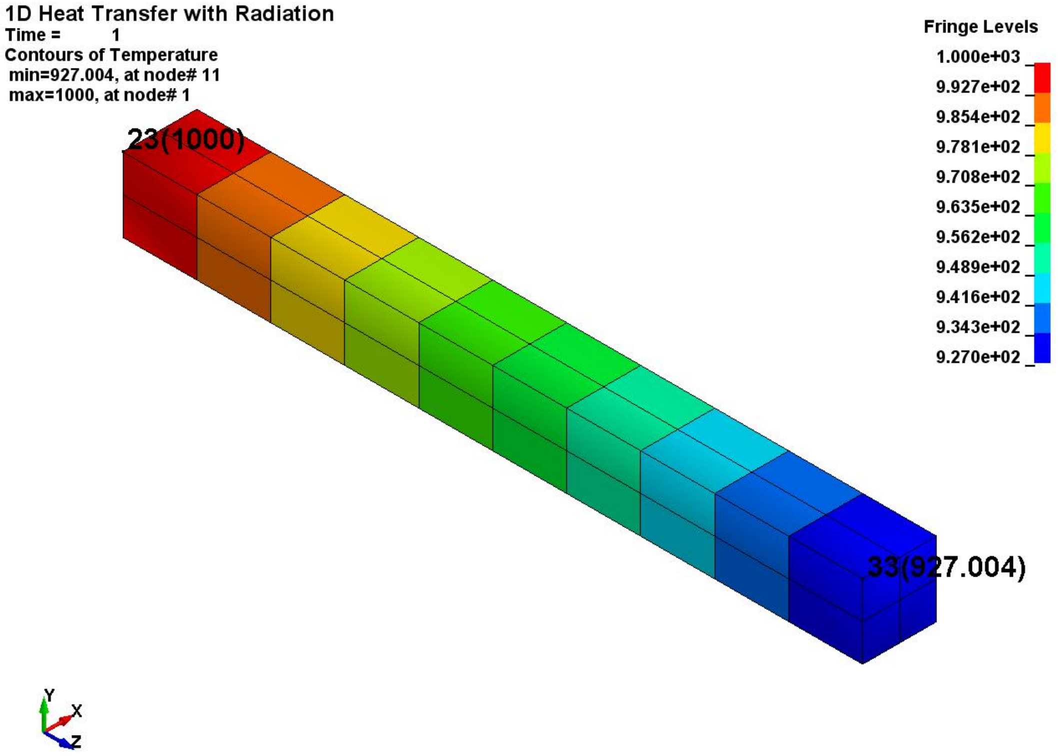

Figure 21.2 -Contour plot of temperatures with nodes 23 and 33 specifications.

Input deck:

```
*KEYWORD
*TITLE
1D Heat Transfer with Radiation
*CONTROL_SOLUTION
$#    soln       nlq     isnan     lcint
         1         0         0         0
*CONTROL_THERMAL_SOLVER
$#   atype     ptype    solver     cgtol       gpt    eqheat     fwork       sbc
         0         1         3  1.00e-06         8  1.000000  1.0000005.6700e-08
*CONTROL_THERMAL_NONLINEAR
$#  refmax       tol       dcp
        20 1.000e-06  0.500000
*CONTROL_THERMAL_TIMESTEP
$#      ts       tip       its      tmin      tmax     dtemp      tscp
         1  0.500000 1.000e-04 1.000e-04  0.100000  1.000000  0.500000
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
   1.00000         0       0.0       0.0       0.0
*DATABASE_TPRINT
$#      dt    binary      lcur     ioopt
  1.000000         0         0         1
*DATABASE_HISTORY_NODE
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
        11        22        33        44        55        66        77        88
        99
*DATABASE_BINARY_D3PLOT
$#      dt      lcdt      beam     npltc    psetid
  1.000000         0         0         0         0
*PART
$# title
Part          1 for TMat        1 and Elem Type         2
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         0         0         0         0         0         1
*SECTION_SOLID
$#   secid    elform       aet
         1         2         1
*MAT_THERMAL_ISOTROPIC
$#    tmid       tro     tgrlc    tgmult
         1  7850.000       0.0       0.0
$#      hc        tc
  460.0000   55.6000
*BOUNDARY_TEMPERATURE_SET
$#    nsid      lcid     cmult       loc
         1         0  1000.000         0
*BOUNDARY_RADIATION_SET
$#    ssid      type
         2         1
$#   flcid     fmult    tilcid    timult       loc
         05.5566e-08         0  300.0000         0
*INITIAL_TEMPERATURE_SET
$#    nsid      temp       loc
         3  300.0000         0
*SET_NODE_LIST_TITLE
A
$#     sid       da1       da2       da3       da4
         1       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1        12        23        34        45        56        67        78
        89
*SET_SEGMENT_TITLE
B
$#     sid       da1       da2       da3       da4
         2     0.000     0.000     0.000     0.000
$#      n1        n2        n3        n4        a1        a2        a3        a4
        55        22        11        44     0.000     0.000     0.000     0.000
        66        33        22        55     0.000     0.000     0.000     0.000
        88        55        44        77     0.000     0.000     0.000     0.000
        99        66        55        88     0.000     0.000     0.000     0.000
```

```
*SET_NODE_LIST_TITLE
central
$#     sid       da1       da2       da3       da4
         3       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1         2         3         4         5         6         7         8
         9        10        11        12        13        14        15        16
        17        18        19        20        21        22        23        24
        25        26        27        28        29        30        31        32
        33        34        35        36        37        38        39        40
        41        42        43        44        45        46        47        48
        49        50        51        52        53        54        55        56
        57        58        59        60        61        62        63        64
        65        66        67        68        69        70        71        72
        73        74        75        76        77        78        79        80
        81        82        83        84        85        86        87        88
        89        90        91        92        93        94        95        96
        97        98        99
*ELEMENT_SOLID
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1      34      45      12       2      35      46      13

      40       1      54      87      98      65      55      88      99      66
*NODE
$#   nid               x               y               z      tc      rc
       1             0.0             0.0             0.0

      99      0.01000000      0.01000000      0.10000000
*END
```

Notes:

1. The problem must be flagged as nonlinear if any boundary condition parameter is a function of temperature. This includes a linear (i.e., straight line) relationship. Iterations are needed to obtain the correct solution. Radiation is a T^4 boundary condition.

2. The *CONTROL_THERMAL_NONLINEAR keyword is optional. For example, the default values for remax (maximum number of iterations allowed per time step), tol (temperature convergence tolerance), and dcp (divergence control tolerance) will be used, if the nonlinear keyword is omitted, with ptype>0 on *CONTROL_THERMAL_SOLUTION keyword.

22. 1D Transient Heat Transfer in a Bar

Keywords:

```
*CONTROL_SOLUTION
*CONTROL_THERMAL_SOLVER
*CONTROL_THERMAL_TIMESTEP
*BOUNDARY_TEMPERATURE_SET
*INITIAL_TEMPERATURE_SET
*MAT_THERMAL_ISOTROPIC
```

Description:

A 0.1 *m* long bar (L_z), with square 0.01 m (L_x) x 0.01 m (L_y) cross-section (Figure 22.1), is subjected at one end (node 6) to a varying thermal with the following law: $T = 100\sin\left(\pi t / 40\right)\,^{\circ}C$. The other end (node 1) is maintained at constant temperature $T = 0\,^{\circ}C$. The bar is perfectly insulated along its length.

Determine the temperature at 0.02 m from the hot end after 32 seconds.

The bar is meshed with 20 elements: five elements along the length and four elements in the cross section.

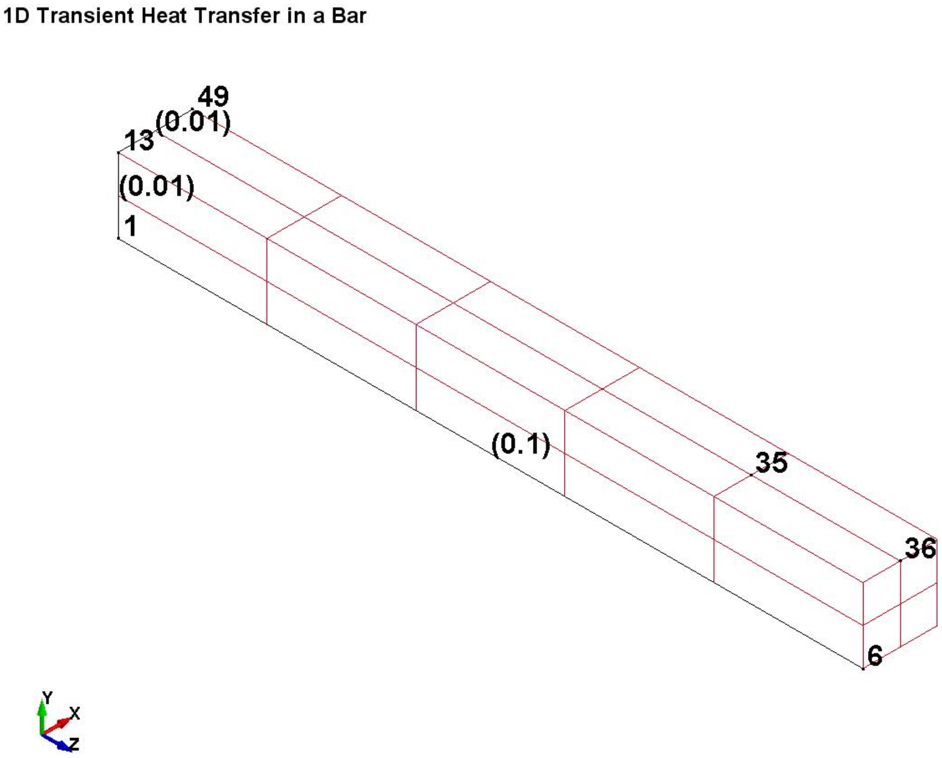

Figure 22.1 - Finite element model with selected nodes and dimensions identified.

Analysis Summary:

Dim.	**Type**	**Load**	**Material**	**Geometry**	**Contact**	**Solver**	**Solution Method**
3D	Thermal Transient	Thermal	Linear	Linear	-	Thermal Linear	3-Diagonal scaled conjugate gradient

Units:

kg, m, s, N, Pa, N-m, $^{\circ}C$ *(kilogram, meter, second, Newton, Pascal, Newton-meter, degree Centigrade) - Joule* (*J*) is a *N-m, Watt* (*W*) is a *J/s*

Dimensional Data:

$L_x = 0.01\ m$, $L_y = 0.01\ m$, $L_z = 0.10\ m$

Material Data:

Mass Density $\rho = 7.200 \times 10^3\ kg / m^3$

Heat Capacity $C_p = 4.405 \times 10^2\ J / kg\ ^{\circ}C$

Thermal Conductivity $K = 3.500 \times 10^1\ W / m\ ^{\circ}C$

Load:

Thermal $T = 100\ ^{\circ}C$ (constant)

Convection $h = 7.500 \times 10^2\ W / m^2\ ^{\circ}C$

Initial Temperature $T_0 = 0\ ^{\circ}C$ (all nodes)

Element Types:

Fully integrated S/R solid (elform=2)

Material Models:

*MAT_T01 or *MAT_THERMAL_ISOTROPIC

Results Comparison:

LS-DYNA bar temperature at x=0.0050 m, y=0.005 m, z= 0.080 m (Node 35) is compared with *NAFEMS Background to Benchmark,* Test T3.

Reference Condition - Point Along Bar 0.2 m (Node 35) from Hot End (Node 36)	**Temperature ($^{\circ}C$)**
NAFEMS Benchmark Test T3	3.6600×10^{1}
Node 35	3.4861×10^{1}

The fully integrated selectively reduced solid element (elform=2) model (Figure 22.2) provides a reasonable temperature comparison for this coarse mesh.

The problem is 1D, although done in 3D. The results are, as expected, the same for all x-y planes along the Z-direction.

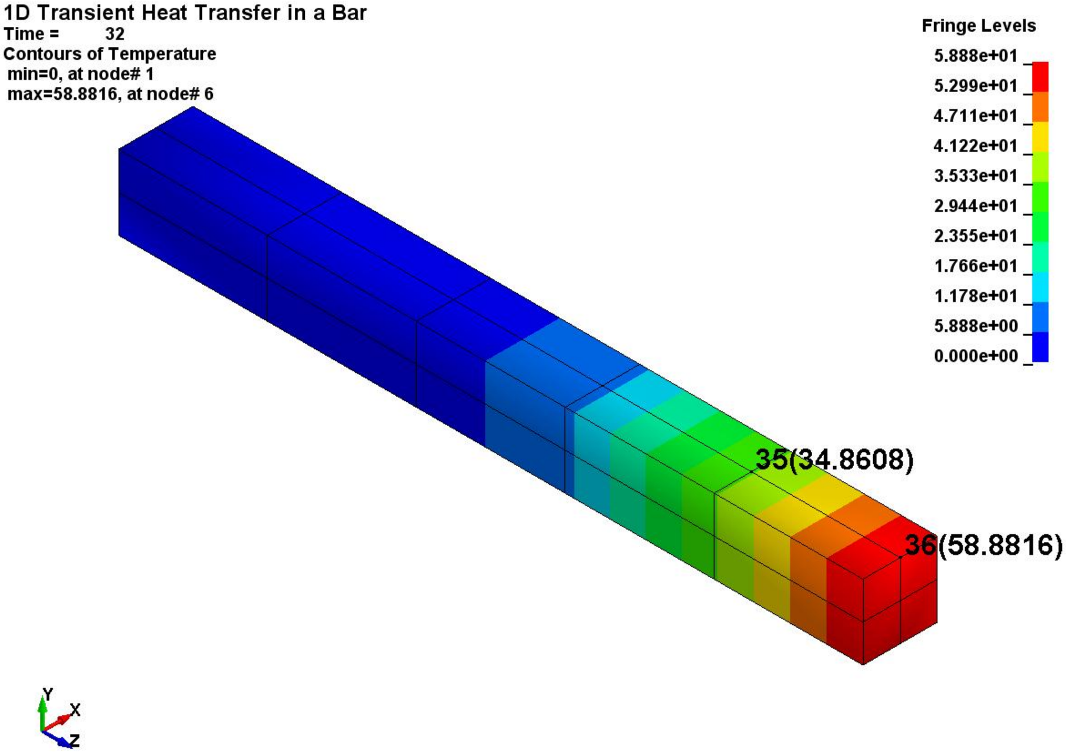

Figure 22.2 -Contour plot of temperatures at time =32.0 seconds with nodes 35 and 36 specifications.

The histories of temperature for two nodes (35 and 36) used in the comparison are shown in Figure 22.3.

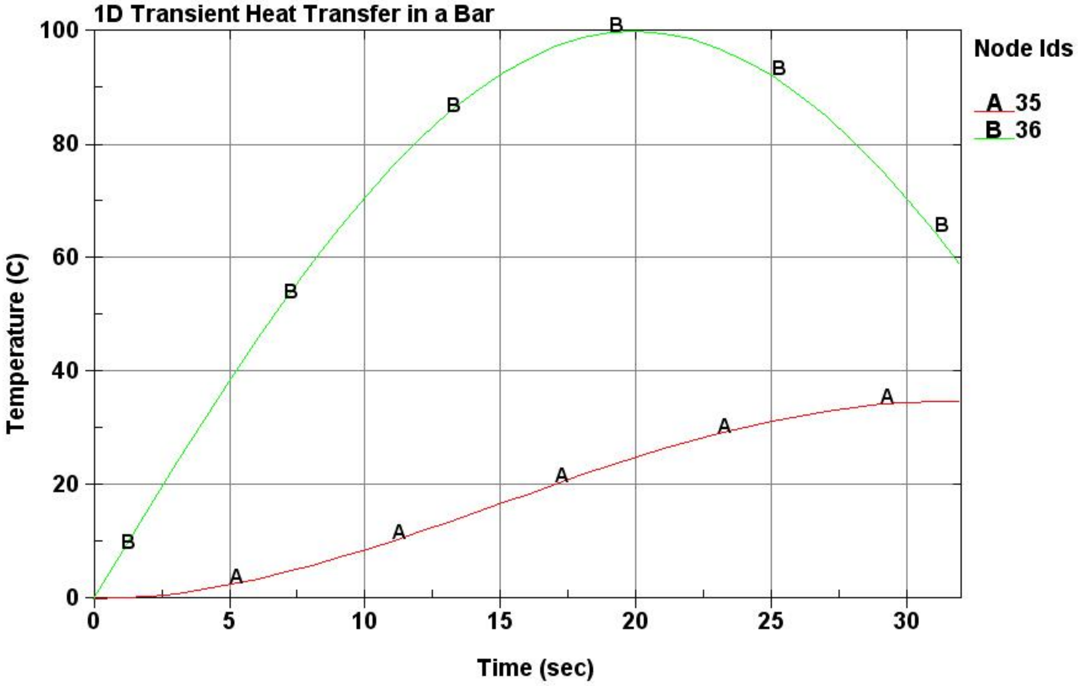

Figure 22.3 - Temperature histories for nodes 35 and 56.

Input deck:

```
*KEYWORD
*TITLE
1D Transient Heat Transfer in a Bar
*CONTROL_SOLUTION
$#    soln       nlq     isnan     lcint
         1         0         0         0
*CONTROL_THERMAL_SOLVER
$#   atype     ptype    solver     cgtol       gpt    eqheat     fwork       sbc
         1         0         3  1.00e-06         8  1.000000  1.000000       0.0
*CONTROL_THERMAL_TIMESTEP
$#      ts       tip       its      tmin      tmax     dtemp      tscp
         1  0.500000 1.000e-04 1.000e-04  0.100000  1.000000  0.500000
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  32.00000         0       0.0       0.0       0.0
*DATABASE_TPRINT
$#      dt    binary      lcur     ioopt
  1.000000         0         0         1
*DATABASE_HISTORY_NODE
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         5         6        11        12        17        18        23        24
        29        30        35        36        41        42        47        48
        53        54
*DATABASE_BINARY_D3PLOT
$#      dt      lcdt      beam     npltc    psetid
  1.000000         0         0         0         0
*PART
$# title
Part          1 for TMat        1 and Elem Type         2
```

```
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         0         0         0         0         0         1
*SECTION_SOLID
$#   secid    elform       aet
         1         2         1
*MAT_THERMAL_ISOTROPIC
$#    tmid       tro     tgrlc    tgmult
         1  7200.000       0.0       0.0
$#      hc        tc
  440.5000   35.0000
*BOUNDARY_TEMPERATURE_SET
$#    nsid      lcid     cmult       loc
         1         0       0.0         0
*BOUNDARY_TEMPERATURE_SET
$#    nsid      lcid     cmult       loc
         2         1  1.000000         0
*INITIAL_TEMPERATURE_SET
$#    nsid      temp       loc
         3       0.0         0
*SET_NODE_LIST_TITLE
A
$#     sid       da1       da2       da3       da4
         1       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1         7        13        19        25        31        37        43
        49
*SET_NODE_LIST_TITLE
B
$#     sid       da1       da2       da3       da4
         2       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         6        12        18        24        30        36        42        48
        54
*SET_NODE_LIST_TITLE
central
$#     sid       da1       da2       da3       da4
         3       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1         2         3         4         5         6         7         8
         9        10        11        12        13        14        15        16
        17        18        19        20        21        22        23        24
        25        26        27        28        29        30        31        32
        33        34        35        36        37        38        39        40
        41        42        43        44        45        46        47        48
        49        50        51        52        53        54
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo    dattyp
         1         0       0.0       0.0       0.0       0.0
$#                  a1                  o1
                 0.0                 0.0
          1.00000000          7.84194040
          2.00000000         15.63558102
          3.00000000         23.33292198
          4.00000000         30.88655281
          5.00000000         38.24995041
          6.00000000         45.37776184
          7.00000000         52.22608948
          8.00000000         58.75275421
          9.00000000         64.91754913
         10.00000000         70.68251801
         11.00000000         76.01214600
         12.00000000         80.87360382
         13.00000000         85.23696136
         14.00000000         89.07533264
         15.00000000         92.36508179
         16.00000000         95.08594513
         17.00000000         97.22116852
         18.00000000         98.75759888
         19.00000000         99.68576813
         20.00000000         99.99996948
         21.00000000         99.69825745
```

```
         22.00000000         98.78250122
         23.00000000         97.25833130
         24.00000000         95.13513947
         25.00000000         92.42600250
         26.00000000         89.14760590
         27.00000000         85.32013702
         28.00000000         80.96717834
         29.00000000         76.11553955
         30.00000000         70.79508972
         31.00000000         65.03861237
         32.00000000         58.88155746
*ELEMENT_SOLID
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1      19      25       7       2      20      26       8

      20       1      29      47      53      35      30      48      54      36
*NODE
$#   nid               x               y               z      tc      rc
       1             0.0             0.0             0.0

      54      0.01000000      0.01000000      0.10000000
*END
```

Notes:

23. 2D Heat Transfer with Convection

Keywords:

```
*CONTROL_SOLUTION
*CONTROL_THERMAL_SOLVER
*CONTROL_THERMAL_TIMESTEP
*MAT_THERMAL_ISOTROPIC
*BOUNDARY_CONVECTION_SET
*BOUNDARY_TEMPERATURE_SET
*INITIAL_TEMPERATURE_SET
```

Description:

A slab ($L_y = 1.00\ m$ in depth) of rectangular cross-section ($L_x = 0.60\ m$ by $L_z = 1.00\ m$) shown in Figure 23.1 is subjected to the following thermal loads for a steady state simulation:

- constant Temperature $T_0 = 100^\circ C$ on the face defined by nodes 78-79-85-95,
- natural convection to the ambient temperature $T_a = 0^\circ C$ on the faces defined by nodes 12-85-79-18 and 1-12-18-2 (convection coefficient $h = 7.50 \times 10^2\ W / m^2\ {}^\circ C$),
- face defined by nodes 1-2-78-95 is adiabatically insulated.

Find the temperature at node 225 (x=0.60 m, y=1.00 m, z=-0.20 m).

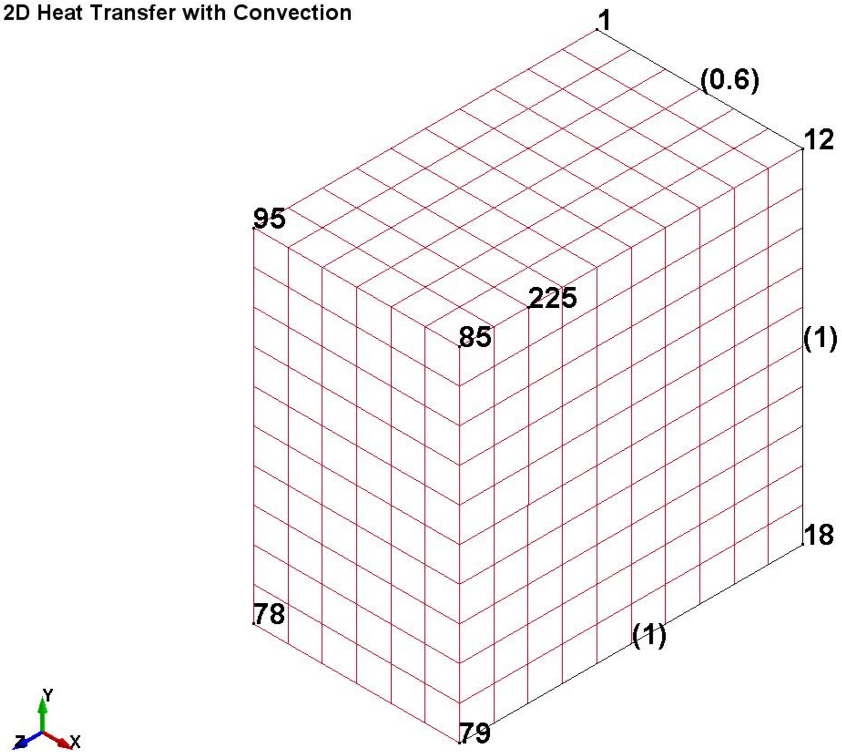

Figure 23.1 - Finite element model with selected nodes and dimensions identified.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Steady State	Thermal	Linear	Linear	-	Thermal	3-Diagonal scaled conjugate gradient

Units:

kg, m, s, N, Pa, N-m, $^\circ C$ *(kilogram, meter, second, Newton, Pascal, Newton-meter, degree Centigrade) - Joule* (*J*) is a *N-m, Watt* (*W*) is a *J/s*

Dimensional Data:

$L_x = 0.60\ m$, $L_y = 1.00\ m$, $L_z = 1.00\ m$

Material Data:

Mass Density	$\rho = 8.000 \times 10^3\ kg / m^3$
Heat Capacity	$C_p = 1.000 \times 10^0\ J / kg\ ^\circ C$
Thermal Conductivity	$k = 5.200 \times 10^1\ W / m\ ^\circ C$

Load:

Thermal	$T = 100\ ^\circ C$ (constant)
Convection	$h = 7.500 \times 10^2\ W / m^2\ ^\circ C$

Element Types:

Fully integrated S/R solid (elform=2)

Material Models:

*MAT_T01 or *MAT_THERMAL_ISOTROPIC

Results Comparison:

LS-DYNA slab edge temperature at x=0.60 m, y=1.00 m, z=-0.20 m (Node 225) are compared with *NAFEMS Background to Benchmark,* Test T4.

Reference Condition - Point Along Slab Edge (Node 225)	**Temperature ($^\circ C$)**
NAFEMS Benchmark Test T4	1.8300×10^1
Node 225	1.7954×10^1

The fully integrated selectively reduced solid element (elform=2) model (Figure 23.2) provides a reasonable temperature comparison for this relatively coarse mesh.

The problem is 2D, although solved in 3D. The results are, as expected, the same for all planes in the 3rd dimension (Y-direction in this case).

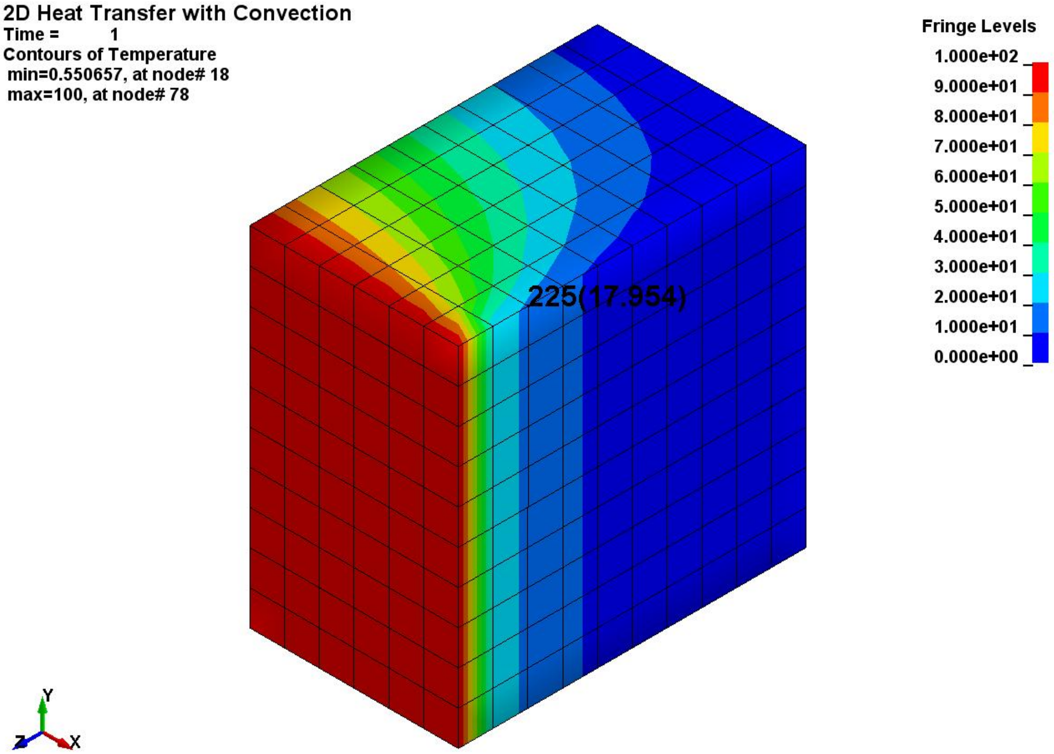

Figure 23.2 -Contour plot of temperatures with node 225 specification.

Input deck:

```
*KEYWORD
*TITLE
2D Heat Transfer with Convection
*CONTROL_SOLUTION
$#    soln       nlq     isnan     lcint
         1         0         0         0
*CONTROL_THERMAL_SOLVER
$#   atype     ptype    solver     cgtol       gpt    eqheat     fwork       sbc
         0         0         3  1.00e-06         8  1.000000  1.000000  0.000000
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  1.000000         0     0.000     0.000     0.000
*DATABASE_TPRINT
$#      dt    binary      lcur     ioopt
  1.000000         0         0         1
*DATABASE_HISTORY_NODE
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       225       171       371       372       373       374       375       376
       377       378       379
*DATABASE_BINARY_D3PLOT
$#      dt      lcdt      beam     npltc    psetid
  1.000000         0         0         0         0
*PART
$# title
Part          1 for Mat         1 and Elem Type         2
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         0         0         0         0         0         1
*SECTION_SOLID
$#   secid    elform       aet
         1         2         0
*MAT_THERMAL_ISOTROPIC
$#    tmid       tro     tgrlc    tgmult      tlat      hlat
         1  8000.000     0.000     0.000     0.000     0.000
$#      hc        tc
     1.000   52.0000
*BOUNDARY_CONVECTION_SET
$#    ssid
         1
$#   hlcid     hmult     tlcid     tmult       loc
         0 750.00000         0     0.000         0
*BOUNDARY_CONVECTION_SET
$#    ssid
         2
$#   hlcid     hmult     tlcid     tmult       loc
         0 750.00000         0     0.000         0
*BOUNDARY_TEMPERATURE_SET
$#    nsid      lcid     cmult       loc
         1         0 100.00000         0
*INITIAL_TEMPERATURE_SET
$#    nsid      temp       loc
         1 100.00000         0
*SET_NODE_LIST_TITLE
frontt100
$#     sid       da1       da2       da3       da4
         1     0.000     0.000     0.000     0.000
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
        78        79        80        81        82        83        84        85
        86        87        88        89        90        91        92        93
        94        95        96        97        98        99       100       101
       102       103       104       105       106       107       108       109
       110       111       112       113       114       115       116       117
       118       119       120       121       122       123       124       125
       126       127       128       129       130       131       132       133
       134       135       136       137       138       139       140       141
       142       143       144       145       146       147       148       149
       150       151       152       153       154         0         0         0
*SET_SEGMENT_TITLE
sidet0
```

```
$#     sid       da1       da2       da3       da4
         1     0.000     0.000     0.000     0.000
$#      n1        n2        n3        n4        a1        a2        a3        a4
        18        19       434       164     0.000     0.000     0.000     0.000

       370       226        85        94     0.000     0.000     0.000     0.000
*SET_SEGMENT_TITLE
backt0
$#     sid       da1       da2       da3       da4
         2     0.000     0.000     0.000     0.000
$#      n1        n2        n3        n4        a1        a2        a3        a4
         2        11        33        28     0.000     0.000     0.000     0.000

        77        13        12        27     0.000     0.000     0.000     0.000
*ELEMENT_SOLID
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       2      28      33      11     163     213     443     289

     600       1     847     370     226     272     154      94      85      96
*NODE
$#   nid               x               y               z      tc      rc
       1           0.000       1.0000000      -1.0000000       0       0

     847       0.5000000       0.9000000      -0.1000000       0       0
*END
```

Notes:

1. A two-dimensional model simulation could be made using the plane stress (x-y plane) element formulation (elform=12 given on *SECTION_SHELL keyword) with a constant temperature through the thickness. Under the keyword *CONTROL_ SHELL, the option tshell allows the user to choose between a constant temperature through the thickness and a 20 node brick formulation which allows heat conduction through the thickness.

24. 3D Thermal Load

Keywords:

```
*CONTROL_IMPLICIT_GENERAL
*CONTROL_IMPLICIT_SOLUTION
*LOAD_THERMAL_VARIABLE_NODE
*MAT_ELASTIC_PLASTIC_THERMAL
```

Description:

The solid cylinder tapering to a sphere geometry depicted below (Figure 24.1) is subjected to a prescribed temperature gradient. Two analyses, one with a coarse mesh 5 x 1 x 3 and one with a fine mesh 10 x 2 x 3 (Figure 24.2), are made. The model represents ¼ of the total geometry. Symmetry conditions on the plane x-z and y-z are enforced. The faces parallel to the plane x-y are simply supported in the Z-direction.

The linear temperature loading (radial and axial direction) (Figure 24.3) is applied by means of temperature dependent material with thermal expansion coefficient and resulting thermal strain $\varepsilon = \alpha T$.

Determine the stress in the Z-direction at Node A (Node 10 for the coarse-mesh model and Node 16 for the fine-mesh model).

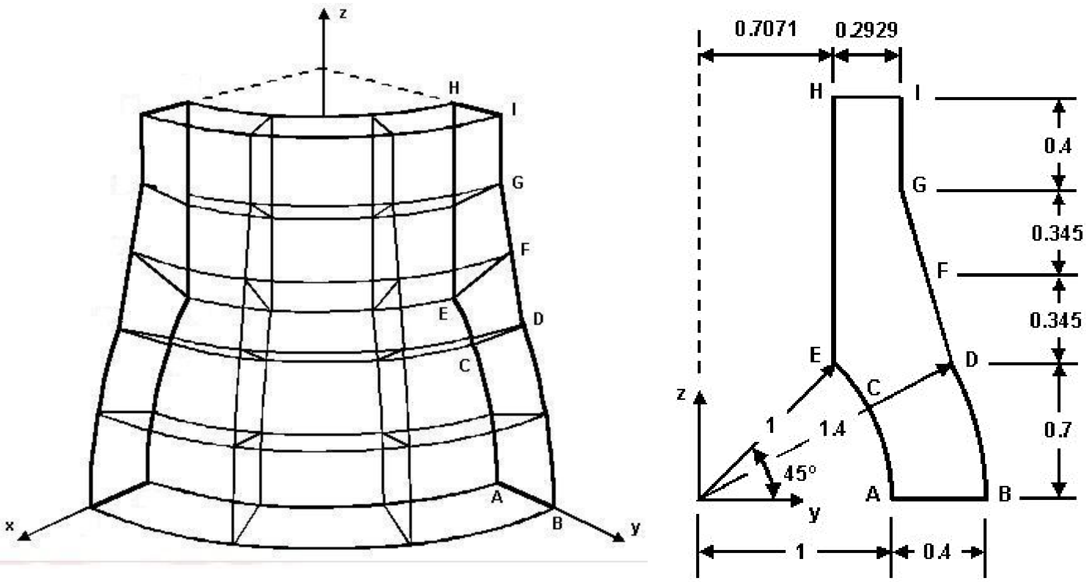

Figure 24.1 - Schematic of ¼ model and cross-section dimensions (all dimensions are in meters).

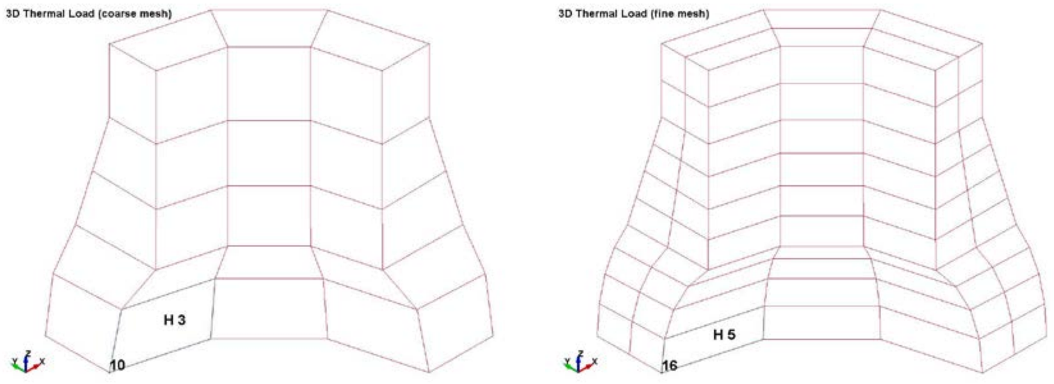

Figure 24.2 - Finite element models with selected node and element identified.

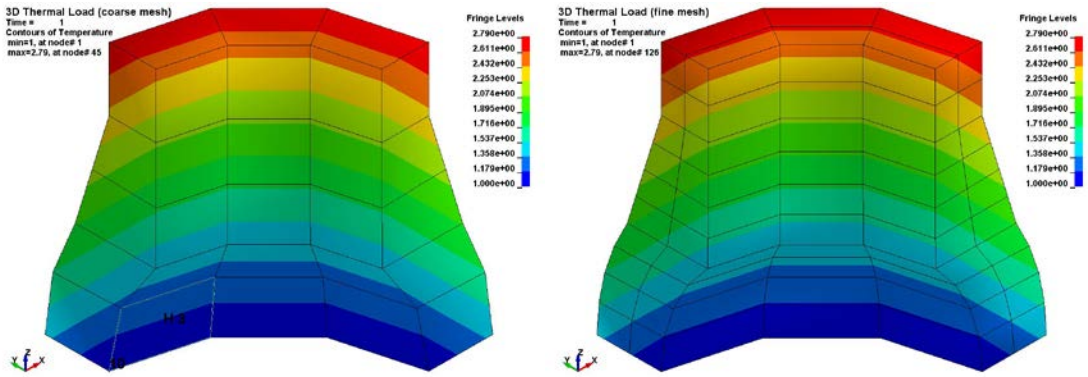

Figure 24.3 - Finite element models with temperature loading.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Static	Thermal	Linear	Linear	-	Implicit	1-Linear

Units:

kg, m, s, N, Pa, N-m, °C (kilogram, meter, second, Newton, Pascal, Newton-meter, degree Centigrade) - Joule (*J*) is a *N-m, Watt* (*W*) is a *J/s*

Dimensional Data:

See Figure 24.1 (all dimensions in meters).

Material Data:

Young's Modulus	$E = 2.10 \times 10^{11}\ Pa$
Poisson's Ratio	$\nu = 0.3$
Linear Expansion	$\alpha = 2.30 \times 10^{-4}\ m/m/^{\circ}C$

Load:

Thermal $T(^{\circ}C) = \sqrt{x^2 + y^2} + z$

Element Types:

Fully integrated S/R solid (elform=2)

Material Models:

*MAT_004 or *MAT_ELASTIC_PLASTIC_THERMAL

Results Comparison:

LS-DYNA global stress σ_{zz} at inner point on yz-symmetry plane (coarse mesh - Node 10, Element 3: fine mesh - Node 16, Element 5) is compared with *NAFEMS Background to Benchmarks*, Test LE11.

Reference Condition - Inner Point on Symmetry Plane	**Coarse Mesh - Global Stress - σ_{zz} (M*Pa*)**	**Fine Mesh - Global Stress - σ_{zz} (M*Pa*)**
NAFEMS Benchmark Test LE11	-1.0500×10^{2}	-1.0500×10^{2}
Element 3 (coarse) - Element 5 (fine) - (an averaged value)	-6.1849×10^{1}	-7.5890×10^{1}
First in-plane integration point - Elem 3 (coarse) - Element 5 (fine)	-7.9107×10^{1}	-8.5217×10^{1}
Node 10 (coarse) - Node 16 (fine)	-9.2071×10^{1}	-9.2102×10^{1}

The global stress σ_{zz} results were generated from *DATABASE_ELOUT (*elout* file) and *DATABASE_EXTENT_BINARY (*eloutdet* file provides detailed element output at integration points and connectivity nodes) keyword entries.

By default, stresses/strains for solids are written to *d3plot* and *elout* in the global coordinate system. The *elout* file contains only the values at the element centroid (average of 8 integration points).

You can set intout=stress or intout=all (*DATABASE_EXTENT_BINARY) and have stresses output for all the integration points to a file called *eloutdet* (*DATABASE_ELOUT governs the output interval and *DATABASE_HISTORY_SOLID governs which elements are output). Setting nodout=stress or nodout=all in *DATABASE_EXTENT_BINARY will write the extrapolated nodal stresses to *eloutdet*.

LS-DYNA stress and strain outputs correspond to integration point locations. Stress at a node is an artifact of the post-processor and represents an average of the surrounding integration point stresses (the value will likely be different with different postprocessors).

Both meshes are rather coarse which makes it difficult to capture the global stress σ_{zz} at the inner point along yz-symmetry plane. The extrapolated nodal stress results (also see Figures 24.4 and 24.5) provide the best comparative value (~12% difference for both meshes), primarily due to the nodal location. As expected, the fine mesh does a better job of capturing the overall contouring. The average global stress of the element provides the least acceptable comparative value results (~40% and ~25% differences), again not unexpected, while the nearest element integration point results provided significant improvement (~25% and ~15% differences) due to the larger integration sample (8 points as compared to 1) and nodal location.

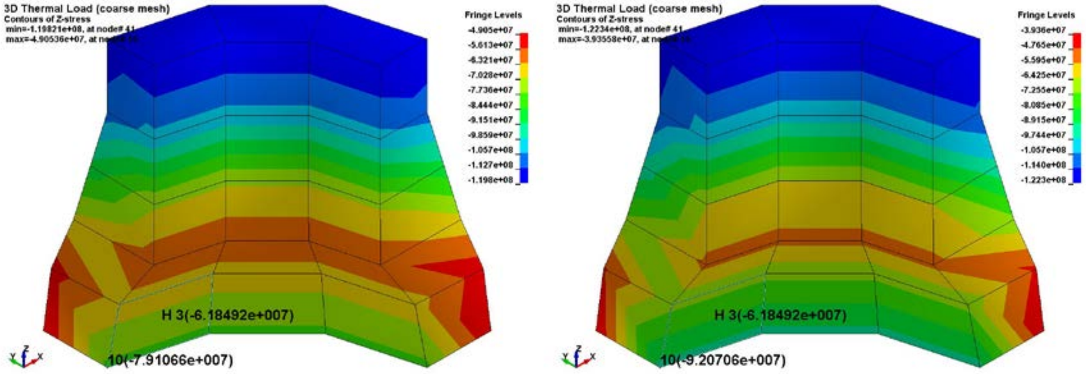

Figure 24.4 - Coarse mesh contour plots of global stress σ_{zz} with average value given for Element 3. On the left is in-plane integration point contouring while on the right is extrapolated nodal stress contouring with specification values given at Node 10.

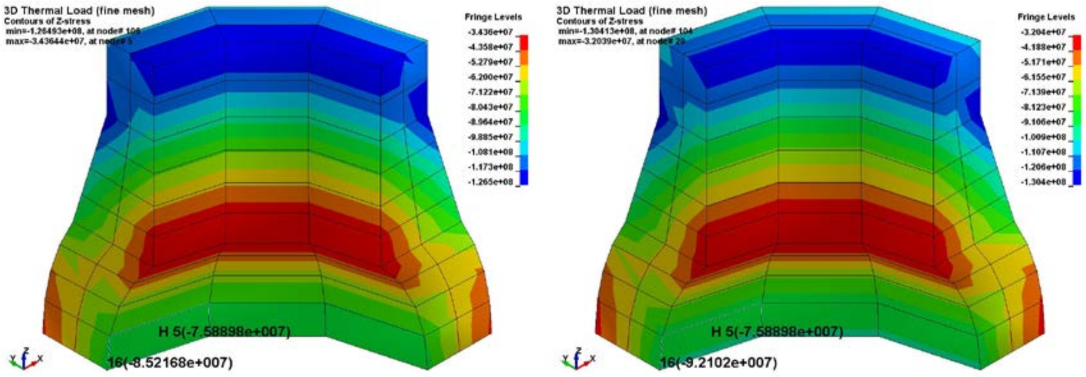

Figure 24.5 - Fine mesh contour plots of global stress σ_{zz} with average value given for Element 5. On the left is in-plane integration point contouring while on the right is extrapolated nodal stress contouring with specification values given at Node 16.

Input deck:

```
*KEYWORD
*TITLE
3D Thermal Load (coarse mesh)
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1  1.000000         2         1         2
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
         1        11        15  0.001000  0.010000   1.0e+10  0.900000  1.000000
$#   dnorm    diverg     istif   nlprint    nlnorm    d3itcl     cpchk
         2         1         1         2         2         0         0
$#  arcctl    arcdir    arclen    arcmth    arcdmp    arcpsi    arcalf    arctim
         0         1       0.0         1         2       0.0       0.0       0.0
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  1.000000         0       0.0       0.0       0.0
*DATABASE_EXTENT_BINARY
$#   neiph     neips    maxint    strflg    sigflg    epsflg     rtflg    engflg
         0         0         0         1         1         1         1         1
$#  cmpflg    ieverp    beamip     dcomp      shge     stssz    n3thdt   ialemat

$# nintsld   pkp_sen      sclp     hydro     msscl     therm    intout    nodout
         8                 1.0                                  stress    stress
*DATABASE_ELOUT
$# dt/cycl
  0.100000
*DATABASE_HISTORY_SOLID
$#    eid1      eid2      eid3      eid4       ei5      eid6      eid7      eid8
         3
*DATABASE_BINARY_D3PLOT
$# dt/cycl
  1.000000
*PART
$# title
Part          1 for Mat         1 and Elem Type          2
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
*SECTION_SOLID
$#   secid    elform       aet
         1         2         1
*MAT_ELASTIC_PLASTIC_THERMAL
$#     mid        ro
         1  1.000000
$#      t1        t2        t3        t4        t5        t6        t7        t8
```

```
       0.0  1000.000       0.0       0.0       0.0       0.0       0.0       0.0
$#      e1        e2        e3        e4        e5        e6        e7        e8
 2.100e+11 2.100e+11       0.0       0.0       0.0       0.0       0.0       0.0
$#     pr1       pr2       pr3       pr4       pr5       pr6       pr7       pr8
  0.300000  0.300000       0.0       0.0       0.0       0.0       0.0       0.0
$#  alpha1    alpha2    alpha3    alpha4    alpha5    alpha6    alpha7    alpha8
 2.300e-04 2.300e-04       0.0       0.0       0.0       0.0       0.0       0.0
$#   sigy1     sigy2     sigy3     sigy4     sigy5     sigy6     sigy7     sigy8
       0.0       0.0       0.0       0.0       0.0       0.0       0.0       0.0
$#   etan1     etan2     etan3     etan4     etan5     etan6     etan7     etan8
       0.0       0.0       0.0       0.0       0.0       0.0       0.0       0.0
*ELEMENT_SOLID
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1      13      16       4       2      14      17       5

      15       1      35      39      40      36      43      47      48      44
*NODE
$#   nid               x               y               z      tc      rc
       1       1.0000000           0.000           0.000       0       0

      48           0.000       1.0000000       1.7900000       0       0
*SET_NODE_LIST_TITLE
base
$#     sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
        19         7         4        16
*SET_NODE_LIST_TITLE
top
$#     sid       da1       da2       da3       da4    solver
         2       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
        43        47        46        42
*SET_NODE_LIST_TITLE
xsymm
$#     sid       da1       da2       da3       da4    solver
         3       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
        23        11        24        12        28        32        40        36
*SET_NODE_LIST_TITLE
ysymm
$#     sid       da1       da2       da3       da4    solver
         4       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         2        14         3        15        25        29        33        37
*SET_NODE_LIST_TITLE
z+ysymm
$#     sid       da1       da2       da3       da4    solver
         5       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1        13        41        45
*SET_NODE_LIST_TITLE
z+xsymm
$#     sid       da1       da2       da3       da4    solver
         6       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
        22        10        48        44
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         0         0         1
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         2         0         0         0         1
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         3         0         1         0         0
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         4         0         0         1         0
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
```

```
         5         0         0         1         1
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         6         0         1         0         1
*LOAD_THERMAL_VARIABLE_NODE
$#     nid        ts        tb      lcid
         1  1.000000       0.0         1

        48  2.790000       0.0         1
*DEFINE_CURVE
$#    lcid      sdir       sfa       sfo      offa      offo    dattyp
         1         0       0.0       0.0       0.0       0.0
$#                a1                  o1
                 0.0          1.00000000
          1.00000000          1.00000000
*END
```

Notes:

1. The fully integrated solid elements are formulated for nonlinear analysis. Although this analysis is linear, it could have been solved with a nonlinear solution method (nsolvr=2). This was done for this simulation and it was found to yield results with some differences (less than 0.01% for coarse mesh and ~4% for fine mesh) from the linear analysis (nsolvr=1). It is not understood why the fine mesh offers this difference.

2. Fully-integrated solid formulations have 8 in-volume integration points for each element. For these formulations, the 8 values of each stress component are averaged at the element centroid before being written to *elout*.

3. If setting nintsld=8 on *DATABASE_EXTENT_BINARY, LS-DYNA will write stresses at all integration points for solid elements (also given in *eloutdet*) to the d3plot file. When this option is set, LS-PrePost applies the stress values to the nodes from the closest integration point and after that, the average value from the contributions are computed and presented in the stress fringe plot.

4. A command line option (extrapolate 1) is added to LS-PrePost, which will linearly extrapolate the values from integration points to the nodes (the extrapolated nodal stresses are also given in *eloutdet*).

5. For elastic bending, two integrations points through the thickness is the minimum number. For plastic bending, three integrations points through the thickness is the minimum.

25. Cooling of a Billet via Radiation

Keywords:

```
*CONTROL_SOLUTION
*CONTROL_THERMAL_SOLVER
*CONTROL_THERMAL_NONLINEAR
*CONTROL_THERMAL_TIMESTEP
*BOUNDARY_TEMPERATURE_SET
*BOUNDARY_RADIATION_SET
*MAT_THERMAL_ISOTROPIC
```

Description:

A billet ($L_z = 4.00\ ft$ in height) of rectangular cross-section ($L_x = 2.00\ ft$ by $L_7 = 2.00\ ft$) shown in Figure 25.1 is initially at temperature $T_0 = 2000\,^{\circ}R$ loses heat by radiation (transient) from all its surfaces to its surroundings at a temperature of $T_e = 530^{\circ}R$. There is zero internal heat generation.

Determine the temperature of the billet (e.g. Node 625) after 3.7 *hours* ($1.3320 \times 10^4\ sec$).

The bar is meshed with 432 elements: 12 elements along the height and 36 elements in the cross section.

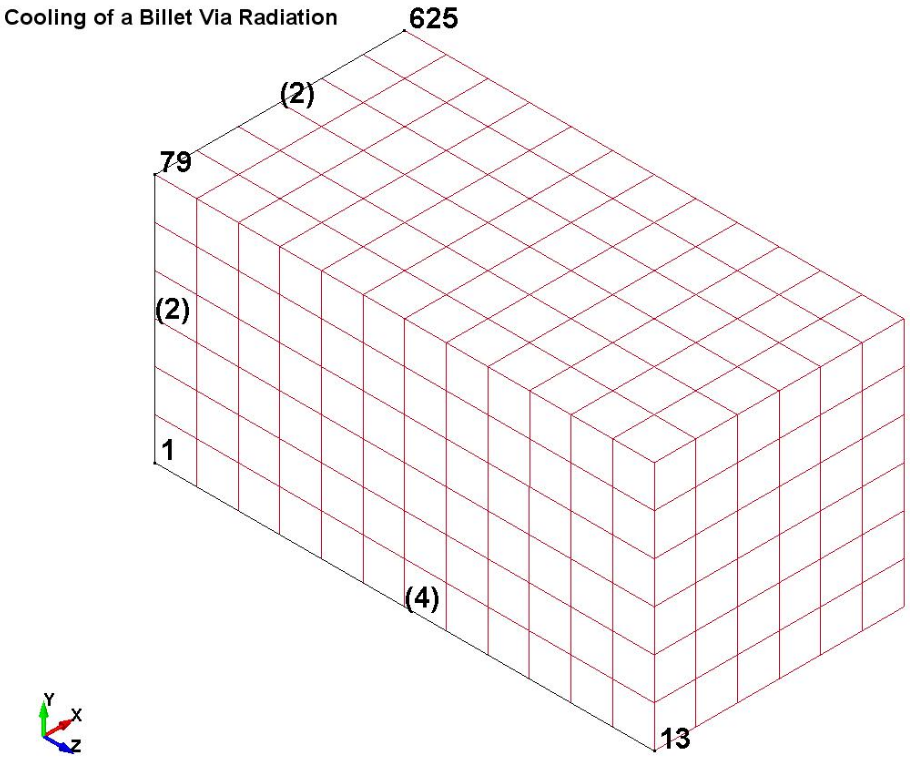

Figure 25.1 - Finite element model with selected nodes and dimensions identified.

Analysis Summary:

Dim.	Type	Load	Material	Geometry	Contact	Solver	Solution Method
3D	Thermal Transient	Thermal	Linear	Linear	-	Thermal Nonlinear	3-Diagonal scaled conjugate gradient

Units:

lbf-s^2*/ft, ft, s, lbf, psf, lbf-ft* (*slug, foot, second, pound force, pound force/foot*^{2}*, pound force-foot*) - *Thermal Energy* is a *Btu*, *Power* is a *Btu/s*, $1^\circ\ \Delta F = 1^\circ\ \Delta R$

Dimensional Data:

$L_x = 2.00\ \ ft$, $L_y = 2.00\ \ ft$, $L_z = 2.00\ \ ft$

Material Data:

Mass Density $\rho = 4.875 \times 10^2\ \ lbf - s^2\ /\ ft^4$
Heat Capacity $C_p = 1.100 \times 10^{-1}\ \ Btu\,/\,lbf -^\circ R$
Thermal Conductivity $k = 1.000 \times 10^4\ \ Btu\,/\ ft -^\circ R$ (arbitrary value)
Emissivity $\varepsilon = 1.000$
Stefan-Boltzman $\sigma = 4.750 \times 10^{-13}\ Btu\,/\,s - ft^2 -^\circ R^4$

Load:

Thermal (billet) $T_0 = 2000.0\ ^\circ R$ (constant)
Thermal (outside) $T_e = 530.0\ ^\circ R$ (surroundings)

Element Types:

Fully integrated S/R solid (elform=2)

Material Models:

*MAT_T01 or *MAT_THERMAL_ISOTROPIC

Results Comparison:

LS-DYNA temperature of the billet at selected point x=2.000 *ft*, y=2.010 *ft*, z= 0.000 *ft* (Node 625) is compared with R. Siegal and J.R. Howell studies in *Thermal Radiation Heat Transfer*, 1981, pg. 229.

Reference Condition - Billet	**Temperature ($^\circ R$) at t=13320 sec**
Siegal and Howell [1981]	1.0000×10^3
Node 625	1.0080×10^3

The fully integrated selectively reduced solid element (elform=2) model provides a reasonable temperature comparison for this mesh (less that 1% difference).

With the given simple geometry plus the initial temperature and radiation heat losses being space invariant, the temperature is uniform throughout the mesh.

The temperature history of Node 625 used in the comparison is shown in Figure 25.2.

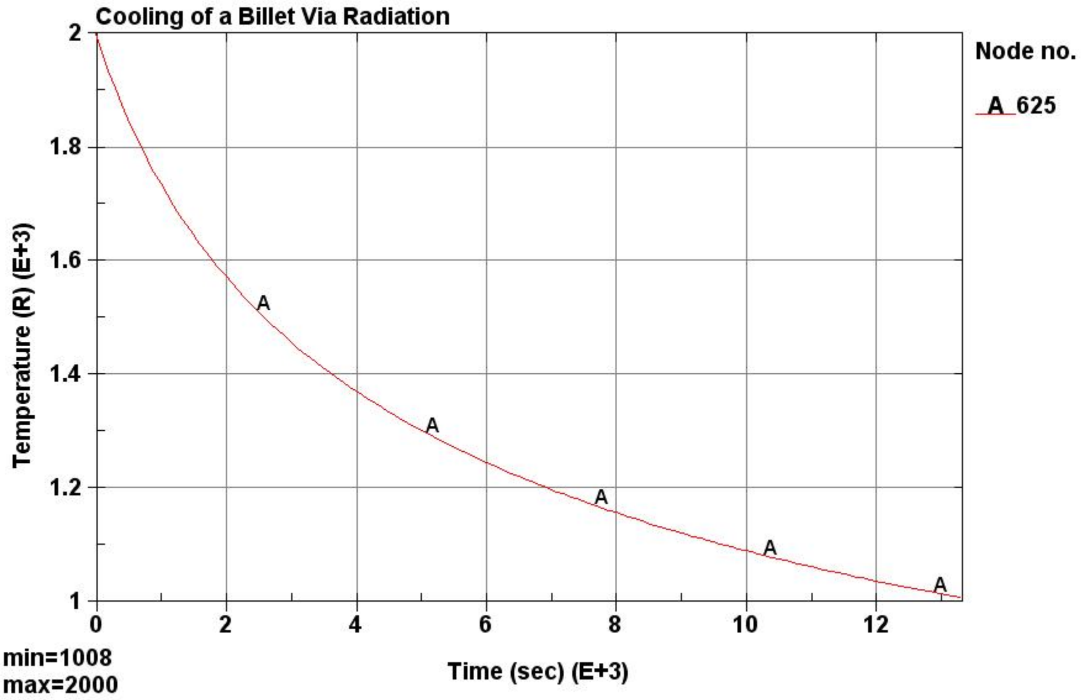

Figure 25.2 - Temperature history for Node 625.

Input deck:

```
*KEYWORD
*TITLE
Cooling of a Billet Via Radiation
*CONTROL_SOLUTION
$#    soln       nlq     isnan     lcint
         1         0         0         0
*CONTROL_THERMAL_SOLVER
$#   atype     ptype    solver     cgtol       gpt    eqheat     fwork       sbc
         1         2         3  1.00e-06         1  1.000000  1.0000004.7500e-13
*CONTROL_THERMAL_NONLINEAR
$#  refmax       tol       dcp
        20 1.000e-06  0.500000
*CONTROL_THERMAL_TIMESTEP
$#      ts       tip       its      tmin      tmax     dtemp      tscp
         1  0.500000  0.100000  0.100000  100.0000  1.000000  0.500000
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
1.3320e+04         0       0.0       0.0       0.0
*DATABASE_TPRINT
$#      dt    binary      lcur     ioopt
  10.00000         0         0         1
*DATABASE_HISTORY_NODE
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1        13        79       265
*DATABASE_BINARY_D3PLOT
$#      dt      lcdt      beam     npltc    psetid
  100.0000         0         0         0         0
*PART
$# title
Part          1 for TMat        1 and Elem Type         2
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         0         0         0         0         0         1
*SECTION_SOLID
$#   secid    elform       aet
         1         2         1
*MAT_THERMAL_ISOTROPIC
$#    tmid       tro     tgrlc    tgmult
         1  487.5000       0.0       0.0
$#      hc        tc
   0.11000 1.000e+04
*BOUNDARY_RADIATION_SET
$#    ssid      type
         1         1
$#   flcid     fmult    tilcid    timult       loc
         04.7500e-13         0  530.0000         0
*INITIAL_TEMPERATURE_SET
$#    nsid      temp       loc
         1 2000.0000         0
*SET_NODE_LIST_TITLE
all
$#     sid       da1       da2       da3       da4
         1       0.0       0.0       0.0       0.0
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1         2         3         4         5         6         7         8

       633       634       635       636       637
*SET_SEGMENT_TITLE
rad_surf
$#     sid       da1       da2       da3       da4
         1       0.0       0.0       0.0       0.0
$#      n1        n2        n3        n4        a1        a2        a3        a4
        92        93         2         1       0.0       0.0       0.0       0.0

       545       546       637       636       0.0       0.0       0.0       0.0
*ELEMENT_SOLID
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1      92     105      14       2      93     106      15
```

```
     432       1      532      623      636      545      533      624      637      546
*NODE
$#   nid               x               y               z      tc      rc
       1             0.0             0.0             0.0

     637      2.00000000      2.00000000      4.00000000
*END
```

Notes:

1. The problem must be flagged as nonlinear if any boundary condition parameter is a function of temperature *T*. This includes a linear (i.e., straight line) relationship. Iterations are needed to obtain the correct solution. Radiation (*q*) is a T^4 temperature boundary condition (usually *F*=1):

$$q = \sigma \varepsilon F (T_{surface}^4 - T_\infty^4)$$

2. The *CONTROL_THERMAL_NONLINEAR keyword is optional. For example, the default values for remax (maximum number of iterations allowed per time step), tol (temperature convergence tolerance), and dcp (divergence control tolerance) will be used, if the nonlinear keyword is omitted, with ptype>0 on *CONTROL_ THERMAL_SOLUTION keyword.

3. This study could have been performed using a single element with the following modifications to the above input deck:

```
*SET_NODE_LIST_TITLE
all
         1       0.0       0.0       0.0       0.0
         1         2         3         4         5         6         7         8
*SET_SEGMENT_TITLE
   ext_surf
$#     sid       da1       da2       da3       da4
         1       0.0       0.0       0.0       0.0
$#      n1        n2        n3        n4        a1        a2        a3        a4
         2         3         7         6       0.0       0.0       0.0       0.0
         7         8         5         6       0.0       0.0       0.0       0.0
         4         8         7         3       0.0       0.0       0.0       0.0
         2         6         5         1       0.0       0.0       0.0       0.0
         4         1         5         8       0.0       0.0       0.0       0.0
         1         4         3         2       0.0       0.0       0.0       0.0
*ELEMENT_SOLID
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1       2       3       4       5       6       7       8
*NODE
       1             0.0             0.0             0.0
       2      2.00000000             0.0             0.0
       3      2.00000000      2.00000000             0.0
       4             0.0      2.00000000             0.0
       5             0.0             0.0      4.00000000
       6      2.00000000             0.0      4.00000000
       7      2.00000000      2.00000000      4.00000000
       8             0.0      2.00000000      4.00000000
```

This single element representation provides the identical temperature result at t=3.7 hours ($1.3320 \times 10^4 \ sec$) as the 432 element mesh.

26. Pipe Whip

Keywords:

```
*CONTROL_CONTACT
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TITLE
*MAT_PLASTIC_KINEMATIC
*MAT_RIGID
*INITIAL_VELOCITY_GENERATION
*CONSTRAINED_EXTRA_NODES_SET
```

Description:

This problem illustrates the capabilities of LS-DYNA in a high speed, large deformation event with complex contact conditions, e.g. a pipe-on-pipe impact.

The pipes are modeled using fully integrated shell elements.

The impacted pipe is fully restrained, translationally and rotationally, at both ends ($x = 0$ and $x = L_1$: $U_X = U_Y = U_Z = R_X = R_Y = R_Z = 0$), while the impacting pipe is rotating at an initial angular speed of 75 *rad/s* about a fixed point at one end (Figures 26.1 and 26.2).

The pipe material is elastic-perfectly plastic, and the material model *MAT_PLASTIC_KINEMATIC with zero tangent modulus is appropriate.

The initial rotational velocity is imposed through the keyword *INITIAL_VELOCITY_GENERATION. A rigid, rotational end joint is defined using the pipe's end ring of nodes which are made rigid using the *CONSTRAINED_EXTRA_NODES_SET and *MAT_RIGID keywords.

The contact is *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE. This contact has the following characteristics:

- it is a two-way contact, in that user-specified slave nodes are checked for penetration of the master segments and then a second time, to check the master-side nodes for penetration through the slave segments,
- the treatment is thus symmetric and the definition of the slave surface and master surface is arbitrary,
- AUTOMATIC contacts check for penetration on either side of a shell element,
- this is a recommended contact type in large deformation application, e.g. in crash simulations, since the orientation of parts relative to each other cannot always be anticipated.

Shell thickness is considered with option shlthk=1. The soft=2 option (segment based contact) is used to distribute the contact forces over the elements.

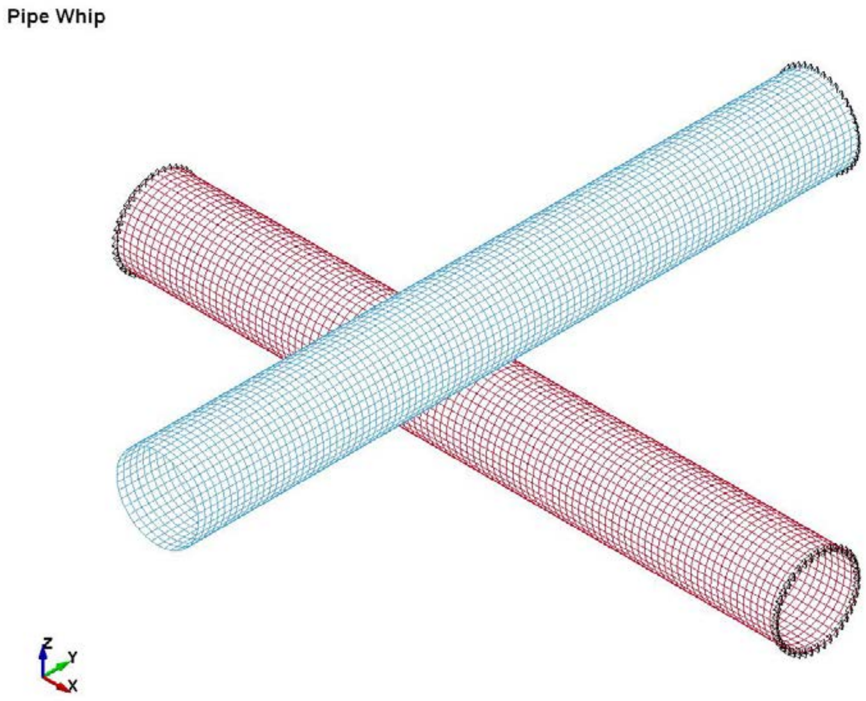

Figure 26.1 – Finite element model with boundary condition nodes (marked with□'s). There are100 elements axially and 40 elements circumferentially.

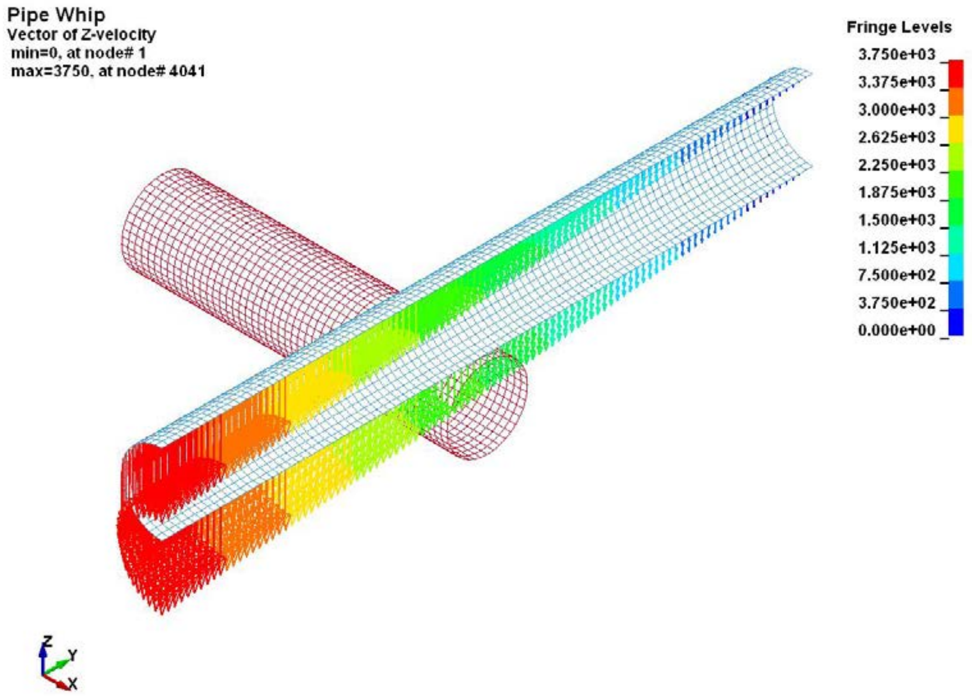

Figure 26.2 – Half-symmetry finite element model of a 50 in pipe with initial angular velocity (75 *rad/s*).

Analysis Summary:

Dim.	**Type**	**Load**	**Material**	**Geometry**	**Contact**	**Solver**	**Solution Method**
3D	Dynamic	Velocity	Non-linear	Linear	3D	Explicit	

Units:

lbf-s^2/in, in, s, lbf, psi, lbf-in (blob, inch, second, pound force, pound force/inch^2, pound force-inch)

Dimensional Data:

$L_1 = L_2 = 5.000\times10^{2}\ in$, $t = 4.320\times10^{-1}\ in$

Material Data:

Mass Density	$\rho = 7.324\times10^{-4}\ lbf - s^2 / in^4$
Young's Modulus	$E = 3.000\times10^{7}\ lbf / in^2$
Poisson's Ratio	$\nu = 0.3$
Yield Stress	$\sigma_y = 4.500\times10^{4}\ lbf / in^2$
Tangent Modulus	$E_t = 0.000\times10^{0}\ lbf / in^2$

Load:

Velocity	$\omega = 7.500\times10^{1}\ rad / s$

Element Types:

Fully integrated shell (elform=16)

Material Models:

*MAT_003 or *MAT_PLASTIC_KINEMATIC

*MAT_020 or *MAT_RIGID

Results Comparison:

The results for deformed shapes taken from R.M. Ferencz studies on *Element-by-Element Preconditioning Techniques for Large-Scale, Vectorized Finite Element Analysis in Nonlinear Solid and Structural Mechanic,* March, 1989 (pg. 142) are reproduced here in Figure 26.3. The LS-DYNA results for deformed shapes at selected times in the simulation (Figures 26.4a and 26.4b) are in good agreement.

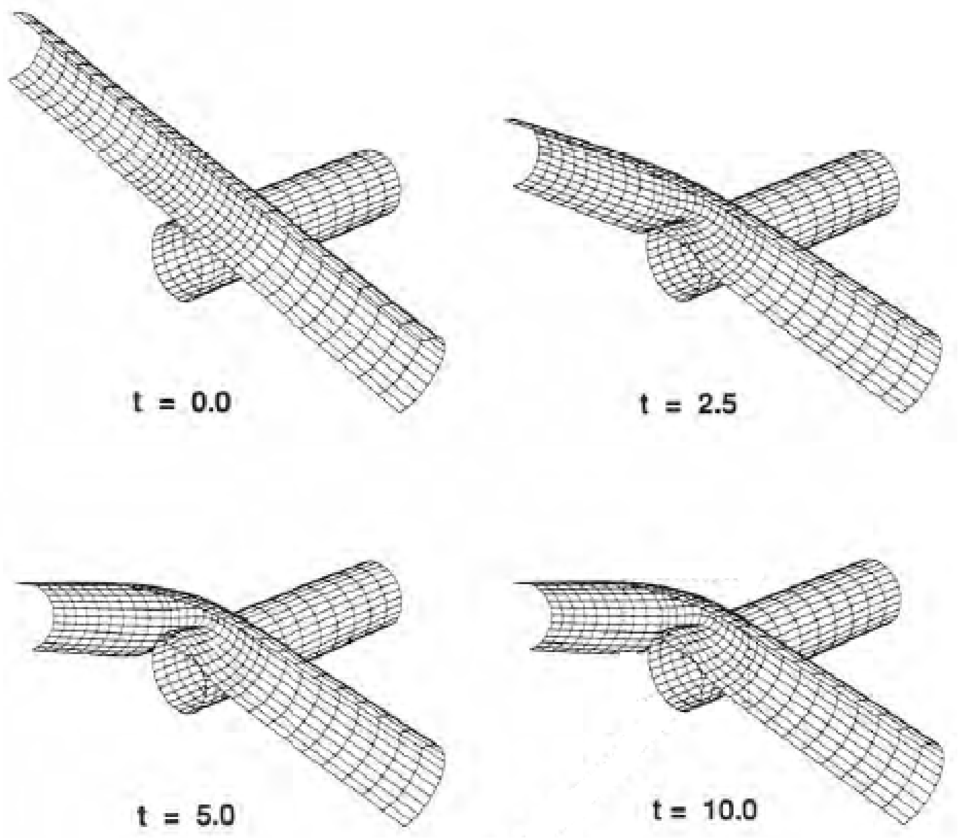

Figure 26.3 – Deformed shapes (Ferencz [1989]) at 0.0, 2.5, 5.0, and 10.0 ms.

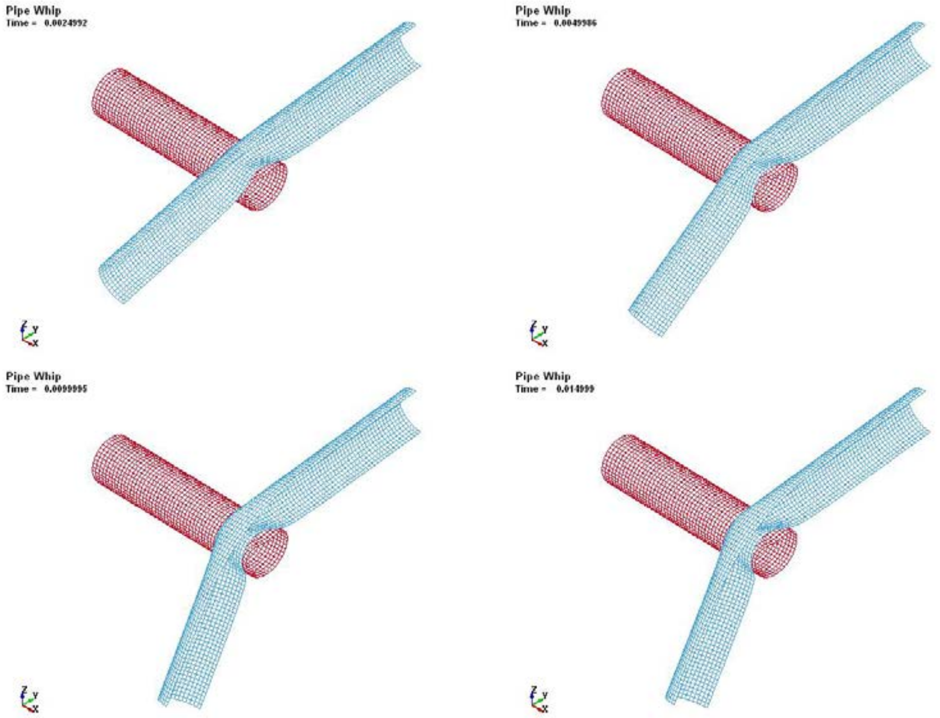

Figure 26.4a – Half-symmetry deformed shapes at 0.0025, 0.0050, 0.0100, and 0.0150 sec (hidden line view).

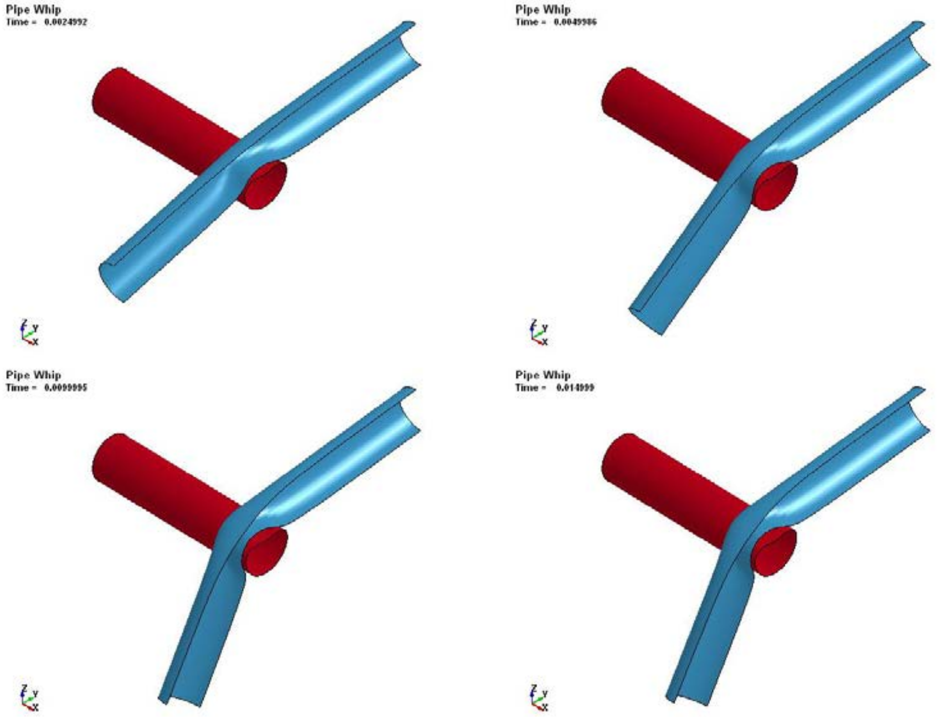

Figure 26.4b – Half-symmetry deformed shapes at 0.0025, 0.0050, 0.0100, and 0.0150 sec (shaded view).

The histories of the kinetic energy, internal energy, sliding energy, and the total energy are given in Figure 26.5.

Nearly all of the initial kinetic energy has been converted into plastic deformation (internal energy) due to the pipe deformation.

There is a small amount of energy dissipated in the contact (sliding energy) between the pipes, which, when included in the output computation, makes for an energy balance.

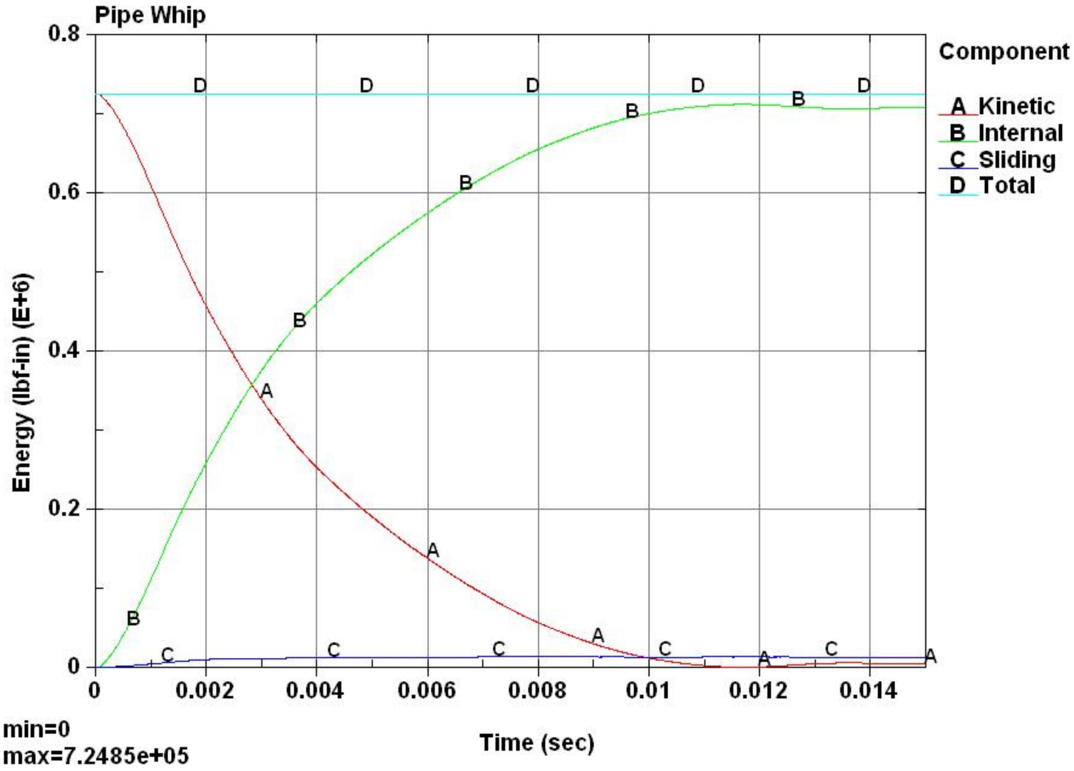

Figure 26.5 – Histories of the kinetic energy, internal energy, sliding energy, and the total energy.

Input Deck:

```
*KEYWORD
*TITLE
Pipe Whip
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  0.015000         0       0.0       0.0       0.0
*CONTROL_TIMESTEP
$#  dtinit    tssfac      isdo    tslimt     dt2ms      lctm     erode     ms1st
       0.0  0.900000
$#  dt2msf   dt2mslc     imscl
       0.0         0         0
*CONTROL_CONTACT
$#  slsfac    rwpnal    islchk    shlthk    penopt    thkchg     orien    enmass
  1.000000       0.0         2         2         0         0         1
$#  usrstr    usrfrc     nsbcs    interm     xpene     ssthk      ecdt   tiedprj
```

```
         0         0         0         0  4.000000
$#   sfric     dfric       edc       vfc        th     th_sf    pen_sf
       0.0       0.0       0.0       0.0       0.0       0.0       0.0
$#  ignore    frceng   skiprwg
         0         0         0
*CONTROL_ENERGY
$#    hgen      rwen    slnten     rylen
         2         2         2         2
*DATABASE_GLSTAT
$#      dt    binary
1.0000e-05         1
*DATABASE_MATSUM
$#      dt    binary
1.0000e-05         1
$#      dt    binary
*DATABASE_SLEOUT
1.0000e-05         1
*DATABASE_BINARY_D3PLOT
$# dt/cycl   lcdt/nr      beam     npltc    psetid
2.5000e-04
*PART
$# title
material type # 3 (Kinematic/Isotropic Elastic-Plastic)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1         0         1
*SECTION_SHELL
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
         1        16   0.83333       5.0         1       0.0         0         1
$#      t1        t2        t3        t4      nloc     marea
  0.432000  0.432000  0.432000  0.432000         0       0.0
*MAT_PLASTIC_KINEMATIC
$#     mid        ro         e        pr      sigy      etan      beta
         1 7.324e-04 3.000e+07  0.300000 4.500e+04       0.0       0.0
$#     src       srp        fs        vp
       0.0       0.0       0.0       0.0
*HOURGLASS
$#    hgid       ihq        qm       ibq        q1        q2        qb        qw
         1         0       0.0         0       0.0       0.0       0.0       0.0
*PART
$# title
material type # 3 (Kinematic/Isotropic Elastic-Plastic)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         2         2         2         0         2         0         1
*SECTION_SHELL
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
         2        16   0.83333       5.0         1       0.0         0         1
$#      t1        t2        t3        t4      nloc     marea
  0.432000  0.432000  0.432000  0.432000         0       0.0
*MAT_PLASTIC_KINEMATIC
$#     mid        ro         e        pr      sigy      etan      beta
         2 7.324e-04 3.000e+07  0.300000 4.500e+04       0.0       0.0
$#     src       srp        fs        vp
       0.0       0.0       0.0       0.0
*HOURGLASS
$#    hgid       ihq        qm       ibq        q1        q2        qb        qw
         2         0       0.0         0       0.0       0.0       0.0       0.0
*PART
$# title
material type # 20 (Rigid)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
        99        99        99
*SECTION_SHELL
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
        99         2   0.83333       1.0         1       0.0         0         1
$#      t1        t2        t3        t4      nloc     marea
  0.432000  0.432000  0.432000  0.432000         0       0.0
*MAT_RIGID
$#     mid        ro         e        pr         n    couple         m     alias
        99 7.324e-04 3.000e+07  0.300000       0.0       0.0       0.0
$#     cmo      con1      con2
  1.000000         7         5
```

```
$#lco or a1       a2        a3        v1        v2        v3
       0.0       0.0       0.0       0.0       0.0       0.0
*CONSTRAINED_EXTRA_NODES_SET
$#     pid      nsid
        99        99
*SET_NODE_LIST_TITLE
rigid ring of nodes
$#     sid       da1       da2       da3       da4    solver
        99     0.000     0.000     0.000     0.000MECH
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
      8041      8042      8043      8044      8045      8046      8047      8048
      8049      8050      8051      8052      8053      8054      8055      8056
      8057      8058      8059      8060      8061      8062      8063      8064
      8065      8066      8067      8068      8069      8070      8071      8072
      8073      8074      8075      8076      8077      8078      8079      8080
*INITIAL_VELOCITY_GENERATION
$#      id      styp     omega        vx        vy        vz     ivatn      icid
         2         2    75.000       0.0       0.0       0.0         0         0
$#      xc        yc        zc        nx        ny        nz     phase    irigid
 25.000000 50.000000  6.725000  1.000000       0.0       0.0         0         0
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TITLE
$#     cid                                                                 title
         1
$#    ssid      msid     sstyp     mstyp    sboxid    mboxid       spr       mpr
         1         2         3         3         0         0         0         0
$#      fs        fd        dc        vc       vdc    penchk        bt        dt
       0.0       0.0       0.0       0.0       0.0         0       0.0       0.0
$#     sfs       sfm       sst       mst      sfst      sfmt       fsf       vsf
       0.0       0.0       0.0       0.0       0.0       0.0       0.0       0.0
$#    soft    sofscl    lcidab    maxpar     sbopt     depth     bsort    frcfrq
         2  0.100000         0     1.025       0.0         2        10         1
$#  penmax    thkopt    shlthk     snlog      isym     i2d3d    sldthk    sldstf
       0.0         0         1         0         0         0       0.0       0.0
$#    igap    ignore
         2         0
*ELEMENT_SHELL
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1       2      42      41       0       0       0       0

    8000       2    8040    8001    8041    8080       0       0       0       0
*NODE
$#   nid               x               y               z      tc      rc
       1           0.000       28.096500           0.000       0       0

    8080       28.058376       50.000000        7.209399       0       0
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4    solver
         1     0.000     0.000     0.000     0.000MECH
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1         2         3         4         5         6         7         8
         9        10        11        12        13        14        15        16
        17        18        19        20        21        22        23        24
        25        26        27        28        29        30        31        32
        33        34        35        36        37        38        39        40
*SET_NODE_LIST
$#     sid       da1       da2       da3       da4    solver
         2     0.000     0.000     0.000     0.000MECH
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
      4001      4002      4003      4004      4005      4006      4007      4008
      4009      4010      4011      4012      4013      4014      4015      4016
      4017      4018      4019      4020      4021      4022      4023      4024
      4025      4026      4027      4028      4029      4030      4031      4032
      4033      4034      4035      4036      4037      4038      4039      4040
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         1         1         1         1         1         1
*BOUNDARY_SPC_SET
$#nid/nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         2         0         1         1         1         1         1         1
*END
```

Notes:

1. The general contact *CONTACT_AUTOMATIC_SINGLE_SURFACE could also have been used. This type of contact has the following characteristics:
   - a single contact surface is created for all the parts included in the contact,
   - self contact is considered,
   - it is robust, reliable and accurate, making it the ideal choice for crashworthiness and impact applications.

   By default, if ssid (slave segment id) is zero or blank, all part IDs are included in the contact. A *PART_SET entry can be used to reduce the size of the part list.

2. The most common contact-related output file, *rcforc*, is produced by including a *DATABASE_RCFORC keyword in the input deck. *rcforc* is an ASCII file containing resultant contact forces for the slave and master sides of each contact interface. The forces are provided in the global coordinate system. Note that *rcforc* data is not provided for single surface contacts as all the contact forces from this contact type come from the slave side (as there is no master side) and thus the net contact forces are zero. To obtain *rcforc* data when single surface contacts are used, one or more *force transducers* should be added via the *CONTACT_FORCE_TRANSDUCER_PENALTY keyword. A force transducer simply measures contact forces produced by other contact interfaces defined in the model.

3. By including a *DATABASE_SLEOUT keyword, individual contact interface energies are written to the ASCII output file *sleout.* The global contact energy is written to the ASCII output file *glstat*.

27. Copper Bar Impacting a Rigid Wall

Keywords:

```
*CONTROL_ALE
*CONTROL_CONTACT
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE
*RIGIDWALL_PLANAR
*INITIAL_VELOCITY_GENERATION
*SECTION_SOLID
*HOURGLASS
```

Description:

This problem is known in the literature as "Taylor Bar Impact Test" and is used to assess material properties (plastic flow) under dynamic conditions. A deformable copper bar impacts a rigid wall at high speed. The deformed length (shortening), spread (widening), and maximum effective plastic strain (ε^p) of the bar is determined.

The contact of the deformable body and the rigid wall can be modeled in one of the following ways:

- rigid wall (*RIGIDWALL_PLANAR), which provides an easy way to treat contact between a rigid-flat surface and the nodes of a deformable body,
- using geometric entities (*CONTACT_ENTITY),
- using a wall modeled with rigid shell elements and a *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE.

Two of these methods are demonstrated for rigid wall contact, (1) the rigid wall is modeled with rigid shell elements (penalty method) and (2) the wall is modeled as a planar rigid boundary. The latter uses a constraint method which represents a perfectly plastic impact since, once penetration into the rigid wall is detected, the acceleration and velocity of the nodes are set to zero. No friction is included.

The material is elastic-plastic with constant tangent stiffness and the material model *MAT_PLASTIC_KINEMATIC is used.

For comparison, different solid hexahedron (elform=1,2,-2,-1) and tetrahedron (elform=10,13) elements, are used to model the bar as shown in Figure 27.1. Hourglass control is used with the default value (ihq=2 - Flanagan-Belytschko viscous form with coefficient qm=0.10) for the one-point quadrature element formulations.

Traditional Lagrangian approaches, for large deformations, often result in highly distorted meshes for the elements close to the impacted region, leading to loss of accuracy and decreasing the critical time step for the simulation. Therefore, a simple Arbitrary Lagrangian-Eulerian (ALE) formulation (elform=5) is also presented. A mix of simple average smoothing and volume weighted smoothing is used for the interior nodal positioning.

The (elform=5) formulation is single material ALE with Lagrange outer boundary node treatment and mesh smoothing effective for only moderate deformation. Thus application of this formulation is limited to mostly academic problems, since there are not many practical applications for use of this feature. In fact, this "Taylor Bar Impact Test" may be the only known practical application.

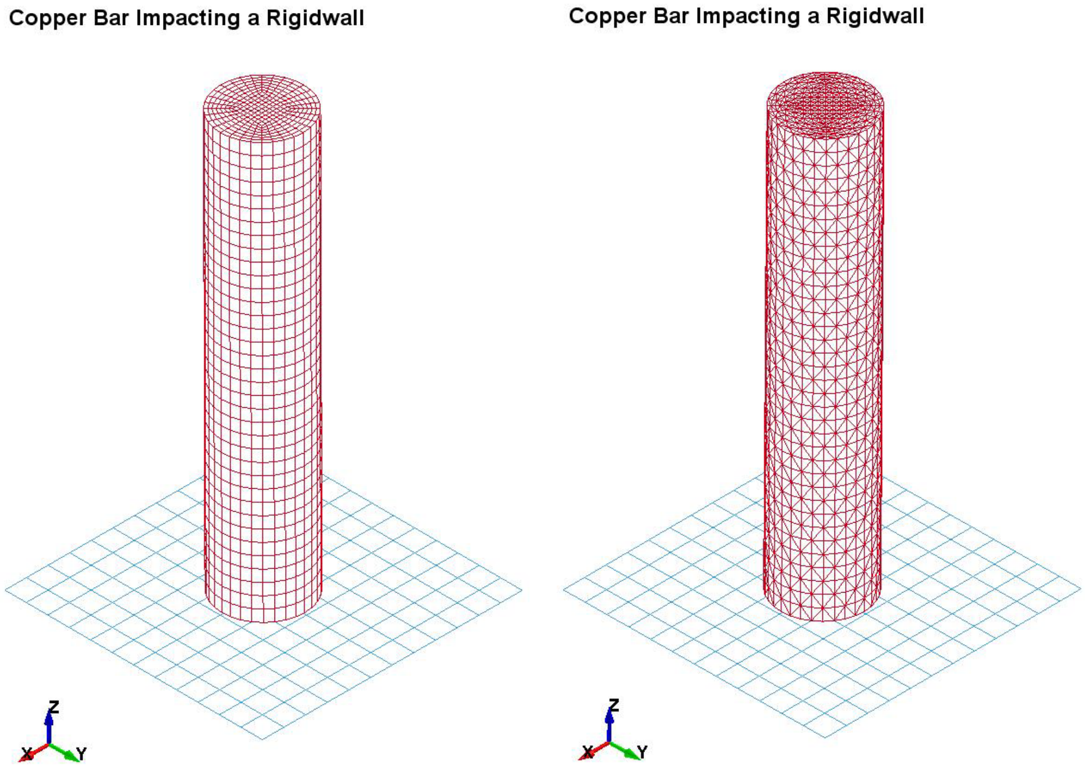

Figure 27.1 – Finite element models of the impacting bar using hexahedron and tetrahedron elements (shell elements were used to model the impacted rigid plate). Each of these solid element meshes has 36 elements axially. There are 288 and 1440 elements per row for the hexahedron and tetrahedron models, respectively.

Analysis Summary:

Dim.	**Type**	**Load**	**Material**	**Geometry**	**Contact**	**Solver**	**Solution Method**
3D	Dynamic	Velocity	Non-linear	Non-linear	3D	Explicit	Lagrangian and ALE

Units:

g, mm, ms, N, MPa, N-mm (gram, millimeter, millisecond, Newton, MegaPascal, Newton-millimeter)

Dimensional Data:

$L = 3.240 \times 10^1\ mm$, $d = 6.400 \times 10^0\ mm$

Material Data:

Mass Density	$\rho = 8.930 \times 10^{-3}\ g / mm^3$
Young's Modulus	$E = 1.170 \times 10^5\ MPa$
Poisson's Ratio	$\nu = 0.35$
Yield Stress	$\sigma_y = 4.000 \times 10^2\ MPa$
Tangent Modulus	$E_t = 1.000 \times 10^2\ MPa$

Load:

Velocity	$V_z = 2.270 \times 10^2\ mm / ms$

Element Types:

Constant stress solid (elform=1)
Fully integrated S/R solid (elform=2)
Fully integrated S/R solid - for poor aspect ratio (acc) - (elform=-2)
Fully integrated S/R solid - for poor aspect ratio (eff) - (elform=-1)
1 point tetrahedron (elform=10)
1 point nodal pressure tetrahedron (elform=13)
1 point ALE (elform=5)

Material Models:

*MAT_003 or *MAT_PLASTIC_KINEMATIC

*MAT_020 or *MAT_RIGID

Results Comparison:

The results for deformed shapes at 0.0, 5, 20, and 80 ms, taken from R.M. Ferencz studies on *Element-by-Element Preconditioning Techniques for Large-Scale, Vectorized Finite Element Analysis in Nonlinear Solid and Structural Mechanics*, March, 1989 (pg. 86), are reproduced here in Figure 27.2. Ferencz [1989] used NIKE3D and its implicit dynamics solver. The LS-DYNA results for deformed shapes at 80.0 ms (Figures 27.3a to 27.3g) using penalty method contact and rigid boundary contact (Figures 27.7a to 27.7g) are in good agreement. The maximum effective plastic strain ($\varepsilon^p = 2.248$) given by Ferencz [1989] differs significantly from the non-ALE results of LS-DYNA

($\varepsilon^p \cong 2.9 \text{ to } 3.9$), even though both use a Lagrangian approach; these values are taken from the highly distorted elements in the vicinity of the rigid wall.

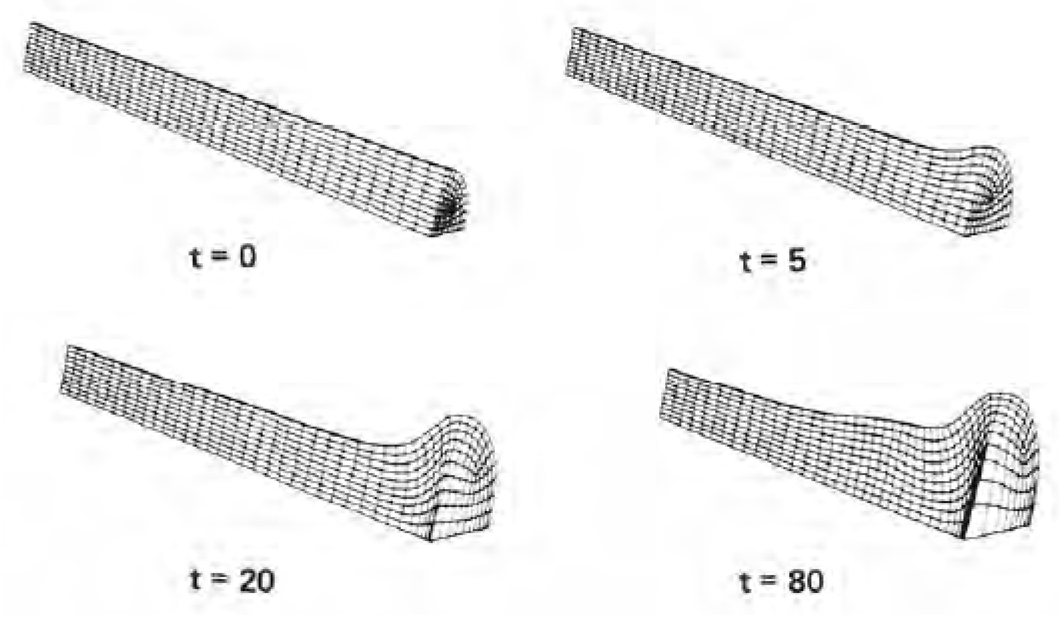

Figure 27.2 – Deformed shapes (Ferencz [1989]) at 0.0, 5, 20, and 80 ms.

Penalty Method (Rigid Mesh Material)	**Shortening (mm)**	**Widening (mm)**	**Max. plastic strain (ε^p)**	**Normalized CPU Time**
Constant stress solid (elform=1)	10.928	8.125	3.523	1.40
Fully integrated S/R solid (elform=2)	10.976	8.395	3.671	5.20
Fully integrated S/R solid (elform=-2)	10.972	8.404	3.663	19.40
Fully integrated S/R solid (elform=-1)	10.976	8.418	3.700	6.40
1 point tetrahedron (elform=10)	11.088	7.537	2.944	3.00
1 point nodal pressure tetrahedron (elform=13)	11.017	8.533	3.924	10.50
1 point ALE (elform=5)	10.892	7.856	2.272	3.70

The above displacement and effective plastic strains results were obtained from the d3plot contour plots at 80.0 ms which were generated by the *DATABASE_BINARY_ D3PLOT keyword.

Normalized CPU times shown in the above Penalty Method results table were normalized using the minimum value (the smallest value for all simulations - other contact type CPU times are to follow).

Large effective plastic strains develop at the impact end of the rod due to the severe local mesh distortion, also resulting in reduced accuracy.

For these simulations, a wide range of CPU times were associated with the different element formulations. The CPU time is controlled by the number of element operations required for that particular formulation, the complexity of the contact-impact approach and the element stable time step.

The one-point quadrature (low order) constant stress solid (elform=1) element formulation (the LS-DYNA default), the higher order, fully integrated selectively reduced solid (elform=2), and the higher order, fully integrated S/R solid (both so-called efficient and accurate formulation choices) intended to address poor aspect ratios (elform=-1 and -2, respectively), provide roughly the same dimensional changes and maximum effective plastic strain.
The one-point quadrature (low order) tetrahedron (elform=10) element formulation provides comparatively stiffer dimensional changes and maximum effective plastic strain than the constant stress solid and fully integrated element formulations. This is probably due to this element formulation being prone to volumetric locking (overly stiff behavior) in incompressible regimes, e.g., as in plasticity.

The one-point quadrature (low order) nodal pressure tetrahedron (elform=13) element formulation provides a less stiff, dimensional changes and maximum effective plastic strain comparison to that of the constant stress solid and fully integrated element formulations. This element formulation has no volumetric locking under plastic incompressible conditions.

The one point ALE (elform=5) element formulation provides similar dimensional changes to other element formulations. With its nodal smoothing capability controlling the aspect ratio of the elements, mesh distortion is reduced, yet a smaller maximum effective plastic strain ($\varepsilon^{\boldsymbol{p}} = 2.272$) is achieved compared to the Lagrangian elements ($\varepsilon^{\boldsymbol{p}} \cong 2.9 \; to \; 3.9$). An explanation for these results is the moderate deformation limitation for the one point ALE formulation.

The LS-DYNA results for deformed shapes at 80.0 ms using penalty method contact with effective plastic strain contouring are given Figures 27.3a to 27.3g.

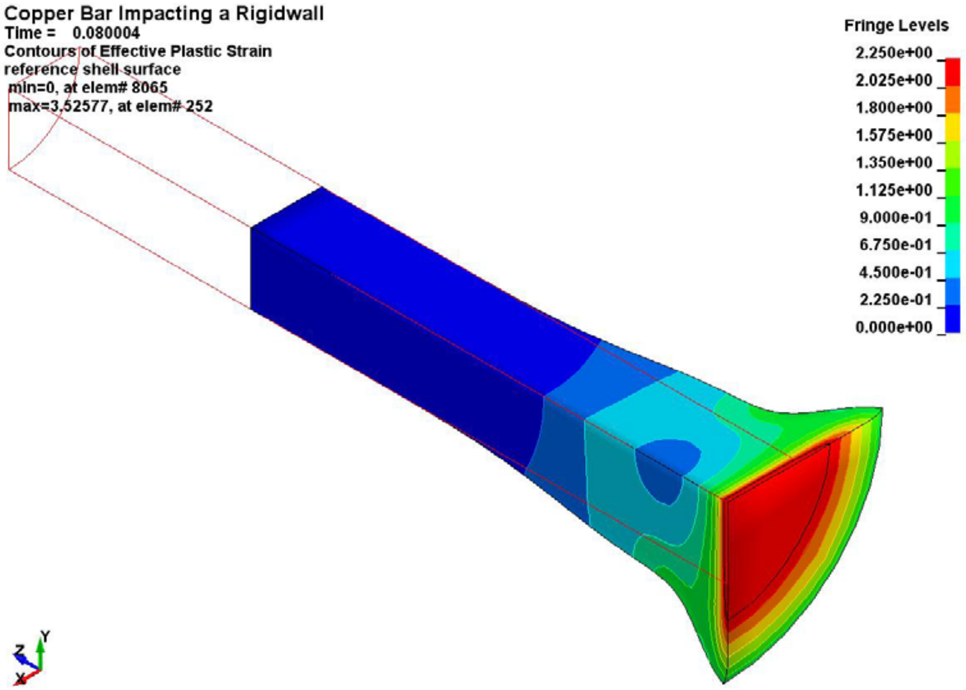

Figure 27.3a – Quarter-symmetry deformed shape (penalty method) with effective plastic strain contouring at 80 ms (elform=1).

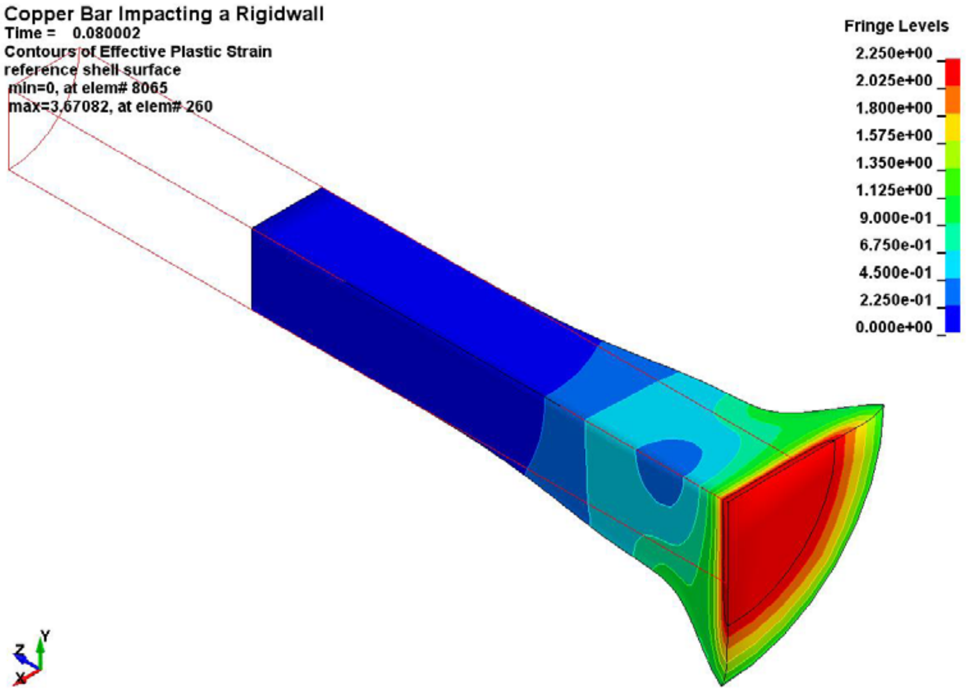

Figure 27.3b – Quarter-symmetry deformed shape (penalty method) with effective plastic strain contouring at 80 ms (elform=2).

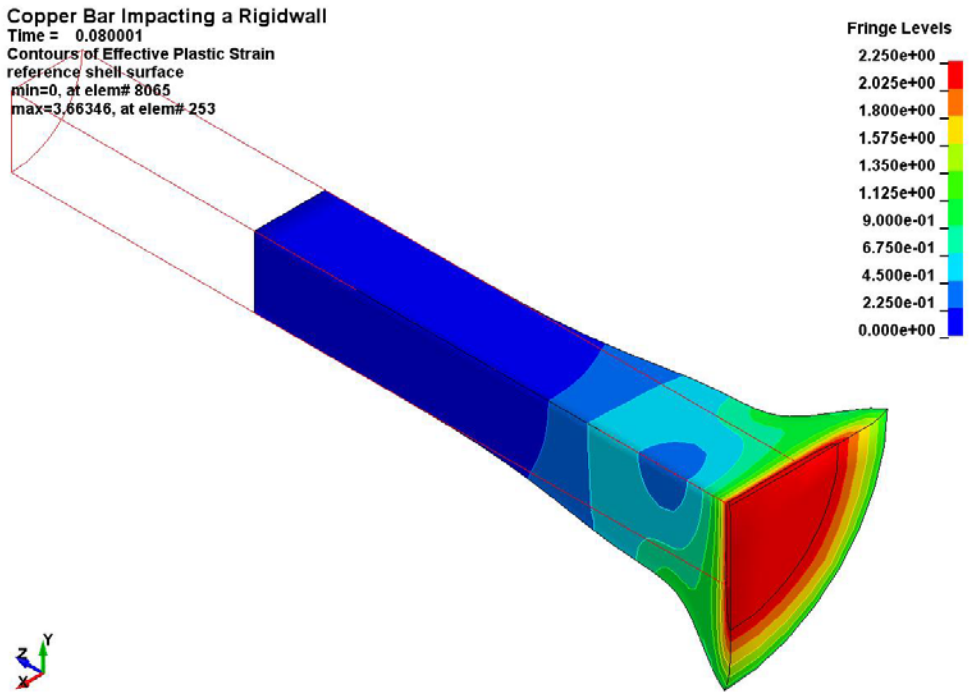

Figure 27.3c – Quarter-symmetry deformed shape (penalty method) with effective plastic strain contouring at 80 ms (elform=-2).

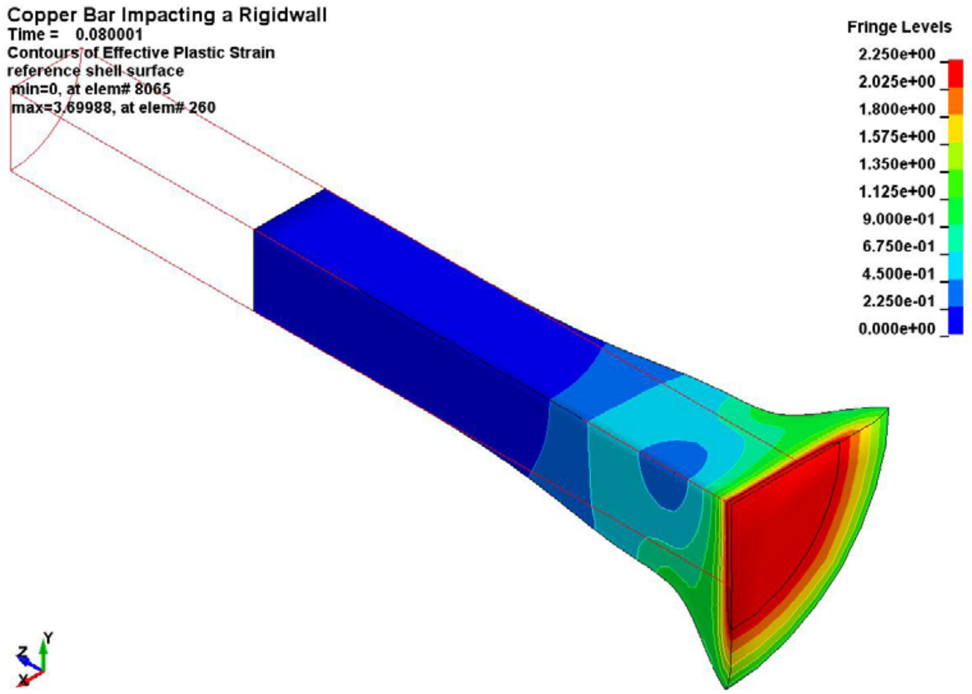

Figure 27.3d – Quarter-symmetry deformed shape (penalty method) with effective plastic strain contouring at 80 ms (elform=-1).

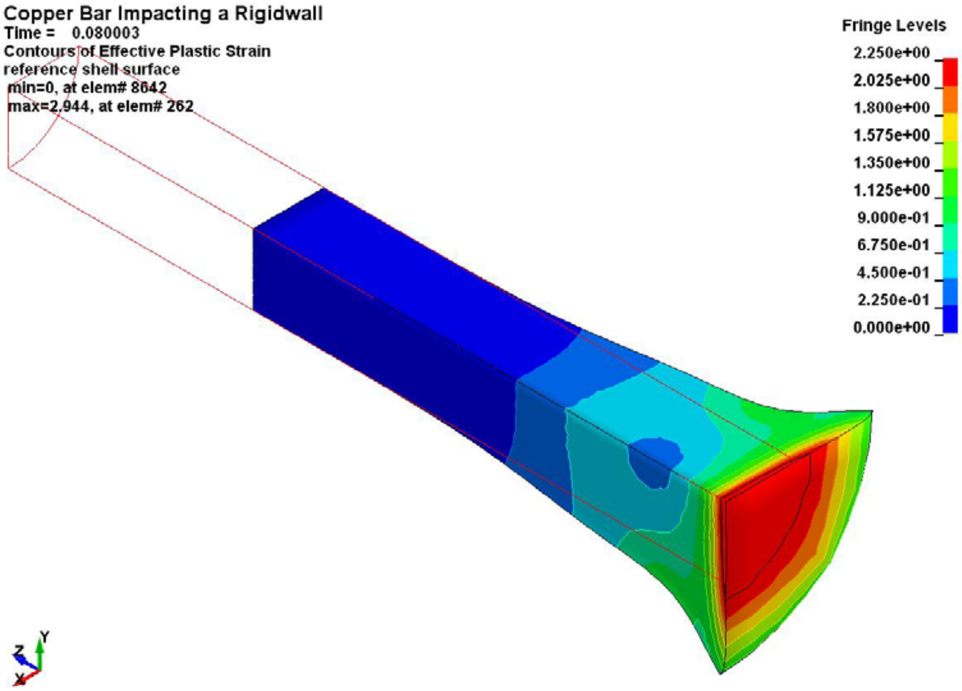

Figure 27.3e – Quarter-symmetry deformed shape (penalty method) with effective plastic strain contouring at 80 ms (elform=10).

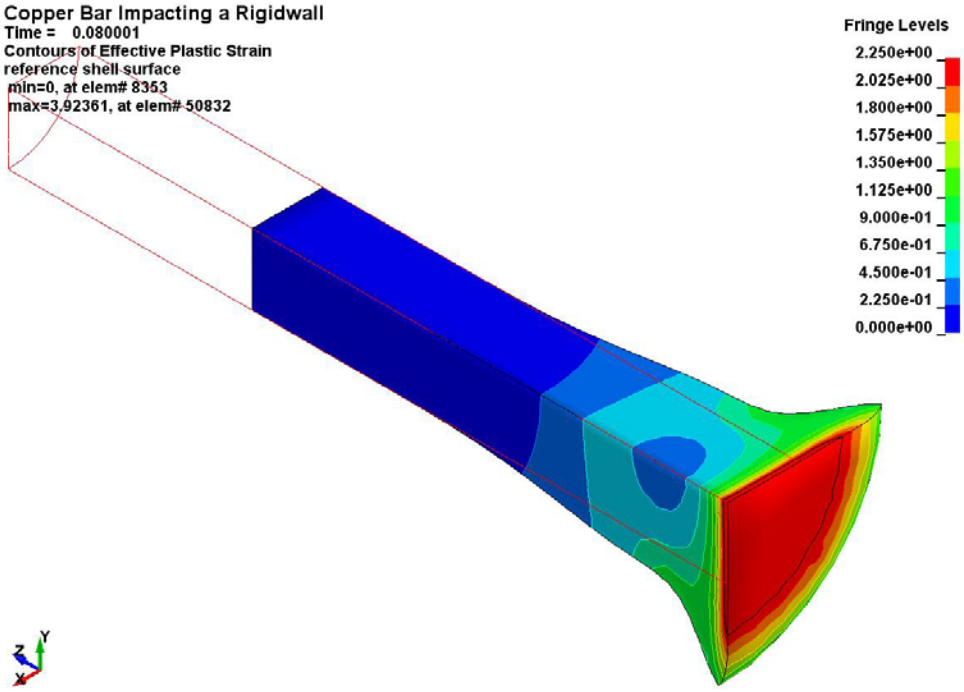

Figure 27.3f – Quarter-symmetry deformed shape (penalty method) with effective plastic strain contouring at 80 ms (elform=13).

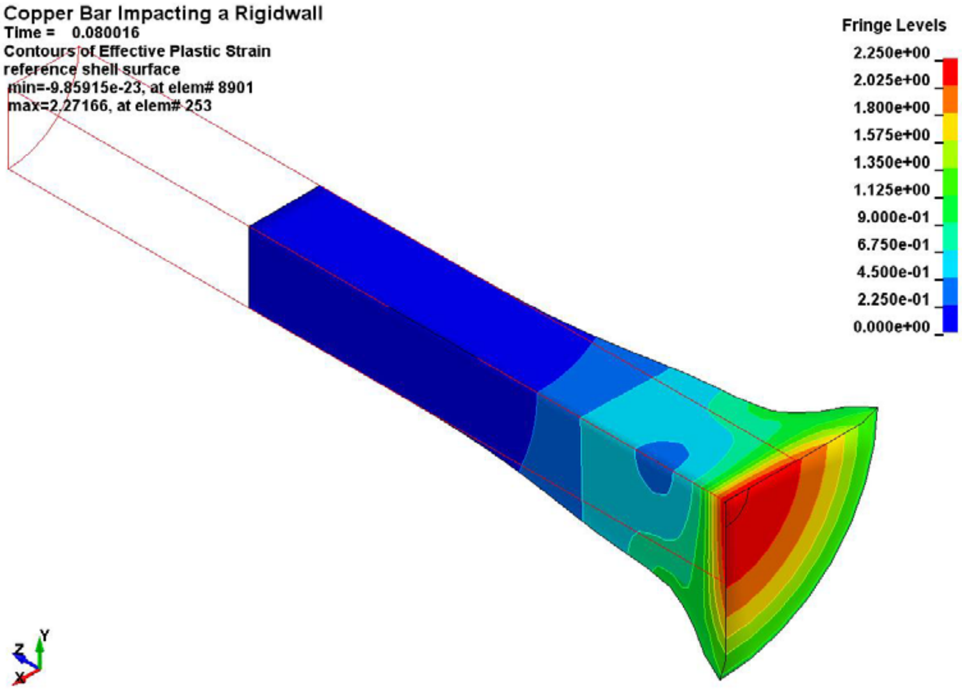

Figure 27.3g – Quarter-symmetry deformed shape (penalty method) with effective plastic strain contouring at 80 ms (elform=5).

The half-symmetry deformed shape (penalty method contact), which illustrates the different element deformation for elform=1 and elform=5 at 80 ms, is given in Figure 27.4.

Figure 27.4 – Half-symmetry deformed shape (penalty method) for elform=1 and elform=5 at 80 ms.

The histories of the kinetic energy, internal energy, hourglass energy, sliding energy, and the total energy for (elform=1) of penalty method impact are given in Figure 27.5, while the histories of the stable time step increment for all elforms (1,2,-2,-1,10,13,5) associated with the penalty method impact are given in Figure 27.6.

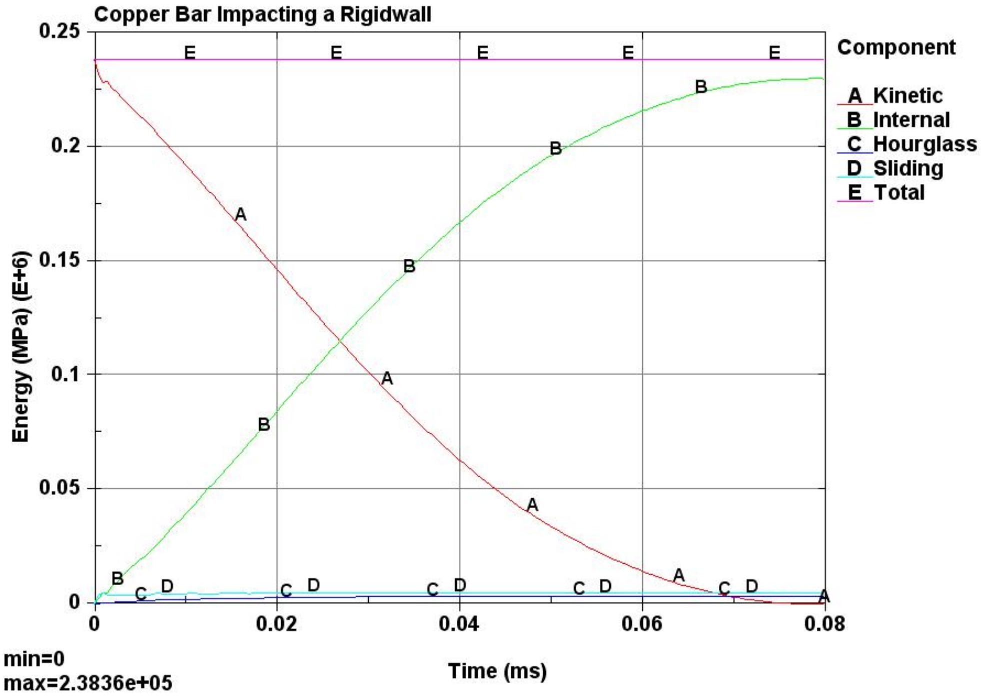

Figure 27.5 – Histories of the kinetic energy, internal energy, hourglass energy, sliding energy, and the total energy for (elform=1) of penalty method impact.

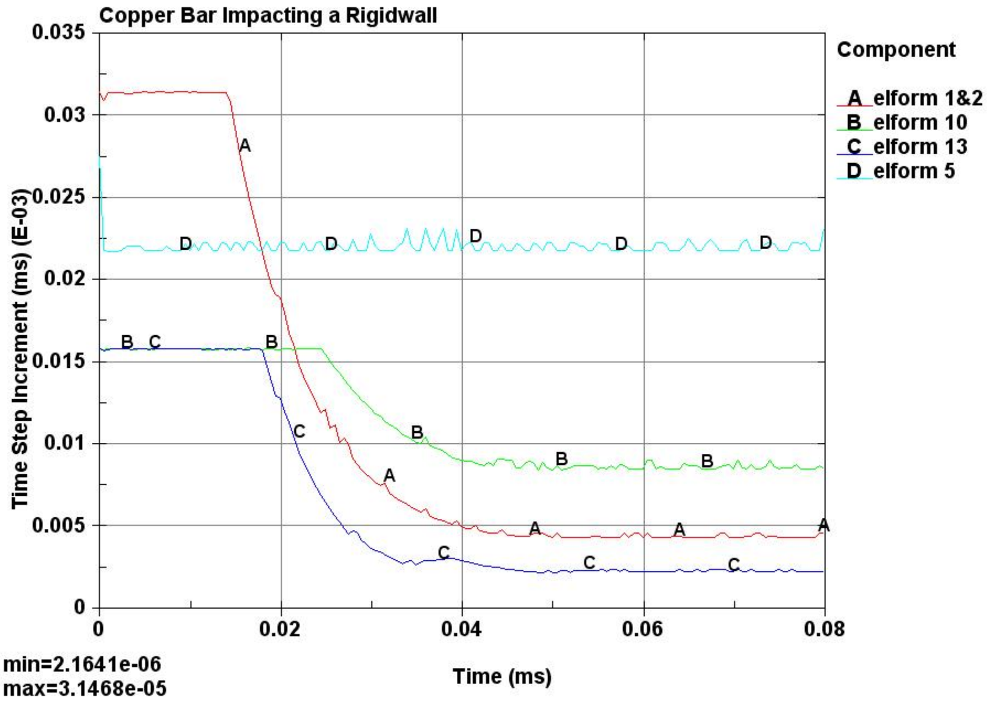

Figure 27.6 – Histories of the stable time step increment for all elforms (1,2,-2,-1,10,13,5) associated with the penalty method impact.

Planar Rigid Boundary	**Shortening (mm)**	**Widening (mm)**	**Max. plastic strain (ε^p)**	**Normalized CPU Time**
Constant stress solid (elform=1)	10.897	7.889	3.243	1.00
Fully integrated S/R solid (elform=2)	10.936	8.139	3.366	3.40
Fully integrated S/R solid (elform=-2)	10.933	8.182	3.394	14.00
Fully integrated S/R solid (elform=-1)	10.936	8.183	3.405	4.10
1 point tetrahedron (elform=10)	11.044	7.201	3.057	2.70
1 point nodal pressure tetrahedron (elform=13)	10.987	8.214	4.288	9.00
1 point ALE (elform=5)	10.886	7.716	2.272	3.60

The above displacement and effective plastic strains results were obtained from the d3plot contour plots at 80.0 ms which were generated by the *DATABASE_BINARY_ D3PLOT keyword.

The more efficient rigid boundary contact procedure requires less CPU time (20% to 60%) than the penalty method for contact-impact. The exception is the one point ALE multi-material formulation, where the CPU times were about the same, probably due to the smoothing operations control.

For all the element formulations (except the 1 point ALE) used, the contact-impact results provided using the penalty method and the planar rigid boundary differ due to the contact methods.

Comments provided for the penalty method results regarding element formulation CPU times, the (elform=1,2,-2,-1) similarities for dimensional changes and maximum effective plastic strain, the (elform=10) stiffer comparison, the (elform=13) less stiff comparison, and the (elform=5) similarity and difference are also appropriate for the rigid boundary contact results.

The LS-DYNA results for deformed shapes at 80.0 ms using rigid boundary contact with effective plastic strain contouring are given in Figures 27.7a to 27.3g.

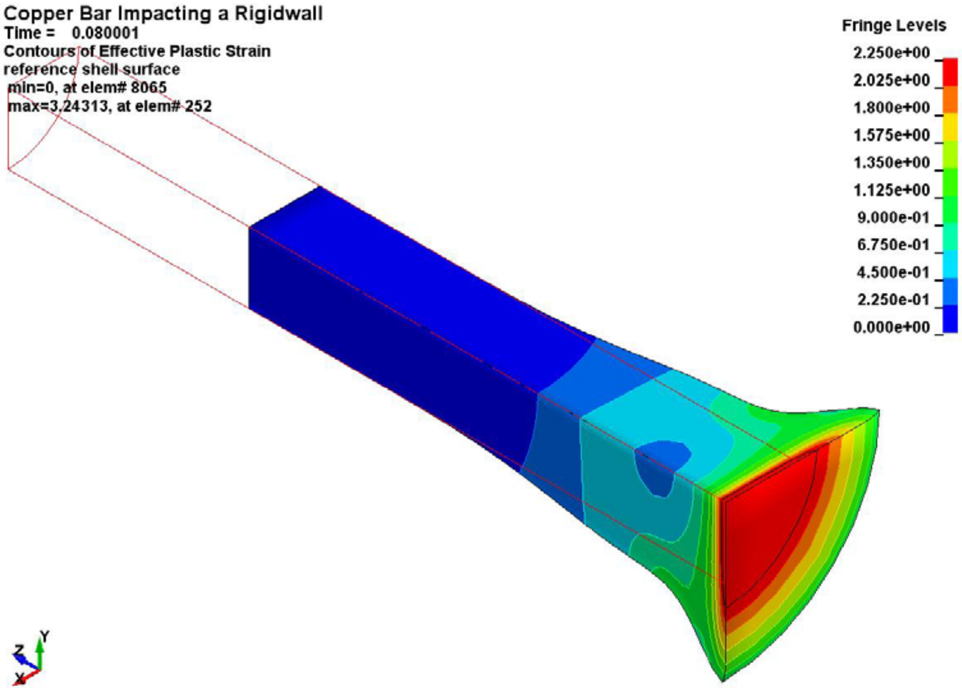

Figure 25.7a – Quarter-symmetry deformed shape (rigid boundary) with effective plastic strain contouring at 80 ms (elform=1).

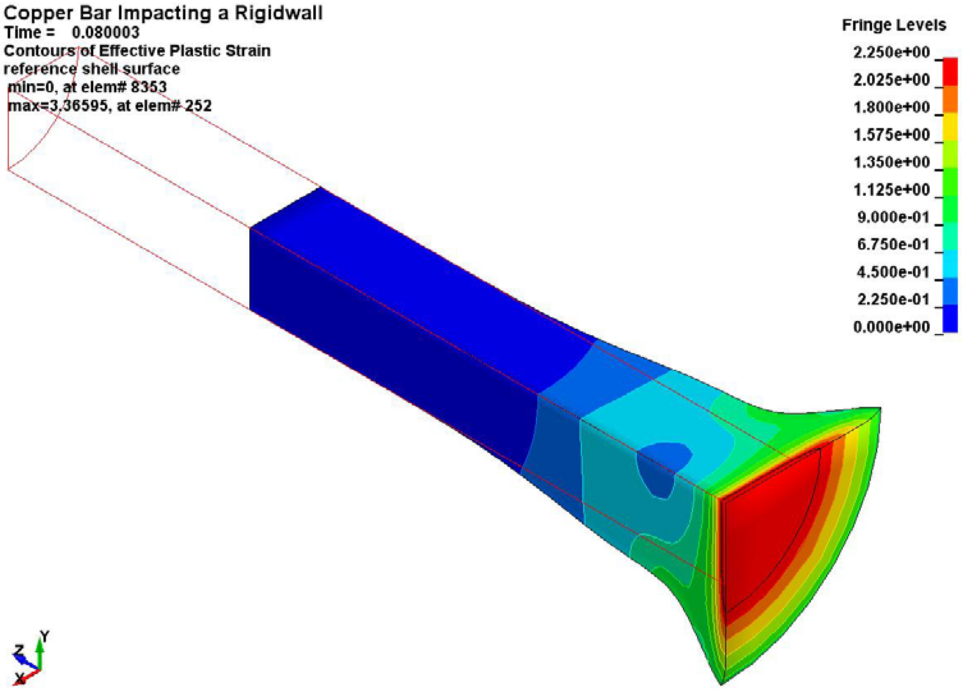

Figure 27.7b – Quarter-symmetry deformed shape (rigid boundary) with effective plastic strain contouring at 80 ms (elform=2).

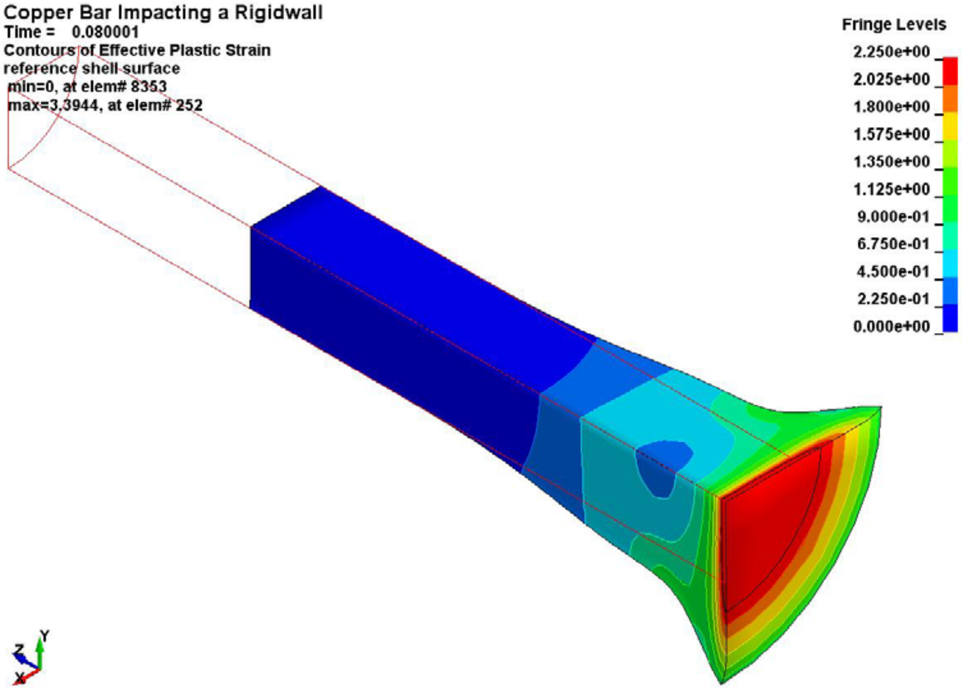

Figure 27.7c – Quarter-symmetry deformed shape (rigid boundary) with effective plastic strain contouring at 80 ms (elform=-2).

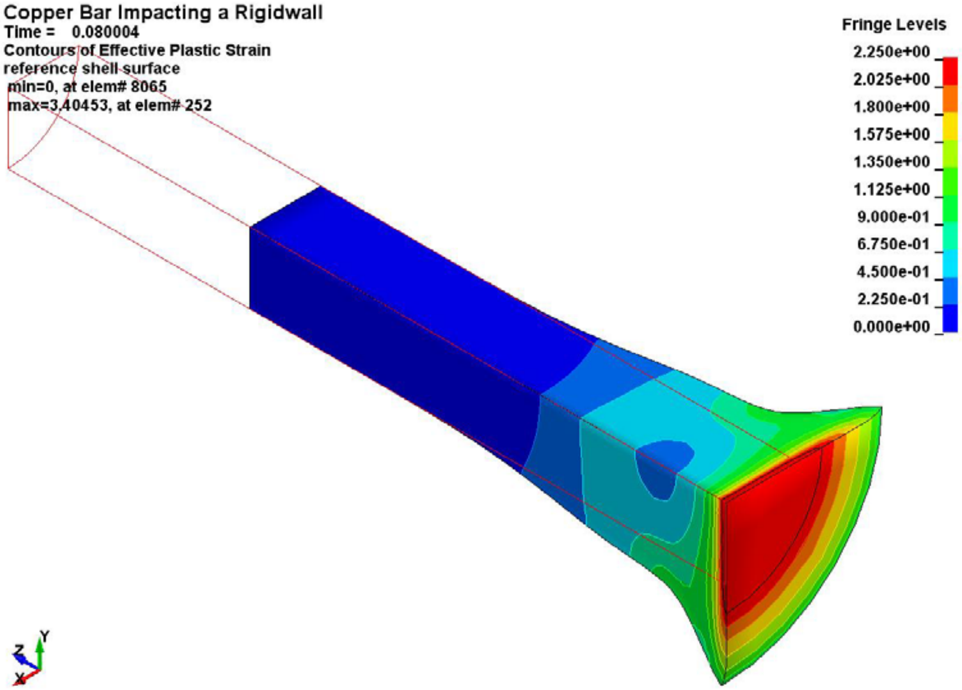

Figure 27.7d – Quarter-symmetry deformed shape (rigid boundary) with effective plastic strain contouring at 80 ms (elform=-1).

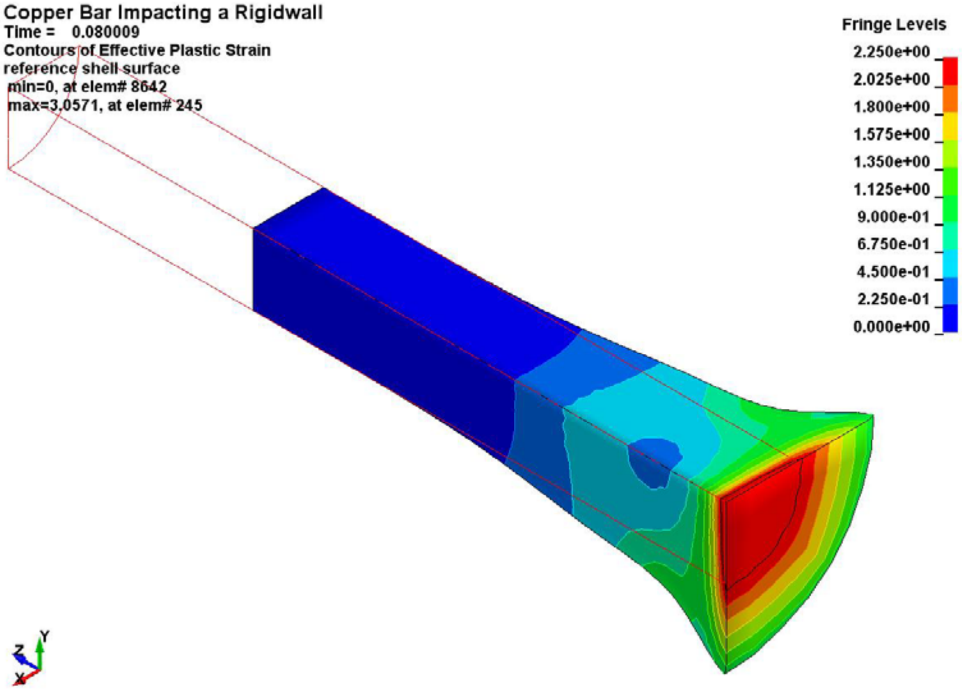

Figure 27.7e – Quarter-symmetry deformed shape (rigid boundary) with effective plastic strain contouring at 80 ms (elform=10).

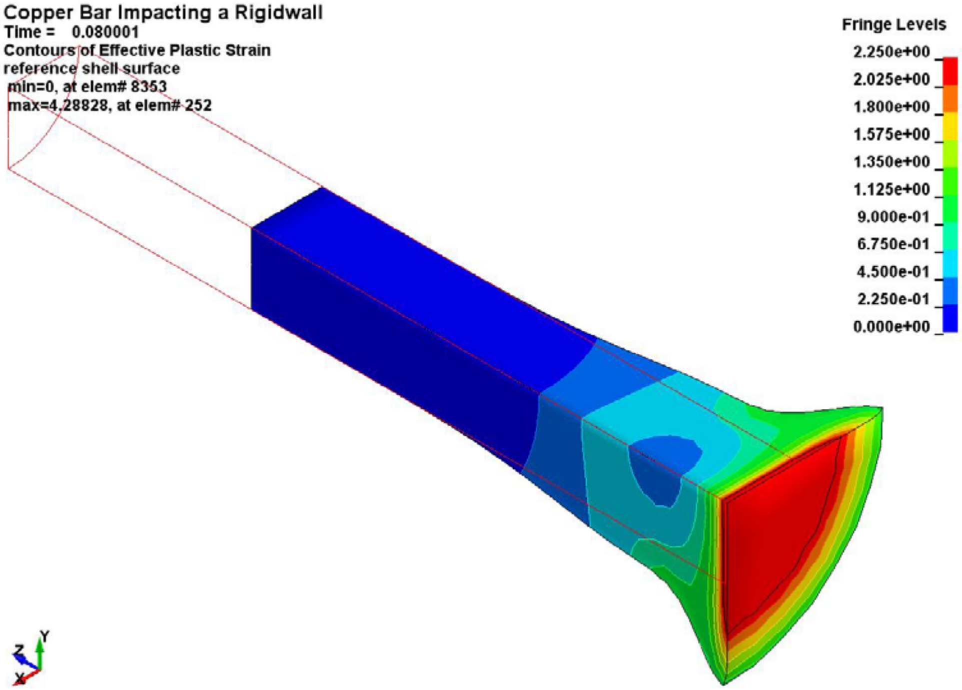

Figure 27.7f – Quarter-symmetry deformed shape (rigid boundary) with effective plastic strain contouring at 80 ms (elform=13).

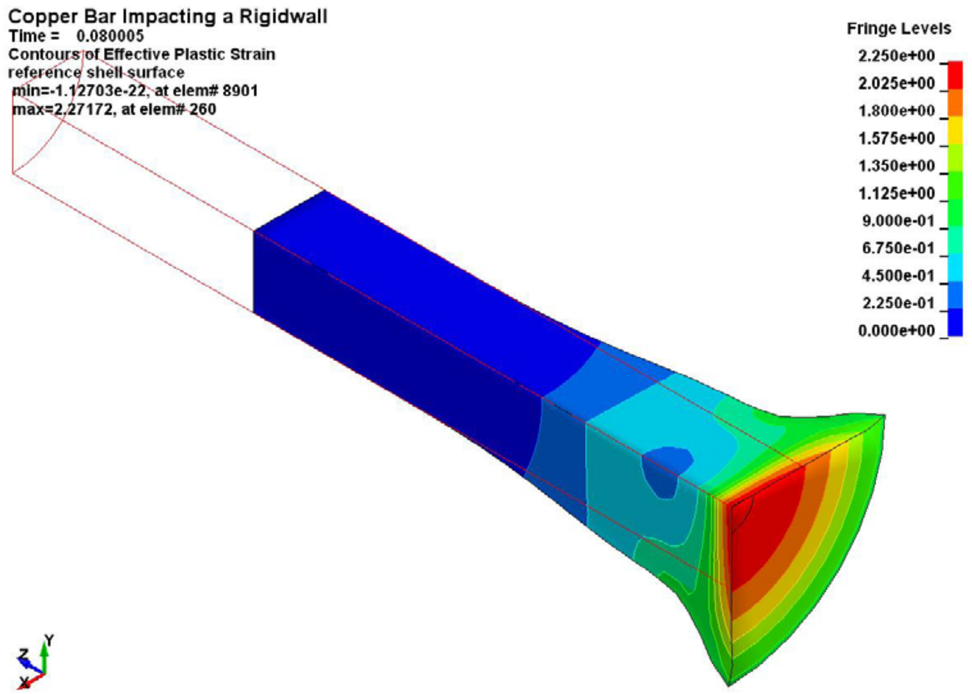

Figure 27.7g – Quarter-symmetry deformed shape (rigid boundary) with effective plastic strain contouring at 80 ms (elform=5).

The half-symmetry deformed shape (planar rigid boundary contact), which illustrates the different element deformation for elform=1 and elform=5 at 80 ms, is given in Figure 27.8.

Figure 27.8 – Half-symmetry deformed shape (rigid boundary) for elform=1 and elform=5 at 80 ms.

The histories of the kinetic energy, internal energy, hourglass energy, sliding energy, and the total energy for (elform=1) of rigid boundary impact are given in Figure 27.9, while the histories of the stable time step increment for all elforms (1,2,-2,-1,10,13,5) associated with the rigid boundary impact are given in Figure 27.10.

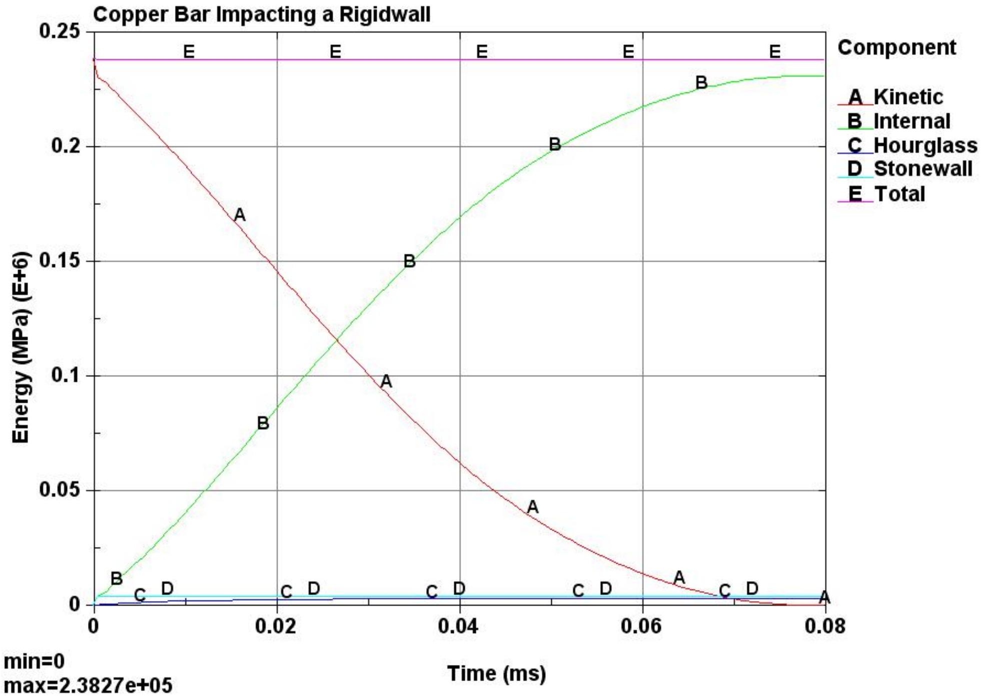

Figure 27.9 – Histories of the kinetic energy, internal energy, hourglass energy, stonewall energy, and the total energy for (elform=1) of rigid boundary impact.

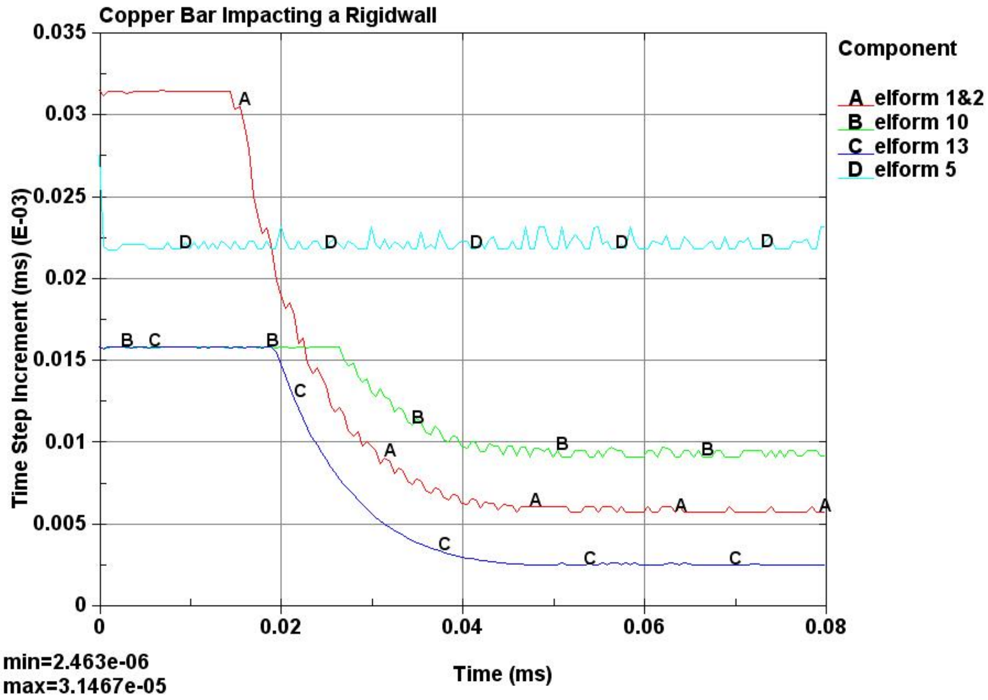

Figure 27.10 – Histories of the stable time step increment for all elforms (1,2,-2,-1,10,13,5) associated with the rigid boundary impact.

Input deck:

```
*KEYWORD
*TITLE
Copper Bar Impacting a Rigidwall
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
 8.000e-02         0       0.0       0.0       0.0
*CONTROL_TIMESTEP
$#  dtinit    tssfac      isdo    tslimt     dt2ms      lctm     erode     ms1st
       0.0  0.800000         0       0.0       0.0
$#  dt2msf   dt2mslc     imscl
       0.0         0         0
*CONTROL_CONTACT
$#  slsfac    rwpnal    islchk    shlthk    penopt    thkchg     orien    enmass
  0.100000       0.0         2         0         0         0         1
$#  usrstr    usrfrc     nsbcs    interm     xpene     ssthk      ecdt   tiedprj
         0         0         0         0   4.00000
$#   sfric     dfric       edc       vfc        th     th_sf    pen_sf
       0.0       0.0       0.0       0.0       0.0       0.0       0.0
$#  ignore    frceng   skiprwg
         0         0         0
*CONTROL_ENERGY
$#    hgen      rwen    slnten     rylen
         2         2         2         2
*DATABASE_GLSTAT
$#      dt    binary
5.0000e-04         1
*DATABASE_MATSUM
$#      dt    binary
5.0000e-04         1
*DATABASE_SLEOUT
$#      dt    binary
5.0000e-04         1
*DATABASE_RWFORC
$#      dt    binary
5.0000e-04         1
*DATABASE_BINARY_D3PLOT
$# dt/cycl   lcdt/nr      beam     npltc    psetid
1.0000e-03
*PART
$# title
material type # 3 (Kinematic/Isotropic Elastic-Plastic)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1         0         1
*SECTION_SOLID
$#   secid    elform       aet
         1         1
*MAT_PLASTIC_KINEMATIC
$#     mid        ro         e        pr      sigy      etan      beta
         1 8.930e-03 1.170e+05   0.35000 4.000e+02 1.000e+02       0.0
$#     src       srp        fs        vp
       0.0       0.0       0.0       0.0
*HOURGLASS
$#    hgid       ihq        qm       ibq        q1        q2        qb        qw
         1         0       0.0         0       0.0       0.0       0.0       0.0
*PART
$# title
material type # 20 (Rigid)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         2         2         2         0         2
*SECTION_SHELL
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
         2         2       0.0         0         0       0.0
$#      t1        t2        t3        t4      nloc     marea
  0.100000  0.100000  0.100000  0.100000         0       0.0
*MAT_RIGID
$#     mid        ro         e        pr         n    couple         m
         2 8.930e-03 1.170e+05   0.35000       0.0       0.0       0.0
$#     cmo      con1      con2
```

```
  1.000000         7         7
$#lco or a1       a2        a3        v1        v2        v3
       0.0       0.0       0.0       0.0       0.0       0.0
*HOURGLASS
$#    hgid       ihq        qm       ibq        q1        q2        qb        qw
         2         0       0.0         0       0.0       0.0       0.0       0.0
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TITLE
$#     cid                                                                 title
         1copper bar-rigidwall interface
$#    ssid      msid     sstyp     mstyp    sboxid    mboxid       spr       mpr
         1         2         3         3
$#      fs        fd        dc        vc       vdc    penchk        bt        dt
       0.0       0.0       0.0       0.0       0.0         0       0.0       0.0
$#     sfs       sfm       sst       mst      sfst      sfmt       fsf       vsf
       0.0       0.0       0.0       0.0       0.0       0.0       0.0       0.0
$#    soft    sofscl    lcidab    maxpar     sbopt     depth     bsort    frcfrq
         2  0.100000         0     1.025       0.0         2        10         1
$#  penmax    thkopt    shlthk     snlog      isym     i2d3d    sldthk    sldstf
       0.0         0         1         0         0         0       0.0       0.0
$#    igap    ignore
         2         0
*INITIAL_VELOCITY_GENERATION
$#      id      styp     omega        vx        vy        vz     ivatn      icid
         1         2       0.0       0.0       0.0   -227.00         0         0
$#      xc        yc        zc        nx        ny        nz     phase    irigid
       0.0       0.0       0.0       0.0       0.0       0.0         0         0
*ELEMENT_SOLID
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1      10      11       2     306     315     316     307

   10368       1   10900   10683   10684   10748   11205   10988   10989   11053
*ELEMENT_SHELL
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
   10369       2   11299   11300   11287   11286       0       0       0       0

   10512       2   11453   11454   11441   11440       0       0       0       0
*NODE
$#   nid               x               y               z      tc      rc
       1        1.131371        1.131371           0.000       0       0

   11285       -0.848528       -0.848528       32.400000       0       0
   11286       10.000000      -10.000000       -0.100000       0       0

   11454      -10.000000       10.000000       -0.100000       0       0
*END
```

Notes:

1. If a part is comprised entirely of tetrahedrons, there are several tetrahedral formulations to choose from, each with various pros and cons. Any of these formulations are preferable to using degenerate, elform=1 tetrahedrons. Two popular choices are (a) elform=10 which is 1 point tetrahedron with 4 nodes, but prone to volumetric locking (overly stiff behavior) in incompressible regimes, e.g., as in metal plasticity, and (b) elform=13 which is a 1 point nodal pressure tetrahedron developed for bulk metal forming; elform=13 is identical with elform=10 with addition of nodal pressure averaging that significantly decreases volumetric locking. There are also two relatively new 10-noded tetrahedron, elform=16 and 17 which have not been widely used.

2. To convert from a Lagrangian simulation to an ALE, the user needs to add the keyword *CONTROL_ALE which defines the ALE control parameters of advection

logic (dct), cycle between advection (nadv), advection method (meth), smoothing weight factors (afac thru efac), etc.:

```
*CONTROL_ALE
$#     dct      nadv      meth      afac      bfac      cfac      dfac      efac
        -1         1         2  0.500000  0.500000       0.0       0.0       0.0
$#   start       end     aafac     vfact    vlimit       ebc      pref
       0.0 1.000e+20  1.000000 1.000e-06       0.0         0       0.0
```

and modify the element formulation choice:

```
*SECTION_SOLID
$#   secid    elform       aet
         1         5
```

3. Hexahedral elements with reasonable aspect ratios should be used for the initial ALE mesh. Degenerate element shapes, such as tetrahedrons and pentahedrons, should be avoided as they lead to reduced accuracy and perhaps numerical instability during the advection.

4. The viscous contact damping parameter, vdc, on card 2 of the *CONTACT_ AUTOMATIC_SURFACE_TO_SURFACE keyword is zero by default. Contact damping is often beneficial in reducing high-frequency oscillation of contact forces in crash or impact simulations. In contacts involving soft materials such as foams and honeycombs, instabilities exist due to contact oscillations. Using a value of vdc between 40-60 (corresponding to 40% to 60% of critical damping), improves stability; however, it may be necessary to reduce the time step scale factor. Generally, a smaller value of vdc, say equal to 20, is recommended when metals, which have similar material stiffnesses, interact.

5. Contact-impact results using penalty method and planar rigid boundaries can possible differ due to their approaches. The penalty method consists of placing normal interface springs between all penetrating nodes and the contact surface. The rigid boundary contact procedure for stopping nodes uses a constraint method which represents a perfectly plastic impact that results in an irreversible energy loss. The total energy dissipated is found by taking the difference between the total kinetic energy of all the nodal points slaved to the rigid wall before and after the impact with the wall. The advantage of the constraint method is that it guarantees the node to lie on the positive side of the rigidwall (no penetration).

6. To move from the contact-impact model used by the penalty method, i.e. that with a meshed rigid wall, to a planar rigid boundary model, the user needs to remove the following entries used to represent the contact-impact and the meshed rigid wall:

```
*CONTROL_CONTACT
$#  slsfac    rwpnal    islchk    shlthk    penopt    thkchg     orien    enmass
  0.100000       0.0         2         0         0         0         1
$#  usrstr    usrfrc     nsbcs    interm     xpene     ssthk      ecdt   tiedprj
         0         0         0         0   4.00000
$#   sfric     dfric       edc       vfc        th     th_sf    pen_sf
       0.0       0.0       0.0       0.0       0.0       0.0       0.0
$#  ignore    frceng   skiprwg
```

```
         0         0         0

*PART
$# title
material type # 20 (Rigid)
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         2         2         2         0         2
*SECTION_SHELL
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
         2         2       0.0         0         0       0.0
$#      t1        t2        t3        t4      nloc     marea
  0.100000  0.100000  0.100000  0.100000         0       0.0
*MAT_RIGID
$#     mid        ro         e        pr         n    couple         m
         2 8.930e-03 1.170e+05   0.35000       0.0       0.0       0.0
$#     cmo      con1      con2
  1.000000         7         7
$#lco or a1        a2        a3        v1        v2        v3
       0.0       0.0       0.0       0.0       0.0       0.0
*HOURGLASS
$#    hgid       ihq        qm       ibq        q1        q2        qb        qw
         2         0       0.0         0       0.0       0.0       0.0       0.0
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TITLE
$#     cid                                                                 title
         1copper bar-rigidwall interface
$#    ssid      msid     sstyp     mstyp    sboxid    mboxid       spr       mpr
         1         2         3         3
$#      fs        fd        dc        vc       vdc    penchk        bt        dt
       0.0       0.0       0.0       0.0      20.0         0       0.0       0.0
$#     sfs       sfm       sst       mst      sfst      sfmt       fsf       vsf
       0.0       0.0       0.0       0.0       0.0       0.0       0.0       0.0
$#    soft    sofscl    lcidab    maxpar     sbopt     depth     bsort    frcfrq
         2  0.100000         0     1.025       0.0         2        10         1
$#  penmax    thkopt    shlthk     snlog      isym     i2d3d    sldthk    sldstf
       0.0         0         1         0         0         0       0.0       0.0
$#    igap    ignore
         2         0

*ELEMENT_SHELL
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
   10369       2   11299   11300   11287   11286       0       0       0       0
   10512       2   11453   11454   11441   11440       0       0       0       0

*NODE
$#   nid               x               y               z      tc      rc
   11286       10.000000      -10.000000       -0.100000       0       0

   11454      -10.000000       10.000000       -0.100000       0       0
```

and replace them with the following planar rigid boundary entry (the user can also include the rigid wall force entry if desired):

```
*DATABASE_RWFORC
$#      dt    binary
5.0000e-04         1

*RIGIDWALL_PLANAR
$#    nsid    nsidex     boxid    offset     birth     death     rwksf
         0         0         0
$#      xt        yt        zt        xh        yh        zh      fric      wvel
  0.000000  0.000000 -0.050000  0.000000  0.000000  1.000000  0.000000  0.000000
```

Implicit Studies:

NIKE3D (implicit dynamics solver) was used by Ferencz [1989] with the computation divided into 80 time steps of 1 microsecond and nodal boundary conditions constraining the impacting face to lie on the global X-Y plane. The half-symmetry deformed shape, at 80 ms (final state), is shown in Figure 27.11.

NIKE3D uses an element formulation, similar to the selected reduced integration of LS-DYNA (elform=2), defined as B-Bar method. The selective reduced integration splits the stress tensor into deviatoric and dilatation (mean) parts, whereas the B-Bar method splits the B matrix (a strain modification) into dilatational and deviatoric parts.

The contact of the deformable body and the rigid wall can be modeled in one of the following ways in this study:

- using nodal boundary conditions which constrain the impacting face to remain on the rigid wall,
- rigid wall (*RIGIDWALL_PLANAR), which provides an easy way to treat contact between a rigid-flat surface and the nodes of a deformable body.

In general, there are two different methods that are available in LS-DYNA to treat nodes impacting a rigid wall. The first method, which is the default method, is the constraint type that is used for all deformable nodes impacting a rigid wall. The second (optional) method is the penalty approach that is used for all rigid nodes or optional deformable nodes impacting the rigid wall. The primary difference between the two methods is in the conservation of energy and momentum. If using the implicit solver, only the penalty approach method is available.

The default constraint method does not conserve momentum and the energy. This is due to the fact that when a deformable node is found to penetrate a rigidwall, its velocity is immediately reset to zero and is moved back onto the surface of the rigidwall. The advantage of the constraint method is that it always guarantees the node to lie on the positive side of the rigidwall (no penetration).

The penalty method (optional for explicit solver/default for implicit solver) for rigid walls uses a scale factor that can be adjusted (default is 1.0) by modifying the rwskf parameter on *RIGIDWALL_PLANAR keyword. This works the same as the contact-impact interface treatment. When a deformable or a rigid node is found to penetrate a rigidwall, the penetrated distance normal to the rigid wall is computed and is resisted by applying a force that is proportional to the computed distance multiplied by a stiffness factor that is based on the material of the impacting node and the dimensions of the attached element. The penalty approach conserves both energy and momentum.

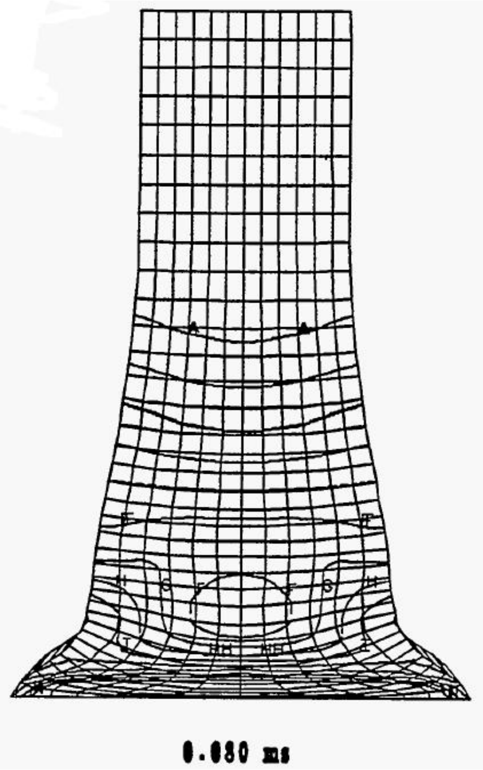

Figure 27.11 – Half-symmetry deformed shape (Ferencz [1989]) at 80 ms (final state).

Analysis Summary:

Dim.	**Type**	**Load**	**Material**	**Geometry**	**Contact**	**Solver**	**Solution Method**
3D	Dynamic	Velocity	Non-linear	Non-linear	SPC's	Implicit	2-Nonlinear w/BFGS

and

Dim.	**Type**	**Load**	**Material**	**Geometry**	**Contact**	**Solver**	**Solution Method**
3D	Dynamic	Velocity	Non-linear	Non-linear	R.Wall (penalty)	Explicit/ Implicit	2-Nonlinear w/BFGS

Element Type:

Fully integrated S/R solid (elform=2)

Different Considerations from Explicit Solver:

The contact of the deformable body and the rigid wall can be modeled in one of the following ways in this study:

- using nodal boundary conditions which constrain the impacting face to remain on the rigid wall,
- rigid wall (*RIGIDWALL_PLANAR), which provides an easy way to treat contact between a rigid-flat surface and the nodes of a deformable body.

Studies 1 and 2:

- NIKE3D and LS-DYNA (each using implicit dynamics solver) Comparison, and
- Implicit LS-DYNA Convergence,

with nodal boundary conditions constraining the impacting face for both studies.

Nodal Boundary (SPC's)	**Shortening (mm)**	**Widening (mm)**	**Max. plastic strain (ε^p)**	**Normalized CPU Time**
B-Bar solid (NIKE3D) - 80 time steps	11.446	7.68 est.	2.248	-
Fully integrated S/R solid 80 time steps	11.313	6.111	2.432	3.35
Fully integrated S/R solid 160 time steps	11.150	7.468	3.126	6.54
Fully integrated S/R solid 320 time steps	11.029	7.730	3.080	13.10
Fully integrated S/R solid 400 time steps	11.005	7.765	3.084	16.84
Fully integrated S/R solid 480 time steps	10.989	7.784	3.084	19.50
Fully integrated S/R solid 640 time steps	10.974	7.798	3.090	25.90
Fully integrated S/R solid 800 time steps	10.967	7.805	3.092	33.38

The above displacement and effective plastic strain results were obtained from the d3plot contour plots at 80.0 ms which were generated by the *DATABASE_BINARY_ D3PLOT keyword.

Normalized CPU times shown in the above Nodal Boundary Condition (SPC's) results table were normalized using the explicit fully integrated S/R solid (elform=2) value.

In the implicit solver direct comparison (80 time steps) of NIKE3D which uses the B-Bar element formulation and the selected reduced integration element formulation of LS-DYNA (elform=2), similar maximum effective plastic strain results ($\varepsilon^{p} = 2.248$ vs. $\varepsilon^{p} = 2.432$) and length shortening (11.446 *mm* vs. 11.313 *mm*) are obtained. For some unexplained reason, the widening profiles (7.680 *mm* vs. 6.111 *mm*) differ significantly.

The maximum effective plastic strain obtained using the LS-DYNA implicit solver ($\varepsilon^{p} = 2.432$) is significantly less than the explicit solver value ($\varepsilon^{p} = 3.366$ - initial work with penalty contact condition). This is believed to be due to the relatively large time step increment used (only 80 steps) which fails to capture the correct dynamics of the simulation. It is shown in above table that increasing the number of time steps (reducing the time step increment) allows the implicit solver to more accurately capture the rate of material deformation (plastic flow) and appears to be converging to a unique solution ($\varepsilon^{p} = 3.100$ and 7.810 *mm*) with a consistent shape profile.

The half-symmetry deformed shape (nodal boundary constraint) with widening profiles and effective plastic strain contouring for selected implicit integration time step sizes at 80 ms, are given in Figures 27.12 (80 time steps), 27.13 (160 time steps), and 27.14 (640 time steps). Figure 27.12 provides a LS-DYNA deformed shape (80 time steps) comparison with the NIKE3D result (80 time steps) shown in Figure 27.11. Together, Figures 27.12 ($\varepsilon^{p} = 2.432$ and 6.111 *mm*), 27.13 ($\varepsilon^{p} = 3.126$ and 7.468 *mm*), and 27.14 ($\varepsilon^{p} = 3.090$ and 7.798 *mm*) illustrate the LS-DYNA converging results with increasing the number of time steps (reducing the time step increment).

Unfortunately, as is shown in above table, the CPU time becomes a deterrent when using implicit dynamics solvers. Thus, the explicit solver is often favored for these types of high deformation, impact simulations due to its ability to provide efficient and stable solutions.

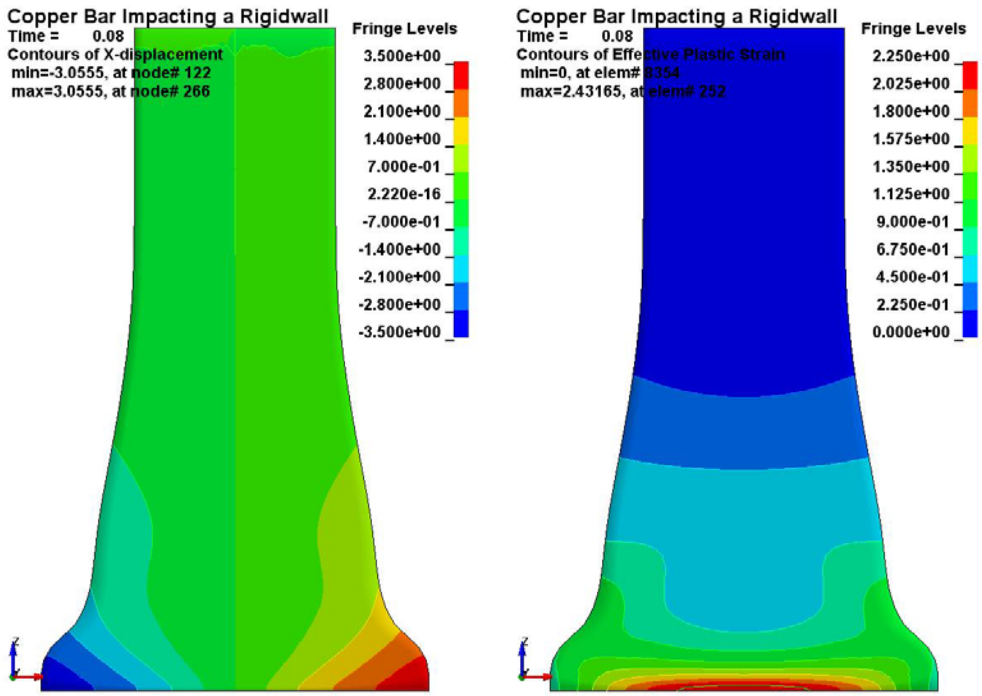

Figure 27.12 – Half-symmetry widening and effective plastic strain contouring with nodal boundary conditions - 80 time steps.

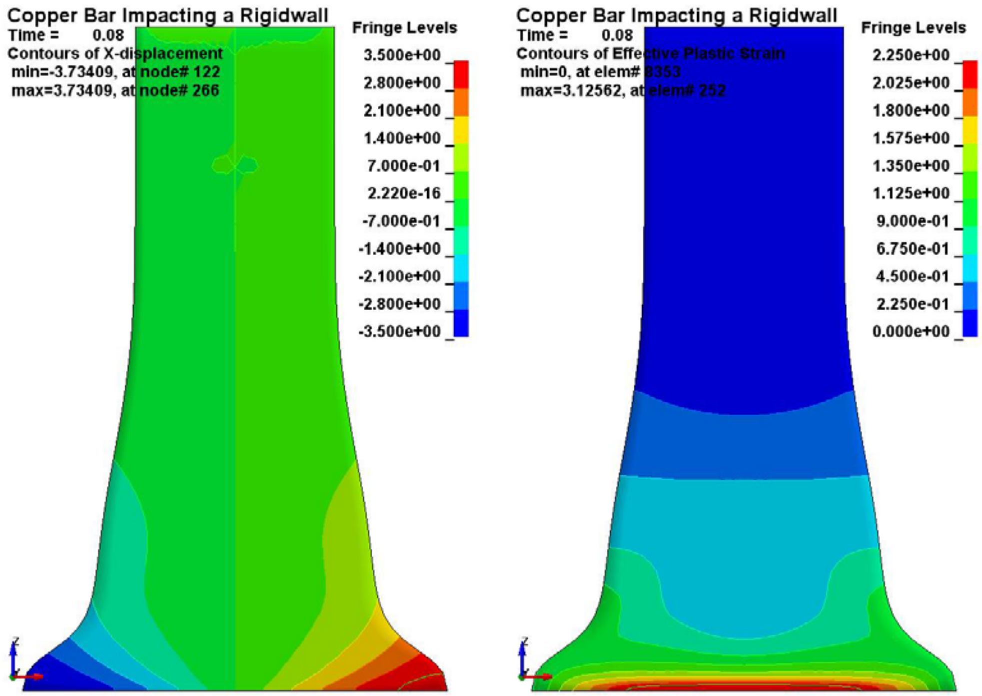

Figure 27.13 – Half-symmetry widening and effective plastic strain contouring with nodal boundary conditions - 160 time steps.

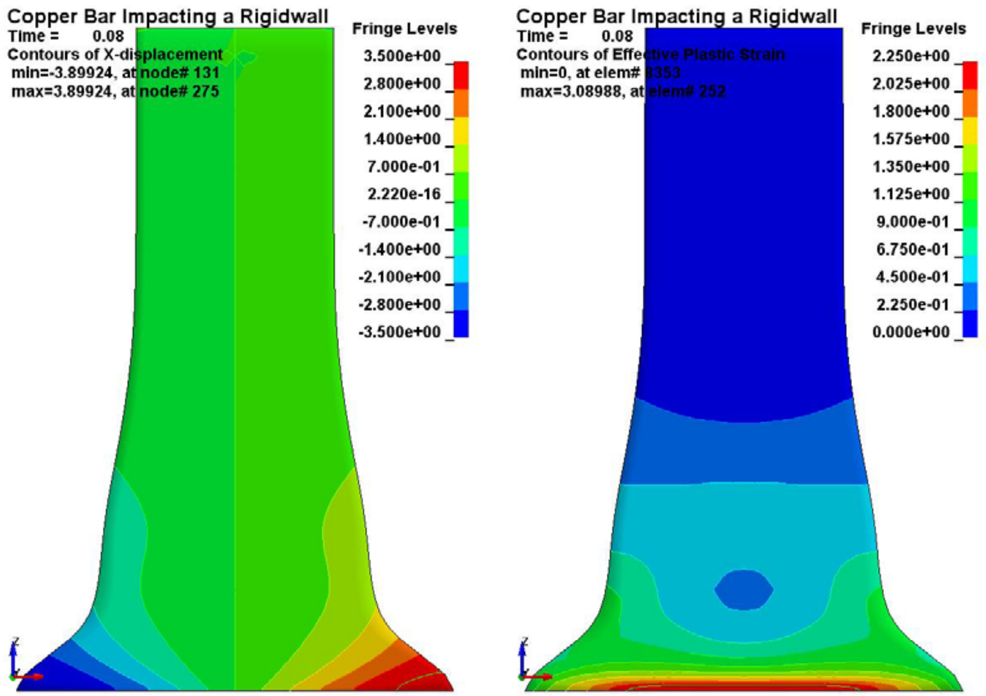

Figure 27.14 – Half-symmetry widening and effective plastic strain contouring with nodal boundary conditions - 640 time steps.

Input deck:

```
*KEYWORD
*TITLE
Copper Bar Impacting a Rigidwall
*CONTROL_IMPLICIT_DYNAMICS
$#   imass     gamma      beta
         1  0.500000  0.250000
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1   0.00100         0         0         0
$         1   0.00050         0         0         0
$         1   0.00025         0         0         0
$         1   0.00020         0         0         0
$         1 0.0001666         0         0         0
$         1 0.0001250         0         0         0
$         1   0.00010         0         0         0
*CONTROL_IMPLICIT_SOLVER
$#  lsolvr   prntflg    negeig     order      drcm    drcprm   autospc    aspctl
         4         2         2         0         1         0         1         0
$#  lcpack
         2
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol     rctol      stol    abstol
         2        11        15    0.0010    0.0100  1.00e+10  0.900000  1.00e-10
$#   dnorm    diverg     istif   nlprint    nlnorm   d3itctl     cpchk
         2         1         1         2
$#  arcctl    arcdir    arclen     rcmth    arcdmp
         0         1       0.0         1         2
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
 8.000e-02         0       0.0       0.0       0.0
```

```
*CONSTRAINED_GLOBAL
$#      tc        rc       dir         x         y         z
         3         0         3       0.0       0.0       0.0

*END
```

Notes:

Studies 3 and 4:

- LS-DYNA Explicit Solver (with rigid wall constraint method contact) and Implicit Dynamics Solver (with rigid wall penalty method contact) Comparison, and
- Implicit LS-DYNA Convergence.

Planar Rigid Boundary	**Shortening (mm)**	**Widening (mm)**	**Max. plastic strain (ε^p)**	**Normalized CPU Time**
Fully integrated S/R solid Explicit - 9883 time steps	10.936	8.139	3.366	1.00
Fully integrated S/R solid Implicit - 80 time steps	11.155	7.083	2.821	3.35
Fully integrated S/R solid Implicit - 160 time steps	11.285	7.326	3.109	6.54
Fully integrated S/R solid Implicit - 320 time steps	11.216	7.502	3.235	13.10
Fully integrated S/R solid Implicit - 400 time steps	11.179	7.534	3.270	16.84
Fully integrated S/R solid Implicit - 480 time steps	11.166	7.550	3.274	19.50
Fully integrated S/R solid Implicit - 640 time steps	11.045	8.503	3.586	25.90
Fully integrated S/R solid Implicit - 800 time steps	11.033	8.517	3.644	33.38
Fully integrated S/R solid Implicit - 1600 time steps	11.023	8.542	3.680	67.30
Fully integrated S/R solid Implicit - 3200 time steps	11.022	8.545	3.696	134.10

The above displacement and effective plastic strain results were obtained from the d3plot contour plots at 80.0 ms which were generated by the *DATABASE_BINARY_ D3PLOT keyword.

Normalized CPU times shown in the above Rigid Wall Planar results table were normalized using the explicit fully integrated S/R solid (elform=2) value.

As for the previous nodal boundary condition method, the maximum effective plastic strain ($\varepsilon^{p} = 2.821$) and widening profile (7.083 *mm*) for the 80 time step solution are roughly 15% less than the explicit results ($\varepsilon^{p} = 3.366$ and 8.139 *mm*). As before, this is believed to be due to the relatively large time step increment used (only 80 steps) which fails to capture the correct dynamics of the simulation. It is shown that increasing the number of time steps (reducing the time step increment) allows the solver to better capture the rate of material deformation (plastic flow) which appears to be converging ($\varepsilon^{p} = 3.274$ and 7.550 *mm*) over a range of time steps studied.

For some unexplained reason, starting with the 640 time step solution, there is a further increase in maximum effective plastic strain and widening results and a distinct change in the widening profile with the outer row of nodes now turning more upward. The corresponding results ($\varepsilon^{p} = 3.696$ and 8.545 *mm*) appear to be converging, though greater than those provided by the explicit solver ($\varepsilon^{p} = 3.366$ and 8.139 *mm*) which also has the outer row of nodes turning slightly upward.

The half-symmetry deformed shape (planar rigid boundary) with widening profiles and effective plastic strain contouring for selected implicit integration time step sizes at 80 ms, are given in Figures 27.15 (explicit), 27.16 (80 time steps), 27.17 (160 time steps), and 27.18 (640 time steps). Figure 27.15 provides a LS-DYNA widening and effective plastic strain results ($\varepsilon^{p} = 3.366$ and 8.139 *mm*) for the explicit solver. Together, Figures 27.16 ($\varepsilon^{p} = 2.821$ and 7.083 *mm*), 27.17 ($\varepsilon^{p} = 3.109$ and 7.326 *mm*), and 27.18 ($\varepsilon^{p} = 3.586$ and 8.503 *mm*) illustrate the LS-DYNA converging results for the implicit solver with increasing the number of time steps (reducing the time step increment).

Unfortunately, as is shown in above table, the CPU time becomes a deterrent when using implicit dynamics solvers. Thus, the explicit solver is often favored for these types of high deformation, impact simulations due to its ability to provide efficient and stable solutions.

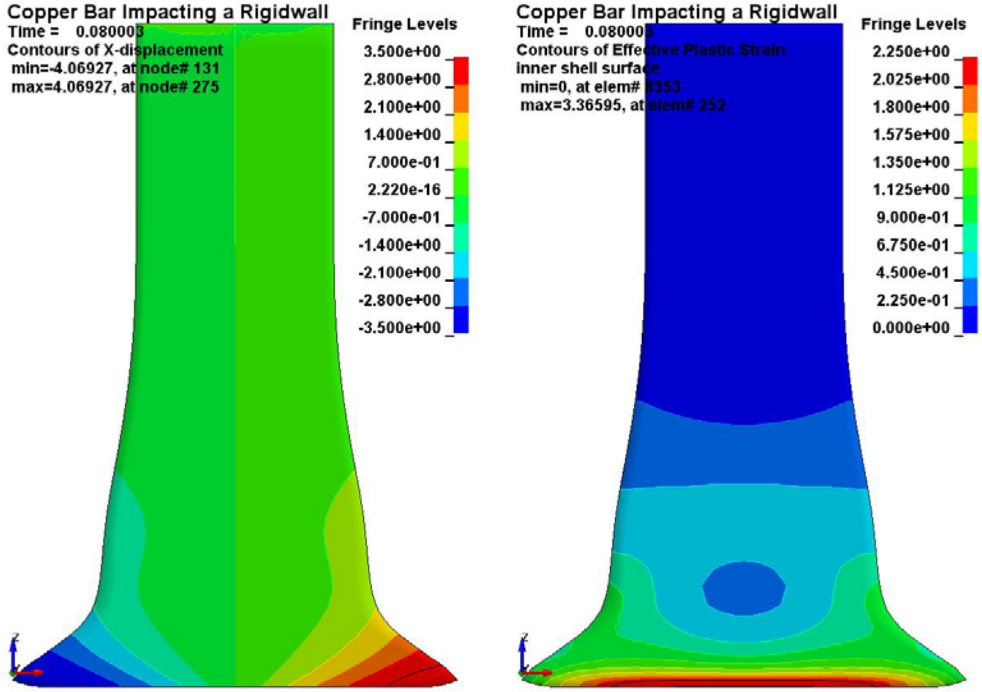

Figure 27.15 – Half-symmetry widening and effective plastic strain contouring with rigid boundary condition - explicit.

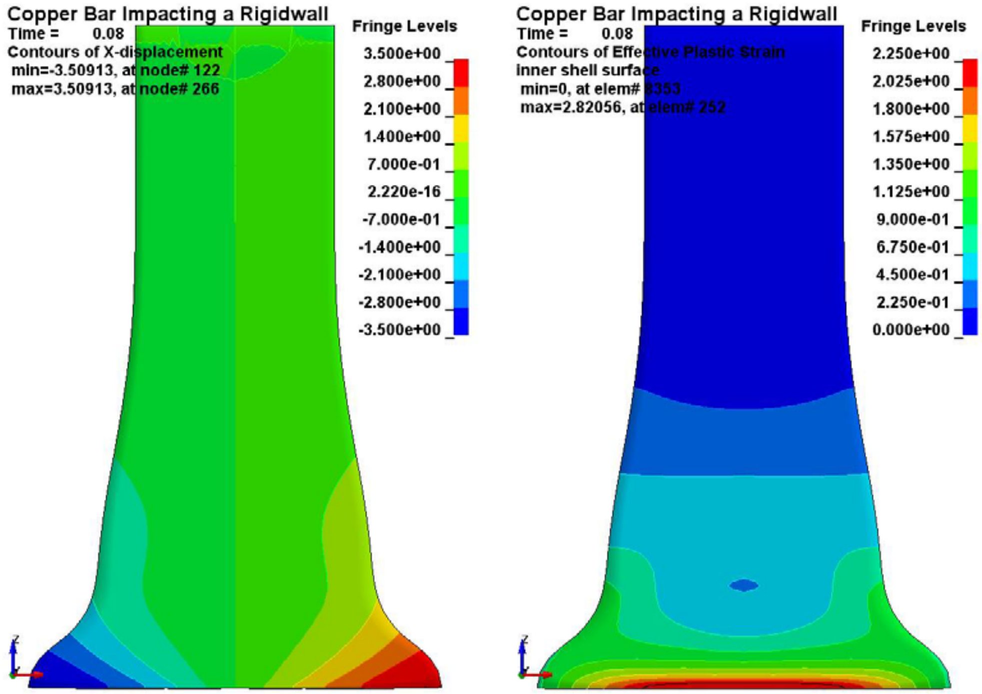

Figure 27.16 – Half-symmetry widening and effective plastic strain contouring with rigid boundary condition - 80 time steps.

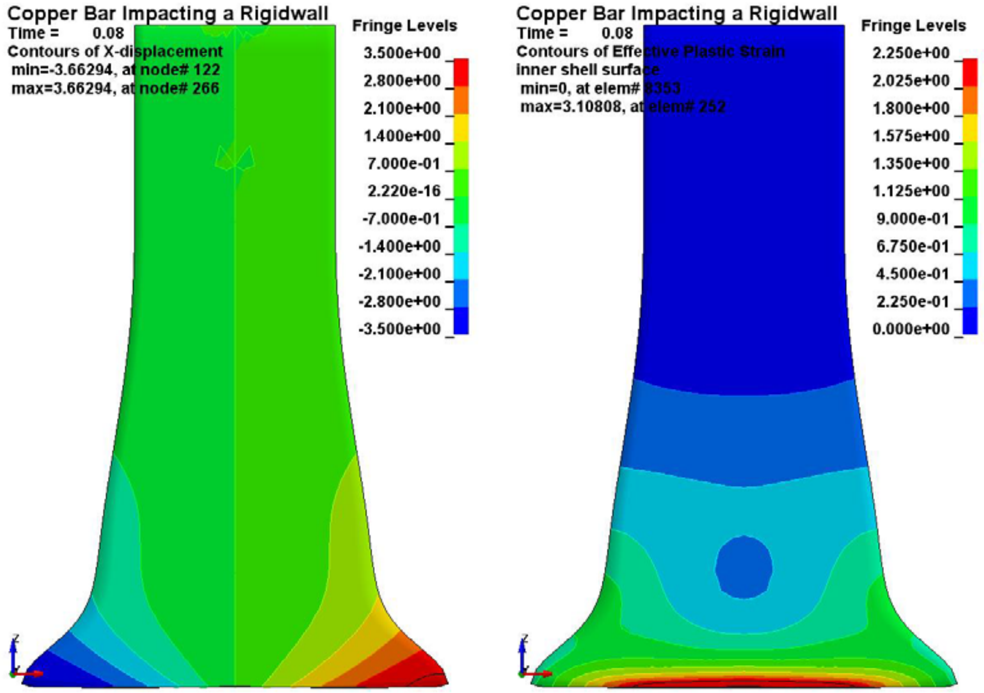

Figure 27.17 – Half-symmetry widening and effective plastic strain contouring with rigid boundary condition - 160 time steps.

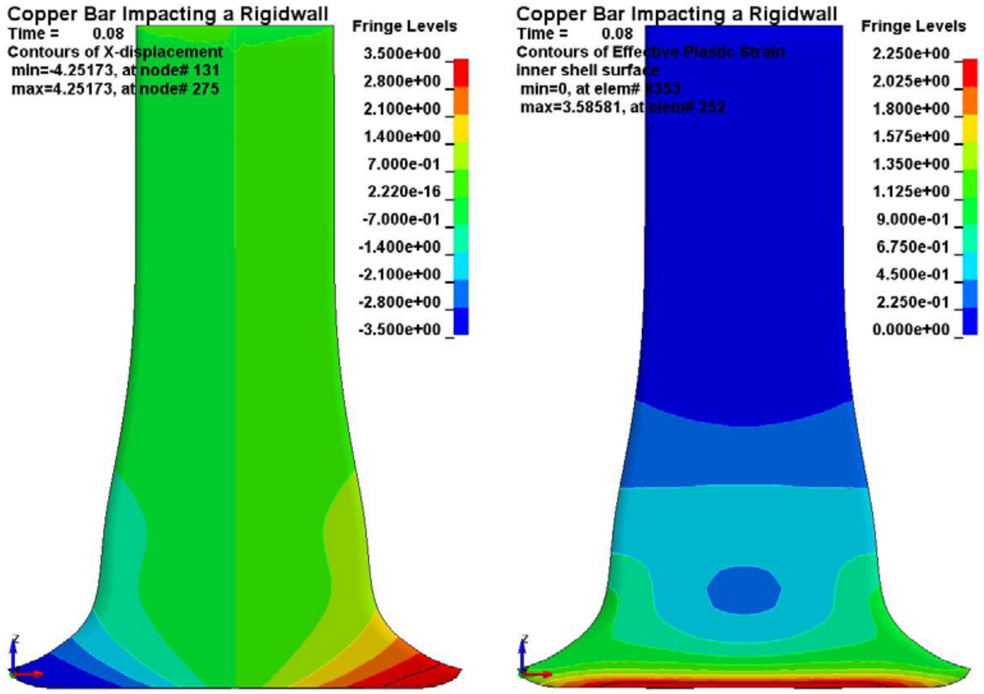

Figure 27.18 – Half-symmetry widening and effective plastic strain contouring with rigid boundary condition - 640 time steps.

Input deck:

```
*KEYWORD
*TITLE
Copper Bar Impacting a Rigidwall
*CONTROL_IMPLICIT_DYNAMICS
$#   imass     gamma      beta
         1  0.500000  0.250000
*CONTROL_IMPLICIT_GENERAL
$#  imflag       dt0    imform      nsbs       igs     cnstn      form
         1   0.00100         0         0         0
$         1   0.00050         0         0         0
$         1   0.00025         0         0         0
$         1   0.00020         0         0         0
$         1 0.0001666         0         0         0
$         1 0.0001250         0         0         0
$         1   0.00010         0         0         0
$         1   0.00005         0         0         0
$         1  0.000025         0         0         0
*CONTROL_IMPLICIT_SOLVER
$#  lsolvr   prntflg    negeig     order      drcm    drcprm    autospc    aspctl
         4         2         2         0         1         0          1         0
$#  lcpack
         2
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol     rctol       stol    abstol
         2        11        15    0.0010    0.0100  1.00e+10   0.900000  1.00e-10
$#   dnorm    diverg     istif   nlprint    nlnorm   d3itctl      cpchk
         2         1         1         2
$#  arcctl    arcdir    arclen     rcmth    arcdmp
         0         1       0.0         1         2
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
 8.000e-02         0       0.0       0.0       0.0

*RIGIDWALL_PLANAR
$#    nsid    nsidex     boxid    offset     birth     death     rwksf
         0         0         0                                     5.0

*END
```

Notes: