NOTICE

**CERTAIN DATA
CONTAINED IN THIS
DOCUMENT MAY BE
DIFFICULT TO READ
IN MICROFICHE
PRODUCTS.**

UCRL-MA--105259

DE91 007594

DYNA3D Example Problem Manual

**Steven C. Lovejoy
Robert G. Whirley**

**University of California
Lawrence Livermore National Laboratory¹**

October 10, 1990

¹Work performed under the auspices of the U.S. Department of Energy by the Lawrence Livermore National Laboratory under contract W-7405-Eng-48.

MASTER

DISTRIBUTION OF THIS DOCUMENT IS UNLIMITED

Contents

ABSTRACT	7
CHAPTER 1: INTRODUCTION.....	8
CHAPTER 2: BAR IMPACTING A RIGID WALL (FILE: D3SAMP1).....	10
CHAPTER 3: IMPACT OF A CYLINDER INTO A RAIL (FILE: D3SAMP2).....	21
CHAPTER 4: IMPACT OF TWO ELASTIC SOLIDS (FILE: D3SAMP3).....	32
CHAPTER 5: SQUARE PLATE IMPACTED BY A ROD (FILE: D3SAMP4).....	40
CHAPTER 6: BOX BEAM BUCKLING (FILE: D3SAMP5).....	54
CHAPTER 7: SPACE FRAME IMPACT (FILE: D3SAMP6).....	65
CHAPTER 8: THIN BEAM SUBJECTED TO AN IMPACT (FILE: D3SAMP7).....	77
CHAPTER 9: IMPACT ON A CYLINDRICAL SHELL (FILE: (D3SAMP8).....	89
CHAPTER 10: SIMPLY SUPPORTED FLAT PLATE (FILE: D3SAMP9).....	101
CHAPTER 11: HOURGLASSING OF SIMPLY SUPPORTED PLATE (FILE: D3SAMP10).....	109
ACKNOWLEDGEMENTS	118
REFERENCES.....	119

List of Figures

Figure 2.1	Initial finite element mesh of quarter symmetry model.....	13
Figure 2.2	Time sequence of deformation.....	14
Figure 2.3	Kinimatic response of node 74.....	16
Figure 2.4	Time sequence of effective plastic strain contours.....	17
Figure 3.1	Drawing of cylinder and rail.....	23
Figure 3.2	Finite element mesh.....	24
Figure 3.3	Time sequence of deforming mesh.....	24
Figure 3.4	Time history of support ring y-displacement.....	27
Figure 3.5	Time history of dent depth.....	27
Figure 3.6	Contours of effective plastic strain at $t = 6.4$ ms.....	28
Figure 4.1	Finite element mesh of bar impact model.....	35
Figure 4.2	X-velocity of nodes 405 and 1.....	35
Figure 4.3	Relative displacement of nodes 1 and 405.....	36
Figure 4.4	Total x-displacements of nodes 405 and 1.....	36
Figure 4.5	X-velocity of nodes 205 and 405.....	37
Figure 4.6	X-velocity of nodes 1 and 201.....	37
Figure 5.1	Finite element mesh.....	44
Figure 5.2	Time sequence of deformation.....	45
Figure 5.3	Velocity time history.....	48
Figure 5.4	Displacement time history.....	48
Figure 5.5	Gap time history.....	49
Figure 5.6	Contours of z-displacement	49
Figure 5.7	Contours of effective plastic strain.....	50
Figure 5.8	Contours of effective stress.....	50

Figure 6.1	Finite element mesh showing boundary conditions.....	59
Figure 6.2	Time sequence of mesh deformation.....	60
Figure 6.3	Z-displacement after buckling.....	62
Figure 7.1	Initial finite element mesh.....	68
Figure 7.2	Time sequence of deforming mesh.....	69
Figure 7.3	Time history of impacting mass z-displacement.....	71
Figure 7.4	Time history of node 54 z-displacement.....	71
Figure 7.5	Effective stress contours of plate at $t = 5.75 \times 10^{-4}$	72
Figure 8.1	Finite element mesh.....	79
Figure 8.2	Mesh deformation sequence.....	80
Figure 8.3	Kinematic responses of node 19.....	82
Figure 8.4	Magnified deformed mesh plots at $t = 0.9$ ms and $t = 1.4$ ms.....	83
Figure 8.5	Effective stress and effective plastic strain of element 9 bottom surface.....	84
Figure 8.6	Effective stress and effective plastic strain of element 9 mid-plane.....	85
Figure 8.7	Effective stress and effective plastic strain of element 9 top surface.....	86
Figure 9.1	Initial finite element mesh.....	91
Figure 9.2	Time sequence of mesh deformation.....	92
Figure 9.3	Kinematic response time histories of center node.....	94
Figure 9.4	Contours of y-displacement.....	95
Figure 9.5	Effective stress and effective plastic strain of contours of inner integration point.....	96
Figure 9.6	Effective stress and effective plastic strain of contours of middle integration point.....	97
Figure 9.7	Effective stress and effective plastic strain of contours of outer integration point.....	98
Figure 10.1	Initial finite element mesh of plate.....	103
Figure 10.2	Kinematic response of node 1.....	104

Figure 10.3	Time history of σ_{xx} in element 1.....	105
Figure 10.4	Contours of z-displacement at $t = 0.55$ ms.....	105
Figure 10.5	Contours of σ_{xx} upper, middle, and lower quadrature points at $t = 0.55$ ms.....	106
Figure 11.1	Example meshes with and without hourglassing.....	112
Figure 11.2	Kinematic response time histories of node 1.....	113
Figure 11.3	Time history of σ_{xx} in element 1.....	114
Figure 11.4	Contours of z-displacement at $t = 0.55$ ms.....	114
Figure 11.5	Contours of σ_{xx} at $t = 0.55$ ms.....	115

List of Tables

Table 1.1	Execution Times for Ten Sample DYNA3D Problems	9
Table 2.1	Material Properties	11
Table 3.1	Material Properties	23
Table 5.1	Material Properties	43
Table 6.1	Material Properties	58
Table 7.1	Material Properties	66
Table 8.1	Material Properties	78
Table 9.1	Material Properties	90
Table 10.1	Material Properties	101

DYNA3D Example Problem Manual

**Steve C. Lovejoy
Robert G. Whirley**

**Applied Mechanics Group
Methods Development Group
Mechanical Engineering**

October 10, 1990

Abstract

This manual describes in detail the solution of ten example problems using the explicit nonlinear finite element code DYNA3D. The sample problems include solid, shell, and beam element types, and a variety of linear and nonlinear material models. For each example, there is first an engineering description of the physical problem to be studied. Next, the analytical techniques incorporated in the model are discussed and key features of DYNA3D are highlighted. INGRID commands used to generate the mesh are listed, and sample plots from the DYNA3D analysis are given. Finally, there is a description of the TAURUS post-processing commands used to generate the plots of the solution. This set of example problems is useful in verifying the installation of DYNA3D on a new computer system. In addition, these documented analyses illustrate the application of DYNA3D to a variety of engineering problems, and thus this manual should be helpful to new analysts getting started with DYNA3D.

Chapter 1

Introduction

DYNA3D [1] is an explicit nonlinear finite element code for analyzing the dynamic response of solids and structures. DYNA3D was originally developed by John Hallquist, and is now under continuous development in the Methods Development Group at Lawrence Livermore National Laboratory. It contains a variety of beam and shell structural elements in addition to the standard 8-node brick continuum element. A large material library containing over 32 material models allows a wide range of options in modeling material stress-strain behavior. Ten types of sliding surfaces permit flexibility in modeling contact, including high pressure interfaces and sliding friction effects. Fully vectorized coding allows DYNA3D to take maximum advantage of computer hardware to produce accurate solutions at minimum cost. DYNA3D is fully operational on CRAY, VAX, and SUN computers, and has been successfully installed on a wide variety of other machines. DYNA3D has been successfully applied to a wide range of dynamic problems including crashworthiness studies, impact calculations, sheet metal forming analyses, and ordinance design.

This manual describes the application of DYNA3D to ten sample engineering analyses, ranging from very simple to very complex. These problems have been chosen to illustrate both modeling techniques and DYNA3D features. Each of the following chapters focuses on a different problem to be studied. First, an engineering description of the physical problem is given. Next, the analytical techniques incorporated in the model are discussed, including salient features of DYNA3D use in the model. Comments are given on alternate modeling strategies that might have been used to achieve the same results. INGRID [2,3,4,5] pre-processing commands required to generate the mesh are listed. To depict the DYNA3D analysis results, each chapter concludes with sample plots and a description of the TAURUS [6] post-processing commands used to generate these plots.

Below is a table of execution times for the ten sample problems on three representative computing platforms. These times represent the execution speeds of the standard 1990 LLNL public domain DYNA3D on each of the three platforms; no attempt was made to optimize either the code or the model for the particular problems under study.

Problem	CRAY Y/MP	SUN 4/310	VAX 8650
D3SAMP1	59.9	3621	4976
D3SAMP2	1298	82,100	112,200
D3SAMP3	3.0	15.4	22.4
D3SAMP4	742	52,160	68,210
D3SAMP5	4956	195,400	-
D3SAMP6	11.2	331	451
D3SAMP7	3.5	20.3	23.8
D3SAMP8	4.8	199	260
D3SAMP9	1.3	6.1	7.4
D3SAMP10	1.3	6.0	7.7

Table 1.1 Execution times for ten sample DYNA3D problems.

This manual is necessarily limited in scope. An attempt has been made to illustrate the most popular features of DYNA3D, but no such manual can be complete. New features are being constantly developed and added to DYNA3D, and many of the newer capabilities are not described in this document. If the following examples are taken as a starting point, the incorporation of improved shell elements, different material models, and other new features can be approached in a step-by-step procedure with a high degree of confidence. Future revisions of this document will expand on both the DYNA3D capabilities illustrated and the breadth of engineering problems to which these tools are applied.

Chapter 2

BAR IMPACTING A RIGID WALL

File: D3SAMP1

2.1 SUMMARY

DYNA3D example 1 simulates a cylindrical bar impacting a rigid wall. The model consists of a long cylindrical bar with an initial velocity in the axial direction which impacts a rigid wall. Key features of this model are the use of quarter symmetry boundary conditions on nodal displacements and the use of an elastic/plastic nonlinear material model. Model results are in good agreement with both experimental and approximate analytical results.

2.2 MODEL DESCRIPTION

Example 1 simulates a cylindrical bar (3.24 cm in length) with a diameter of 0.32 cm impacting a rigid wall at a right angle (normal impact). The finite element model has three planes of symmetry. The first two planes correspond to the x-z and y-z surfaces (see Figure 2.1 for finite element mesh). These two symmetry planes yield a quarter section model which reduces the number of elements by a factor of four over a full model with no loss in accuracy. Eight node continuum brick elements are used.

The third symmetry plane corresponds to the lower x-y surface of the mesh, and simulates a rigid wall. This could have been modeled using either a rigid wall or sliding surface definitions at greater cpu cost.

A bilinear elastic/plastic material model (model 3) was used with the properties of copper. Isotropic strain hardening is included. The material properties used are summarized in Table 2.1.

Material Model	3
Density	8.93 g/cm ³
Elastic Modulus	1.17 g/μs ² cm
Tangent Modulus	1.0x10 ⁻³ g/μs ² cm
Yield Strength	4.0x10 ⁻³ g/μs ² cm
Poisson's Ratio	3.3x10 ⁻¹
Hardening Parameter	1.0

Table 2.1 Material Properties.

The bar is given an initial velocity of 2.27×10^{-2} cm/μsec in the negative z-direction. Figure 2.2 shows a time sequence of the deforming mesh. Figure 2.3 shows the time history kinematic response of node 74 in the z-direction. Node 74 is centrally located on the top x-y surface of the mesh. The velocity response shows that the bar is fully decelerated just after 75 μsec. The oscillations in the early portion of the response can be attributed to longitudinal shock waves which rapidly damp out. If the plot files had been generated at a higher frequency the abrupt points on the response graph would appear more rounded.

The displacement response shows a total z-displacement of -1.087 cm. Thus the final length of the 3.24 cm long bar is 2.15 cm. Wilkins and Guinan developed an analytic prediction of the ratio of final to initial lengths for cylindrical rods impacting a rigid wall, which was verified by experimental results.[?] Equation 2.1 expresses the length ratio in terms of the impact speed U_0 , initial mass density ρ_0 , initial yield strength Y_0 , and parameter h/L_0 . The parameter h/L_0 is the distance h the plastic wave moves from the rigid boundary in the deforming body non-dimensionalized with the initial bar length L_0 . They found this ratio to be independent of the impact velocity.

$$\frac{L^{final}}{L^0} = \left(1 - \frac{h}{L^0}\right) \exp\left(-\rho^0 \frac{U^2}{2Y^0}\right) + \frac{h}{L^0} \quad (1)$$

Using the suggested value of 0.12 for h/L_0 with the material properties of the model, Equation 2.1 predicts a final length L_{final} of 1.99 cm. Thus the F.E. prediction is within 8% of the analytic prediction, which in turn is in good agreement with experimental data. In fact, the analytic solution tends to predict final lengths that are on the order of 10% shorter than experimental results, which suggests that the DYNA3D numerical model is closer to experimental results.

Figure 2.4 shows a time sequence of the deforming mesh with contours of effective plastic strain. Note that the boundary of plastic deformation (contour A) moves up the bar in time. Also note the extreme plastic strain near the impact surface. The model predicts a maximum plastic strain of almost 300% in this localized region. This superplastic behavior is physically possible at elevated temperatures in metals such as copper. An impact such as this can produce high enough temperatures for such deformation, but the material model used does not account for temperature dependence of the properties. One must question the validity of the material model in this local region of large strain.

Overall, this example models the physical impact with good results. Using a thermal elastic/plastic material model may give better insight into the behavior of the deformations in the region of very large plastic flow.

bar impact problem (gm cm microsec)
time = 0.

Figure 2.1 Initial finite element mesh of quarter symmetry model.

bar impact problem (gm cm microsec)
time = 2.00000e+01

bar impact problem (gm cm microsec)
time = 3.00000e+01

Figure 2.2 Time sequence of deformation.

bar impact problem (gm cm microsec)
time = 4.00000e+01

disp. scale factor = 1.000e+00 (default)

bar impact problem (gm cm microsec)
time = 5.00000e+01

disp. scale factor = 1.000e+00 (default)

bar impact problem (gm cm microsec)
time = 6.00000e+01

disp. scale factor = 1.000e+00 (default)

bar impact problem (gm cm microsec)
time = 8.00000e+01

disp. scale factor = 1.000e+00 (default)

Figure 2.2 (continued)

Figure 2.3 Kinematic response of node 74.

bar impact problem (gm cm microsec)
time = 1.00000e+01
contours of eff. plastic strain
min= 0. in element 972
max= 9.983e-01 in element 7

contour values
A= 8.39e-02
B= 1.88e-01
C= 2.91e-01
D= 3.95e-01
E= 4.99e-01
F= 6.03e-01
G= 7.07e-01
H= 8.11e-01
I= 9.14e-01

bar impact problem (gm cm microsec)
time = 4.00000e+01
contours of eff. plastic strain
min= 0. in element 972
max= 2.743e+00 in element 1

contour values
A= 2.30e-01
B= 5.16e-01
C= 8.01e-01
D= 1.09e+00
E= 1.37e+00
F= 1.66e+00
G= 1.94e+00
H= 2.23e+00
I= 2.51e+00

Figure 2.4 Time sequence of effective plastic strain contours.

bar impact problem (gm cm microsec)
time = 6.00000e+01
contours of eff. plastic strain
min= 0. in element 972
max= 2.805e+00 in element 1

contour values
A= 2.36e-01
B= 5.27e-01
C= 8.19e-01
D= 1.11e+00
E= 1.40e+00
F= 1.69e+00
G= 1.99e+00
H= 2.28e+00
I= 2.57e+00

bar impact problem (gm cm microsec)
time = 8.00000e+01
contours of eff. plastic strain
min= 0. in element 972
max= 2.805e+00 in element 1

contour values
A= 2.36e-01
B= 5.27e-01
C= 8.19e-01
D= 1.11e+00
E= 1.40e+00
F= 1.69e+00
G= 1.99e+00
H= 2.28e+00
I= 2.57e+00

Figure 2.4 (continued)

INGRID input file for Example 1

```
bar impact problem (gm cm microsec)
dn3d
term 80.0
plti 1.0
prti 81.
velocity 0 0 -0.0227
c Define symmetry planes
plane 3 0 0 0 0 -1 0 .001 symm
      0 0 0 -1 0 0 .001 symm
      0 0 0 0 0 1 .001 symm
c Define bar.
start
c Define index space
1 4 7 10 13;1 4 7 10 13;1 37;
c Define coordinate of index space
-.16 -.16 0 .16 .16 -.16 -.16 0 .16 .16 0 3.24
c Delete corners for cylindrical mapping
d 0 1 0 0 3 0
d 1 0 0 3 0 0
di 1 2 0 4 5;1 2 0 4 5;;
c Capture surfaces to be mapped and map in cyld. space
sfi -1 -5;-1 -5;;
cy 0 0 0 0 0 1 .32
mate 1
end
c Use stp 0.001 interactively in Ingrid
c Define material properties
mat 1 3
ro 8.93
sigy 0.004
e 1.17
etan 0.001
beta 1.0
pr 0.33
endmat
end end
```

TAURUS input file for Example 1

reso 1024 1024 triad

c Figure 2.1

g

rx -90 g

c Figure 2.2

time 0 g time 10 g time 20 g time 30 g time 40 g time 50 g time 60 g
time 80 g

c Figure 2.3

phs2 nodes 1 74 gather ntime 3 1 74 ntime 6 1 74

c Figure 2.4

phs1

time 10 contour 7 time 40 contour 7 time 60 contour 7 time 80 contour 7
end

Chapter 3

Impact of a Cylinder into a Rail

File: D3SAMP2

3.1 SUMMARY

DYNA3D example 2 models a hollow circular cylinder impacting a rigid rail in the radial direction. Key features of this model are the use of brick elements with an elastic/plastic material model, a stonewall definition, and the use of a dummy slide surface in INGRID. A time sequence of the deforming mesh is shown and compared to experimental results.

3.2 MODEL DESCRIPTION

Example 2 is an example problem taken from the DYNA3D User's Manual, Revision 5, entitled "Impact of a Cylinder Into a Rail." Figure 3.1 is a drawing of the physical problem that is modeled. The cylinder is 9 inches in diameter by 12 inches long with a 1/4 inch wall thickness. A rigid ring is added to each end to increase stiffness and mass. The cylinder is given an initial velocity of 660 in/sec toward the rail.

One quarter of the cylinder was modeled using two planes of symmetry. Figure 3.2 shows the finite element mesh. The first plane of symmetry is the x-y plane on the right side of the mesh. The second plane of symmetry is the y-z plane. The rail is modeled using a stonewall plane on the top surface. The other surfaces of the rail are added for graphic display clarity and serve no other purpose. Approximately 70 nodes on the cylinder in the vicinity of the rail are slaved to the stonewall. A dummy slide surface is used in INGRID between the stonewall and the 70 slaved nodes to prevent any nodal merges between the cylinder and the rail during mesh generation.

The cylinder model has three brick elements through the wall thickness. This is the minimum number required to capture bending stresses with plasticity. Note the higher element density in the vicinity of the rail. The modeler anticipated that this region would undergo the most deformation and decreased element density away from the rail to minimize the cost of the analysis.

The cylinder uses an isotropic elastic/plastic material model (model 12) with the elastic/perfectly plastic material properties of steel. The rigid support ring on the end of the cylinder uses material model 1, to represent a perfectly elastic material with twice the stiffness of steel. The density of this material is approximately 20 times that of steel. Table 3.1 gives a summary of the material properties.

Figure 3.3 shows a time sequence of the deforming mesh; the x-y symmetry plane has been reflected in the post-processor TAURUS for graphical clarity. Figure 3.4 is a time history of the rigid body displacement of the support ring in the y-direction. A maximum displacement of -1.69 in. occurs at 4.4 ms, after which the structure loses its elastic strain energy and rebounds upward.

Figure 3.5 is a time history of the difference in nodal displacements between nodes 205 and 860. Node 205 is located on the outside surface of the cylinder near the center of the rail. Node 860 is located on the outside of the cylinder near the lower end of the support ring. The difference between the y-displacements of these nodes is a measure of the depth of the dent in the cylinder. Figure 3.5 shows a maximum relative displacement of 1.65 inches which then stabilizes to a 1.45 inch dent after the elastic strain energy is recovered. Experimental measurements recorded a maximum residual dent of 1.44 inches as reported in [1]. The post-peak oscillations in Figure 3.5 are due to elastic vibration of the cylinder about its deformed shape.

Figure 3.6 shows contours of effective plastic strain after the impact. Most of the contours shown represent less than 16% plastic strain. Some very localized plastic strain of up to 32% is predicted on the outer surface

very localized plastic strain of up to 32% is predicted on the outer surface at the center of the rail.

	<u>Steel cylinder</u>	<u>Added mass</u>
Material Model	12	1
Density (lb-s ² /in ⁴)	7.346×10^{-4}	1.473×10^{-2}
Shear Modulus (psi)	1.133×10^5	N/A
Yield Strength (psi)	1.90×10^5	N/A
Hardening Modulus (psi)	0.0	N/A
Bulk Modulus (psi)	2.4×10^7	N/A
Elastic Modulus (psi)	N/A	60×10^6

Table 3.1 Material properties.

Figure 3.1 Drawing of cylinder and rail.

Cylinder drop calculation (in,sec,lb s²/in)
time = 0.

Figure 3.2 Finite element mesh.

Cylinder drop calculation (in,sec,lb s²/in)
time = 0.

Figure 3.3 Time sequence of deforming mesh.

Cylinder drop calculation (in,sec,lb s²/in)
time = 1.00000e-03

Cylinder drop calculation (in,sec,lb s²/in)
time = 2.00000e-03

Figure 3.3 (continued)

Cylinder drop calculation (in,sec,lb s²/in)
time = 3.0000e-03

Cylinder drop calculation (in,sec,lb s²/in)
time = 6.4000e-03

Figure 3.3 (continued)

Figure 3.4 Time history of support ring y-displacement.

Figure 3.5 Time history of dent depth.

Cylinder drop calculation (in,sec,lb s²/in)
time = 6.40000e-03
contours of eff. plastic strain
min= 0. in element 3667
max= 3.170e-01 in element 103

contour values
A= 2.66e-02
B= 5.96e-02
C= 9.26e-02
D= 1.25e-01
E= 1.59e-01
F= 1.91e-01
G= 2.24e-01
H= 2.57e-01
I= 2.90e-01

Figure 3.6 Contours of plastic strain at t = 6.4 ms.

INGRID input file for Example 2

```
Cylinder drop calculation (in,sec,lb s^2/in)
dn3d
term 0.013
plti 0.0001
prti 1.0
c Define symmetry planes and stonewall
plane 3
0 0 0 0 1 0.001 symm
0 0 0 -1 0 0 0.001 symm
0 -4.5 0 0 1 0 .001 ston
si 1 dummy;
c use tp 0.001 interactively in Ingrid
c Define rail
start
c Define index space in corresponding coordinates
1 2;1 2;1 2;
0 5 -7.5 -4.5 0 -.75
c Define master side of slide surface. The rail has fewer nodes in the
c slide surface definition therefore it should be the master side.
sii 1 2;2 2;1 2; 1 m
c Note that no material has been assigned to this part
end

velocity 0 -660 0
c Define cylinder
start
c Define index space and corresponding coordinates. this space is in
c cylindrical coordinates
1 4;1 13 40;1 5 9 27;
4.25 4.5 -90 -34.6153846 90 0 -.75 -1.5 -6
cyli
c Define slave nodes of stonewall
sw 2 1 1 2 2 2 3
c Define slave nodes of slide surface
sii 2 2;1 2;1 2; 1 s
mate 1
end
c Define support ring
start
c Define index space and corresponding coordinates
1 5; 1 13 40; 1 5;
4.25 3.25 -90 -34.6153846 90 -5 -6
cyli
mate 2
end
```

```
c Define material properties
c Cylinder
mat 1 12
c steel
ro 0.7346e-03
g 11.33e+06
sigy 1.90e+05
eh 0.0
k 24e+06
endmat
c Support ring
mat 2
ro 0.147333e-01
e 60.0e+06
endmat
end
```

TAURUS input file for Example 2

c figure 3.1 is a hand sketch

c Figure 3.2

reso 1024 1024 triad state 1

ry 45 g

c Figure 3.3

center raxy ry 90 g time 0.001 g time 0.002 g time 0.003 g

time 6.4e-03 g

c Figure 3.4

phs2 matls 1 2 nodes 2 205 860 gather

mtime 2 1 2

c Figure 3.5

nrtime 2 205 860

c Figure 3.6

phs1

rx 60 contour 7

end

Chapter 4

Impact of Two Elastic Solids

File: D3SAMP3

4.1 SUMMARY

DYNA3D example 3 investigates the uniaxial strain wave propagation developed by two elastic solids under normal impact. The behavior of slide surfaces is also discussed and related to the example.

4.2 MODEL DESCRIPTION

Example 3 investigates uniaxial strain propagation through an elastic solid. The finite element mesh (see Figure 4.1) is a column of 100 brick elements arranged as a one-dimensional bar. The cross-section is square, one unit of length by one unit of length with one element in each of the sectional directions. At the mid-length section the model is separated by a sliding with voids (type 3) slide surface which divides the bar into two pieces.

All nodal translational displacements are constrained in both the y and z directions, thus only allowing translation in the x, or "length-wise," direction. This generates a uniaxial strain state within the bar to represent the behavior of two impacting half spaces. The left half of the model is given an initial x-velocity of 0.1 length/time, while the right half is initially at rest.

The dynamics resulting from this collision are best seen by examining kinematic response time histories of each of the two pieces of the model. The left piece begins with node 205 (leftmost end) and ends with node 405 (rightmost end). The right piece begins with node 1 (leftmost end) and ends with node 201 (rightmost end).

Figure 4.2 shows an x-velocity time history of nodes 405 and 1. Node 405 (left piece) impacts node 1 (right piece) in a very short time. The initial shock from the impact has a rise time of approximately 0.10 time units. During this time node 405 decelerates and node 1 accelerates until a common velocity is attained. This common velocity is maintained as the strain wave travels down each section of the bar. The strain wave in the left piece propagates from negative x direction, reflects off the free end and comes back towards the interface of the two pieces traveling a distance equal to the length of the whole bar or twice the length of each piece. The strain wave in the right piece travels from left to right and then returns back to the interface. The time needed for the strain wave to propagate to the free surface, reflect, and propagate back to the interface is approximately 1.0 time units. The wave velocity c in an elastic solid can be approximated by Equation 4.1,

$$c = \sqrt{\frac{\lambda + 2G}{\rho}} \approx \sqrt{\frac{E}{\rho}} \quad \text{for } \nu = 0.0 \quad (4.1)$$

where λ is Lamé's first constant, E is the elastic modulus, G is the shear modulus, ρ is the mass density, and ν is Poisson's ratio. The elastic material model specifies that $E = 100$ and $\rho = 0.01$, yielding a strain wave velocity of 100 (length/time). The time required for the strain wave to travel a distance L is given by Equation 4.2,

$$t = \frac{L}{c} \quad (4.2)$$

In the present example, $L = 100$ and $c = 100$, thus the time required for each of the two strain waves to travel the length of each piece and reflect back is 1 unit of time. This agrees well with the DYNA3D analysis results.

The two halves of the bar separate when the reflected strain waves reach the interface. The left piece loses its kinetic energy to the right piece. As can be seen in the velocity plot of Figure 4.2, the system is conservative since the right piece gains all of the velocity lost by the left piece due to their equal masses.

Also of interest is the overshoot in velocity seen when the two pieces first impact. This is partially due to the penalty formulation of the slide surface, and partially due to the finite spatial discretization and sharp strain wave front. This effect is damped out quite rapidly and could be made as small as desired through mesh refinement.

Figure 4.3 shows the difference in nodal displacements between nodes 1 and 405. This quantity can be interpreted as the gap between the two pieces. During the collision when the two pieces are mated, the gap distance is shown to be a small negative quantity. Of course, a physical distance cannot be negative, and in fact should be zero in this case. This type of response is typical of penalty-type slide surfaces in contact, and should not be cause for concern. This negative gap can be decreased by increasing the penalty scale factor in DYNA3D. Increasing the penalty parameter over the default value can decrease the maximum allowable time step, requiring the user to input a "time step scale factor" < 1.0 and thus increasing the cost of the calculation. This may result in a larger amplitude on the overshoot discussed above. Depending on the particular application, it is often best to accept a small amount of overlap or negative gap when using slide surfaces instead of using too high of a penalty parameter. The default penalty parameter has proven an effective choice for a wide range of applications.

Figure 4.4 shows total displacement time histories of nodes 405 and 1. Figure 4.5 shows x-velocity time histories of nodes 205 and 405, and Figure 4.6 shows x-velocity time histories of nodes 1 and 201.

Bar Impacting bar (non-dimens
time = 0.00000E+00

disp. scale factor = 0.100E+01 (default)

Figure 4.1 Finite element mesh of bar impact model.

Figure 4.2 X-velocity of nodes 405 and 1.

Figure 4.3 Relative displacements of nodes 1 and 405.

Figure 4.4 Total x-displacements of nodes 405 and 1.

Figure 4.5 X-velocity of nodes 205 and 405.

Figure 4.6 X-velocity of nodes 1 and 201.

INGRID input file for Example 3

Bar impacting bar (non-dimensional)

dn3d

term 1.5

plti .01

prti 2.0

si 1 sv;

c Define index space and corresponding coordinates for right half

start

52 102;1 2;1 2;

50 100 0 1 0 1

c Fix y and z translation

b 1 1 1 2 2 1 011000

b 1 1 2 2 2 2 011000

c Define slave side of slide surface

si 1 1 1 1 2 2 1 S

mate 1

end

velocity 0.1 0 0

c Define index space and corresponding coordinates for left half

start

1 51;1 2;1 2;

0 50 0 1 0 1

c Fix y and z translation

b 1 1 1 2 2 1 011000

b 1 1 2 2 2 2 011000

c Define master side of slide surface

si 2 1 1 2 2 2 m

mate 1

end

c Define material properties

mat 1 1

ro 0.01

e 100

pr 0.0

endmat

end

TAURUS input file for Example 3

reso 1024 1024

c Figure 4.1
triad g

c Figure 4.2
phs2 nodes 4 1 201 205 405 gather
ntime 4 2 405 1

c Figure 4.3
nrtme 1 1 405

c Figure 4.4 ntime 1 2 405 1

c Figure 4.5
ntime 1 2 205 405

c Figure 4.6
ntime 4 2 1 201
end

Chapter 5

Square Plate Impacted by a Rod

FILE: D3SAMP4

5.1 SUMMARY

DYNA3D example 4 simulates a square plate impacted by a solid rod. The rod impacts and deforms the plate at the center. The plate geometry is such that the rod rebounds after impact. Key features of DYNA3D used in this example are four node shell elements, symmetry boundary conditions, a rigid material, and a "sliding with voids" slide surface definition. This model simulates a realistic response to such an impact.

5.2 MODEL DESCRIPTION

Example 4 simulates a solid rod, 4 centimeters in diameter by 25 centimeters long, impacting a 62 centimeter by 62 centimeter square plate in the center. The plate is supported near the edges by a plate frame that elevates the main plate 5 centimeters from the reference ground. The main plate is 0.79 centimeter thick and the plate frame 0.5 centimeter thick. Both parts are modeled using four-node Belytschko-Tsay shell elements. Figure 5.1 shows the finite element mesh of the model. Table 5.1 lists the material properties of the rod, main plate and plate frame respectively.

The impacting rod is given a rigid material model with eight node brick elements and an initial velocity of $1.8 \times 10^{-3} \text{ cm}/\mu\text{s}$ (18 m/s) into the center of the main plate which is initially at rest. The elastic modulus specified for the rigid material is used only for slide surface calculations. Quarter symmetry boundary conditions were used on the rod.

The main plate is modeled using quarter symmetry boundary conditions. Quadrilateral shell elements are used with an elastic/plastic material model. Both the rod and main plate are given symmetric

boundary conditions on the x-z and y-z surfaces to utilize the symmetries of the problem and hence reduce the number of elements by a factor of four.

A sliding with voids (type 3) slide surface is defined between the rod and the center of the main plate as previously mentioned. This allows the rod to impart loads and deformations onto the plate without node penetration.

The nodes of the innermost 4 square centimeters of the quarter model of the plate are slaved to the bottom end of the rod which acts as the master surface for the slide surface definition. By limiting the slave region as mentioned, the computation time can be greatly reduced. The vertical support plates are attached 25 centimeters out from the center of the target plate. The nodes of the support plates are merged with the nodes of the main plate, thus simulating a welded union between the main plate and support plates.

Figure 5.2 shows the time sequence of the rod impacting the plate. The sequence lasts for $10^4 \mu\text{s}$. Note that the rod begins rebounding from the plate, reversing its velocity near $t = 3 \times 10^3 \mu\text{s}$. This event is more clearly seen in the time history velocity plot of Figure 5.3. Node 1 (line A) corresponds to the front left node of the main plate, node 4970 (line B) corresponds to the lower center node of the rod. One can see that in the early and later stages of the impact the plate oscillates relative to the rod.

Figure 5.4 shows the corresponding z-displacement of the rod and plate. The maximum deflection occurs at $3 \times 10^3 \mu\text{s}$ after which both the plate and rod rebound back. At $t = 4.5 \times 10^3 \mu\text{s}$ the plate oscillates about its final deflection of approximately 2.5 centimeters and the rod rebounds at a velocity of 7.0 m/s in the positive z-direction. The initial and final kinetic energies of the rod are 0.98 kJ and 0.15 kJ, respectively. Thus, the rod lost approximately 85% of its energy to the plastic deformation and motion of the target plate.

Figure 5.5 shows the gap between the rod and the plate as a function of time. Note the positive finite gap of 0.1 centimeter during the simulated contact. This is due to the measured displacements being on the rod centerline, and the target plate cupping below the centerline of the rod. Contact is maintained between the outer edge of the rod and the plate until separation occurs (see Figure 5.2). This "cupping" phenomenon is frequently observed experimentally and is accurately predicted by DYNA3D.

Figure 5.6 shows the contours of z-displacement of the main plate at $t = 1 \times 10^4$ ms. Note that even though the simulation is terminated at $t = 1 \times 10^4$ ms the plate is still responding dynamically i.e., it has not yet reached static equilibrium. Figure 5.7 shows contours of effective plastic strain in the main plate at $t = 1 \times 10^4$ ms. The majority of the plastic strain occurs in the vicinity of the impact, with a small zone along the 45° diagonal of the plate due to strain wave focusing effects. Figure 5.8 shows contours of effective stress in the target plate. Many of the contours represent the effects of transient strain waves in the plate at this time.

Overall, this model is a good example of the robust dynamic impact capabilities of DYNA3D.

Rod

Material Model	2.0
Density (g/cm ³)	1.9218x10 ¹
Elastic Modulus (g/μs ² cm)	2.1
Poisson's Ratio	0.0

Main plate

Material Model	3
Density (g/cm ³)	7.85
Elastic Modulus (g/μs ² cm)	2.1
Tangent Modulus (g/μs ² cm)	1.24x10 ⁻²
Yield Strength (g/μs ² cm)	4.0x10 ⁻³
Hardening Parameter	1.0
Poisson's Ratio	3.0x10 ⁻¹

Plate frame

Material Model	3
Density (g/cm ³)	7.85
Elastic Modulus (g/μs ² cm)	2.1
Tangent Modulus (g/μs ² cm)	1.24x10 ⁻²
Yield Strength (g/μs ² cm)	2.15x10 ⁻³
Hardening Parameter	1.0
Poisson's Ratio	3.0x10 ⁻¹

Table 5.1 Material Properties of Model.

Plate impact by rod (cm,gm,microsec)
time = 0.

Plate impact by rod (cm,gm,microsec)
time = 0.

Figure 5.1 Finite element mesh.

Plate Impact by rod (cm,gm,microsec)
time = 0.

Plate Impact by rod (cm,gm,microsec)
time = 1.00000e+03

Figure 5.2 Time sequence of deformation.

Plate Impact by rod (cm,gm,microsec)
time = 3.00000e+03

Plate Impact by rod (cm,gm,microsec)
time = 5.00000e+03

Figure 5.2 (continued)

Plate Impact by rod (cm,gm,microsec)
time = 6.75000e+03

Plate Impact by rod (cm,gm,microsec)
time = 1.00000e+04

Figure 5.2 (continued)

Figure 5.3 Velocity time history.

Figure 5.4 Displacement time history.

Figure 5.5 Centerline gap time history.

Plate impact by rod (cm,gm,microsec) INGDY4.dat
time = 1.00000e+04
contours of z-displacement
min=-2.611e+00 in element 1
max= 4.976e-01 in element 489

contour values
A=-2.35e+00
B=-2.03e+00
C=-1.70e+00
D=-1.38e+00
E=-1.06e+00
F=-7.33e-01
G=-4.10e-01
H=-8.68e-02
I= 2.36e-01

disp. scale factor = 1.000e+00 (default)

Figure 5.6 Contours of z-displacement.

Plate Impact by rod (cm,gm,microsec)
time = 1.00000e+04
contours of eff. plastic strain
min= 0. in element 3844
max= 4.580e-02 in element 8

contour values
A= 3.85e-03
B= 6.61e-03
C= 1.34e-02
D= 1.81e-02
E= 2.29e-02
F= 2.77e-02
G= 3.24e-02
H= 3.72e-02
I= 4.20e-02

Figure 5.7 Contours of effective plastic strain.

Plate Impact by rod (cm,gm,microsec)
time = 1.00000e+04
contours of eff. stress (v-m)
min= 2.393e-05 in element 3844
max= 2.288e-03 in element 1871

contour values
A= 2.14e-04
B= 4.50e-04
C= 6.85e-04
D= 9.21e-04
E= 1.16e-03
F= 1.39e-03
G= 1.63e-03
H= 1.86e-03
I= 2.10e-03

Figure 5.8 Contours of effective stress.

INGRID input file for Example 4

```

Plate impact by rod (cm,m,microsec)
dn3d
term 1.0e+04
plti 1.0e+03
prti 1.0e+05
c Define symmetry planes
plane 2
0 0 0 0 -1 0 .01 symm
0 0 0 -1 0 0 .01 symm
si 1 sv ;
c Define main plate
start
1 9 51 63;1 9 51 63;-1;
0 4 25 31 0 4 25 31 5
th 1 1 1 4 4 1 0.79
si+ 1 1 1 4 4 1 0.79
si+ 1 1 1 2 2 1 1 s 0 0 10
c Orientate plate slave surface towards impacting rod surface.
c Always check shell surface normal vectors!
mate 1
end
c Define left support plate
start
-1 ;1 50;1 2 11;
25 0 24.5 5 4.5 0
b 1 1 3 1 2 3 111000
c Fix translational DOF on bottom edge of support plate
mate 2
end
c Define upper support plate
start
1 50;-1;1 2 11;
0 24.5 25 5 4.5 0
b 1 1 3 2 1 3 111000
mate 2
end c Define rod
velocity 0 0 -1.8e-03
start
1 4 7 10 13;1 4 7 10 13;1 51;
-2 -2 0 2 2 -2 -2 0 2 2 5.005 30
d 0 1 0 0 3 0
d 1 0 0 3 0 0
di 1 2 0 4 5;1 2 0 4 5;;
sfi -1 -5;-1 -5;;
cy 0 0 0 0 0 1 4
si 1 1 1 5 5 1 1 m
c Define bottom surface of rod as the master side of slide surface.
c This surface has fewer nodes than the surface of the plate, i.e. a
c master has many slaves.
mate 3
end

```

c Set tolerance for merge between main plate and supports
c Use tp 0.0001 interactively before the continue command
bptol 1 2 0.001
bptol 1 3 0.001
c Define material properties

c Main plate
mat 1 3 shell
ro 7.85
e 2.1
etan 0.0124
sigy 0.004
pr 0.3
beta 1.0
endmat
c Support plates
mat 2 3 shell
ro 7.85
e 2.1
etan 0.0124
sigy 0.00215
pr 0.3
beta 1.0
thick 0.5
endmat
c Rod
mat 3 20
ro 19.218
e 2.1
pr 0.0
endmat
end

TAURUS input file for Example 4

reso 1024 1024

c Figure 5.1

state 1 triad g ytrans 5 rx -90 g

c Figure 5.2

time 0 g time 1.0e+03 g time 3.0e+03 g time 5.0e+03 g time 6.75e+03 g
time 1.0e+04 g

c Figure 5.3

phs2 nodes 2 1 4970 gather ntime 6 2 1 4970

c Figure 5.4

ntime 3 2 1 4970

c Figure 5.5

nrtime 3 4970 1

c Figure 5.6

phs1 restore triad m 1 time 1.0e+04 contour 19

c Figure 5.7

contour 7

c Figure 5.8

contour 9

end

Chapter 6

Box Beam Buckling

File: D3SAMP5

6.1 SUMMARY

Example 5 is an excellent example of the non-linear capabilities of DYNA3D. By using a specific combination of nodal boundary conditions, shell elements, and a slide surface, a highly non-linear and unstable buckling problem is modeled with realistic results. Key features of this model include the use of a "single surface contact" slide surface and four node shell elements with elastoplastic material behavior.

6.2 MODEL DESCRIPTION

Example 5 investigates the buckling of a slender beam. The beam, made of 0.06 inch thick sheet metal, is 12 inches long and its cross-section measures 1.75 by 1.75 inches. A quarter symmetric model is used in this analysis. The upper 2 inches of the length of the beam is loaded by a constant velocity field, which acts in a direction parallel to the beams longitudinal axis.

Figure 6.1 shows the finite element mesh used with the specified nodal boundary conditions. The mesh is composed of 1800 four node shell elements using three integration points through the thickness. The material model used is bilinear elastic/plastic with isotopic hardening and the (model 3) material properties of steel. A summary of the material properties is given in Table 6.1.

Buckling is an unstable physical phenomena which complicates the development of a realistic numeric model. Physically, buckling is sensitive to imperfections in a structure, which must be incorporated in some way into the numerical model to obtain meaningful results. This model uses a carefully constructed mesh incorporating nodal displacement constraints

for quarter symmetry, slide surfaces to prevent element interpenetration, and initial displacements to model geometric imperfections. The mesh uses 900 elements for each side of the quarter sector: 10 elements for the flange width and 90 elements for the flange length.

The nodes located at the base of the model are given a completely fixed displacement constraint to prevent rigid body motion when loaded. Note from Figure 6.1 that the length of the part (z-axis) is divided into two sections. The upper section has all nodal displacements constrained with the exception of z-translation. The top edge is given a prescribed constant velocity in the negative z-direction of 273 inches/seconds. These two kinematic features of the upper portion allow it to act as rigid ram, causing the lower portion into buckling.

The right lengthwise edge has symmetry boundary conditions (nodal displacement constraints in the translational y, rotational x and z directions). The left lengthwise edge has the translational x, rotational y and z displacements constrained. All internal nodes have no displacement restrictions on the lower portion of the part.

The most unstable stage of the buckling is the initiation of lateral deflection. This is numerically stabilized in the model by using a small crease or initial displacement in the part at the interface between the upper and lower portions. This crease starts the buckling in a predetermined direction, thus eliminating the initial numeric instability. Physically, parts exhibit buckling behavior that can, in some cases, be quite sensitive to initial imperfections. The appropriate inclusion of initial imperfections is one of the most important modeling choices in a buckling analysis.

Figure 6.2 shows the sequential deformation of the model. Note that the box beam walls folds onto itself in a distorted sinusoid pattern. To prevent the contacting surfaces from penetrating each other a slide surface is defined. The particular slide surface used is the single surface contact (type 4) slide surface. The key feature of this type of slide surface is that every node in the definition is a slave to all other nodes. The advantage of

using this type of slide surface lies in the fact that any portion of the defined area can contact any other portion without undesirable penetration. The disadvantage is that the computation time required for such a slide surface is somewhat longer than for the other slide surfaces. Even though both the outside and inside surfaces of the model may fold into contact, only one type 4 slide surface needs to be defined. This surface is chosen to have normal vectors pointing toward the center or longitudinal axis of the box beam, although outward normal vectors would yield the same solution.

In the single surface contact algorithm, every segment in the definition must check every other segment in the definition for penetration. Thus, computation time increases greatly with the number of segments in the definition. When using this type of slide surface, extra time spent by the analyst in reducing the number of segments in the definition will substantially reduce computation time and hence cost.

Many times the modeler can use engineering intuition to eliminate areas from the slide surface definition that will not contact other areas. A few such examples can be found in this model. The upper portion used as the ram contains 300 elements, 200 of which do not contact any other portion. These upper 200 elements could therefore be excluded from the slide surface definition without degrading the results. In the initial analysis, contact of these elements in the vicinity of the buckle may have been questionable. However, if parameter studies were to be conducted, these elements could be deleted from the slide surface definition for all subsequent runs resulting in a substantial decrease in run time. Additionally, this upper portion should not contact the lower 200 or 300 elements due to the imposed displacement constraints. Here, two or three separate slide surface definitions could be used. By dividing the slide surface definition into three parts (upper, middle, and lower), one could use the intuition that the upper portion might contact the middle but not the bottom portion and the middle portion may contact both the upper and lower portions. Computation time could be saved by using a single surface contact definition on the middle section while the upper and lower sections are separately slaved to the middle using a less costly type of slide surface.

The extent of the middle section would decrease with increased intuition of the behavior. With the insight gained from this model one could probably limit the slide surface definition to the middle section only.

Also of interest in this calculation is the use of four node Belytschko-Tsay shell elements with three integration points through the thickness. Three integration points is the minimum number required to capture bending with plasticity. Purely elastic bending can be captured by two points through the thickness due to the linear stress distribution. Of course, the more integration points used the larger the computation time, with increased accuracy in capturing a complex stress distribution through the thickness.

This part could have been modeled using eight node brick elements. Since brick elements have only one integration point, they would have to be layered at least three deep to capture a stress distribution due to bending, thus substantially increasing the number of elements needed. Another consideration is the ratio of maximum to minimum lengths of the three sides of a brick element. This aspect ratio is best kept less than four for reliable accuracy. Using three elements through the thickness for a given plate thickness will thus severely reduce the in-plane dimensions of the element, hence requiring a very large number of small elements to be used. The formulation of the shell element does not constrain the in-plane dimensions of the element regardless of the thickness, except that the thickness must be sufficiently small that shell theory is applicable. Thus, for problems where the stress gradients through the thickness are small relative to the in-plane stress gradients, as is the case in thin shells and membranes, the shell element will permit fewer elements to be used when compared to brick elements. Also worth noting is the fact that a three node Belytschko-Tsay shell element with three integration points through the thickness is only slightly higher in cpu cost than an eight node brick element which has one integration point.

Another advantage of the shell element is the time step computed by DYNA3D. For the brick element, the time step has a linear dependence on the minimum side length, which in the present case would be the

thickness. The time step computed for the shell element has a much weaker time dependence on the thickness, thus allowing larger time steps to be used for a given element thickness. If wave propagation through the thickness of the structure is not of major concern, then the shell element can be used with greater efficiency and substantial savings in cost over a comparable model with brick elements.

Overall, this problem is an excellent example of the non-linear buckling simulation capabilities of DYNA3D. Figure 6.3 shows the z-displacement of the model after buckling. The top or ram portion of the model has displaced almost half the original height of 12 inches with realistic deformation. This model requires 5 cpu hours to complete on a Cray XMP.

Material Model	3
Density (lbf s ² /in ⁴)	7.1 x 10 ⁻⁴
Elastic Modulus (psi)	3.0 x 10 ⁷
Tangent Modulus (psi)	6.0 x 10 ⁴
Poisson's Ratio	3.0 x 10 ⁻¹
Yield Strength (psi)	3.0x10 ⁴
Hardening Parameter	1.0

Table 6.1 Material Properties.

Figure 6.1 Finite element mesh showing boundary conditions.

Beam buckle #1
time = 0.

Beam buckle #1
time = 1.20000e-03

Beam buckle #1
time = 2.80000e-03

Beam buckle #1
time = 4.40000e-03

Beam buckle #1
time = 6.00000e-03

Figure 6.2 Time sequence of mesh deformation.

Beam buckle #1
time = 9.20000e-03

Beam buckle #1
time = 1.10000e-02

Beam buckle #1
time = 1.40000e-02

Beam buckle #1
time = 1.60000e-02

Beam buckle #1
time = 1.70000e-02

Figure 6.2 (continued)

Beam buckle #1 (in,sec,lb s²/in) INGDY5.dat
time = 1.72000e-02
contours of z-displacement
min=-4.696e+00 in element 1660
max=-1.277e-04 in element 910

contour values
A=-4.30e+00
B=-3.81e+00
C=-3.32e+00
D=-2.84e+00
E=-2.35e+00
F=-1.86e+00
G=-1.37e+00
H=-8.83e-01
I=-3.94e-01

disp. scale factor = 1.000e+00 (default)

Figure 6.3 Z-displacement after buckling.

INGRID input file for Example 5

Beam buckle #1 (in,sec,lb s²/in)

dn3d

term 1.72e-02

plti 1.72e-03

prti 1.0

lcd 1 2 0 1 1 1

plane 2

1.375 0 0 2 0 0 0.01 symm

0 1.375 0 0 2 0 0.01 symm

si 1 single;

start

1 11; -1; 1 74 75 76 91;

0 1.375 0 0 9.7333 9.8667 10 12

si 1 0 1 2 0 5 1 s

b 1 1 1 2 1 1 111111

b 1 1 4 2 1 5 110111

fv 1 1 4 2 1 5 1 273 0 0 -1

mb 1 0 3 2 0 3 2 -.02

pa 1 0 3 1 -.02

mate 1

end

start

-1; 1 11; 1 74 75 76 91;

0 0 1.375 0 9.7333 9.8667 10 12

si 0 1 1 0 2 5 1 s

b 1 1 1 1 2 1 111111

b 1 1 4 1 2 5 110111

fv 1 1 4 1 2 5 1 273 0 0 -1

mb 0 1 3 0 2 3 1 -.02

pa 0 1 3 2 -.02

mate 1

end

c use tp 0.001 interactively in Ingrid

mat 1 3 shell

ro 7.1e-04

e 30e+06

etan 60e+03

pr 0.3

sigy 30e+03

beta 1.0

thick 0.06

tsti 3

endmat

end

TAURUS input file for Example 5

```
reso 1024 1024
state 1
c Figure 6.1
triad rx -90 g
c Figure 6.2
restore rz -180 xtrans 1.374 razx rx -90 ry 60 g
time 1.2e-03 g time 2.8e-03 g time 4.4e-03 g time 6.0e-03 g
time 9.2e-03 g time 1.1e-02 g time 1.4e-02 g time 1.6e-03 g
time 1.72e-03 g
c Figure 6.3
contour 19
end
```

Chapter 7

Space Frame Impact

File: D3SAMP6

7.1 SUMMARY

DYNA3D example 6 uses a combination of beam, shell, and brick elements to model an impact. The beam elements are loaded dynamically by impact along the longitudinal axis causing the first mode of buckling to occur. A time sequence of the deforming mesh as well as time histories of pertinent parameters are presented to quantify the results.

7.2 MODEL DESCRIPTION

Example 6 models the impact of a rigid mass onto a thin plate supported by a space frame. Figure 7.1 shows the quarter symmetry finite element mesh. The lower portion is a space frame 2 inches in diameter and 2 inches tall, composed of beam elements. Rigidly connected to the top of the space frame is a thin plate. A 5 lb. mass, initially 0.2 inches above the plate, is given an initial velocity of 1000 in/sec towards the plate.

The space frame is constructed with three main components. The first component is the lower ring. This uses 3 Belytschko-Schwer beam elements for the quarter model. The end nodes of each element are given fixed boundary conditions, hence these elements experience no loads and are for visual effect only. The second component is the upper ring, also composed of three beam elements. The end nodes of these beam elements are merged to the local nodes of the plate, thus receiving both translational and rotational stiffness from the plate. The third component of the space frame is the vertical columns connecting the lower and upper rings. Each column has ten elements in order to capture the anticipated bending. These columns are not perfectly straight but are slightly bowed out at midspan. This geometric feature was incorporated as a perturbation to help initiate and numerically stabilize the buckling behavior. The beam

elements have the cross-sectional properties of a 1/4 inch solid cylindrical rod. The material properties of all parts are given in Table 7.1.

Beam Elements

Material Model	3
Density (lbf s ² /in ⁴)	2.77x10 ⁻⁴
Elastic Modulus (psi)	3.0x10 ⁷
Tangent Modulus (psi)	3.0x10 ⁴
Poisson's Ratio	3.0x10 ⁻¹
Yield Strength (psi)	5.0x10 ⁴
Hardening Parameter	1.0

Impacting Mass

Material Model	1
Density (lbf s ² /in ⁴)	2.77x10 ⁻³
Elastic Modulus (psi)	3.0x10 ⁸
Poisson's Ratio	3.0x10 ⁻¹

Plate

Material Model	1
Density (lbf s ² /in ⁴)	2.77x10 ⁻⁴
Elastic Modulus (psi)	3.0x10 ⁷
Poisson's Ratio	3.0x10 ⁻¹

Table 7.1 Material properties

The plate is circular with a 1 inch inner diameter, a 3 inch outer diameter, and 1/4 inch thickness. The impacting mass is a 1.8 inch long thick tube. The inner and outer diameters match that of the plate. The mass is constructed of brick elements and given a very stiff elastic material model. All nodes of this part have constrained translational degrees of freedom in the x and y directions. A sliding with voids (type 3) slide surface is defined between the mass and the plate to prevent node penetration between the two parts.

Figure 7.2 shows a time sequence of the deforming mesh. Contact between the mass and the plate is made at time 2.25×10^{-4} sec, after which the columns of the space frame begin to buckle. All columns buckle outward due to the geometric perturbation.

Figure 7.3 shows a time history plot of the rigid body z-displacement of the mass. Maximum displacement occurs at 5.75×10^{-4} sec, after which the mass rebounds upward. Figure 7.4 shows a time history of node 54 z-displacement, which is located on the plate near the upper end of one of the space frame columns. Since the deformations are symmetric and the plate quite rigid, this can be interpreted as the vertical deflection of the columns. Deflection begins at 2.25×10^{-4} seconds and reaches a maximum of 0.155 inches or 8% of the column length at 5.75×10^{-4} seconds. The columns regain a small portion of the deformation and oscillate about the 0.152 inch permanent vertical deflection imparted by the impact. It is apparent from Figure 7.4 that most of the deformation is plastic.

Figure 7.5 shows the effective stress on the plate at the time of maximum deflection. The regions of highest stress occur where the columns attach to the plate.

Beam element example (in,sec,lb-s²/in)Quarter m
time = 0.

Figure 7.1 Initial finite element mesh.

Beam element example (in,sec,lb-s²/in)Quarter m
time = 0.

Beam element example (in,sec,lb-s²/in)Quarter m
time = 2.00000e-04

Figure 7.2 Time sequence of deforming mesh.

Beam element example (in,sec,lb-s²/in)Quarter m
time = 5.75000e-04

Beam element example (in,sec,lb-s²/in)Quarter m
time = 1.00000e-03

Figure 7.2 (continued)

Figure 7.3 Time history of impacting mass z-displacement.

Figure 7.4 Time history of node 54 z-displacement.

Beam element example (in,sec,lb-s²/in)Quarter m

time = 5.75000e-04

contours of eff. stress (v-m)

min= 1.347e+03 in element 10

max= 2.526e+04 in element 10

contour values

A= 3.36e+03

B= 5.84e+03

C= 8.33e+03

D= 1.08e+04

E= 1.33e+04

F= 1.58e+04

G= 1.83e+04

H= 2.08e+04

I= 2.33e+04

Figure 7.5 Effective stress contours of plate at $t = 5.75 \times 10^{-4}$ seconds.

INGRID input file for Example 6

```

Beam element example (in,sec,Ib-s^2/in)Quarter model
dn3d
term 1.0e-03
plti 1.0e-05
prti 9.0e-04
c Define symmetry planes
plane 2
0000-1 0 0.01 symm
000-1 0 0 0.01 symm
c Define slide surface type 3
si 1 sv;
c Begin node definition for beam elements
c Bottom ring nodes
beam
cy 111111 1 0 0 c node 1
cy 000000 1 15 10 c node 2
cy 111111 1 30 0 c node 3
cy 000000 1 45 10 c node 4
cy 111111 1 60 0 c node 5
cy 000000 1 75 10 c node 6
cy 111111 1 90 0 c node 7
c nodes 2,4,6, and 16 are used for aligning cross-sectional properties of
c beam elements

c middle nodes
cy 000110 1.01 0 1 c node 8
cy 000110 1.01 30 1 c node 9
cy 000110 1.01 60 1 c node 10
cy 000110 1.01 90 1 c node 11

c top ring nodes
cy 000000 1 0 2 c node 12
cy 000000 1 30 2 c node 13
cy 000000 1 60 2 c node 14
cy 000000 1 90 2 c node 15
c normal vector node for vertical elements
cy 000000 0 0 1 c node 16
0
c Begin beam element definition
c elements for bottom ring
131112
351114
571116
c elements for top ring
12131112
13141114
14151116
c vertical elements from bottom to middle
1851116
3951116
51051116
71151116

```

```

c vertical elements from top to middle
12 8 5 1 1 16
13 9 5 1 1 16
14 10 5 1 1 16
15 11 5 1 1 16
0
end
c Define top plate
start
1 3 5;1 4;-1;
0.5 1 1.5 0 90 2.0
cyli
c Point normal vector of slide surface towards solid ram part
si+ 1 1 1 3 2 1 1 s 0 0 3
mate 3
end

bptol 1 2 0.001
c Define solid ram
velocity 0 0 -1000
start
1 3 5;1 4;1 4;
0.5 1 1.5 0 90 2.20 3.0
cyli
si 1 1 1 3 2 1 1 m
c Fix x and y displacement of ram
b 1 1 1 3 2 2 110000
mate 2
end

c Define material properties

c Beam properties
mat 1 3 beam
e 30.e+06
etan 30.e+03
sigy 50000
pr 0.3
ro 2.77e-04
beta 1.0
sthi 0.25
tthi 0.25
endmat

c Ram properties
mat 2 1
e 30.e+07
pr 0.3
ro 2.77e-03
endmat

c Plate properties
mat 3 1 shell

```

e 30.e+06
ro 2.77e-04
pr 0.3
thick 0.125
endmat

end

TAURUS input file for Example 6

```
reso 1024 1024
c Figure 7.1
state 1 rx -90 ry 15 triad g
restore razx rayz triad rx -90 ry 15
c Figure 7.2
state 1 g
time 2.0e-04 g
time 5.75e-04 g
time 1.0e-03 g
c Figure 7.3
phs2 mats 1 2 nodes 1 54 gather
mtime 3 1 2
c Figure 7.4
ntime 3 1 54
c Figure 7.5
phs1 time 5.75e-04 dms 1 3 rx 90 contour 9
end
```

Chapter 8

Thin Beam Subjected to an Impact

File: D3SAMP7

8.1 SUMMARY

DYNA3D example 7 is a simple model of a thin beam subjected to a transverse impact load. The key features of this model are the use of four node Belytschko-Tsay shell elements and the modeling of dynamic bending response. Stresses and strains through the thickness of the beam, as well as kinematic responses, are plotted and examined as a function of time. The dynamic responses compare favorably to approximate analytical results.

8.2 MODEL DESCRIPTION

Example 7 models a thin rectangular beam 0.6 inches wide by 10 inches long with a thickness of 0.125 inches. Symmetry is used about the plane in the center of the span thus reducing the number of elements by one half. Figure 8.1 shows the finite element mesh. The end boundary condition is fixed with the displacements on the x-z surfaces constrained in the y-direction.

Ten four node shell elements are used, with five evenly spaced integration points through the thickness (trapezoidal integration). Using the trapezoidal integration option with three or more points in odd increments allows the surface and mid-plane stresses and strains to be captured exactly, as opposed to using Gauss quadrature which requires these stresses and strains to be extrapolated or interpolated. The shell elements are given the elastic/perfectly plastic material properties of 6061-T6 aluminum using material model 3 in DYNA3D. These properties are listed in Table 8.1.

Material Model	3
Density (lbf s ² /in ⁴)	2.61×10^{-4}
Elastic Modulus (psi)	1.04×10^7
Tangent Modulus (psi)	0.0
Poisson's Ratio	3.3×10^{-1}
Yield Strength (psi)	4.14×10^4
Hardening Parameter	1.0

Table 8.1 Material Properties.

The middle 2 inches of the ten inch span are given an initial velocity of 5,000 in/sec in the negative z-direction. The response is simulated for 1.75 milliseconds. Figure 8.2 shows a time sequence of the deforming mesh with its symmetric reflection. Figure 8.3 shows the kinematic responses of node 19, which is at the center of the span. The simulated impact produces a maximum deflection of 0.78 inches at the center. This deflection, more than four times the shell thickness, is sufficient to make large deformation effects important in this problem.

The low frequency transverse structural vibration resulting from the impact can be seen most clearly in the displacement response. Note that the center of the span is oscillating in time. The period of oscillation is approximately 0.6 milliseconds. Figure 8.4 shows the deformed mesh at 0.9 milliseconds and 1.4 milliseconds with the z-displacements amplified by a factor of 5. The deformed mesh at 0.9 milliseconds has three troughs and four crests over the span. This shape occurs again at 1.4 milliseconds which is in the second cycle of structural vibration. These transverse waves propagate from the center of the span to the fixed ends where they are reflected back towards the center for another cycle.

The shapes in Figure 8.4 are characteristic of the third mode of vibration. Elastic transverse vibration theory for a fixed end beam with similar stiffness and mass properties predicts a third mode natural period of 0.7 milliseconds. Even though the model experiences plastic strains, the elastic theory can be used for an approximate comparison. The first and second modes are not distinguishable in the given time interval. Higher

modes can be seen in the velocity and acceleration responses but they are indistinguishable in the deformed geometry plots, because their amplitudes are relatively smaller.

Figures 8.5-8.7 show the effective stress and effective plastic strain of the center element. Figure 8.5 corresponds to the bottom surface of the element. Figure 8.6 corresponds to the internal mid-plane surface, and Figure 8.7 to the top surface. The bending stresses add to the membrane stresses at the bottom surface and subtract from the membrane stresses at top surface, thus the bottom fibers suffer the most plastic strain. The membrane stresses appear to be significantly larger than the compressive bending stresses on the top surface.

Stresses and strains through the thickness of a shell element are selected in post-processor TAURUS by using the Shell inner, Shell middle, and Shell outer commands prior to the stress or strain display command. For this model, three trapezoidal rule integration points through the thickness would have been sufficient to capture the three stress planes discussed.

Impulsively loaded thin beam (
time = 0.00000E+00
element numbers

Figure 8.1 Finite element mesh.

Impulsively loaded thin beam (
time = 0.00000E+00

Impulsively loaded thin beam (
time = 0.10000E-03

Impulsively loaded thin beam (
time = 0.20000E-03

Figure 8.2 Mesh deformation sequence.

Impulsively loaded thin beam (
time = $0.30000E-03$

Impulsively loaded thin beam (
time = $0.70000E-03$

Impulsively loaded thin beam (
time = $0.90000E-03$

Figure 8.2 (continued)

Figure 8.3 Kinematic responses of node 19.

Impulsively loaded thin beam (
time = 0.90000E-03

disp. scale factors: x = 0.100E+01 y = 0.100E+01 z = 0.500E+01

Impulsively loaded thin beam (
time = 0.14000E-02

disp. scale factors: x = 0.100E+01 y = 0.100E+01 z = 0.500E+01

Figure 8.4 Magnified deformed mesh plots at $t = 0.9$ ms and $t = 1.4$ ms.

Figure 3.5 Effective stress and effective plastic strain of element 9 bottom surface.

Figure 8.6 Effective stress and effective plastic strain of element 9 mid-plane.

Figure 8.7 Effective stress and effective plastic strain of element 9 top surface.

INGRID input file for Example 7

Impulsively loaded thin beam (in,sec,lb sec²/in)

dn3d
term 2.0e-03
prti 1.0
plti 1.0e-05

c Define index space and coordinates for outer portion

start
4 11;1 2;-1;
1.5 5.0 0. 0.6 0.

c Set lengthwise edge boundary conditions

b 1 1 1 2 1 1 010101
b 1 2 1 2 2 1 000101

c Set fixed end boundary condition

b 2 1 1 2 2 1 111111

mate 1

end

c Define index space and corresponding coordinates for center portion

c that is given initial velocity

start
1 4;1 2;-1;
0 1.5 0. 6 0

c Set symmetry boundary conditions on center

b 1 1 1 1 2 1 110111

c Set symmetry boundary conditions on center

b 1 1 1 1 2 1 110111

c Set lengthwise edge boundary conditions

b 1 1 1 2 1 1 010101

b 1 2 1 2 2 1 000101

velocity 0. 0. -5000.

mate 1

end

c Use tp 0.001 interactively in INGRID

c Define material properties

mat 1 3 shell
c 6061-T6 AL
ro .261e-03
e 10.4e+03
pr 0.33
sigy 41.4e+03
etan 0.0
beta 1.0
thick 0.125
tsti 5
endmat
end

TAURUS input file for Example 7

reso 1024 1024

Figure 8.1

time 0 elplt

c Figure 8.2

rayz rx -90 g time .1e-03 g time .2e-03 g time .3e-03 g time .5e-03 g
time .7e-03 g time 0.9e-03 g time 1.6e-03 g

c Figure 8.3

phs2 nodes 1 19 gather

ntime 3 1 19 ntime 6 1 19

c Figure 8.4

phs1 dsfs 1 1 5 time .9e-03 g time 1.4e-03 g

c Figure 8.5

phs2 elements 1 9 gather

shell inner etime 9 1 9 etime 7 1 9

c Figure 8.6

shell middle etime 9 1 9 etime 7 1 9

c Figure 8.7

shell outer etime 9 1 9 etime 7 1 9

end

Chapter 9

Impact on a Cylindrical Shell

File: D3SAMP8

9.1 SUMMARY

DYNA3D example 8 simulates an impact on a section of a cylindrical shell. The key feature of this example is the large plastic deformation of four node Belytschko-Tsay shell elements. Contour plots of stress, strain, and displacements as well as time histories of nodal displacement, velocity, and acceleration are presented. The resulting displaced shape as well as through-thickness stresses and strains are representative of a physical impact of such a structure.

9.2 MODEL DESCRIPTION

Example 8 models a section of a circular cylindrical shell with a radius of 2.938 inches, length of 12.56 inches, and thickness of 0.125 inches, subjected to an impact load that causes large deformation in the radial direction. Figure 9.1 shows the finite element mesh used in this model. Symmetry is used about the y-z plane by constraining the nodal x-displacements as well as y and z rotations. The ends of the cylinder have the x and y displacements constrained while the bottom edge has all displacements and rotations constrained.

The elements used in the model are four node Belytschko-Tsay shell elements with 5 gauss integration points through the thickness and the material properties of an elastic/perfectly plastic 6061-T6 aluminum. Each element has a uniform thickness of 0.125 in. A summary of the material properties can be seen in Table 9.1.

An initial velocity of 5650 in/sec in the negative y-direction is given to 65 interior nodes. The resulting deformation can be seen in the time

sequence of Figure 9.2. The deformed mesh is shown with its symmetric reflection for clarity.

Material Model	3
Density (lbf s ² /in ⁴)	2.50x10 ⁻⁴
Elastic Modulus (psi)	1.05x10 ⁷
Hardening Modulus (psi)	0.0
Poisson's Ratios	3.3x10 ⁻¹
Yield Strength (psi)	4.40x10 ⁴
Hardening Parameter	1.0

Table 9.1 Material properties.

Figure 9.3 shows the kinematic response time histories of node 8, which is centrally located on the top of the shell. The maximum deflection of 1.27 inches occurs at 0.425 milliseconds in the 1.0 milliseconds simulation. All plots show structural vibration as a result of the impact. The lowest mode appears to have a period of approximately 0.7 milliseconds as seen in the displacement response. Higher modes can be found in the velocity time history.

Figure 9.4 shows contours of y-displacement at 1.0 milliseconds. The deformed shape is representative of a real impact on such a structure. Figures 9.5-9.7 show contours of effective stress and effective plastic strain of the inner, middle, and outer integration points through the thickness, respectively.

Note that the maximum effective plastic strain of 42.1% occurs on the inner surface of element 78, and 25.2% on the outer surface of element 82, while the midpoint maximum effective plastic strain is less than 13.4%. This strain distribution is the result of both membrane and bending stresses. These high strains occur near the lengthwise crease in the shell. This model is a good example of the use of four node shell elements combined with an elastic/plastic material model to analyze a thin-walled structure under impact loads.

Figure 9.1 Initial finite element mesh.

Cylindrical shell (in,sec,lb s²/in)
time = 0.

Cylindrical shell (in,sec,lb s²/in)
time = 1.00000e-04

Cylindrical shell (in,sec,lb s²/in)
time = 3.00000e-04

Figure 9.2 Time sequence of mesh deformation.

Cylindrical shell (in,sec,lb s²/in)
time = 5.00000e-04

Cylindrical shell (in,sec,lb s²/in)
time = 7.00000e-04

Cylindrical shell (in,sec,lb s²/in)
time = 1.00000e-03

Figure 9.2 (continued)

Figure 9.3 Kinematic response time histories of center node.

Cylindrical shell (in,sec,lb
time = 0.10000E-02
contours of y-displacement
min=-0.113E+01 in element 21
max= 0.518E-01 in element 118

contour values
A=-1.83E+00
B=-5.06E-01
C=-7.83E-01
D=-6.60E-01
E=-5.38E-01
F=-4.15E-01
G=-2.93E-01
H=-1.70E-01
I=-4.73E-02

disp. scale factor = 0.100E+01 (default)

Figure 9.4 Contours of y-displacement.

Cylindrical shell (in,sec,lb
time = 0.10000E-02
contours of eff. stress (psi)
min= 0.345E+04 in element 25
max= 0.440E+05 in element 102

contour values
A= 6.85E+03
B= 1.11E+04
C= 1.33E+04
D= 1.55E+04
E= 1.77E+04
F= 1.99E+04
G= 2.21E+04
H= 2.43E+04
I= 2.65E+04

disp. scale factor = 0.100E+01 (default)

Cylindrical shell (in,sec,lb
time = 0.10000E-02
contours of eff. plastic strain
min= 0.000E+00 in element 127
max= 0.421E+00 in element 78

contour values
A= 0.054E-02
B= 7.91E-02
C= 1.23E-01
D= 1.67E-01
E= 2.10E-01
F= 2.54E-01
G= 2.98E-01
H= 3.42E-01
I= 3.86E-01

disp. scale factor = 0.100E+01 (default)

Figure 9.5 Effective stress and effective plastic strain contours of inner integration point.

Cylindrical shell (in,sec,lb
time = 0.10000E-02
contours of eff. stress (v-m)
min= 0.460E+04 in element 78
max= 0.395E+05 in element 125

contour values
D= 7.60E+03
B= 1.12E+04
C= 1.40E+04
D= 1.85E+04
TF= 2.21E+04
W= 3.77E+04
IG= 5.99E+04
I= 7.29E+04
I= 8.55E+04

disp. scale factor = 0.100E+01 (default)

Cylindrical shell (in,sec,lb
time = 0.10000E-02
contours of eff. plastic strain
min= 0.000E+00 in element 127
max= 0.134E+00 in element 81

contour values
D= 1.11E-02
B= 2.51E-02
C= 3.90E-02
D= 5.20E-02
TF= 6.60E-02
W= 8.00E-02
IG= 9.47E-02
I= 1.09E-01
I= 1.23E-01

disp. scale factor = 0.100E+01 (default)

Figure 9.6 Effective stress and effective plastic strain contours of middle integration point.

Cylindrical shell (in,sec,lb
time = 0.10000E-02
contours of eff. stress (v-m)
min= 0.547E+04 in element 95
max= 0.440E+05 in element 78

contour values
A= 0.71E+03
B= 1.27E+04
C= 1.67E+04
D= 1.87E+04
E= 1.47E+04
F= 0.87E+04
G= 0.20E+04
H= 0.60E+04
I= 4.00E+04

disp. scale factor = 0.100E+01 (default)

Cylindrical shell (in,sec,lb
time = 0.10000E-02
contours of eff. plastic strain
min= 0.399E-04 in element 126
max= 0.252E+00 in element 82

contour values
A= 2.12E-02
B= 4.74E-02
C= 7.96E-02
D= 9.97E-02
E= 1.20E-01
F= 1.32E-01
G= 1.70E-01
H= 2.05E-01
I= 2.31E-01

disp. scale factor = 0.100E+01 (default)

Figure 9.7 Effective stress and effective plastic strain contours of outer integration point.

INGRID input file for Example 8

```
Cylindrical shell (in,sec,lb s^2/in)
dn3d
term 1.0e-03
plti 1.0e-05
prti 1.0
c Define symmetry plane
plane 1
0 1 0 -1 0 0 0.0001 symm
c Define inner portion of cylinder with initial velocity
c Index space and corresponding cylindrical coordinates
velocity 0.0 -5650 0
start
-1;1 5;2 14;
2.938 90 52.5 -0.785 -10.205
cyli
mate 1
end
c Define outer portion of cylinder, note velocities are reset to zero.
c If this were not the case all elements in this part would have the
c above initial velocity.
velocity 0 0 0
start
-1;1 5 9; 1 2 14 17;
2.938 90 52.5 30 0 -.785 -10.205 -12.56
cyli
c Fix all displacements on bottom edge
b 1 3 1 1 3 4 111111
c Fix x and y translation on ends
b 1 1 1 1 3 1 1100000
b 1 1 4 1 3 4 110000
c Delete region that corresponds to the first part definition
d 1 1 2 1 2 3
mate 1
end
c use tp 0.001 interactively in INGRID
c Define material properties
c 6061 t-6 AL
mat 1 3 shell
thick 0.125
ro 2.5e-04
e 10.5e+06
etan 0.0
pr 0.33
sigy 44.0e+03
beta 1.0
tsti 5
endmat
end
```

TAURUS input file for Example 8

reso 1024 1024

triad

c Figure 9.1

state 1 rx 15 g rx -15 ry 90 g

c Figure 9.2

restore traid

rayz ry 45 g time .1e-03 g time .3e-03 g time .5e-03 g time .7e-03 g

time 1.e-03 g

c Figure 9.3

phs2 nodes 1 8 gather

ntime 2 1 8 ntime 5 1 8

c Figure 9.4

phs1 restore rayz rx25 ry25 time1.0e-03

contour 18

c Figure 9.5

shell inner contour 9 contour 7

c Figure 9.6

shell middle contour 9 contour 7

c Figure 9.7

shell outer contour 9 contour 7

end

Chapter 10

Simply Supported Flat Plate

File: D3SAMP9

10.1 SUMMARY

DYNA3D example 9 investigates the response of a simply supported flat plate subjected to a rapidly applied uniform pressure load. Response parameters such as maximum deflection, natural frequency, and maximum stress are compared with analytical results. The DYNA3D results are in close agreement with the analytical solutions.

10.2 MODEL DESCRIPTION

Example 9 models the response of a simply supported flat plate subjected to a rapidly applied uniform pressure load. The 10 inch by 10 inch, 1/2 inch thick plate is modeled using two planes of symmetry: the x-z plane and the y-z plane as seen in Figure 10.1. A total of 16 elements are used in the quarter model, each having five gauss integration points through the thickness. Material model 3 (elastic/plastic) is used with the properties of a perfectly elastic aluminum (the yield stress is set artificially high to prevent plasticity). A summary of the material properties is shown in Table 10.1.

Material Model	3
Density (lbf s ² /in ⁴)	2.588x10 ⁻⁴
Elastic Modulus (psi)	1.0x10 ⁷
Hardening Modulus (psi)	0.0
Poisson's Ratio	3.0x10 ⁻¹
Yield Strength (psi)	1.00x10 ⁵
Hardening Parameter	1.0

Table 10.1 Material properties

A uniform pressure load of 300 psi is applied on the top surface instantaneously at time zero and held constant for the entire 1.2 millisecond simulation. Figure 10.2 shows the displacement, velocity, and acceleration time histories of node 1, which is located at the center of the plate (lower left corner of quarter model). The maximum deflection of 0.2186 inches in the negative z-direction occurs at 0.55 milliseconds.

Now consider an approximate analytical estimate of the deflection. Equation 10.1 expresses the maximum deflection of a square plate in terms of the uniform pressure load q , side length a , flexural rigidity D , and semiempirical coefficient α . This equation, derived from elastic plate theory, assumes the plate consists of perfectly elastic, homogeneous, isotropic material with uniform thickness which is small in comparison to the edge lengths. Deflections are assumed small in comparison to the thickness as well as the load being static.

$$\delta_{\max} = \frac{\alpha q a^4}{D} \quad (10.1)$$

This equation predicts a maximum static deflection at the center of the plate of 0.11 inches for the given configuration. Dynamic load deflections in general amplify the static deflection for a given load by an amount equal to the dynamic load factor. Such a load factor is not easily calculated for a plate under large deflections, but a reasonable approximation is 2.0. The DYNA3D results agree well with the analytical estimate based on this assumed value of dynamic load factor.

Also of interest is the natural free vibration frequency of the plate. The displacement response of Figure 10.2 predicts a fundamental period of 1.10 milliseconds (frequency of 909 Hz). Equation 10.2 expresses the fundamental period of a square plate in terms of the side length a , flexural rigidity D , mass density ρ , and plate thickness t . The same assumptions that applied to Equation 10.1 also apply to Equation 10.2.

$$T = \frac{a^2}{\pi} \sqrt{\frac{\rho t}{D}} \quad (10.2)$$

This equation predicts a fundamental period of 1.07 milliseconds (frequency of 935 Hz), which is in excellent agreement with the DYNA3D results.

Figure 10.3 shows a time history plot of the stress σ_{xx} in element 1, which is located at the center of the plate. This plot corresponds to the bottom or tension surface of the plate. The response of the stress σ_{yy} is identical due to symmetry. The peak stress occurs 0.03 milliseconds prior to the maximum deflection with a value of 67.7 ksi. Using a maximum deflection of 0.22 inches in Equation 10.1 and solving for the load q gives 619 psi. Equation 10.3 expresses the maximum stress $\sigma_{xx, \max}$ in the plate in terms of the load q , side length a , thickness t , and semiempirical coefficient β . This equation is also based on elastic theory. Using a load of 619 psi in Equation 10.3 yields a maximum stress of 71.2 ksi, which agrees well with the numerical analysis.

$$\sigma_{xx, \max} = \frac{\beta q a^2}{t^2} \quad (10.3)$$

Figure 10.4 shows a contour plot of the z -displacement at $t = 0.55$ ms. The mesh shown has the x - z and y - z planes reflected symmetrically for clarity. The displaced shape is in good agreement with analytical contour plots. Figure 10.5 shows the σ_{xx} contour plots for the upper, middle, and lower quadrature points through the thickness at time equal 0.55 ms.

Simply supported plate (in, sec, lb s²/in)
time = 0.

Figure 10.1 Initial finite element mesh of plate.

Simply supported plate (in,sec,lb s²/in)

Simply supported plate (in,sec,lb s²/in)

Simply supported plate (in,sec,lb s²/in)

Figure 10.2 Kinematic response of node 1.

Figure 10.3 Time history of σ_{xx} in element 1.

Figure 10.4 Contours of z-displacement at $t = 0.55$ ms.

disp. scale factor = 1.000e+00 (default)

disp. scale factor = 1.000e+00 (default)

disp. scale factor = 1.000e+00 (default)

Figure 10.5 Contours of σ_{xx} at upper, middle, and lower quadrature points at $t = 0.55$ ms.

INGRID input file for Example 9

```
simply supported plate (in,sec.lb s^2/in)
dn3d
term 1.2e-03
plti 2.0e-05
prti 2.0e-04
lcd 1 2 0 1.0 1.0 1.0
c Define symmetry planes
plane 2
1.0 0.0 0.0 -1 0 0.001 symm
0.0 1.0 0.0 -1 0 00 0.001 symm
c Define index space and corresponding coordinates
start
1 5;1 5;-1;
0 5.0 0 5.0 0.0
c Fix y displacement on outside edges
b 2 1 1 2 2 1 001000
b 1 2 1 2 2 1 001000
c Define applied pressure region
pr 1 1 1 2 2 1 1 300.0 0 0 -1
c Specify element printout blocks
epb 0.0 0.0 0.0 0.625 0.625 0;
mate 1
end
c Specify hourglassing parameters
qh 0.05
q1 1.2
c Define material
mat 1 3 shell
c aluminum
ro .2588e-03
e 10.0e+06
pr 0.3
sigy 100.0e+03
etan 0.0
beta 1.0
thick 0.5
tsti 5
endmat
end
```

TAURUS input file for Example 9

reso 1024 1024

c Figure 10.1

state 1 rx -75 g

c Figure 10.2

phs2 nodes 1 1 elements 1 1 gather ntime 3 1 1 ntime 6 1 1 ntime 9 1 1

c Figure 10.3 shell inner

etime 1 1 1

cFigure 10.4

phs1 restore razx rayz

time 0.55e 03 contour 19

c Figure 10.5

shell outer contour 1 shell middle contour 1

shell inner contour 1

end

Chapter 11

Hourglassing of Simply Supported Plate

File: D3SAMP10

11.1 SUMMARY

DYNA3D example 10 is a rerun of example 9 with a different value of hourglass viscosity. A brief description of hourglassing of finite elements is presented as well as a comparison to the response of example 9.

11.2 MODEL DESCRIPTION

Example 10 is an exact duplicate of example 9 with the exception of the hourglass viscosity coefficient value. Figure 11.1 shows two corner supported plates. The plate on the left has undergone deformation with no appreciable hourglassing of the elements. The plate on the right has experienced hourglassing of its elements in the so-called "w-mode" or "eggcrate mode," named for the alternate up and down displacements of the nodes. There are several other modes of hourglassing that can occur, including both in-plane and out-of-plane modes. In general, hourglassing involves the nodal deformations of finite elements that do not contribute to the strain energy of the element.

Hourglass modes arise from the use of single point Gauss quadrature to evaluate integrals appearing in the shell element formulation. It is necessary to use single point integration in an explicit code like DYNA3D, and therefore some techniques for stabilizing the spurious hourglass modes must be implemented. DYNA3D offers both viscous hourglass control (the default) and stiffness hourglass control. The default parameters have been chosen to give acceptable performance over a wide range of problems.

Hourglass modes tend to form over a time duration that is typically much shorter than the time duration of the structural response, and they

are often observed to be oscillatory. Hourglass modes that are a stable kinematic component of the global deformation modes occur over a much larger time frame and must be admissible. Therefore, DYNA3D resists undesirable hourglassing with viscous damping capable of stopping the formation of anomalous modes but having a negligible affect on the stable global modes. Since the hourglass modes are orthogonal to the real deformations, work done by hourglass resistance is neglected in the energy equation. This can lead to a slight loss of energy, however, hourglass viscosity should always be used.[7]

The default value for the hourglass coefficient is 0.10. The recommended range is 0.05 to 0.15. These values apply equally to the shells and eight-node brick element. The values used in examples 9 and 10 are 0.05 and 0.005 respectively. The `qh` command in the pre-processor INGRID is used to specify this value when different from the default.

Figure 11.2 shows the kinematic responses of the center node of the plate. As a result of reducing the hourglass coefficient an order of magnitude, the displacement of the center node has increased slightly in amplitude. The maximum deflection of -0.2213 inches occurs at 0.55 milliseconds, compared to the maximum deflection of D3SAMP9, -0.2186 inches, also occurring at 0.55 milliseconds. This node then rebounds, reaching a maximum positive deflection of 0.0036 inches. The response of example 9 rebounded to 0.0000 inches. Both of the maximum rebound deflections occur at 1.1 milliseconds. The difference is small (4%), and it is not apparent which is more accurate.

The velocity response of the center node shows a similar amplitude increase. Example 10 with the lower hourglass coefficient shows a 4% larger amplitude than example 9. Evidence of high frequency oscillation is apparent on the example 10 velocity plot which is not the case with example 9, although this oscillation is not necessarily a direct result of hourglassing.

An 8% increase in acceleration amplitude can be found in example 10 when compared to example 9. Figure 11.3 show the x-direction stress time

history of the center element. A 2% increase in peak stress can be found in example 10 response over example 9.

Thus, although small, the damping effect of the hourglass coefficient can be seen, especially in the velocity and acceleration responses. Note from the z-displacement and x-stress contour plots (Figures 11.4 and 11.5) that no hourglass modes are apparent. This example problem demonstrates the more subtle aspects of hourglass control, i.e., the effect of hourglass control parameters on the various response parameters as opposed to outright element hourglassing. As mentioned above, the hourglass control is not intended to affect normal modes of deformation, but from this example it is seen that it can. The difference in responses between example 9 and example 10 are quite small. Any adjustment of this parameter is best left to the experienced user.

Without hourglassing

With hourglassing

Figure 11.1 Example meshes illustrating the effect of hourglassing.

Figure 11.2 Kinematic response time histories of node 1.

Figure 11.3 Time history of σ_{xx} in element 1.

Figure 11.4 Contours of z-displacement at $t = 0.55$ ms.

Simply supported plate (ln,se
time = 0.55000E-03

contours of x-stress

min= 0.232E+04 in element 16
max= 0.668E+05 in element 1

contour values

A= 7.74E+03
B= 1.44E+04
C= 2.11E+04
D= 2.78E+04
E= 3.45E+04
F= 4.13E+04
G= 4.80E+04
H= 5.47E+04
I= 6.14E+04

disp. scale factor = 0.100E+01 (default)

Figure 11.5 Contours of σ_{xx} at $t = 0.55$ ms.

INGRID input file for Example 10

Simply supported plate (in,sec,lb s²/in)

dn3d

term 1.2e-03

plti 2.0e-05

prti 2.0e-04

lcd 1 2 0 1.0 1.0 1.0

plane 2

1.0 0.0 0.0 0 -1 0 0.001 symm

0.0 1.0 0.0 -1 0 0.001 symm

start

1 5;1 5;-1;

0 5.0 0000 5.0 0.0

b 2 1 1 2 2 1 001000

b 1 2 1 2 2 1 001000

pr 1 1 1 2 2 1 1 300.0 0 0 -1

epb 0.0 0.0 0.0 0.625 0.625 0;

mate 1

end

qh 0.00S

q1 1.2

mat 1 3 shell

al

ro .2588e-03

e 10.0e+06

pr 0.3

sigy 100.0e+03

etan 0.0

beta 1.0

thick 0.5

tsti 5

endmat

end end

TAURUS input file for Example 10

reso 1024 1024

c Figure 11.1 is a hand sketch

c figure 11.2

phs2 nodes 1 1 elements 1 1 gather ntime 3 1 1 ntime 6 1 1 ntime 9 1 1

c Figure 11.3 Shell inner

etime 1 1 1

c Figure 11.4

phs1 restore razx rayz

time 0.55e-03 contour 19

c Figure 11.5

contour 1

end

Acknowledgements

The authors wish to acknowledge the many DYNA3D users who, over the years, have contributed their models for use as example problems. It is through interaction with code users, both inside and outside LLNL, that the inspiration and material for this manual originated. In particular, the analysts in the LLNL Applied Mechanics Group and Advanced Engineering Analysis Group have been especially generous in sharing their work for the benefit of others. Special thanks should also be given to Todd Slavik for his thoroughness in reviewing this manual.

References

- [1] Hallquist, J. O., and Whirley, R. G., *DYNA3D USER'S MANUAL: Nonlinear Dynamic Analysis of Structures in Three Dimensions*, University of California, Lawrence Livermore National Laboratory, Report UCID-19592, Rev. 5, May, 1989.
- [2] Wilkins, M. L., and Guinan, M. W., "Impact of Cylinders on a Rigid Boundary", *J. of Appl. Phys.*, 44 (1973).
- [2] Stillman, D. W., and Hallquist, J. O., *INGRID: A Three-Dimensional Mesh Generator for Modeling Nonlinear Systems*, University of California, Lawrence Livermore National Laboratory, Report UCID-20506, July, 1985.
- [3] Rainsberger, R., *INGRID by Example: A Pictorial Tutorial*, University of California, Lawrence Livermore National Laboratory, Report UCID-21566, November, 1988.
- [4] Rainsberger, R., *INGRID Features and Updates*, University of California, Lawrence Livermore National Laboratory, Report UCID-21620, January, 1988.
- [5] Wayne, B., *INGRID: Entering Material Specifications*, University of California, Lawrence Livermore National Laboratory, Report UCID-21798, July, 1989.
- [6] Brown, B. E., and Hallquist, J. O., *TAURUS: An Interactive Post-Processor for the Analysis Codes NIKE3D, DYNA3D, TACO3D, and GEMINI*, University of California, Lawrence Livermore National Laboratory, Report UCID-19392, Rev. 1, May, 1984.
- [7] Hallquist, J. O., *Theoretical Manual for DYNA3D*, March, 1983, University of California, Lawrence Livermore National Laboratory, pp. 19-20.

END

DATE FILMED

03 / 05 / 91