DYNA3D

A Nonlinear, Explicit, Three-Dimensional Finite Element Code For Solid and Structural Mechanics-User Manual

Robert G. Whirley Bruce E. Engelmann

Methods Development Group Mechanical Engineering

> Originated by: J. O. Hallquist

November, 1993



•

1

DISTRIBUTION OF THIS DOCUMENT IS UNLIMITED

DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, make any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.

DISCLAIMER

Portions of this document may be illegible in electronic image products. Images are produced from the best available original document. (

4 F2

5

(;

/

**

Table of Contents

ABSTRACT			viii
1.0	INTR	ODUCTION	1
2.0	OVEF	RVIEW OF DYNA3D	5
	2.1	NOMENCLATURE AND NOTATIONAL CONVENTIONS	8
	2.2	UNITS	9
	2.3	GENERAL FORMULATION	9
	2.4	MATERIAL MODELS	10
	2.5	EQUATIONS OF STATE	11
	2.6	FINITE ELEMENT FORMULATION	12
	2.7	GENERALIZED RAYLEIGH DAMPING	14
	2.8	LUMPED PARAMETER ELEMENTS	15
	2.9	INITIAL AND BOUNDARY CONDITIONS	15
	2.10	INITIAL VELOCITIES FOR ROTATING BODIES	16
	2.11	LOADS	17
	2.12	GENERAL INTERFACE CONTACT	18
	2.13	CONSTRAINT EQUATIONS	24
	2.14	CROSS SECTION FORCES	24
	2.15	TOTAL INTERFACE FORCES AND REACTION FORCES	24
	2.16	HIGH EXPLOSIVE PROGRAMMED BURN MODELS	25
	2.17	RIGID BODY DYNAMICS	25
	2.18	ANALYSIS OPTIONS	26
	2.19	STATIC INITIALIZATION FROM NIKE3D	27
	2.20	DYNAMIC RELAXATION FOR STATIC INITIALIZATION	27
	2.21	DYNAMIC RELAXATION FOR STRESSES IN ROTATING BODIES	28
	2.22	RESTART	29
	2.23	MATERIAL MODEL DRIVER	30
	2.24	COUPLING WITH CRASH VICTIM SIMULATION (CVS) CODES	30
3.0	ANAI	LYSIS WITH DYNA3D	33
	3.1	PRE-PROCESSING AND MODEL GENERATION	33
	3.2	STARTING A NEW DYNA3D ANALYSIS	33

TABLE OF CONTENTS

DYNA3D User Manual

•

1

	3.3	INTERACTIVE ANALYSIS STATUS QUERY	36
	3.4	RESTARTING A DYNA3D ANALYSIS	36
	3.5	POST-PROCESSING AND RESULTS DISPLAY	38
	3.6	INTERFACE FORCES AND NODAL REACTION FORCES	39
4.0	INPU'	Г FORMAT	41
	4.1	CONTROL CARDS	41
		Problem Definition Notes	51
		Problem Definition Notes	52
	4.2	MATERIALS	59
		Material Control Card Notes	62
		Material Type 1 (Elastic)	65
		Material Type 2 (Orthotropic Elastic)	66
		Material Type 3 (Kinematic/Isotropic Elastic-Plastic)	71
		Material Type 4 (Thermo-Elastic-Plastic)	74
		Material Type 5 (Soil and Crushable Foam)	77
		Material Type 6 (Viscoelastic)	82
		Material Type 7 (Blatz - Ko Hyperelastic Rubber)	83
		Material Type 8 (High Explosive Burn)	84
		Material Type 9 (Fluid)	86
		Material Type 10 (Isotropic-Elastic-Plastic-Hydrodynamic)	87
		Material Type 11 (Steinberg-Guinan High Rate Elastic-Plastic)	91
		Material Type 12 (Isotropic-Elastic-Plastic)	95
		Material Type 13 (Elastic-Plastic with Failure)	96
		Material Type 14 (Soil and Crushable Foam with Failure)	9 8
		Material Type 15 (Johnson/Cook Elastic-Plastic)	102
		Material Type 16 (Concrete/Geological Material)	106
		Material Type 17 (Isotropic Elastic-Plastic with Oriented Cracks)	113
		Material Type 18 (Power Law Isotropic Elastic-Plastic)	115
		Material Type 19 (Strain Rate Dependent Isotropic Elastic-Plastic)	116
		Material Type 20 (Rigid)	119
		Material Type 21 (Thermal Orthotropic Elastic)	121
		Material Type 22 (Fiber Composite with Damage)	124

•

~

(.

ű.

()

TABLE OF CONTENTS

.

Material Type 23 (Thermal Orthotropic Elastic with Variable Properties)	128
Material Type 24 (Rate-Dependent Tabular Isotropic Elastic-Plastic)	131
Material Type 25 (Extended Two Invariant Geologic Cap)	134
Material Type 26 (Metallic Honeycomb)	139
Material Type 27 (Compressible Mooney-Rivlin Hyperelastic Rubber)	142
Material Type 28 (Resultant Plasticity)	144
Material Type 29 (Not Currently Available)	145
Material Type 30 (Closed-Form Update Elastic-Plastic for Shells)	146
Material Type 31 (Frazer-Nash Hyperelastic Rubber)	147
Material Type 32 (Ramberg-Osgood Elastic-Plastic)	149
Material Type 33 (General Anisotropic Elastic-Plastic)	151
Material Type 34 (Normal Anisotropic Elastic-Plastic for Shells)	155
Material Type 35 (Elastic-Plastic with Forming Limit Diagram)	157
Material Type 36 (Brittle Damage Model)	165
Material Type 37 (Three-Invariant Viscoplastic Cap Model)	166
Material Type 38 (Bammann Plasticity Model)	169
Material Type 39 (Sandia Damage Model)	173
Material Type 40 (Fahrenthold Brittle Damage)	178
Material Type 41 (Fabric with Damage)	180
Equation-of-State Form 1 (Linear Polynomial)	185
Equation-of-State Form 2 (JWL)	186
Equation-of-State Form 3 (Sack)	187
Equation-of-State Form 4 (Gruneisen)	188
Equation-of-State Form 5 (Ratio of Polynomials)	189
Equation-of-State Form 6 (Linear Polynomial with Energy Deposition)	191
Equation-of-State Form 7 (Ignition and Growth of Reaction in HE)	192
Equation-of-State Form 8 (Tabulated with Compaction)	194
Equation-of-State Form 9 (Tabulated)	197
Equation-of-State Form 10 (Not Available)	199
Equation-of-State Form 11 (Pore Collapse)	200
Cross Section Property Definition for Structural Elements	203
Truss Element Cross Section Properties	205

TABLE OF CONTENTS

DYNA3D User Manual

-

	Hughes-Liu Beam Element Cross Section Properties		
	Hughes-Liu Beam Cross Section Property Notes	207	
	Belytschko-Schwer Beam Element Cross Section Properties	209	
	Belytschko-Schwer Beam Cross Section Property Notes	209	
	Shell Element Cross Section Properties	211	
	Shell Element Cross Section Property Notes	212	
	Thick Shell Element Cross Section Properties	213	
	Thick Shell Element Cross Section Property Notes	213	
4.3	HUGHES-LIU BEAM USER DEFINED INTEGRATION RULES	214	
4.4	SHELL USER DEFINED INTEGRATION RULES	215	
	Shell User-Defined Integration Rule Notes	216	
4.5	NODES	217	
4.6	SOLID ELEMENTS	219	
4.7	BEAM AND TRUSS ELEMENTS	221	
	Hughes-Liu Beam Element	221	
	Belytschko-Schwer Beam Element	221	
	Truss Element	222	
4.8	SHELL AND MEMBRANE ELEMENTS	224	
4.9	THICK SHELL ELEMENTS	226	
4.10	INTERFACE SAVE SEGMENT DEFINITION	228	
4.11	NODAL SINGLE POINT CONSTRAINTS	230	
4.12	SLIDING BOUNDARY PLANES	232	
4.13	SYMMETRY PLANES WITH FAILURE	233	
4.14	NODE TIME HISTORY BLOCKS	236	
4.15	ELEMENT TIME HISTORY BLOCKS	237	
4.16	GRAVITY STRESS INITIALIZATION	239	
4.17	BRODE FUNCTIONS	241	
4.18	CROSS SECTION DEFINITIONS FOR FORCE OUTPUT	242	
4.19	LOAD CURVES	245	
4.20	NODAL FORCES AND FOLLOWER FORCES	247	
4.21	PRESSURE LOADS	249	
4.22	PRESCRIBED VELOCITIES AND ACCELERATIONS	251	
4.23	RIGID WALLS	253	

н с,

()

(

	4.24	NODAL CONSTRAINTS	257
	4.25	INITIAL CONDITIONS	259
	4.26	SLIDING INTERFACE DEFINITIONS	261
	4.27	TIE-BREAKING SHELL SLIDELINES	274
	4.28	TIED NODE SETS WITH FAILURE	276
	4.29	RIGID BODY MERGES	278
	4.30	EXTRA NODES FOR RIGID BODIES	279
	4.31	RIGID BODY JOINTS	280
	4.32	PRESCRIBED BASE ACCELERATIONS	282
	4.33	PRESCRIBED ANGULAR VELOCITIES	283
	4.34	MOMENTUM DEPOSITION IN SOLID ELEMENTS	285
	4.35	DETONATION POINTS	286
	4.36	SHELL-SOLID INTERFACES	287
	4.37	DISCRETE SPRINGS, DAMPERS, AND MASSES	289
	4.38	RIGID BODY INERTIAL PROPERTIES	295
	4.39	NONREFLECTING BOUNDARY SEGMENTS	297
	4.40	TEMPERATURE INPUT OPTION I	298
	4.41	TEMPERATURE INPUT OPTION II	299
	4.42	ONE DIMENSIONAL SLIDELINES	300
	4.43	GENERALIZED RAYLEIGH DAMPING	303
	4.44	MATERIAL INITIALIZATION FOR ROTATIONAL MOTION	304
	4.45	BODY FORCE LOADS BY MATERIAL	305
	4.46	CVS (MADYMO/ATB) COUPLING DATA	308
5.0	REST	ART INPUT FORMAT	309
	5.1	TITLE CARD	310
	5.2	RESTART CONTROL	311
	5.3	DELETED SLIDING INTERFACES	313
	5.4	DELETED MATERIALS	314
	5.5	DELETED SOLID ELEMENT BLOCKS	315
	5.6	DELETED BEAM ELEMENT BLOCKS	316
	5.7	DELETED SHELL ELEMENT BLOCKS	317
	5.8	DELETED THICK SHELL ELEMENT BLOCKS	318

TABLE OF CONTENTS

DYNA3D User Manual

۳_

	5.9	CHANGED BOUNDARY CONDITIONS	319
6.0	MATI	ERIAL MODEL DRIVER	321
	6.1	OVERVIEW	321
	6.2	INPUT DEFINITION	321
	6.3	INTERACTIVE COMMANDS	324
7.0	EXAN	1PLES	327
ACKNOWLEDGEMENTS		348	
REFERENCES			349

÷

-

Preface

DYNA3D has seen wide application to a variety of problems over the past fourteen years. As a public code, DYNA3D has been applied to transient dynamic problems from crash dynamics to human artery simulations by analysts both at LLNL and elsewhere. DYNA3D was originated by Dr. John O. Hallquist of the Methods Development Group at LLNL. During the period 1984-1987, he was joined by Dr. David J. Benson, who is now on the Engineering faculty at the University of California, San Diego. Dr. Hallquist continued as Lead Developer on DYNA3D until 1988, when he left LLNL to pursue a career in private business. During his time at the Laboratory, Dr. Hallquist made innumerable contributions to the field of computational mechanics; many of them by demonstration. It is with great respect for these scientific accomplishments embodied in DYNA3D that the authors continue the expansion of DYNA3D's capabilities to meet new challenges as they arise.

DYNA3D USER'S MANUAL

ABSTRACT

This report is the User Manual for the 1993 version of DYNA3D, and also serves as a User Guide. DYNA3D is a nonlinear, explicit, finite element code for analyzing the transient dynamic response of three-dimensional solids and structures. The code is fully vectorized and is available on several computer platforms. DYNA3D includes solid, shell, beam, and truss elements to allow maximum flexibility in modeling physical problems. Many material models are available to represent a wide range of material behavior, including elasticity, plasticity, composites, thermal effects, and rate dependence. In addition, DYNA3D has a sophisticated contact interface capability, including frictional sliding and single surface contact. Rigid materials provide added modeling flexibility. A material model driver with interactive graphics display is incorporated into DYNA3D to permit accurate modeling of complex material response based on experimental data. Along with the DYNA3D Example Problem Manual, this document provides the information necessary to apply DYNA3D to solve a wide range of engineering analysis problems.

1.0 INTRODUCTION

DYNA3D is an explicit finite element code for analyzing the transient dynamic response of threedimensional solids and structures. The element formulations available include one-dimensional truss and beam elements, two-dimensional quadrilateral and triangular shell elements, and threedimensional continuum elements. Many material models are available to represent a wide range of material behavior, including elasticity, plasticity, composites, thermal effects, and rate dependence. In addition, DYNA3D has a sophisticated contact interface capability, including frictional sliding and single surface contact, to handle arbitrary mechanical interactions between independent bodies or between two portions of one body. Also, all element types support rigid materials for modeling rigid body dynamics or for accurately representing the geometry and mass distribution of a complex body at minimum cost. A material model driver with interactive graphics display is integrated into DYNA3D to allow computation of the stress response to any prescribed strain history without inertial effects. This feature allows accurate assessment of the representation of complex material behavior by the numerical constitutive model in DYNA3D.

Over the last twelve years, DYNA3D has been used extensively at LLNL and in industry. It has been applied to a wide spectrum of problems, many involving large inelastic deformations and contact. The code has evolved rapidly to meet changing engineering analysis requirements and to fully exploit current technology in computing hardware. Algorithms have been optimized, and nearly all of the code is now vectorized. Versions of DYNA3D are now available for several computing platforms, including CRAY/UNICOS, VAX/VMS, SUN, SILICON GRAPHICS, and the IBM RS/6000. The code has been ported to many other machines, and the use of X-Windows graphics allows the SUN version to port easily to other 32-bit UNIX-based machines. The use of a "single-source" development system assures that all new developments appear simultaneously in all supported code versions.

Recent developments in DYNA3D span many areas, including both code capability and user convenience. For modeling rotating machinery problems there is now an improved dynamic relaxation (DR) option for solving static problems, velocity initialization by material for rotational motion on pre- or post-initialization geometry, body force loads by material on deformable or rigid bodies, and improved stress initialization from NIKE3D with exact mass conservation. For simulating penetration and material failure a new type of contact, Slidesurfaces with Adaptive New Definitions (SAND), has been developed and implemented for a a wide range of material models. This new capability is functionally similar to "erosion" algorithms but allows arbitrary combina-

INTRODUCTION

DYNA3D User Manual

tions of shell and solid elements in the failing part of the model. Failure criteria may be based on material variables or on time-step decay of an element. Other contact developments include improved robustness for the single-surface option, significant enhancements to the tie-breaking shell slideline capability, and an option to enforce the contact constraint using a Lagrangian projection rather than a penalty method. A new fully automatic contact formulation has also been developed which completely eliminates the need for user specification of contact surfaces. New material models which are now available include a brittle damage model, a three-invariant viscoplastic cap model, the Bammann plasticity model, and the Sandia damage model. In addition, substantial enhancements have been made to the composite material models (types 22 and 23), the general anisotropic plasticity model (type 33), and the forming limit diagram model (type 35). Beam element performance has been improved for elastic-plastic materials, body force loads may now be applied to rigid bodies on either a global or a material-by-material basis, and the nonreflecting boundary has been improved to work with a wider range of materials. A link between DYNA3D and the crash victim simulation (CVS) codes ATB and MADYMO has been implemented in this release. New user-convenience features include improved performance of the double-precision version, a greatly-improved option for calculation and output of forces across slidesurfaces. In addition, output of reaction forces at nodes with prescribed velocities and restart options to change any type of nodal boundary condition applied to either deformable or rigid bodies have been recently added. The material model driver now supports all shell materials, and the CPU time breakdown in the printed output file has been enhanced. In general, the code's reliability and crash resistance has been significantly improved in this version, along with better error diagnostics and continuing mprovements to input checking and overall code speed. Collectively, these developments should enlarge the class of problems which can be solved, and continue to improve the accuracy, efficiency, and user convenience of DYNA3D.

DYNA3D is part of a set of public codes developed in the Methods Development Group at LLNL. Other analysis codes include the three-dimensional implicit NIKE3D code (Maker, Ferencz, and Hallquist, 1991), and the two dimensional codes NIKE2D (Engelmann and Hallquist, 1990) for implicit analysis and DYNA2D (Engelmann, Whirley, and Hallquist, 1992) for explicit analysis. TOPAZ2D (Shapiro and Edwards, 1990) and TOPAZ3D (Shapiro, 1985) are finite element codes for nonlinear heat transfer and field problem analysis. PALM2D (Engelmann, Whirley, and Shapiro, 1990) is a rapidly developing code for fully coupled thermomechanical analysis. Interactive graphics pre-processors and postprocessors include MAZE (Hallquist, 1983) and ORION (Hallquist and Levatin, 1985) for the two-dimensional codes and INGRID (Christon, Dovey, and

DYNA3D User Manual

Hallquist, 1992), TAURUS (Spelce and Hallquist, 1991), and GRIZ (Dovey and Spelce, 1993) for the three-dimensional codes. All plotting (except in GRIZ) is accomplished using the DIGLIB public domain graphics library developed by Hal Brand at LLNL.

As a public code, the use of DYNA3D by outside firms has been widespread, and this has played an important role in its development. Many code shortcomings have been discovered and remedied as a direct result of dialog with outside users in industry. In addition, many capabilities have been suggested or inspired by feedback obtained from Collaborators outside LLNL. It is hoped that the DYNA3D user community will continue to expand and provide feedback to the author at LLNL, and that improvements made by others will be made available for possible incorporation into future versions of DYNA3D. This active participation provides important information for future development directions of DYNA3D.

INTRODUCTION

.s. . .

· · · · ·

N. .

.

: • .

-

DYNA3D User Manual

2

~_

1.11

2 . .

•,`

DYNA3D is an explicit, nonlinear, finite element code for the transient dynamic response of threedimensional solids and structures. As an explicit code, DYNA3D is appropriate for problems where high rate dynamics or stress wave propagation effects are important. For static and low rate dynamic problems, the implicit NIKE3D code (Maker, Ferencz, and Hallquist, 1991) may be more suitable. DYNA3D may be applied to quasistatic problems by either using the dynamic relaxation option or by simply applying the external loads slowly and integrating the dynamics equations until all significant transients have died out. In contrast to NIKE3D, DYNA3D uses a large number of relatively small time steps, with the solution being explicit (and inexpensive) at each step. Thus, DYNA3D does not form and solve the large matrix equation typical of implicit codes such as NIKE3D, and does not require iteration at each time step. This often leads DYNA3D to be compute-bound, with modest memory requirements, whereas NIKE3D is often memory or I/O bound due to the assembly of a large stiffness matrix at each time step.

The algorithms and architecture are designed for speed and robustness. Nearly all of the code is vectorized for optimal performance on vector computers such as the CRAY. DYNA3D has element formulations that include:

- 2-node truss elements
- 2-node integrated beam elements
- 2-node resultant beam elements
- 3-node triangular shell elements
- 4-node quadrilateral shell elements
- 8-node quadrilateral thick shell elements
- 8-node continuum elements.

These element formulations all handle geometric nonlinearities and do not lock for incompressible materials.

DYNA3D has material models that include:

- elasticity and plasticity (isotropic and anisotropic)
- finite elasticity
- volumetric compaction
- rate dependence
- thermal effects
- damage and failure of ductile and brittle solids.

To model nonlinear pressure-volume behavior, DYNA3D has equations of state that include:

1997 - NA

- polynomial functions
- high explosive models
- tabulated functions.

A variety of boundary conditions are available, including:

- prescribed velocities
- non-reflecting (transmitting) boundaries
- sliding boundaries along arbitrary planes
- symmetry planes with failure.

Methods of prescribing loads include:

- nodal forces and moments
- follower forces
- surface pressure loads
- body force loads
- loads due to thermal expansion
- loads arising from momentum deposition
- Brode function airblast loads.

DYNA3D has a general interface contact capability which includes:

- frictional sliding
- single surface contact
- nodes impacting on a surface
- tied interfaces
- one-dimensional slidelines
- rigid walls
- material failure along interfaces
- penalty and Lagrangian projection options for constraint enforcement
- fully automatic contact.

The constraint modeling capabilities include:

- single point constraints
- arbitrary nodal constraints.

Several methods for modeling failure are available, including:

- reduction of material stiffness and/or element stresses to zero after reaching a criterion
- element deletion for solid or shell elements based on a failure criterion
- SAND adaptive contact for penetration or material failure on contact surfaces
- tie-breaking shell slidelines to allow the mesh to split along predefined lines
- tied node sets with failure to allow the mesh to split along arbitrary element boundaries.

DYNA3D can be interfaced to the Crash Victim Simulation codes MADYMO and ATB via:

- a rigid body motion link file for uncoupled analysis
- a PVM-based data exchange for fully coupled analysis.

DYNA3D has the ability to model rigid body dynamics using features such as:

- rigid materials for trusses, beams, shells, and solids
- general rigid body joint definitions between rigid bodies
- inertial property specification by rigid body
- automatic calculation of rigid body inertial properties
- merging multiple rigid materials to form one rigid body
- nonlinear springs and dampers to connect rigid or deformable bodies.

The analysis capabilities of DYNA3D include:

- transient dynamic analysis
- static analysis using dynamic relaxation
- dynamic analysis with static initialization from a NIKE3D implicit analysis.

The code may be restarted with a variety of modifications to the analysis, including:

- changes in termination time
- deletion of portions of the model by element or by material
- deletion of sliding interfaces
- modification of boundary conditions on deformable or rigid materials.

There are no inherent limits on the size of a DYNA3D analysis model, and storage allocation is dynamic within the code. Problem size is constrained only by the memory available on the computer. Current generation supercomputers have solved DYNA3D problems with more than 500,000 elements, and computing capabilities continue to expand as new generations of hardware become available. Upcoming massively parallel processing (MPP) versions of DYNA3D may enlarge these capacities by an order of magnitude on appropriate hardware.

The following sections briefly discuss each of these capabilities and its applicability to engineering analysis problems.

2.1 NOMENCLATURE AND NOTATIONAL CONVENTIONS

This section will briefly describe the notational conventions used in the remainder of this manual. Vectors and tensors are denoted by boldface type (v, s), or in components by subscripted italic characters (v_i, s_{ij}) , and matrices are generally indicated by boldface capital letters (**M**, **K**). The distinction between vectors and matrices or tensors will be clear from the context in which the symbol is used.

A set of typeface conventions is followed throughout this manual to allow the reader to easily distinguish between **commands**, *parameters*, and computer generated text. **Commands** appear in bold type, and should be entered verbatim. *Parameters* appear in italics, and should be given values when included in the input. Computer generated text, such as error messages or default file names, is printed in a typewriter-like font.

In the descriptions of input format, "column" numbers refer to character positions in the ASCII input file, "field" refers to a group of character positions which contains one item of data, and "format" specifiers suggest the FORTRAN format edit descriptor used to read that item of data. This form of input description is historical in nature, and evolves from the days when paper punch cards where the predominant medium for data input to the computer. Although those days have long past, this notation and general structure of the input definition has proved convenient and are therefore retained. Further, as modern model generation software progresses, the need for the analyst to directly interact with the DYNA3D input file diminishes. Currently, the most common reason for directly editing this file is to adjust an analysis control parameter, and these quantities are easily and quickly located in the current style of input.

In the input definitions described throughout the remainder of this manual, column numbers are sometimes followed by another set of column numbers enclosed in parenthesis: this second set of column numbers applies to input files using the "large" input option. This feature allows the use of node numbers greater than 99,999, and is the default format for DYNA3D input files generated using the INGRID mesh generator.

2.2 UNITS

There is no unit system embedded in DYNA3D. Problems may be defined in any convenient, consistent set of units. The units must be consistent in that mathematical operations directly yield the correct units for the result quantity; no units conversion is done internally in the code. For example, Newton's law states that force equals mass times acceleration, so when mass (in the chosen units) is multiplied by acceleration (derived from the chosen units), the resulting quantity must be force in the chosen units. For example, if the force unit is lb_f , the length unit is *in*, and the

time unit is s, then the units of acceleration are $\frac{in}{s^2}$ and the mass unit is $\frac{lb_f \cdot s^2}{in}$. This approach allows complete freedom in the choice of a units system in which to describe a problem.

2.3 GENERAL FORMULATION

DYNA3D is based on a finite element discretization of the three spatial dimensions and a finite difference discretization of time. The explicit central difference method is used to integrate the equations of motion in time. The central difference method is conditionally stable, and stability is governed by the Courant limit on the time step Δt . For solid elements, this limit is essentially the time required for an elastic stress wave to propagate across the shortest dimension of the smallest element in the mesh. Equivalently, this maximum time step may be related to the period of the highest free vibration mode of the finite element mesh. DYNA3D automatically calculates the maximum time step size at each step of the solution, and adjusts the time step accordingly to minimize the number of time steps used in a solution. This feature minimizes the cost of the analysis while assuring that stability is maintained. Time step considerations suggest the use of structural elements (beams and shells) rather than solid elements for modeling structures that are "thin" in some dimension, since this thin direction has been analytically incorporated into the element formulation. The time step for the structural element model is therefore many times larger than it would be for a corresponding solid element model of the same structure.

DYNA3D uses a lumped mass formulation for efficiency. This produces a diagonal mass matrix M, which renders the solution of the momentum equation

$$Ma_{n+1} = \mathbf{f}^{ext} - \mathbf{f}^{int} \tag{1}$$

trivial at each step in that no simultaneous system of equations must be solved. In the above equation, f^{ext} are the applied external forces, and f^{int} are the element internal forces. The new accelerations a_{n+1} are easily found, from which the updated velocity and coordinates are calculated using the central difference integration formulas. Additional details on the general formulation of DYNA3D are given in (Whirley and Engelmann, 1991a).

2.4 MATERIAL MODELS

DYNA3D includes a large number of material models to represent a wide range of physical behavior. Some models may be used with an equation-of-state to represent complex pressure-volume-energy behavior in a material; these are termed hydrodynamic models. The material models presently implemented are:

- elastic
- orthotropic elastic
- kinematic/isotropic plasticity
- thermo-elastic-plastic
- soil and crushable foam
- viscoelastic
- Blatz-Ko hyperelastic rubber
- high explosive burn
- fluid
- isotropic-elastic-plastic hydrodynamic
- Steinberg-Guinan high rate elastic-plastic
- isotropic elastic-plastic
- elastic-plastic with failure
- soil and crushable foam with failure
- Johnson/Cook elastic-plastic
- concrete/geological material
- elastic-plastic with oriented cracks
- power law isotropic elastic-plastic
- strain rate dependent isotropic elastic-plastic
- rigid
- thermal orthotropic elastic
- fiber composite with damage
- thermal orthotropic elastic with variable properties

- rate-dependent tabular isotropic elastic-plastic
- extended two invariant geologic cap
- metallic honeycomb
- compressible Mooney-Rivlin hyperelastic rubber
- resultant plasticity
- closed form update elastic-plastic for shells
- Frazer-Nash hyperelastic rubber
- Ramberg-Osgood elastic-plastic
- general orthotropic elastic-plastic
- normal anisotropic elastic-plastic for shells
- elastic-plastic with forming limit diagram
- brittle damage
- three-invariant viscoplastic cap
- Bammann plasticity
- Sandia damage
- Fahrenthold brittle damage
- fabric

2.5 EQUATIONS OF STATE

A hydrodynamic material model requires an equation-of-state to define the pressure-volume relationship. The equations of state available in this release are:

- linear polynomial
- JWL high explosive
- Sack high explosive
- Gruneisen
- ratio of polynomials
- linear polynomial with energy deposition,
- ignition and growth of reaction in HE
- tabulated compaction
- tabulated
- pore collapse

2.6 FINITE ELEMENT FORMULATION

The finite element formulations used in DYNA3D have been chosen for their accuracy, speed and robustness in large deformation nonlinear problems. In an explicit analysis code, there are many small time steps, so it is important that the number of operations performed at each time step be minimized. This consideration has largely motivated the use of elements with one-point Gauss quadrature for the element integration. This approach gives rise to spurious zero energy deformation modes, or "hourglass modes," within the element. The element must then be stabilized to eliminate the spurious modes while retaining legitimate deformation modes. This stabilization is effectively accomplished in DYNA3D. A selection of stabilization methods is available to handle almost any situation, and the default algorithms have proven effective for most applications. All element operations are fully vectorized, including the assembly of the element force vector (as described in Ginsberg and Johnson, 1989), for optimal performance on vector computers.

The basic continuum finite element in DYNA3D is the eight-node "brick" solid element. This element optionally uses either one-point integration (default) or the constant stress formulation of (Flanagan and Belytschko, 1981) with exact volume integration. Spurious hourglass modes are stabilized using an "hourglass viscosity" or an "hourglass stiffness." Displacements within the element are interpolated using trilinear interpolation functions, and the constitutive equations are evaluated once based on the state at the center of the element. This element is valid for large displacements and large strains. The element may be degenerated to a wedge or tetrahedral element, but at the expense of accuracy. Thus, these degenerated elements should be avoided whenever possible.

DYNA3D fully supports truss, beam, and shell structural elements. For each of these element types, there are several elements available based on different formulations. There is usually no best element formulation for all problems. Some element formulations perform better on certain problem classes, while in other cases the distinguishing feature may be execution speed/cost. Element formulations may be mixed in a single model. The default shell element formulation may be specified in the master control section of the DYNA3D input. Any beam or shell element formulation may lation may be specified for a specific material on the Material Control Card, and this specification overrides any defaults.

The eight-node thick shell element is an experimental capability, and should be used cautiously. It is integrated using one point in the center for the in-plane contribution, and using an arbitrary number of through-thickness integration points. This multi-point integration through the thickness,

with one constitutive evaluation at each point, permits the element to capture complex stress states associated with bending deformations. Hourglass modes are controlled using techniques similar to those for the solid element.

This release of DYNA3D supports three quadrilateral 4-node shell elements, a quadrilateral 4-node membrane element, two triangular 3-node shell elements, two 2-node beam elements, and a 2-node truss element. The shell elements use three-dimensional plane stress constitutive models where the stress component normal to the shell midsurface is zero. One constitutive evaluation is made for each integration point through the shell thickness. All shell elements use one integration point in the center for the in-plane integration, and a pre-defined or user-defined integration rule for the numerical integration through the thickness.

The three quadrilateral shell element formulations are the Hughes-Liu theory, the Belytschko-Tsay theory, and the YASE theory. The Hughes-Liu formulation is based on degenerate shell theory, as described in (Hughes and Liu, 1981a), (Hughes and Liu, 1981b) and (Hughes, Liu and Levit, 1981). The DYNA3D implementation of the Hughes-Liu shell element is described in (Hallquist, Benson and Goudreau, 1985) and (Hallquist and Benson, 1986), and uses hourglass control procedures based on those reported in (Belytschko and Tsay, 1984). The Hughes-Liu shell uses normal vectors defined at each node for the element calculations. In addition, the Hughes-Liu shell has a user-defined reference surface, which may be specified as the shell midsurface, or the upper or lower surface. This feature is often useful when modeling stiffened shells or shell-beam intersections. The Hughes-Liu shell is the most expensive of the DYNA3D shell elements. The Belytschko-Tsay element formulation is based on resultant shell theory, and is described in (Belytschko and Tsay, 1981), (Belytschko and Tsay, 1983) and (Belytschko and Tsay, 1984). This is currently the default shell element in DYNA3D. The YASE shell element is a new element based on resultant shell theory and a 3-field variational principle, and is described in (Engelmann and Whirley, 1990) and (Engelmann and Whirley, 1991). Additional details of the formulation are given in (Engelmann, Whirley, and Goudreau, 1989). Both the Belytschko-Tsay and YASE element formulations use element average normals for all computations. Also, the reference surface must always be the midsurface for these formulations. The Belytschko-Tsay and YASE elements differ substantially in the treatment of the higher order variations in the stress field. The YASE shell is somewhat more accurate on some problems and is more resistant to hourglassing, while the Belytschko-Tsay element is slightly faster. The membrane element is a specialization of the Belytschko-Tsay shell element which has only in-plane stiffness, and is slightly faster than the other shell elements.

DYNA3D User Manual

The two triangular shell elements implemented in this release of DYNA3D are denoted the C^0 element and the BCIZ element. The C^0 triangular shell is based on the work of Belytschko et. al., and is described in (Belytschko and Marchertas, 1974), and (Belytschko, Stolarski and Carpenter, 1984). The BCIZ triangular shell element is described in (Bazeley, Cheung, Irons, and Zienkiewicz, 1965). Although it is possible to collapse a quadrilateral shell element into a triangle, elements in this configuration lock due to excessive transverse shear, and therefore this procedure is strongly discouraged.

The two beam element formulations now available in DYNA3D are the Hughes-Liu beam and the Belytschko-Schwer beam. The Hughes-Liu beam is an extension of the ideas in the Hughes-Liu shell to produce a degenerated beam element. This element uses nodal fiber vectors for orientation, and has a reference surface which may be defined at the midplane, or on the upper or lower beam surface. This feature is frequently useful when modeling stiffened shells or beam-shell intersections. One point integration is used along the length, and a choice of pre-defined or user-defined integration rules is used for numerical integration over the cross section. When used with a userdefined integration rule, this element can model almost any cross section geometry. The Belytschko-Schwer beam is based on a resultant beam theory, and is described in (Belytschko, Schwer, and Klein, 1977). The Belytschko-Schwer beam formulation is based on hermite cubic shape functions with one integration point along the length. No numerical integration is performed over the cross section; constitutive models for this element are formulated directly in terms of forces and moments. One set of element-average orientation vectors is computed and used for all element calculations. The reference surface for the Belytschko-Schwer beam is always the midsurface. The Hughes-Liu beam element is considerably more expensive than the Belytschko-Schwer beam, but may be more accurate for some problems, especially if the cross section is complex in shape.

The DYNA3D truss element is adapted from the Belytschko-Schwer beam element, and is slightly less expensive. The truss element has only axial stiffness, and may be used for large-displacement problems.

2.7 GENERALIZED RAYLEIGH DAMPING

A global damping capability is implemented in DYNA3D based on concepts from Rayleigh damping in linear analysis. In traditional Rayleigh damping, a global damping matrix C is constructed as a linear combination of the mass matrix M and stiffness matrix K as

 $\mathbf{C} = \alpha \mathbf{M} + \beta \mathbf{K},$

(2)

where α and β are damping coefficients. This approach can be used to introduce a specified fraction of critical damping at two frequencies, since two parameters are available.

The Rayleigh damping concept is extended to apply to nonlinear analysis with large displacements and strains and nonlinear behavior in (Whirley and Engelmann, 1992). The above damping characteristics are preserved in the nonlinear implementation, but no global damping matrix is actually formed. When used with nonlinear material models, the stiffness proportional damping is found using the elastic part of the material stiffness, i.e., the stiffness proportional damping contribution is computed as though the material was linearly elastic with the elastic properties of the nonlinear material. This approach prevents the amount of damping from varying with the material behavior (such as plastic yielding), and allows accurate computation of the degree of damping introduced into the model.

2.8 LUMPED PARAMETER ELEMENTS

DYNA3D contains lumped parameter elements such as discrete springs and dampers and lumped nodal masses. This feature allows DYNA3D to be used to solve simple spring-mass models, or to couple these simple models with complex finite element models. Lumped parameter elements may be used to connect any combination of deformable and rigid elements, and are often useful in building rigid body models.

2.9 INITIAL AND BOUNDARY CONDITIONS

A transient dynamic problem requires the specification of initial conditions in order to be completely defined. In DYNA3D, initial conditions are specified as initial velocities. All initial velocities may be set to zero by setting a flag in the input, or the initial velocity of every node or a subset of nodes may be explicitly defined.

The time variation of quantities in DYNA3D is specified by "load curves." An arbitrary number of load curves may be defined, and any number of boundary conditions or loads may reference one load curve. Each load curve may have an arbitrary number of points.

DYNA3D contains a number of options for modeling the wide range of boundary conditions encountered in engineering analysis. Nodes may be constrained from translation or rotation in the global coordinate system. Alternatively, a local coordinate system may be defined and nodes constrained from translation or rotation in this local system using Single Point Constraints. Any

DYNA3D User Manual

number of local coordinate systems may be defined in a model. Nodes may be constrained to move only along an arbitrary but specified line or only on an arbitrarily oriented plane using "Sliding Boundary Plane" definitions. "Symmetry Planes with Failure" may be defined to constrain a group of nodes to move only in a specified plane. In addition, a tensile failure stress σ_f may be defined such that if the tensile stress normal to the plane exceeds σ_f in an element, the nodes of that element are released from the symmetry plane. These released nodes are allowed to move in arbitrary directions, but may not pass through the Symmetry Plane. This feature is useful for modeling some oblique penetration problems.

Nodes may be given prescribed velocities as a function of time in any global coordinate direction, or in an arbitrary direction specified by a given vector. In cases where prescribed velocities at t = 0 are not equal to defined initial velocities, significant dynamic loads are induced in the model due to the incompatibility of boundary and initial conditions.

Nonreflecting boundaries, also called "transmitting boundaries," may be defined to simulate semiinfinite domains and prevent spurious wave reflections due to the finite extent of the mesh. These boundaries are quite effective, but assume essentially linear material behavior. Thus, it is necessary to discretize the domain sufficiently far from the region of interest to contain all significant nonlinear material behavior in the analysis model.

2.10 INITIAL VELOCITIES FOR ROTATING BODIES

The transient dynamic analysis of rotating bodies arises in many areas, including tire and wheel dynamics, turbomachinery, flywheels, and rotating machine tools. For these problems it is often required to initialize velocities for some part of the model to be consistent with rotation about an arbitrary axis. To address this need, DYNA3D allows initial velocities to be computed for a subset of materials composing a model based on a given angular velocity vector through the origin. This feature is activated by specifying the number of materials to receive velocity initialization on Control Card 9 in section 4.1 on page 41, and then giving the angular velocity and list of rotating materials as described in section 4.43 on page 303. If static initialization (via a NIKE3D stress initialization file or via a static dynamic relaxation solution within DYNA3D) has been performed, then the initial velocities computed with this option are evaluated on the statically deformed geometry.

2.11 LOADS

Several types of applied loads may be defined in DYNA3D. Most load definitions include time variation by reference to a load curve, and multiply the load curve value by a scale factor to evaluate the load to be applied. DYNA3D supports surface loads, body force loads, and thermal loads.

Surface loads may be specified as either nodal forces, follower forces, or surface pressures. Nodal forces may be defined in arbitrary directions, and may include both point forces and point moments defined as a function of time. The direction of concentrated nodal loads is fixed throughout the problem and does not evolve with the deformation. Follower forces may be defined which always act normal to a plane defined by three nodes. The direction of this force evolves with the deformation to remain normal to the specified plane throughout the analysis. Surface pressure loads may be defined, and these always act normal to the surface in the current configuration. Pressures may vary linearly over an element surface, and may be an arbitrary function of time. Alternatively, pressures may be defined to follow a Brode airblast time history shifted by a given shock wave arrival time based on the distance from the Brode origin.

Body force loads can be specified to represent physical phenomena such as gravity, electromagnetic forces, or centrifugal loads due to an angular velocity. Body force loads may be specified as arising from translational "base acceleration" or from an angular velocity of the model about any fixed axis of rotation. In DYNA3D, body force loads may be specified by a translational base acceleration a, where a has components defined in the global x, y, and z directions. The body force density b is found from

$$b = -\rho a, \tag{3}$$

where ρ is the material mass density. In addition, body force loads arising from rotation about an arbitrary axis through the global origin may be defined by specifying the angular velocity components about the global x, y, and z axes. Body force density **b** at a point is then found from the angular velocity vector w and the radius vector r (from the origin to the point) by

$$b = \rho w \times (w \times r) \,. \tag{4}$$

The time variation of angular velocity components ω_i may defined using a load curve, but note that the time derivative of the angular velocity is *not* included in the body force density calculation.

Body force loads can be specified for an entire model, or can be specified by material for a subset of a model. This is particularly useful when computing initial stresses in rotating parts which subsequently interact with nonrotating parts, such as the impact of a stationary workpiece on a spinning machine tool.

Loads due to thermal expansion are applied by specifying temperature changes with material models which include thermal expansion. Nodal temperatures may be read from a TOPAZ3D (Shapiro, 1985) plot database file, or may be individually specified and scaled by a load curve in the DYNA3D input file.

Finally, loads may be defined by directly specifying the momentum transferred into the structure by an external entity. Using the Momentum Deposition option, three components of linear momentum may be added to an individual solid element at a specified time. This capability can be used to represent the physical problem of the impact of many small particles onto a larger body, for example.

2.12 GENERAL INTERFACE CONTACT

DYNA3D contains a robust and efficient capability for modeling the mechanical interaction of two bodies or two parts of a single body. Fourteen different options are offered for defining the behavior of "slide surfaces" in a wide variety of situations. Twelve of these options are actually general slide surface definitions, and the other two are special purpose capabilities. These options include facilities for treating interactions between two surfaces, between discrete nodes and a surface, or between a body and a rigid wall. One option allows material failure along the material surface. All DYNA3D slide surfaces accommodate arbitrarily large relative motions, and slide surfaces may be defined between surfaces of differing element types (e.g., a shell structure may interact with a solid body). Beam element contact on shells or solids is modeled by defining the nodes of the beam as discrete nodes impacting a surface, since the beam element itself is mathematically a line and thus has no area with which to define a surface. Interface pressure distributions on a slidesurface may be written to a special TAURUS database, or interface forces for each node on a slidesurface may be written to the "FORCES" output file, as described in section 2.15 on page 24. The interface forces file also contains the total force across the interface at at given time. There are no limits on the type or total number of slide surfaces defined for a DYNA3D model. In the remainder of this section, first each of the twelve slide surfaces is briefly discussed, and modeling suggestions are given. The two special purpose capabilities, rigid walls and one-dimensional slidelines, are then presented.

The twelve types of slide surfaces supported in this release of DYNA3D are:

- 1. sliding only
- 2. tied
- 3. sliding with separation and friction
- 4. single surface
- 5. discrete nodes impacting surface
- 6. discrete nodes tied to surface
- 7. shell edge tied to shell surface
- 8. node spotwelded to surface
- 9. tied with failure
- 10. one-pass algorithm for sliding with separation and friction
- 11. SAND contact (Slidesurfaces with Adaptive New Definitions)
- 12. automatic contact

Slidesurface types 1 through 11 are "defined-surface" options, wherein the user specifies the nodes or segments on each side of the contact interface. Slidesurface type 12 is automatic contact, wherein DYNA3D automatically identifies surfaces which may potentially come into contact and treats them accordingly. This is a new capability which should progress rapidly in the near future.

A defined-surface sliding interface is defined by specifying one master surface and either a slave surface or a set of slave nodes, as required by the type of slide surface chosen. Slide surfaces are defined by listing groups of triangular or quadrilateral segments, and each segment is defined by three or four corner nodes. These segments are actually element faces which define the surface. Outward normals are internally computed for segments of solid elements, but for segments of shell elements the outward normal is defined by the "right-hand rule" from the list of defining nodes. *Care should be taken in defining slide surfaces on shell elements to ensure that the correct side of the shell element is chosen.* Each slide surface is designated either a "master surface" or a "slave surface." The importance of this distinction is discussed below.

Sliding interfaces and contact in DYNA3D are formulated using a "node on surface" concept, as described in (Hallquist, 1977) and (Hallquist, Goudreau, and Benson, 1985). *Slave* nodes are restricted from penetrating *master* surfaces. Conceptually, the two-surface algorithms could thus be thought of as looping over all slave nodes, and for each slave node checking that there is no penetration through any of the master segments defined for this sliding interface. In the symmetric treatments, the designation of the two surfaces as slave or master are interchanged, and the

algorithm is applied a second time. Thus, each surface goes through the algorithm once as a master surface and once as a slave surface. This symmetric approach has been found to greatly increase the robustness and reliability of slidesurfaces when both surfaces are deformable.

Sec. 1

1.3

The contact formulations used in single surface (type 4) and SAND (type 11) account for the thickness of a shell element in contact calculations. All other contact options operate using the midplane definition of shell elements for the surface definition. Situations arise in modeling complex objects when both capabilities are needed, and the slide surface type can be chosen to obtain the desired behavior.

The "sliding only" (type 1) and "tied" (type 2) options are based on a nodal constraint formulation, and are not symmetric, as described in (Hallquist, 1976) and (Hallquist, 1978). For these types of slide surfaces, it is important that the more coarsely meshed side be defined as the master surface. All other slide surface types may use either a penalty formulation or a Lagrangian projection formulation and are completely symmetric; the distinction between master and slave surfaces for these types is for convenience only.

When the penalty formulation is chosen to enforce the contact constraint, a restoring force is placed on a penetrating node to return it to the surface. This force is proportional to the depth of penetration, the bulk modulus of the penetrated material, the dimensions of the penetrated element, and a scale factor specified by the user. The amount of penetration observed is inversely proportional to the chosen scale factor. Using a larger penalty scale factor reduces the observed penetration, but may make the problem more difficult to solve. Using a smaller penalty scale factor makes the problem easier to solve, but may lead to unacceptably large interpenetrations. One global penalty scale factor is defined in the Control Section of the DYNA3D input file, and another penalty scale factor may be defined separately for each surface of a slide surface pair. The default value of the global penalty scale factor is 0.10, and the default for the local penalty scale factor used for computations.

When the Lagrangian formulation is chosen to enforce the contact constraint, a restoring force is again placed on a penetrating node to return it to the surface, but is computed differently. This option computes the force by determining the predicted location of a node on the interface at the end of the time step, and computing the contact force required to place the node on the surface at the end of the time step. This method yields exact enforcement of the contact constraint for nodeon-node contact, and good approximations for most reasonable mesh refinements. Although this is

a new option in this release of DYNA3D, early use at LLNL has shown considerable promise. This option works particularly well for contact between very thin shells, such as sometimes occurs in sheet forming problems. It addition, it is also advantageous in modeling contact betweem materials of widely varying stiffness.

During the initialization phase of DYNA3D, nodes on a slide surface which are initially interpenetrating are put back on the surface before the dynamic calculation is begun. A message is written to the screen and a summary printout is provided in the "hsp" file when this occurs. Initial interpenetration may result from errors in the definition of the model, or simply from curved parts of different mesh densities placed in close proximity to each other. In the latter case, the maximum distance a node was moved should be a small fraction of the minimum element dimension. An exception to this procedure is that nodes which are part of a NIKE3D static initialization model are not moved during DYNA3D initialization.

The sliding only (type 1) slide surface is a two-surface method based on a nodal constraint formulation. The two surfaces are allowed to slide arbitrarily large distances without friction, but are not permitted to separate or interpenetrate. This option performs well when extremely high interface pressures are present.

The tied (type 2) slide surface is not really a slide surface at all, but is a feature for joining two parts of a mesh with differing mesh refinement. This is also a nodal constraint formulation, so the more coarsely meshed surface should be chosen as the master surface. Tied interfaces may only be defined between surfaces defined from brick elements.

The sliding with separation and friction (type 3) slide surface is the most generally applicable option. This implementation is a penalty formulation, and allows two bodies to be either initially separate or in contact. Large relative motions are permitted, and Coulomb friction is included. Surfaces may separate and come together in a completely arbitrary fashion. Contact between shell surfaces and solid body surfaces is permitted.

The single surface contact (type 4) slide surface is a penalty formulation used for modeling two portions of the same body which may come into contact. This situation often arises in buckling problems, where one surface often develops folds and comes into contact with itself. This option may be used with either solid or shell element surfaces. Frictional sliding is permitted between surfaces in contact. When used with shell element surfaces, it is only necessary to define *one* side of the surface in the slide surface definition; internal logic accounts for the possibility that contact

may occur on either side. Due to the complexity of this algorithm, it is slightly more expensive than the other slide surface options, and therefore should only be used where necessary. The formulation of single surface contact is described in (Benson and Hallquist, 1990).

The **discrete nodes impacting surface** (type 5) slide surface is a penalty formulation which allows separate definition of the slave nodes and master surface. This option is necessarily not symmetric, since no slave segments are defined. This feature can be used to model the impact of beams or shell edges onto a solid or shell surface, for example. The target surface is specified as the master surface, and the nodes of the beams or the shell edge nodes are defined as the slave nodes. Coulomb friction may be defined between the nodes and the master surface.

The discrete nodes tied to surface (type 6) slide surface is analogous to the "discrete nodes impacting surface" option described above except that the slave nodes are required to remain rigidly attached to the master surface at their initial location. This feature is sometimes useful in joining two parts of a difficult mesh.

The shell edge tied to shell surface (type 7) slide surface is another specialization of the tied slide surface idea, although the implementation is somewhat different. This option allows the easy definition of intersections between a shell edge and a shell surface when the mesh spacing prevents merging nodes.

The node spotwelded to surface (type 8) slide surface is a combination of the type 5 and type 6 slide surfaces. Slave nodes are constrained to remain on the master surface until a prescribed failure criterion is satisfied. Once the spotweld has "failed," the node is released and then the slide surface functions as a type 5 slide surface.

The **tied with failure** (type 9) slide surface is an experimental penalty method formulation. This option functions as a tied slide surface until a prescribed failure criterion is satisfied, thereafter the slide surface functions as a type 3 slide surface. This option may only be used between surfaces defined from solid elements.

The one-pass algorithm sliding with separation and friction (type 10) slide surface is an unsymmetric implementation of the algorithm described for type 3 above. This option may effectively be used when one surface is defined as part of a rigid body, as often occurs in metal forming problems. This algorithm is somewhat less expensive than the symmetric type 3 algorithm, but may yield poor results if used between two deformable bodies.

The SAND (type 11) slide surface is a new capability for modeling material failure along interfaces. SAND is an acronym for Slidesurfaces with Adaptive New Definitions. A failure criterion is defined for a volume of material adjacent to a SAND contact surface; these volumes of potentially failing material are called SAND volumes. As material within an element on the contact surface fails, the failed element is removed from the calculation and the slidesurface definition adapts to the new exterior boundary of the unfailed material. SAND contact may include solid elements or shell elements or any combination thereof. A SAND volume may be composed entirely of solid elements or shell elements for simple structures, or may include an external layer of shell elements followed by layers of solid elements and shell elements as needed to model a honeycomb or composite structure. Only the outside surface needs to be included in the initial sliding interface definition. SAND volumes are defined by listing the DYNA3D material numbers (not material types) for all materials which comprise the SAND volume. A SAND contact interface may have a SAND volume on only the master side, on only the slave side, or on both the master and slave sides. This capability allows efficient modeling of problems where failure is expected to occur only on one side and problems where failure may occur on both sides of a sliding interface. SAND contact interfaces allow improved modeling of many penetration and failure problems with DYNA3D. Note: SAND volume materials are currently permitted to be of Material Type 24 for shell elements, or Material Types 10, 11, 13, 24, or 35 for solid elements.

The **automatic** (type 12) slide surface is an experimental capability new in this version of DYNA3D. This feature eliminates the need for the user to define any contact surfaces or segments. DYNA3D internally finds the outer boundary of all bodies in the model, and defines the appropriate contact segments to handle the most general contact conditions, including both multibody and single surface contact. The formulation of automatic contact is described in (Whirley and Engelmann, 1993) and (Whirley and Engelmann, 1993a).

"Rigid walls," also known as "stonewalls," may be defined to represent a rigid planar surface. This surface may be stationary, or may be given a mass and a specified initial velocity. Rigid walls permit modeling unilateral contact without requiring discretization of the target surface, and thus are considerably less expensive than defining a type 3 slide surface.

Finally, "one-dimensional slidelines" allow truss or beam elements to slide along predefined lines of nodes. This capability is sometimes useful to represent the pullout of reinforcing bars from a concrete continuum model, for example. Friction coefficients may be specified to incorporate the effect of the stress state in the surrounding material on the required pullout force.

2.13 CONSTRAINT EQUATIONS

Two types of constraint equations may be explicitly defined in DYNA3D: nodal single point constraints, and nodal constraint equations. Nodal single point constraints restrict the translational or rotational motion of a specified node in a specified direction. These constraints may be specified using an arbitrary number of local coordinate system definitions, and many nodes may reference one local coordinate system. General translational or rotational constraints may be defined using this option. Nodal single point constraints enforce the constraint exactly, as do global constraints. This approach is slightly more expensive than constraints defined with respect to global axes, so global constraints should be used whenever possible. Nodal constraint equations simply specify that two or more nodes share a global degree-of-freedom.

4 J.

2.14 CROSS SECTION FORCES

DYNA3D allows a cross section to be defined through a structural model, and at selected intervals will output the total resultant force on this cross section. This feature is often used to quickly evaluate a global response quantity, such as the total force transmitted from one part of a model to another.

2.15 TOTAL INTERFACE FORCES AND REACTION FORCES

The total force across a slidesurface as a function of time can be optionally output from DYNA3D into either an ASCII print file and/or a binary plot file which can be read by TAURUS or other postprocessing software. This interface force information is available for all types of slidesurfaces, including tied slidesurfaces. Thus, it is possible to easily obtain the total cross-section force in a solid element model by simply defining a tied interface and examining the interface force on this surface. Details on this capability are described in section 3.6 on page 39. Interface force output has proven quite useful in many types of engineering analysis, and has been significantly enhanced in this version of DYNA3D.

Reaction forces at each prescribed velocity degree-of-freedom may be output at the state plot interval into a file FORRCT. This feature is activated by an option in the control section.

2.16 HIGH EXPLOSIVE PROGRAMMED BURN MODELS

DYNA3D contains several options for simulating the initiation and burning of high explosives. Beta burn defines a compression criteria for the ignition of HE. Programmed burn options allow the specification of lighting times or detonation points for problems where these are known. An HE ignition and growth equation-of-state allows the study of impact or temperature initiation of energetic materials, and is useful when the existence and location of detonation points must be determined by the solution of the boundary value problem. These features have proved useful on a wide range of problems from explosive metal forming to enhanced oil recovery. Each of the DYNA3D burn options is described below.

There are two options for determining the lighting time of HE materials, including "beta burn" and programmed burn. The beta burn option does not use the concept of a lighting time, and computes a burn fraction F directly from

$$F = \beta (1 - V) = \frac{1 - V}{1 - V_{CI}},$$
(5)

where V is relative volume and V_{CJ} is the Chapman-Jouguet relative volume. The programmed burn option determines an element lighting time based on specified detonation point locations. Single or multiple detonation points may be defined, and the lighting time of an element is computed based on the distance from the center of the element to the nearest detonation point. These burn options allow considerable flexibility in defining the detonation geometry for high explosive problems.

DYNA3D contains an equations-of-state for modeling the ignition and growth of detonation waves in high explosive materials. Equation-of-state 7, developed by Tarver and coworkers, models the behavior of single component explosives. This equation-of-state allows an analysis to determine whether an explosive will ignite and burn due to pressures and temperatures present in the solution of the boundary value problem. This model is quite complex, and care is urged in their use.

2.17 RIGID BODY DYNAMICS

DYNA3D has the ability to do limited rigid body dynamics modeling. The rigid body formulation is described in (Benson and Hallquist, 1986). Rigid body nodal lumped masses, springs, and dashpots are available. Rigid materials can be defined for all element types. This rigid body feature
OVERVIEW OF DYNA3D

2

provides an inexpensive method for modeling portions of a structure that are much stiffer than the regions of interest. This feature is also convenient in many metal forming problems, where the punch and die are often effectively rigid.

Two separate rigid bodies may not share any nodes in common. In other words, no one node may be used in the definition of two elements each of which use a different material if both materials are defined as rigid. In this case, the two rigid materials must be merged to form one rigid body using the Rigid Body Merge feature. Alternatively, Rigid Body Joints may be defined which couple the two rigid bodies leaving specified degrees of freedom unconstrained.

Inertial properties for rigid materials may be defined in one of two ways. By default, the inertial properties are calculated from the geometry of the elements of the rigid material and the density specified in the material properties definition. Alternatively, the inertial properties and initial velocity for a rigid body may be directly specified, and this overrides data calculated from the material property definition and nodal initial velocity definitions.

Analysis results for rigid bodies may be post-processed in the usual way. Displacements, velocities, and accelerations are stored in the plot database for each node of a rigid body. It is not meaningful to plot stresses for elements defining a rigid body, however.

2.18 ANALYSIS OPTIONS

DYNA3D may be used for transient dynamic analysis, or for quasistatic analysis, or for transient dynamic analysis with a quasistatic initialization from NIKE3D. A transient dynamic analysis, using the central difference method, is the default option. A quasistatic analysis may be performed using the dynamic relaxation option. Although this capability has proven its value in production calculations, caution should be exercised in its use since monotonic convergence is not guaranteed. This solution procedure may introduce errors into the solution *if* history dependent materials are present in the model. Alternatively, a quasistatic analysis may be performed by simply applying the loads very slowly and using the transient dynamic analysis option. If the analysis is run until all significant transient oscillations die out, then the remaining solution is the quasistatic solution. This procedure may confidently be used with history dependent materials if dynamic oscillations are kept to a minimum (i.e., loads are applied sufficiently slowly). Finally, a transient dynamic analysis may be performed using stresses and geometry initialized from a NIKE3D analysis.

DYNA3D User Manual

2.19 STATIC INITIALIZATION FROM NIKE3D

The NIKE3D static initialization option provides a versatile capability for incorporating initial stress and deformation effects in a transient dynamic analysis. Typical applications requiring this capability include failure analysis of rotating disks and transient dynamic analysis of internally pressurized thin-wall vessels. Rotating disks often build up large stresses before failure occurs due to centrifugal forces, and these initial stress fields can influence the post-failure dynamic interactions of the fragments with surrounding structures. Internally pressurized thin-wall tanks experience considerable pressure-stiffening, and therefore it is important to have the correct quasistatic equilibrium stresses in the wall material before initiating any studies of transient dynamic events, such as impact or fracture.

To perform a DYNA3D analysis using static initialization from a NIKE3D analysis, the first step is to run the NIKE3D analysis and request the output of a stress initialization file. This file is then specified on the DYNA3D input line, as described in section 3.2 on page 33. The DYNA3D model may be larger than the NIKE3D model, but the part of the model in common must be the lowest numbered nodes and elements. The portion of the model to be initialized must have the same node numbers and element connectivities in both the NIKE3D and DYNA3D models. In addition, multiple shell element formulations may *not* be mixed in a DYNA3D model when static initialization is used. If shell elements are used, the same element formulation and integration rule must be used in both the NIKE3D and DYNA3D models.

The ability to enlarge the DYNA3D model above that used for the NIKE3D analysis is quite useful if only a small part of the dynamic model needs to be initialized, as would be the case with an internally pressurized tank incorporated into a large structure, for example. The stress initialization capability is valid for elastic quasistatic response, or for elastic-plastic quasistatic response if isotropic strain hardening is used.

2.20 DYNAMIC RELAXATION FOR STATIC INITIALIZATION

DYNA3D contains a limited capability for performing quasistatic analysis using a dynamic relaxation algorithm. This feature is primarily intended to be used to generate a static stress solution as an initial condition for a transient dynamic analysis, but it has been applied with some success to the solution of more general static problems.

OVERVIEW OF DYNA3D

DYNA3D User Manual

The dynamic relaxation (DR) method is based on the observation that the long-time limit of a damped dynamic solution is the quasistatic solution. Damping is introduced through a "dynamic relaxation factor" (default value=0.995) which multiplies the velocities computed at each step of a DR solution. This factor can be adjusted by the user if required; increasing the factor decreases the effective damping while decreasing this factor increases the effective damping.

.

During then DR static solution process, "time" is really just a parameter to describe the solution process, and does not correspond to physical time. The current implementation uses a DR time step equal to the standard dynamic time step. Thus, if it is desired to slowly apply the static loads to minimize overshoot in the solution, then a short trial dynamic run can be made to determine the time step size. The static loads to be applied during the DR solution can then be applied over some number of time steps (typically 2000-5000 but problem dependent), and this determines the time points to be used on the load curve controlling the static loads.

The current implementation of DR in DYNA3D is susceptable to dynamic overshoot if static loads are applied too quickly. If only history-independent material models (such as elasticity) are used, then the resulting solution will still be correct and this overshoot behavior is of little consequence. If history-dependent material models (such as plasticity) are used, however, this dynamic overshoot can cause yielding which is erroneous, and therefore an incorrect static solution is obtained. Thus, the DR static solution capability can be used with confidence for elastic initialization, but must be carefully used with slowly applied loads to prevent overshoot and inaccuracy in history-dependent static problems.

A DR static solution is activated by specifying the "load curve option" for a load curve to control the application of the static loads, as discussed in section 4.19 on page 245. The dynamic relaxation factor, the DR time step, and the DR convergence tolerance can be modified from default values on Control Card 8 as described in section 4.1 on page 41.

2.21 DYNAMIC RELAXATION FOR STRESSES IN ROTATING BODIES

In many applications such as flywheel design, machine tool safety, or turbine engine containment, it is important to solve a transient dynamic problem beginning with a stress state induced by rotational motion. This problem is easily solved in this release of DYNA3D using the dynamic relaxation option for computing the initial stresses. The part of the model which is rotating should be identified, by material, as receiving a body force load due to prescribed angular velocity. These body force loads should reference a load curve which begins at zero and increases to a value

DYNA3D User Manual

commensurate with the rotational velocity, and remains constant at that value out to some large time. This load curve should be marked as active for static initialization only. Initial velocities for these rotating bodies, computed on their post-initialization deformed geometry, may be generated by specifying on Control Card 9 the number of materials to initialize for rotational motion, and then listing these materials as described in section 4.43 on page 303. This approach will allow a smooth transition from the body-force-based calculation of the initial stresses into the transient dynamic phase where the bodies actually rotate in space. During the transient portion of the analysis, the rotating bodies may be allowed to rotate freely, or may have rotational velocities prescribed for them using load curves flagged to be active for "transient dynamic analysis only." Also, other loads may be added to the rotational body during the transient dynamic phase, such as impact with stationary object. The breakaway of part of the rotating body may be easily simulated by various techniques using restart input files to change nodal boundary condition codes or to delete elements which tie the broken fragments to the remainder of the rotating body. This is a new capability for DYNA3D, and it is expected that users will find many innovative ways to use this feature.

2.22 RESTART

DYNA3D supports two types of restart: standard restart and running restart. Standard restart files are written at an interval specified in the input file, and are written sequentially. Standard restart files are never overwritten, and therefore the analysis may be restarted from any standard restart dump state. Since these files accumulate, the frequency of writing standard restart files is often limited by the amount of available disk space. To alleviate this problem, running restart capability was developed. Running restart files are also written at an interval specified in the input file, but each running restart dump overwrites the previous dump. Thus, a running restart dump may be written as often as desired without requiring excessive disk storage.

DYNA3D allows many changes to the model and analysis at a restart. These changes are specified in an optional "restart input file," which is specified in addition to the binary "restart dump file" which contains the data from the first part of the run. The restart dump file may be either a standard restart file or a running restart file, and all restart options are supported from either choice. The restart input file need not be supplied if nothing is to be changed. Analysis options which can be changed at restart include termination time, plot data dump interval, and restart dump intervals. In addition, the model itself may be changed during a restart, including deletion of any number of materials, elements, or slide surfaces, and alterations of nodal boundary conditions for rigid or deformable materials.

OVERVIEW OF DYNA3D

2.23 MATERIAL MODEL DRIVER

and the second

DYNA3D incorporates a Material Model Driver to simplify fitting material models to experimental data. The driver computes the stress response to a prescribed strain path, and allows the results to be displayed in an interactive graphics environment. Any stress, strain, or history variable may be plotted against any other component to allow clear insight into the material model behavior. In the Material Model Driver, the stress response is computed without inertial effects, so the true constitutive response is shown. This capability is available for all DYNA3D material models for both shell elements and solid elements, and the material input format is identical to that used for dynamic analysis. The Material Model Driver is described in detail in Chapter 6 of this manual.

2.24 COUPLING WITH CRASH VICTIM SIMULATION (CVS) CODES

Two types of coupling are available between DYNA3D and crash victim simulation (CVS) codes: data link and full coupling.

In the link option, a DYNA3D rigid material representing the occupant compartment is selected, and its displacement or acceleration time history is written to the file CVS.LNK. This file can then be easily inserted into a CVS input file to prescribe the motion of the occupant environment for detailed injury studies. The data link option is activated by specifying the DYNA3D material number which represents the occupant cage on Control Card 4.

The full coupling option allows the CVS representation of the occupant to impact interior surfaces in the passenger compartment, and thus yields a more realistic simulation. In the current implementation of this coupling, during a time step the CVS program gives to DYNA3D the positions and orientations of each ellipsoid. DYNA3D then uses its contact algorithms to evaluate the total force and moment arising from impact between the ellipsoid and the occupant environment, and transmits this information back to the CVS code. To use this full coupling, the DYNA3D model should contain a geometric representation of the ellipsoids in the CVS model, and each ellipsoid should be defined as a separate DYNA3D rigid material.

The CVS coupling has been implemented to two specific CVS codes: MADYMO and ATB. MADYMO is a commercial code developed and distributed by TNO, and is described in (TNO, 1992). ATB is a U.S. government-developed CVS code (Obergefell, Gardner, Kaleps, and Fleck, DYNA3D User Manual

COUPLING WITH CRASH VICTIM SIMULATION (CVS) CODES

1983). The coupling between DYNA3D and these CVS codes uses the public domain PVM message passing library to implement the interface, and therefore should allow new versions of either DYNA3D or the CVS code to be used in the coupling with little or no additional effort..

As DYNA3D's use has grown rapidly in the crashworthiness area, it is expected that this CVS link should be useful to analysts in evaluating safety aspects of a variety of vehicles including automobiles, airplanes, and helicopters.

.

DYNA3D User Manual

,

-

2

)

/

.

3.0 ANALYSIS WITH DYNA3D

The typical engineering analysis process begins with a physical description of some problem or system to be studied. First, a pre-processor or mesh generator is used to construct the model. This program writes a DYNA3D input file. This input file is a standard ASCII text file, so it may be edited or modified at this stage, if desired. Next, the DYNA3D analysis is run, which generates the ASCII printout file and a number of binary plot and restart files. The analysis may be conducted all in one run, or there may be multiple DYNA3D terminations and restarts in this part of the process. Finally, a post-processor is used to read the binary plot files and create display and hardcopy graphics output of desired quantities. Each of these steps is described in the sections that follow.

3.1 PRE-PROCESSING AND MODEL GENERATION

DYNA3D does not contain any significant model generation capability, and relies totally on external software for this task. The LLNL mesh generator INGRID (Christon, Dovey, and Hallquist, 1992) is widely used at LLNL and provides full support for all DYNA3D analysis options. In addition, many of the widely-used commercial pre/post-processing codes are now supporting DYNA3D. The format of the DYNA3D input file is described in Chapter 4 of this manual. Since the DYNA3D input file is in ASCII text format, many users find it convenient to do all model generation on an engineering workstation, and then transfer the DYNA3D input file to a larger computer to run the analysis.

3.2 STARTING A NEW DYNA3D ANALYSIS

The execution line for DYNA3D varies slightly depending on the computing platform. On all systems except VAX/VMS, the execution line is:

dyna3d i=inf, o=otf, g=ptf, d=dpf, f=thf, t=tpf, a=rrd, m=sif, s=iff, u=isf1, v=isf2, b=rlf

where

*inf=*input file name

otf=printed output file name

ptf=binary state data plot file name (for post-processing)

dpf=binary standard dump file name (for restarting)

ANALYSIS WITH DYNA3D

DYNA3D User Manual

thf=binary time history data plot file name (for post-processing) tpf=binary TOPAZ3D plot file name (for temperature profiles) rrd=binary running dump file name (for restarting) sif=binary NIKE3D initialization file name (for static initialization) iff=binary interface force plot file name (for post-processing) isf1=binary interface save segment file name for writing data isf2=binary interface save segment file name for reading data rlf=binary plot file name for dynamic relaxation data (for post-processing)

On VAX/VMS systems, the execution line is:

run dyna3d

The user then types the file name specifications when prompted:

i=inf, o=otf, g=ptf, d=dpf, f=thf, t=tpf, a=rrd, m=sif, s=iff, u=isf, v=isf2, b=rlf

File names must be unique and can have up to six characters. When starting an analysis, the input file name must be specified. For example,

dyna3d i=inf

is a valid execution line. The specification of other file names is optional, except as noted below.

A TOPAZ3D plot file name, t=tpf, must be specified if nodal temperatures are to be read from a TOPAZ3D plot file. A stress initialization file name, m=sif, must be specified if it is desired to initialize the DYNA3D model using the stresses and deformations from a NIKE3D analysis. DYNA3D creates an interface force plot file for post-processing when an interface force plot file name, s=iff, is specified on the command line. Similarly, DYNA3D only creates an interface save segment file when an interface save segment file name, u=isf1, is given on the command line. An interface save segment file name for reading, v=isf2, must be specified when stored data is to be read from an existing interface save segment file.

Due to the large volume of data generated by a typical DYNA3D analysis, printed results output is of limited use and is not generated by default. Printed results may be obtained for nodes and elements selected in time history blocks by specifying **f=hsp** on the DYNA3D execution line. This flag will signal DYNA3D to write the results for the selected nodes and elements into the printed output file instead of into a binary time history data plot file.

The default file names for DYNA3D files are given in Table 1. It is expected that all file names will have a file extension of .DAT on VAX/VMS systems, however, this extension should never be specified in defining a file name. No file name extensions are expected on non-VMS systems.

identifier	default file name	purpose
inf	(none)	input file
otf	d3hsp	printed output file
ptf	d3plot	binary plot state data file
dpf	d3dump	binary standard restart dump file
thf	d3thdt	binary time history plot data file
tpf	(none)	TOPAZ3D plot file containing temperature data
rrd	runrst	binary running restart dump file
sif	(none)	stress initialization file from NIKE3D
iff	(none)	interface force plot data file
isf1	(none)	interface segment save file to be created
isf2	(none)	interface segment save file to be used
rlf	d3drlf	binary plot data file for dynamic relaxation

Binary file sizes are set depending on the computing platform being used. Like the other codes in the MDG set, DYNA3D uses a familied file system to control the length of individual files. In this system, a root file is augmented by additional "family members" when data to be written exceeds the set file length. New family members are named by appending a two digit number to the root name. For example, following an analysis with several state dumps written to the plot state database, the resulting family of binary files might be

d3plot d3plot01 d3plot02 d3plot03.

ANALYSIS WITH DYNA3D

DYNA3D User Manual

4

If, instead of the default, the binary state data plot file name g=pltdat had been given on the command line, the resulting file family would be

pltdat pltdat01 pltdat02 pltdat03.

Note that file families are specified by their root name, and that there is a maximum of 100 files in a family.

3.3 INTERACTIVE ANALYSIS STATUS QUERY

DYNA3D offers several "sense switches" to allow the brief interruption of an analysis to request a status report, request that a plot state or restart file be written at the current time, or to cleanly terminate the run. To use sense switches on CRAY/NLTSS, simply type the desired switch while DYNA3D is executing. On CRAY/UNICOS, VAX/VMS, and SUN/UNIX machines, first interrupt execution by typing <ctrl>c, and then enter the desired sense switch at the prompt. The following four sense switches are available:

Туре	Response
sw1.	A restart file is written and DYNA3D terminates.
sw2.	DYNA3D responses with time and cycle numbers.
sw3.	A restart file is written and DYNA3D continues.
sw4.	A plot state is written and DYNA3D continues.

3.4 RESTARTING A DYNA3D ANALYSIS

The execution line for restarting a DYNA3D analysis from either a standard restart dump file or a running restart dump file is

```
dyna3d i=irf, r=rtf, o=otf, g=ptf, d=dpf, f=thf, t=tpf, a=rrd, s=iff, u=isf1, v=isf2, b=rlf
```

where

rtf=binary restart dump file name, including the full family name (e.g., d3dump03) *irf*=optional restart input file name

and the other file designations are as given in Section 3.2. The restart file name specified by $\mathbf{r}=rtf$ may be either a standard restart file or a running restart file. In either case, care must be taken to specify the entire name of the file, and not just the root name (e.g., d3dump03, not just d3dump). This is necessary since family members in a standard restart dump sequence represent different points in an analysis, and it is necessary to specify which restart dump is to be used as the starting point for the current restart run. Some very large problems require two standard restart dump file family members to store the required data for one restart dump. This case is easily recognized since DYNA3D writes to the screen the name of the first file in which a specific restart dump is written. In most cases, the first three restart dumps would produce messages to the screen like

restart file d3dump01 written, 186350 words restart file d3dump02 written, 186350 words restart file d3dump03 written, 186350 words

For very large problems, the first three restart dumps would produce messages to the screen like

restart file d3dump01 written, 1586232 words restart file d3dump04 written, 1586232 words restart file d3dump07 written, 1586232 words

When restarting these very large problems, the restart dump file name specified by r=rtf on the command line should name the first restart file family member for the desired restart dump. For example, to restart the very large problem above from the second restart dump state, a valid execution line might be

dyna3d r=d3dump04

Specifying the name of restart files is the *only* time a family file member name should be specified including the two digit suffix.

3.5 POST-PROCESSING AND RESULTS DISPLAY

DYNA3D writes up to three binary plot databases. The state data plot file family contains information for complete states at relatively infrequent intervals; 50 to 100 states of data are typical in a state database. The time history data plot file family contains information for only selected nodes and elements, but at much more frequent intervals; 1000 to 10,000 states of data are typical in a time history database. The interface force plot file family, if requested, contains information about the normal and shear forces on DYNA3D slide surfaces.

There are two LLNL codes which can be used to visualize results from DYNA3D: TAURUS (Spelce and Hallquist, 1991) and GRIZ (Dovey and Spelce, 1993). In addition, there are several commercial pre/post-processors which can read and display the DYNA3D database, and more are coming. The LLNL TAURUS and GRIZ codes are briefly described below.

TAURUS is the older of the two LLNL postprocessing codes, and uses a command-line based user interface with X-Windows graphics for portability. TAURUS can read any of the three binary plot databases produced by DYNA3D. TAURUS allows plotting of color contours, fringes, deformed shapes, and time histories in an interactive graphics environment. TAURUS fully supports unstructured meshes with arbitrary combinations of beam, shell, and solid element classes. TAURUS can compute a variety of strain measures, momenta, and other response quantities of interest. An interactive help package describes new commands and provides assistance to new or infrequent users. TAURUS is supported for the same computing platforms as DYNA3D: CRAY, VAX/VMS, SUN, SILICON GRAPHICS, and the IBM RS/6000. TAURUS uses the public domain graphics library DIGLIB, developed by Hal Brand at LLNL. DIGLIB supports a large number of display and hardcopy graphics devices, including X-Windows and Postscript (black and white or color) for hardcopy output.

GRIZ is a ground-up new LLNL code for visualization of finite element results. It is under intensive development at the time of this writing, so the following information is subject to change. GRIZ is currently oriented toward hardware-based graphics systems and uses the GL library, although future versions may remove this requirement. GRIZ provides support for modern 3-D visualization techniques such as isosurface display, cutting planes, and display of vector data, all within a menu-driven graphical user environment. GRIZ also provides support for animating data over time and for storing animation frames to a video disk. Initial GRIZ development is aimed at supporting the DYNA3D state database, with time history database and interface force database support to follow in subsequent releases.

DYNA3D User Manual

3.6 INTERFACE FORCES AND NODAL REACTION FORCES

The total force across a slidesurface as a function of time can be optionally output from DYNA3D into either an ASCII print file and/or a binary plot file which can be read by TAURUS or other postprocessing software. This interface force information is available for all types of slidesurfaces, including tied slidesurfaces. Thus, it is possible to easily obtain the total cross-section force in a solid element model by simply defining a tied interface and examining the interface force on this surface. Details on this capability are described in section 3.6 on page 39. Interface force output has proven quite useful in many types of engineering analysis, and has been significantly enhanced in this version of DYNA3D.

Interface forces can be written to an ASCII printout file called FORCES if flags are set on the Slidesurface Control Card described in section 4.26 on page 261. This FORCES file contains interface force data at the time interval specified for state plot output. At each output time, a block of interface force data is written for each side (slave/master) of each slidesurface for which output has been requested. For each block of slidesurface data, the total force acting on the side of the interface is first written in global components. Next, three lines of data are given for each node on the interface. The first line is the interface force at that node in global coordinates. The second line is the same interface force in a local coordinate system oriented with a local normal n to the surface and two local basis vectors v and c in the plane of the interface. The third line gives the global coordinates of this interface node. It is hoped that this data format in the FORCES file will be convenient for users who want to import this data into another program for display or plotting.

Interface force data can also be written to a binary plotfile by specifying a filename using the s=filename on the command line, as described in section 3.2 on page 33. This option writes contact segments into the database as though they were shell elements, and they can therefore be viewed as a surface at a fixed time in the analysis. Time history plots of total force on an interface, in global components, can be generated with the TAURUS postprocessor, and possibly with other codes as well.

In TAURUS Phase 2, time history plots of interfaces are generated using the

FTIME c n s1 s2 ... sn

command, where c is the interface component number, n is the number of interface sides to plot, and sI to sn are the interface sides to be plotted. Interface component numbers are 1 for x-force, 2 for y-force, and 3 for z-force, and may be displayed using the FCOMP command. Interface sides are numbered by beginning with the slave side of slidesurface one as interface side one, the master

2.1. 或時間的目的方面為正式的基本的範疇的方面的。

ANALYSIS WITH DYNA3D

. DYNA3D User Manual

side of slidesurface one as interface side two, and proceeding sequentially through all slidesurfaces in the DYNA3D model. Thus, the slave side of DYNA3D slidesurface k is numbered 2k-1, and the master side has interface side number 2k. Note that only a slave side exists for single surface contact, and thus this numbering scheme must be modified if single-surface slidesurfaces are present. Caution should be exercised in determining interface side numbers as no internal validity checking is done in TAURUS, and erroneous plots will be generated if invalid interface side numbers are requested.

4.0 INPUT FORMAT

The following sections describe the input for DYNA3D. Notational conventions are described in section 2.1 on page 8. Numerous notes and explanations are given to describe the purpose and application of specific features. The DYNA3D input file is entirely ASCII, and is completely portable across all computer platforms.

4.1 CONTROL CARDS

Following is the input description for the ten control cards. A set of notes on control card entries follows at the end of this section.

	Card 1	
Columns	Quantity	Format
1-72	Heading or problem title	A72
73-74	Code input format EQ.87: input follows format of User Manual, Rev. 3, 1987 EQ.88: input follows format of User Manuals since 1988	A2
76-80	Large format option (default in INGRID, use if node numbers exceed 99999); input "large"	A5

INPUT FORMAT

. ...

DYNA3D User Manual

.

.

{

Columns	Quantity	Format
1-5	Number of materials, NUMMAT	15
6-15	Number of nodes, NUMNP	110
16-25	Number of solid (brick) elements, NUMELH	110
26-35	Number of 2-node beam and truss elements, NUMELB	I10
36-45	Number of 4-node shell elements, NUMELS	I10
46-55	Number of 8-node thick shell elements, NUMELT	I10
56-65	Number of interface segments for linking, NUMIFS	I10
66-75	Output interval for interface segment save file (Δt)	E10.0
76-80	Scale factor for minimum shell element time step	E5.0

ţ

<u>Columns</u>	Quantity	Format
1-5	Number of node time history blocks (maximum of 2000)	15
6-10	Number of solid element time history blocks (maximum of 2000)	15
11-15	Number of beam element time history blocks (maximum of 2000)	15
16-20	Number of shell element time history blocks (maximum of 2000)	15
21-25	Number of thick shell element time history blocks (maximum of 2000)	15
26-30	Number of steps between problem status reports	15
31-35	Reaction forces print flag EQ.0: no printing EQ.1: reactions at nodes with prescribed velocities are written to the file FORBCT	15
36-40	Discrete (lumped parameter) element forces print flag EQ.0: no printing EQ 1: forces in all discrete element are written to the file EOBDIS	15
41-45	Element deletion/SAND database flag EQ.0: failed elements not deleted, no SAND info in plot database EQ.1: failed elements are deleted, SAND info in plot database	15

INPUT FORMAT

DYNA3D User Manual

(* a

;

Columns	Quantity	Format
1-5	Unused at this time	
6-10	Number of sliding boundary planes	15
11-15	Number of symmetry planes with failure	15
16-20	Number of points in density vs. depth curve, NUMDP	15
21-25	Brode function flag EQ.0: Brode functions are not used. EQ.1: Brode function parameters are defined in input.	15
26-30	Number of rigid body merge cards, NRBC	15
31-35	Nodal coordinate format: either E10.0 or E20.0	A5
36-40	Number of cross section definitions for force output, NUMCSD	15
41-50	Time interval between output of cross section forces	E10.0
51-55	ATB/MADYMO rigid body code link flag EQ.0: No link file is written. EQ.N: Link file containing displacements is written for rigid mate EQN: Link file containing accelerations is written for rigid mate	I5 rial N. rial N.
56-60	Number of ellipsoids for coupled CVS analysis, NMADEL	15

DYNA3D User Manual

٩

í

CONTROL CARDS

Columns	Quantity	Format
1-5	Number of load curves	15
6-10	Number of concentrated nodal loads	15
11-15	Number of element faces having pressure loads applied	15
16-20	Number of prescribed velocity/acceleration cards	15
21-25	Number of rigid walls	15
26-30	Number of nodal constraint cards	15
31-35	Initial condition flag EQ.0: initialize velocities to zero EQ.1: initial velocities are read from the input file	15
36-40	Number of sliding interfaces, NUMSI	15
41-45	Base acceleration in x-direction EQ.0: no EQ.1: yes	I5 .
46-50	Base acceleration in y-direction EQ.0: no EQ.1: yes	15
51-55	Base acceleration in z-direction EQ.0: no EQ.1: yes	15
56-60	Angular velocity about x-axis EQ.0: no EQ.1: yes	15
61-65	Angular velocity about y-axis EQ.0: no EQ.1: yes	15
66-70	Angular velocity about z-axis EQ.0: no EQ.1: yes	15
71-75	Number of solid elements for momentum deposition	15
76-80	Number of detonation points	15

INPUT FORMAT

DYNA3D User Manual

• • •

٦,

. . .

___/

Columns	Quantity	Format
1-10	Termination time	E10.0
11-20	Time interval between writes of time history plot data	E10.0
21-30	Time interval between writes of state plot data and interface force data	E10.0
31-35	Number of time steps between writes of restart file	15
36-40	Number of time steps between writes of running restart file	15
41-50	Initial time step size EQ.0.0: DYNA3D determines initial step size	E10.0
51-60	Global scale factor for sliding interface penalty stiffness EQ.0.0: default =.10	E10.0
61-65	Thermal effects option EQ.0: no thermal effects EQ.n: temperature-time history is defined by load curve n EQ1: nodal temperatures are defined in TOPAZ3D generated plot files EQ2: nodal temperatures use Temperature Data Option 2 EQ9999: nodal temperatures use Temperature Data Option 1	15
66-70	Reset default hourglass and bulk viscosities flag (IRQ) EQ.1: new defaults are read on Control Card 10	15
71-80	Time step scale factor, <i>SCFT</i> (Default =.90, if high explosive materials are used, the default is lowered to.67)	E10.0

(

٠

Columns	Quantity	Format
1-5	Number of rigid body joint definitions	15
6-10	Number of rigid bodies for which extra nodes are defined, NRBEN	15
11-15	Number of shell-solid element interface definitions	I5
16-20	Number of tie-breaking shell slidelines, NTBSL	15
21-25	Number of tied node sets with failure, NTNWF	15
26-30	Load curve number that limits maximum time step size (optional)	15
31-35	Flag for lumped parameter springs, dampers, and mass input, <i>INPSD</i> EQ.0: no discrete springs, dampers, or masses EQ.1: discrete springs, dampers, or masses are input	15
36-40	Number of rigid bodies for which optional inertial properties are defined	15
41-45	Flag for output shell strains tensors at inner and outer surface EQ.0: no additional data written EQ.1: data written to plot database	15
56-60	Hughes-Liu shell normal update option EQ2: unique nodal fibers EQ1: compute normals each time step EQ.0: default set to -1 EQ.1: compute on restarts EQ.n: compute every n time steps	15
61-65	Shell thickness change option EQ.0: no change EQ.1: membrane straining causes thickness change	15
66-70	Default shell element formulation EQ.1: Hughes-Liu EQ.2: Belytschko-Tsay (default) EQ.3: BCIZ (triangle) EQ.4: C^0 (triangle) EQ.5: Membrane EQ.6: YASE	15
71-75	Number of nonreflecting boundary segments, NNRBS	15

INPUT FORMAT

~

DYNA3D User Manual

ļ

1.1

Card 8

<u>Columns</u>	Quantity	Format
1-5	Number of nodal single point constraints, NODSPC	15
6-10	Number of coordinate systems for single point constraints, NSPCOR	15
11-20	Reduction factor for initial time step size to determine minimum time step size (TSMIN). $\Delta t_{min} = \Delta t_{initial} \times TSMIN$ where $\Delta t_{initial}$ is the initial step size determined by DYNA3D. When Δt_{min} is reached, DYNA3D terminates with a restart dump.	E10.0
21-25	Number of user specified beam integration rules, NUSBIR	15
26-30	Maximum number of integration points required in the user specified beam integration rules	. I5
31-35	Number of user specified shell integration rules, NUSSIR	15
36-40	Maximum number of integration, points required in the user specified shell integration rules	15
41-45	Number of iterations between convergence checks for dynamic relaxation option (default=250)	15
46-55	Convergence tolerance for dynamic relaxation option (default=0.001)	E10.0
56-65	Dynamic relaxation factor (default=.995)	E10.0
66-75	Scale factor for computed time step during dynamic relaxation; if zero, the value is set to <i>SCFT</i> defined on Control Card 6. After converging, the scale factor is reset to <i>SCFT</i> .	E10.0
76-80	Basis of time step size calculation for 4-node shell elements EQ.0: characteristic length=area/(longest side) EQ.1: characteristic length=area/(longest diagonal) EQ.2: based on bar wave speed and	15

max (shortest side, area/longest side)

DYNA3D User Manual

.

Columns	Quantity	Format
1-5	Plane stress plasticity algorithm option for shell elements EQ.1: iterative plasticity with 3 secant iterations (default) EQ.2: full iterative plasticity EQ.3: stress scaling noniterative plasticity	15
6-10	Printout flag for element time step sizes on the first cycle EQ.1: the governing time step size for each element is printed	15
11-15	Number of one-dimensional slideline definitions, NUMSL	15
16-20	Dynamic relaxation database flag EQ.1: write plot database at every convergence check during dynamic relaxation into the DR database	15
21-30	Rayleigh damping mass proportional coefficient, α	E10.0
31-35	No. of materials with stiffness proportional Rayleigh damping, NDAMP	15
36-40	No. of materials to initialize for rotational motion, NMROT	15
41-45	No. of materials with body force loads, NMBDF	I5

INPUT FORMAT

DYNA3D User Manual

Card 10 Optional Default Viscosity Reset Card (define only if IRQ=1)

Columns	Quantity	Format
1-5	Hourglass stabilization method, <i>IHQ</i> EQ.0: default=1 EQ.1: standard DYNA3D (viscous form) EQ.2: Flanagan-Belytschko (viscous form) EQ.3: Flanagan-Belytschko with exact volume integration (viscous for EQ.4: stiffness form of type 2 (Flanagan-Belytschko) EQ.5: stiffness form of type 3 (Flanagan-Belytschko exact volume)	15 rm)
5-15	Hourglass stabilization coefficient (QH , default=.10) Values of QH that exceed .15 may cause instabilities. The recommended default applies to all options and element types. The stiffness forms can stiffen the response, especially if deformations are large, and therefore should be used with care.	E10.0
16-20	Bulk viscosity type for shock capture, <i>IBQ</i> EQ.0: default=1 EQ.1: standard DYNA3D	15
21-30	Quadratic bulk viscosity coefficient (default=1.5)	E10.0
31-40	Linear bulk viscosity coefficient (default =.06)	E10.0

For all 4-node shell elements except YASE, $IHG \leq 3$ gives the viscous form of Belytschko-Tsay hourglass stabilization, and IHG = 4 or 5 gives the stiffness form. Currently, only these two stabilization algorithms are available for 4-node Belytschko-Tsay and Hughes-Liu shell elements and membrane elements. The above stabilization algorithms do not apply to the 4-node YASE shell element formulation.

A bulk viscosity method is used to smear shock fronts across a small number of solid elements. This procedure improves numerical behavior and minimizes spurious oscillations which may appear in the vicinity of strong shocks in numerical models. , ``

.

1

.

Problem Definition Notes

(card):(field)	Comments
1:2	Although many new features have been added, the input format has remained unchanged in the DYNA3D manuals of 1988, 1989, 1991, and 1993.
1:3	When specifying the LARGE format option, "large" must be input in lower case.
2:1	Materials are defined as described in section 4.2 on page 59.
2:2	Nodal coordinates and attributes are defined as described in section 4.5 on page 217.
2:3	Solid (brick) element connectivities and attributes are defined as described in section 4.6 on page 219.
2:4	Beam and truss element connectivities and attributes are defined as described in section 4.7 on page 221.
2:5	Shell element connectivities and attributes are defined as described in section 4.8 on page 224.
2:6	Thick shell element connectivities and attributes are defined as described in section 4.9 on page 226.
2:7	Interface save segments are defined as described in section 4.10 on page 228.
2:8	Time interval between writes of the interface segment save file.
2:9	If nonzero, this option allows the specification of a minimum time step size for thin shell elements using material types 3, 19, or 24. When a shell element controls the time step, element material properties will be modified such that the time step does not fall below this factor times the initial time step size.
3:1	Node time history blocks are defined as described in section 4.14 on page 236.
3:2-5	Element time history blocks are defined as described in section 4.15 on page 237.
3:6	Sense switch "sw2." data summarizing current problem time, time step, energies, and momenta are written to the hsp file at this interval. This is often useful for monitoring the status of a run in batch mode.

INPUT FORMAT

DYNA3D User Manual

j

ĩ

Problem Definition Notes

~

(card):(field)	Comments
3:7	If set, this flag writes the total reaction forces at all degrees-of-freedom with prescribed velocities into the file FORRCT at the state plot interval. Note that reactions at nodes with nodal boundary conditions (specified on the node cards) are <i>not</i> included.
3:8	If set, this flag writes the internal force in each discrete (lumped parameter) element into the file FORDIS at the state plot interval.
3:9	This flag controls what happens when "failure" is detected in a material for which SAND contact is supported. If this flag is input as "1" then the element is deleted from the calculation and marked as deleted in the plot database once failure has occurred regardless of whether or not the element is actually in a SAND volume. If this flag is input as "0" then the element remains in the calculation and is marked active in the plot database, and the subsequent post-failure mechanical behavior is as described for that particular material.
4:2	Sliding boundary planes are used to constrain a node to move on an arbitrarily oriented line or plane. These are defined as described in section 4.12 on page 232.
4:3	Symmetry planes with failure are used to constrain a node to move on a plane until a critical tensile stress is reached, after which the constraint is released. This feature is frequently used to model oblique penetration. Symmetry planes with failure are defined as described in section 4.13 on page 233.
4:4	Density vs. depth curves are used to permit initialization of a hydrostatic stress state due to overburden and gravity. These curves are defined as described in section 4.16 on page 239.
4:5	This option is used to define a pressure time history using the Brode function definition. Define the Brode parameters as described in section 4.17 on page 241, and the surfaces to receive the pressure loading as described in section 4.21 on page 249.
4:6	Two distinct materials which each use the rigid material model may not share nodes in common. These two materials may be merged into one rigid body using the rigid body merge feature described in section 4.29 on page 278.
4:7	Defines the format in which nodal coordinates are read from the input file. Use of the E20.0 format is recommended when high precision is required in the geometry definition.

(

ι,

5

(card):(field)	Comments
4:8	To obtain cross section forces for structural elements, define as described in section 4.18 on page 242.
4:10	When a nonzero rigid body material number is specified, the time, followed by the rigid body center-of-mass translational and rotational displacements or accelerations are written to the file named "CVS.LNK" in the current directory.
4:11	A kinetic interface coupling between DYNA3D and the crash victim simulation codes MADYMO or ATB may be defined. Rigid representations of the CVS ellipsoids should be be defined in the DYNA3D model as described in section 4.43 on page 303.
5:1	Define load curves as described in section 4.19 on page 245.
5:2	Define concentrated nodal loads and follower forces as described in section 4.20 on page 247.
5:3	Define pressure loads on element faces as described in section 4.21 on page 249.
5:4	Define prescribed velocity (or acceleration) boundary conditions as described in section 4.22 on page 251.
5:5	Rigid walls are used to define unilateral contact, such as the impact of a body into a large, rigid object. Define rigid walls as described in section 4.23 on page 253.
5:6	Nodal constraints are used to constrain two or more nodes to move together (i.e., share a common degree of freedom). Define nodal constraint sets as described in section 4.24 on page 257.
5:7	Specify nonzero initial velocities as described in section 4.25 on page 259.
5:8	Sliding interfaces allow general contact between bodies or parts of bodies. Define sliding interfaces as described in section 4.26 on page 261.
5:9-11	Prescribed base acceleration may be used to apply load to an entire structure. Define the three translational acceleration components as described in section 4.32 on page 282. Base acceleration may alternatively be applied to only a subset of the model, as described in section 4.43 on page 303.

INPUT FORMAT

DYNA3D User Manual

¥ .1

(card):(field)	Comments
5 : 12-14	Prescribed angular velocities about any axis through the origin may be used to apply loads to an entire structure. Define the three angular velocity components as described in section 4.33 on page 283. Prescribed angular velocity may alternatively be specified for only a subset of the model, as described in section 4.43 on page 303.
5:15	Momentum may be deposited directly into solid elements. Define momentum deposition as described in section 4.34 on page 285.
5:16	Detonation points may be specified for the "programmed HE burn" options described in section 2.16 on page 25. Specify detonation points as discussed in section 4.35 on page 286.
6:1	Termination time is the simulation time at which the run is to terminate.
6:2	The time history database allows plotting information to be stored frequently for a selected group of nodes and elements, thus allowing resolution of high frequency response components. Alternatively, nodes and elements selected for the time history database may be printed into the hsp file by specifying f=hsp on the DYNA3D execution line, as described in section 3.2 on page 33. Nodes and elements are selected for inclusion in the time history database as described in section 4.14 on page 236.
6:3	The state plot database contains data for all nodes and elements of a model. The interface force plot database is optional (specified as described in section 3.6 on page 39), and contains interface forces, pressures and shear tractions for all sliding interfaces of a model.
6:4	Restart files are written consecutively and are not overwritten, thus allowing restart from any point in an analysis at which a restart file was written. The use of restart is described in section 3.4 on page 36.
6:5	Running restart files are continuously overwritten to conserve disk space and allow frequent updates. The use of running restart is described in section 3.4 on page 36.

."

i

(card):(field)	Comments
6:7	This factor is used in computing penalty stiffnesses for sliding interface calculations, and is global. Local penalty scale factors may be specified on the sliding interface control card described in section 4.26 on page 261. In general, increasing this factor decreases interpenetration, but may adversely affect stability if the time step scale factor is not also reduced. NIKE3D uses a softer default penalty factor than DYNA3D, so in order to achieve force equilibrium when slidesurfaces are involved in a model in which initial stresses are computed in NIKE3D, the penalty scale factor should be set to 0.046, which is about a factor of two softer than the DYNA3D default of 0.10.
6:8	Temperatures must be defined if temperature-dependent material properties are used. Several options are available for specifying temperatures for DYNA3D. Care should be taken in the definition of the reference, or stress- free temperature.
6:9	This flag allows the global default values of the hourglass viscosity and shock viscosities to be reset. These values may also be altered for a specific material in the Material Cards, as described in section 4.2 on page 59.
6:10	This factor is multiplied by the maximum stable time step to determine the time step size actually used by DYNA3D. This is useful for the (rare) situations in which the time step size is chosen based on other than stability considerations.
7:1	Rigid body joints allow the connection of multiple rigid bodies with specified degrees-of-freedom remaining free at the connection. Define rigid body joints as described in section 4.31 on page 280.
7:2	Extra nodes can be defined and placed anywhere on a rigid body. This is often useful for defining rigid body joints. Define extra nodes for rigid bodies as described in section 4.30 on page 279.
7:3	One technique for transitioning between shell elements and solid elements in a mesh is the shell-solid interface. This is an experimental capability which enforces appropriate constraints on solid element nodes to provide moment resistance at the interface with a 4-node shell element. Define shell- solid interfaces as described in section 4.36 on page 287.
7:4	Tie-breaking shell slidelines allow shell edges to separate once a critical effective plastic strain is exceeded. Define tie-breaking shell slidelines as described in section 4.27 on page 274. Note that tie-breaking shell slidelines may not cross.

INPUT FORMAT

DYNA3D User Manual

1

Ĵ

÷

•

(card):(field)	Comments
7:5	Tied node sets allow arbitrary failure and separation of nodes for 4-node shell elements once a critical plastic strain is exceeded. Tied node sets are similar to tie-breaking shell slidelines except that they are defined purely on a nodal level, and there is no restriction on the topology. Define tied node sets with failure as described in section 4.28 on page 276.
7:6	If desired, define a load curve giving the maximum time step size as a function of time. DYNA3D will use the smaller of this value or the internally computed time step size. This feature is useful when the time step size must be limited to provide adequate time resolution of an externally applied load.
7:7	Set this flag to 1 if discrete springs, dampers, and masses are specified as described in section 4.37 on page 289.
7:8	This option allows rigid body inertial properties to be specified directly rather than calculated from the given geometry and mass density. Define rigid body inertial properties as described in section 4.38 on page 295.
7:9	The through-thickness location of shell result quantities in TAURUS is selected using the SHELL INNER, SHELL MIDDLE, and SHELL OUTER commands.
7:10	The Hughes-Liu shell element formulation uses normal vectors defined at each node. Several algorithms are provided for the definition and evolution of these nodal normal vectors. The "unique nodal fibers" algorithm (nodal normal update option -2) is most nearly the procedure described in the original papers, but is somewhat more expensive than the default option.
7:12	This option sets the default shell element formulation to be used for 4-node shell elements. The shell element formulation may also be specified by material on the material control card as described in section 4.2 on page 59.
7:13	Nonreflecting boundaries prevent artificial stress wave reflections arising from a finite model of an infinite domain. Define nonreflecting boundary segments as described in section 4.39 on page 297.
8:1,2	Nodal single point constraints are used to specify a nodal boundary condition in a local coordinate system. Define nodal single point constraints and single point constraint local coordinate systems as described in section 4.11 on page 230.
8:3	This options terminates DYNA3D when a minimum time step size is reached, and prevents the code from consuming large amounts of computer time if the time step falls to a very small value.

.

(card):(field)	Comments
8 : 4,5	In addition to predefined Gauss and Lobotto integration rules, arbitrary integration rules may be specified for integration over the cross section in the Hughes-Liu beam element. These are mainly useful in describing complex cross section shapes. Define user-specified beam integration rules as described in section 4.3 on page 214.
8:6,7	In addition to predefined Gauss and Lobotto integration rules, arbitrary integration rules may be specified for integration through the thickness of shell elements. These are mainly useful for describing laminated composite shells where material properties (such as fiber orientation) vary by lamina. Define user-specified shell integration rules as described in section 4.4 on page 215.
8:8	Dynamic relaxation may be used for the static solution of some problems. This feature is activated by the flag on the load curve definition card as described in section 4.19 on page 245. This an experimental capability and caution should be exercised in its use.
8:10	The dynamic relaxation factor is the factor by which velocities are reduced at each time step in a dynamic relaxation static analysis.
8:12	This option permits alternative algorithms to be used for approximating the maximum stable time step for shell elements.
9:1	Several algorithms are available for performing the plane stress constitutive integration for elastoplastic shell analysis. These options only apply to the elastoplastic shell material models.
9:2	This option is useful for determining which elements in a model control the time step and thus the cost of the solution.
9:3	One-dimensional slidelines allow beam elements to slide along a predefined row of nodes. Define one-dimensional slidelines as described in section 4.42 on page 300.
9:4	This flag controls the output of successive unconverged states into the dynamic relaxation plot file during a dynamic relaxation solution. The name of this binary plot file may be chosen by the user, as described in section 3.2 on page 33.

INPUT	FORMA	l
-------	-------	---

DYNA3D User Manual

Sterry 1.

•. 77

Problem Definition Notes (continued)

2.....

(card):(field)	<u>Comments</u>
9 : 5,6	Generalized Rayleigh damping may be used to introduce global dissipation into a model. The mass proportional damping coefficient is global, but the stiffness proportional coefficients may be defined on a material basis. All damping computations use only the elastic part of the material constitutive response. Generalized Rayleigh damping is described in section 4.43 on page 303.
9:7	Materials are initialized for rotational motion by computing initial velocities from $\mathbf{v} = \boldsymbol{\omega} \times \mathbf{r}$ for all nodes of all elements using the listed materials. If static initialization is used (via either input from NIKE3D or dynamic relax- ation in DYNA3D), \mathbf{r} is evaluated using the post-initialization geometry. Define materials to be initialized for rotational motion as described in section 4.43 on page 303.
9:8	Body forces arising from base acceleration (gravity) or from rotational motion may be specified for a selected set of materials using this option. This capability is especially useful for static initialization of stresses in rotating bodies prior to a transient analysis. Define materials to receive body force loads as described in section 4.43 on page 303.
10:1,2	Several algorithms are available for the stabilization of hourglass modes in the solid and shell elements. These algorithms do not apply for the YASE shell element formulation.
10:3-5	Bulk viscosity methods are used for added stability and resolution when shock waves are present.

......

4.2 MATERIALS

Repeat the following set of cards for each material definition until *NUMMAT* materials have been defined. Materials may be input in any convenient order, but all materials must be defined. Materials are referenced by their material number, which is used for identification only. Material type refers to the material model number, such as Material Type 1 for elastic behavior. Note that a material number can only apply to one type of element (beam, shell, thick shell, or solid).

A material model typically requires 8 cards for input, and exceptions are noted as appropriate. Not all material models are available for all element types. Valid element types are shown for each material model on the Material Control Card (Material Card 1) description. A brief list of notes follows the description of the Material Control Card.

Some material models, called "hydrodynamic" models, define only the deviatoric behavior of the material. An equation of state must be defined in conjunction with these hydrodynamic models to specify the volumetric behavior of the material. Following the material cards of a hydrodynamic material, the equation of state is defined. Note that different equations of state may require a different number of input cards. Also note that an equation of state is required by some material models and is not permitted by others. Equations of state are applicable to solid elements *only*, and are not used with beam or shell elements.

For structural elements, the cross sectional properties are defined following the material definition using the last 3 cards in this section. Cross sectional properties are defined differently depending on the element type (truss, beam, or shell) and element formulation (Hughes-Liu, Belytschko-Tsay, etc.). Many structural element formulations also permit cross sectional properties to be defined uniquely for each element on the element card. In this case, the cross sectional properties specified in the material cards define default cross section properties for that material, and are overridden by cross sectional property values specified on the element cards.

INPUT FORMAT

.

. . .

DYNA3D User Manual

}

1

olumns	Quantity	Format
1-5	Material identification number	15
6-10	Material type. The numbers in brackets indicate the element types for which the material is available: 0-solid, 1-beam, 2-shell, and 3-thick shell. An asterisk * indicates that an equation of state is required when that model is used with solid elements. EQ. 1: elastic [0,1,2,3] EQ. 2: orthotropic elastic [0,2,3] EQ. 3: kinematic/isotropic elastic-plastic [0,1,2,3] EQ. 4: thermo-elastic-plastic [0,2] EQ. 5: soil and crushable foam [0] EQ. 6: viscoelastic [0] EQ. 7: Blatz-Ko hyperelastic rubber [0] EQ. 8: high explosive burn* [0] EQ. 9: fluid* [0]	15
	 EQ.10: isotropic elastic-plastic hydrodynamic* [0] EQ.11: Steinberg-Guinan high rate elastic-plastic* [0] EQ.12: isotropic elastic-plastic [0,2] EQ.13: isotropic elastic-plastic with failure [0] EQ.14: soil and crushable foam with failure [0] EQ.15: Johnson/Cook elastic-plastic* [0,2] EQ.16: concrete/geologic material* [0] EQ.17: isotropic elastic-plastic with oriented cracks* [0] EQ.18: power law isotropic elastic-plastic [0,2] EQ.19: strain rate dependent elastic-plastic [0,2] EQ.20: rigid [0,1,2,3] EQ.21: thermal orthotropic elastic [0,2] 	
	 EQ.21: thermal offilotiopic etastic [0,2] EQ.22: fiber composite with damage [0,2] EQ.23: thermal orthotropic elastic with variable properties [0,2] EQ.24: rate dependent tabular isotropic elastic-plastic [0,2] EQ.25: extended two invariant geologic cap [0] EQ.26: metallic honeycomb [0] EQ.27: compressible Mooney-Rivlin hyperelastic rubber [0] EQ.28: resultant plasticity [1,2] EQ.29: not currently used EQ.30: closed form update elastic-plastic for shells [2] EQ.31: Frazer-Nash hyperelastic rubber [0] EQ.32: Ramberg-Osgood elastic-plastic [0] EQ.33: general orthotropic elastic-plastic [0,2] EQ.34: normal anisotropic elastic-plastic for shells [2] EQ.35: elastic-plastic with forming limit diagram [2] 	

-

ŗ

l

<u>Columns</u>	Quantity	Format
	Material type. The numbers in brackets indicate the element types for which the material is available: 0-solid, 1-beam, 2-shell, and 3-thick shell. An asterisk * indicates that an equation of state is required when that model is used with solid elements. (list of material models continued from previous page) EQ.36: brittle damage [0] EQ.37: 3-invariant viscoplastic cap [0] EQ.38: Bammann plasticity [0,2] EQ.39: Bammann plasticity with damage [0,2] EQ.40: Fahrenthold brittle damage [0] EQ.41: Fabric [2]	
11-20	Mass density, $ ho$	E10.0
21-25	Equation of state type. Define only for material types 8, 9, 10, 11, 15, 16, and 17 when these materials are used with solid elements. EQ. 1: linear polynomial EQ. 2: JWL high explosive EQ. 3: Sack high explosive EQ. 4: Gruneisen EQ. 5: ratio of polynomials EQ. 6: linear polynomial with energy deposition EQ. 7: ignition and growth of reaction in HE EQ. 8: tabulated compaction EQ. 9: tabulated EQ.10: not currently used EQ.11: pore collapse	15
26-30	Hourglass stabilization method (IHQ) EQ. 0: default=1 EQ. 1: standard DYNA3D (viscous form) EQ. 2: Flanagan-Belytschko integration (viscous form) EQ. 3: Flanagan-Belytschko with exact volume integration (viscous EQ. 4: stiffness form of type 2 (Flanagan-Belytschko) EQ. 5: stiffness form of type 3 (Flanagan-Belytschko exact volume)	I5 form)
31-40	Hourglass stabilization coefficient (QH, default = .10). Values of QH above .15 may cause instabilities if the time step is not decreased. The default value applies to all options and element types. The stiffness forms can stiffen the response, especially if deformations are large, and therefore should be used with care.	E10.0
41-45	Bulk viscosity type for shock capture (IBQ) EQ. 0: default=1 EQ. 1: standard DYNA3D	15
. - .

DYNA3D User Manual

5

i

Quantity	Format
Quadratic bulk viscosity coefficient, Q1 (default = 1.5)	E10.0
Linear bulk viscosity coefficient, Q2, (default = .06)	E10.0
Element type for which this material model is valid EQ. 0: solid (brick) EQ. 1: beam EQ. 2: shell EQ. 3: thick shell	15
Material initialization for gravity loading EQ. 0: all materials are initialized EQ. 1: only current material is initialized	15
Element formulation if other than default For beam elements: EQ. 1: Hughes-Liu EQ. 2: Belytschko-Schwer EQ. 3: Truss For shell elements: EQ. 1: Hughes-Liu EQ. 2: Belytschko-Tsay EQ. 3: BCIZ (triangle) EQ. 4: C ^o (triangle) EQ. 5: membrane EQ. 6: VA SE	15
	Quantity Quadratic bulk viscosity coefficient, Q1 (default = 1.5) Linear bulk viscosity coefficient, Q2, (default = .06) Element type for which this material model is valid EQ. 0: solid (brick) EQ. 1: beam EQ. 2: shell EQ. 3: thick shell Material initialization for gravity loading EQ. 0: all materials are initialized EQ. 1: only current material is initialized Element formulation if other than default For beam elements: EQ. 1: Hughes-Liu EQ. 2: Belytschko-Schwer EQ. 3: Truss For shell elements: EQ. 1: Hughes-Liu EQ. 2: Belytschko-Tsay EQ. 3: BCIZ (triangle) EQ. 4: C ⁰ (triangle) EQ. 5: membrane EQ. 6: VA SE

Material Control Card Notes

field_	Comments
1	The material identification number should be between 1 and NUMMAT, the number of materials in the problem. Materials may be input in any order, but all materials must be defined. It is permissible to have unused materials defined in the input.
2	The material type defines the mathematical model used to evaluate the material stress-strain behavior. Note that not all material types are available for all element types.
3	Define the mass density for the material in consistent units. No unit conversions are done internally in DYNA3D (see discussion in section 2.2 on page 9).

62

•

(

ł

•

field	Comments
4	If a hydrodynamic material model is selected, then an equation-of-state must be selected to define the material pressure-volume behavior.
5, 6	The default hourglass stabilization algorithm and parameters may be modified for this material, if desired. These options do not apply to truss or beam elements or to the YASE shell element.
7-9	Bulk viscosity methods are used for added stability and resolution in solid elements when shock waves are present.
10	A material definition may only be used for one element type.
11	If density vs. depth curves are used to initialize materials with hydrostatic stresses due to overburden, all materials or selected materials may be chosen for initialization. If only selected materials are to be initialized, this flag indicates whether the current material is to be initialized.
12	Element formulations for an element type may be arbitrarily mixed in an analysis. The default shell element formulation is Belytschko-Tsay, and the default beam element formulation is Hughes-Liu. A non-default element formulation applies only to elements using the current material.



This material title will appear in the printed output. It is often helpful to use this heading to identify the physical material for which the DYNA3D material model was constructed.

DYNA3D User Manual

Cards 3, ..., 8

Define material cards 3, ..., 8 as described below for the desired material type.

Material Type 1 (Elastic)

Columns		Quantity	Format
1-10	Card 3	Young's modulus, E	E10.0
1-10	Card 4	Poisson's ratio, v	E10.0
	Card 5	Blank	E10.0
	Card 6	Blank	E10.0
	Card 7	Blank	E10.0
	Card 8	Blank	E10.0

This model produces linear elastic material behavior.

65

· .-*

DYNA3D User Manual

--

ţ

Material Type 2 (Orthotropic Elastic)

Columns		Quantity	Format
1-10	Card 3	Elastic modulus, E_a (see Figure 1 and Figure 2)	E10.0
11-20		Elastic modulus, E_b	E10.0
21-30		Elastic modulus, E_c	E10.0
1-10	Card 4	Poisson's ratio, V_{ba}	E10.0
11-20		Poisson's ratio, V_{ca}	E10.0
21-30		Poisson's ratio, V_{cb}	E10.0
1-10	Card 5	Shear modulus, G_{ab}	E10.0
11-20		Shear modulus, G_{bc}	E10.0
21-30		Shear modulus, G_{ca}	E10.0
1-10	Card 6	Material axes definition option, AOPT	E10.0
	EQ.0.0:	locally orthotropic with material axes determined by element nodes n_1 , n_2 , and n_4 as shown in Figure 1. Cards 7 and 8 must be blank with this option.	
	EQ.1.0:	locally orthotropic with material axes determined by a point in space P and the global location of the element ce as shown in Figure 1. Note that d is parallel to the global Card 8 below is blank.	nter, z-axis.
	EQ.2.0:	globally orthotropic with material axes determined by vectors defined on Cards 7 and 8. (See Figure 2).	
	EQ.3.0:	applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the materia axis a by an angle β from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector (See Figure 2). The angle β is defined on Card 8, and may be overridden by specifying a value on the element card.	1 • v
1-10	Card 7	x_{p} , define only if AOPT = 1.0	E10.0
11-20		y_p , define only if AOPT = 1.0	E10.0
21-30		z_n , define only if AOPT = 1.0	E10.0
1-10	Card 7	a_r , define only if AOPT = 2.0	E10.0

Columns	<u></u>	Quantity	Format
11-20		a_y , define only if AOPT = 2.0	E10.0
21-30		a_z , define only if AOPT = 2.0	E10.0
1-10	Card 7	v_x , define only if AOPT = 3.0	E10.0
11-20		v_y , define only if AOPT = 3.0	E10.0
21-30		v_z , define only if AOPT = 3.0	E10.0
1-10	Card 8	d_x , define only if AOPT = 2.0	E10.0
11-20		d_y , define only if AOPT = 2.0	E10.0
21-30		d_z , define only if AOPT = 2.0	E10.0
1-10	Card 8	Material angle β , define only if AOPT=3.0 (may be overridden on the element card)	E10.0

The constitutive matrix C that relates increments in global components of stress to increments in global components of strain is defined as:

$$\mathbf{C} = \mathbf{T}^T \mathbf{C}_L \mathbf{T},\tag{6}$$

where T is the transformation matrix between the local material coordinate system and the global coordinate system, and C_L is the constitutive matrix defined in terms of the material constants of the local orthogonal material axes a, b, and c.

$$C_{L}^{-1} = \begin{pmatrix} \frac{1}{E_{a}} & -\frac{v_{ba}}{E_{b}} & -\frac{v_{ca}}{E_{c}} & 0 & 0 & 0 \\ -\frac{v_{ab}}{E_{a}} & \frac{1}{E_{b}} & -\frac{v_{cb}}{E_{c}} & 0 & 0 & 0 \\ -\frac{v_{ac}}{E_{a}} & -\frac{v_{bc}}{E_{b}} & \frac{1}{E_{c}} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{ab}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{bc}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{ca}} \end{pmatrix}$$
(7)

Note that symmetry of the elastic compliance C_L^{-1} implies

$$\frac{V_{ab}}{E_a} = \frac{V_{ba}}{E_b}, \ \frac{V_{ca}}{E_c} = \frac{V_{ac}}{E_a}, \text{ and } \frac{V_{cb}}{E_c} = \frac{V_{bc}}{E_b}.$$
(8)

1 - X

<u>.</u>

Further, positive definiteness of C_L yields the following restrictions on the elastic constants:

$$v_{ba} < \left(\frac{E_b}{E_a}\right)^{\frac{1}{2}}, v_{ca} < \left(\frac{E_c}{E_a}\right)^{\frac{1}{2}}, \text{ and } v_{cb} < \left(\frac{E_c}{E_b}\right)^{\frac{1}{2}}.$$
 (9)

Nonphysical energy growth may appear in the solution if these restrictions on the elastic constants are not observed.

Note that the values of Poisson's ratio are defined as

$$v_{ij} = \frac{-\varepsilon_j}{\varepsilon_i} \tag{10}$$

where ε_i is the strain in the j^{ih} direction and ε_i is the strain in the i^{ih} direction in a uniaxial stress test in the i^{ih} direction.

Į.



DYNA3D User Manual



Columns		Quantity	Format
1-10	Card 3	Young's modulus, E	E10.0
1-10	Card 4	Poisson's ratio, v	E10.0
1-10	Card 5	Yield stress, σ_0	E10.0
1-10	Card 6	Tangent modulus, E_T	E10.0
1-10	Card 7	Hardening parameter, β	E10.0
	Card 8	Blank	

Material Type 3 (Kinematic/Isotropic Elastic-Plastic)

The material behavior is elastoplastic and includes linear strain hardening. The hardening parameter β specifies an arbitrary combination of kinematic and isotropic hardening; $\beta = 0.0$ yields purely kinematic hardening, while $\beta = 1.0$ gives purely isotropic hardening. Figure 3 illustrates the effect of β on the uniaxial stress-strain curve. For modeling elastoplastic material behavior with purely isotropic hardening, Material Type 12 requires less storage and is somewhat more efficient while producing exactly the same behavior as Material Type 3 with $\beta = 1.0$. The numerical algorithms used in this model are adapted from (Krieg and Key, 1976).

The yield condition can be written

$$\phi = \overline{\sigma} - \sigma_{y}(\overline{\varepsilon}^{p}), \qquad (11)$$

where $\overline{\sigma}$ is the effective stress and σ_y is the current yield stress, which may be a function of the effective plastic strain $\overline{\varepsilon}^p$ if strain hardening is included. For isotropic hardening, the effective stress $\overline{\sigma}$ is given by

$$\overline{\sigma} = \left(\frac{3}{2}s_{ij}s_{ij}\right)^{\frac{1}{2}},\tag{12}$$

where s_{ii} is the deviatoric stress tensor. For kinematic hardening,

$$\overline{\sigma} = \left(\frac{3}{2}\eta_{ij}\eta_{ij}\right)^{\frac{1}{2}} \tag{13}$$

where the translated stress η_{ij} is defined as

$$\eta_{ij} = s_{ij} - \alpha_{ij}, \tag{14}$$

and α_{ii} is the (deviatoric) back stress tensor.

DYNA3D User Manual

ŝ



਼

1. 1.

The linear isotropic hardening law has the form

$$\sigma_{y} = \sigma_{0} + \beta E_{p} \overline{\varepsilon}^{p}, \qquad (15)$$

where σ_y is the current yield stress, σ_0 is the initial yield stress, and E_p is the plastic modulus.

The effective plastic strain $\overline{\varepsilon}^{p}$ is given by

$$\overline{\varepsilon}^{p} = \int_{0}^{1} d\overline{\varepsilon}^{p}, \qquad (16)$$

where the incremental effective plastic strain $d\overline{\varepsilon}^{p}$ is found from the incremental plastic strain tensor $d\varepsilon_{ij}$ as

$$d\overline{\varepsilon}^{p} = \left(\frac{2}{3}d\overline{\varepsilon}^{p}_{ij}d\overline{\varepsilon}^{p}_{ij}\right)^{\frac{1}{2}}.$$
(17)

The plastic modulus is found from Young's modulus E and the tangent modulus E_T using

$$E_p = \frac{EE_T}{E - E_T}.$$
 (18)

The plastic hardening modulus E_p is the slope of the inelastic portion of the effective stress $\overline{\sigma}$ vs. effective plastic strain $\overline{\epsilon}^p$ curve. Similarly, the tangent modulus E_T is the slope of the inelastic part of a uniaxial stress vs. strain curve (or equivalently, the effective stress vs. effective strain curve).

Kinematic and isotropic hardening elastoplastic models yield identical behavior under monotonic loading. Under reversed loading from a maximum stress σ_{max} , kinematic hardening predicts reverse yielding when the stress has unloaded by an amount $2\sigma_0$, and isotropic hardening predicts that reverse yielding occurs when the stress reaches $-\sigma_{max}$. Thus, under cyclic loading conditions where many stress reversals may occur, kinematic hardening predicts a hysteretic energy dissipation, while isotropic hardening predicts no energy dissipation after the first cycle. The isotropic model is slightly faster in computation speed, however.

73

INPUT FORMAT

DYNA3D User Manual

•

Material Type 4 (Thermo-Elastic-Plastic)

Columns		Quantity	Format
1-10	Card 3	First temperature, T_1	E10.0
11-20		Second temperature, T_2	E10.0
•		•	•
•		•	•
•		•	•
71-80		Eighth temperature, T_8	E10.0
1-10	Card 4	Young's modulus at first temperature, E_1	E10.0
11-20		Young's modulus at second temperature, E_2	E10.0
•		•	•
٠		•	•
•			
71-80		Young's modulus at eighth temperature, E_8	E10.0
1-10	Card 5	Poisson's ratio at first temperature, v_1	E10.0
11-20		Poisson's ratio at second temperature, v_2	E10.0
		•	•
•		•	•
•			•
71-80		Poisson's ratio at eighth temperature, v_8	E10.0
1-10	Card 6	Secant coefficient of thermal expansion, $\overline{\alpha_1}$	E10.0
11-20		Secant coefficient of thermal expansion, $\overline{\alpha_2}$	E10.0
٠			•
•		•	
•			•
71-80		Secant coefficient of thermal expansion, $\overline{\alpha_8}$	E10.0
1-10	Card 7	Yield stress at first temperature, σ_{y1}	E10.0
		Yield stress at second temperature, σ_{y2}	E10.0

DYNA3D	User Ma	nual
--------	---------	------

Columns	.	Quantity	Format
•		•	•
•		•	•
71-80		Yield stress at eighth temperature, σ_{y8}	E10.0
1-10	Card 8	Plastic modulus at first temperature, E_{p1}	E10.0
11-20		Plastic modulus at second temperature, E_{p2}	E10.0
			•
		•	•
•			
71-80		Plastic modulus at eighth temperature, E_{p8}	E10.0

At least two temperatures and their corresponding material properties must be defined. The analysis will be terminated if a material temperature falls outside the range defined in the input. If a thermo-elastic material is desired (i.e., no plasticity effects), leave Cards 7 and 8 blank.

The plastic hardening modulus E_p is the slope of the effective stress vs. effective plastic strain curve (or equivalently, the uniaxial stress vs. effective plastic strain curve). The plastic hardening modulus may be found from the tangent modulus E_T as

$$E_{p} = \frac{EE_{T}}{E - E_{T}},\tag{19}$$

where the tangent modulus E_T is the slope of the post-yield portion of the uniaxial stress - strain curve.

Thermal expansion due to temperature change is included when nonzero values of $\overline{\alpha}$ are specified. The *secant* coefficient of thermal expansion $\overline{\alpha}$ can also be a function of temperature, and is defined with respect to the reference temperature at the beginning of the calculation for that material. Total thermal strain ε_{ij}^{T} is defined in terms of the secant thermal expansion coefficient $\overline{\alpha}$ as

$$\varepsilon_{ij}^{T} = \overline{\alpha} \left(T - T_{ref} \right) \delta_{ij}, \qquad (20)$$

where T is the current temperature and T_{ref} is the reference temperature. Therefore, temperature dependent secant coefficients of thermal expansion should be defined as the value to that temperature, not the value at that temperature. The secant coefficient $\overline{\alpha}$ is related to the tangent coefficient of thermal expansion α by

$$\overline{\alpha} = \frac{1}{T - T_{ref}} \int_{T_{ref}}^{T} \alpha(T) dT.$$
(21)

For temperature independent coefficients of thermal expansion, $\overline{\alpha}$ is identical to α , and the classical definition of thermal expansion is valid.

Since this model is temperature dependent, the thermal effects option on Control Card 6 must be nonzero. Care should be taken to define a reference temperature consistent with the specified secant coefficients of thermal expansion.

This model is applicable to materials exhibiting elastic or elastoplastic behavior where thermal effects are important. Both thermal strains and temperature-dependent material properties are included.

Material Type 5 (Soil and Crushable Foam)

<u>Columns</u>	·	Quantity	Format
1-10	Card 3	Shear modulus, G	E10.0
11-20		Bulk unloading modulus, K_{μ}	E10.0
21-30		First yield function constant, a_0	E10.0
31-40		Second yield function constant, a_1	E10.0
41-50		Third yield function constant, a_2	E10.0
51-60		Pressure cutoff for tensile fracture, p_{cut}	E10.0
1-10	Card 4	First (most tensile) tabulated volumetric strain, ε_{v1} (see Figure 4)	E10.0
11-20		First tabulated pressure, p_1 ($p_1 < 0$)	E10.0
21-30		Second tabulated volumetric strain, $\varepsilon_{\nu 2}$	E10.0
31-40		Second tabulated pressure, p_2	E10.0
1-10	Card 5	Third tabulated volumetric strain, ε_{v3}	E10.0
11-20		Third tabulated pressure, p_3	E10.0
21-30		Fourth tabulated volumetric strain, ε_{v4}	E10.0
31-40		Fourth tabulated pressure, p_4	· E10.0
1-10	Card 6	Fifth tabulated volumetric strain, $\mathcal{E}_{\nu 3}$	E10.0
11-20		Fifth tabulated pressure, p_3	E10.0
21-30		Sixth tabulated volumetric strain, \mathcal{E}_{v4}	E10.0
31-40		Sixth tabulated pressure, p_4	E10.0
1-10	Card 7	Seventh tabulated volumetric strain, $\varepsilon_{\nu 3}$	E10.0
11-20		Seventh tabulated pressure, p_3	E10.0
21-30		Eighth tabulated volumetric strain, ε_{v4}	E10.0
31-40		Eighth tabulated pressure, p_4	E10.0
1-10	Card 8	Ninth tabulated volumetric strain, ε_{v7}	E10.0
11-20		Ninth tabulated pressure, p_7	E10.0
21-30		Modified elliptical surface flag, AMOD	E10.0

This model is based on the formulation suggested in (Key, 1974).

7

Pressure is positive in compression, and volumetric strain is negative in compression, $\varepsilon_v = ln\left(\frac{V}{V_0}\right)$. Volumetric strain is given by the natural logarithm of the relative volume. The tabulated pressure-volumetric strain data may contain up to nine pairs of points, and must be given in order of *increasing* compression. If the pressure drops below (i.e., becomes more tensile than) the cutoff value p_{cut} , then the pressure is reset to the cutoff value.



The deviatoric perfectly plastic yield function ϕ is defined as

$$\phi = J_2 - [a_0 + a_1 p + a_2 p^2], \qquad (22)$$

where a_0 , a_1 , and a_2 are constants, p is pressure, and J_2 is the second invariant of the deviatoric stress tensor s given by

$$J_2 = \frac{1}{2} s_{ij} s_{ij}.$$
 (23)

The variation of σ_{y_i} as a function of p has three conical forms which depend on the parameter a_2 : elliptic ($a_2 < 0$), parabolic ($a_2 = 0$), or hyperbolic ($a_2 > 0$). These three forms are shown in Figure 5 in terms of J_2 vs. p. Figure 6 shows the corresponding forms in a more familiar engineering form of σ_{yt} vs. p. The elliptic yield function curves back toward the p axis at higher pressures and predicts a softening behavior which is not often observed in test data. If the modified elliptical yield surface flag AMOD is nonzero, then the elliptical yield surface is used up to the point of maximum J_2 . For higher pressures, the yield surface is extended as a von Mises surface. The resulting yield surface is depicted in Figure 5 and Figure 6. This modification yields much improved agreement with a test data for many geologic materials such as concrete, as discussed in (Schwer, Rosinsky, and Day, 1988).



Figure 5 Material type 5 failure surface forms in J_2 vs. p space.



Figure 6 Material type 5 failure surface forms in σ_y vs. p space.

The unmodified elliptic yield surface crosses the $\sigma_y = 0$ at a pressure p_0 , as shown in Figure 6. Thus, p_0 is the maximum pressure at which the unmodified elliptic failure surface may be used. It is easily shown that

$$p_0 = \frac{-a_1 + \sqrt{a_1^2 - 4a_0 a_2}}{2a_2}.$$
 (24)

The modified elliptic yield surface transitions to a von Mises surface at pressures greater than p_{crit} , where

$$p_{crit} = -\frac{a_1}{2a_2}.$$
 (25)

Plastic flow is nonassociative if a_1 or a_2 are nonzero.

On the yield surface, $J_2 = \frac{1}{3}\sigma_y^2$, where σ_y is the uniaxial yield stress. Thus, the yield stress at any pressure p is given by

$$\sigma_{y} = \left[3\left(a_{0} + a_{1}p + a_{2}p^{2}\right)\right]^{\frac{1}{2}}.$$
(26)

There is no strain hardening in this model, so the yield stress is completely determined by the pressure.

To eliminate the pressure dependence of the yield strength, set $a_1 = a_2 = 0$ and $a_0 = \frac{1}{3}\sigma_y^2$. This approach is useful when a von Mises type elastic-plastic model is desired for use with tabulated volumetric data.

This material model is useful as a simple representation of pressure hardening, where the deviatoric yield strength of a material increases as the hydrostatic pressure increases. At any constant pressure, the deviatoric behavior is elastic perfectly-plastic. This model permits a general representation of the material volumetric behavior via a tabulated pressure-volumetric strain curve. This flexibility makes this model useful for representing materials with a stiffening pressure-volume curve, such as porous materials like wood or foam. This model is also often used as a simplified model for concrete or soil.

DYNA3D User Manual

Material Type 6 (Viscoelastic)

Columns		Quantity	Format
1-10	Card 3	Elastic bulk modulus, K	E10.0
1-10	Card 4	Short-time shear modulus, G_0	E10.0
1-10	Card 5	Long-time shear modulus, G_{∞}	E10.0
1-10	Card 6	Time parameter MFLAG.EQ.0.0: decay constant, β MFLAG.EQ.1.0: time relaxation constant, τ	E10.0
1-10	Card 7	Model formulation flag, MFLAG EQ.0.0: standard DYNA3D formulation EQ.1.0: Kelvin viscoelastic formulation	E10.0
	Card 8	Blank	

Two formulations of viscoelasticity are available. In the standard DYNA3D model (Key, 1974), the deviatoric stresses are found from

$$s_{ij} = 2 \int_{0}^{t} G(t-\tau) \dot{e}_{ij} d\tau, \qquad (27)$$

where the shear relaxation behavior is described by

$$G(t) = G_{\infty} + (G_0 - G_{\infty}) e^{-\beta t}$$
(28)

and \dot{e}_{ij} is the deviatoric strain rate. In the Kelvin viscoelastic formulation, the evolution of deviatoric stress is governed by

$$\dot{s}_{ij} + \frac{1}{\tau} s_{ij} = (1 + \delta_{ij}) G_0 \dot{e}_{ij} + (1 + \delta_{ij}) \frac{G_{\infty}}{\tau} \dot{e}_{ij} \quad \text{(no sum)},$$
(29)

followed by a projection back into deviatoric stress space. The model parameters may be related to shear stiffness and damping properties over a frequency range of interest. This model is primarily intended as a simple viscous shear model for use in seismic studies.

The volumetric response is elastic in both formulations, so the pressure is computed from the current volumetric strain using

$$p = -K\varepsilon_{\nu}, \tag{30}$$

where K is the elastic bulk modulus. The viscoelastic model is useful for modeling rate-dependent elastic materials. It is also effectively used whenever viscous disipation is desired with deviatoric (shear) deformations.

Material Type 7 (Blatz - Ko Hyperelastic Rubber)

Columns		Quantity	Format
1-10	Card 3	Shear modulus, G	E10.0
	Card 4	Blank	
•			
•			•
•			. •
	Card 8	Blank	

This model is a hyperelastic model based on the implementation in (Key, 1974), and is appropriate for materials undergoing moderately large strains. In this formulation, the second Piola-Kirchhoff stress t is computed as

$$\tau_{ij} = G\left(\frac{1}{\nu}C_{ij} - V^{\frac{-1}{1-2\nu}}\delta_{ij}\right),$$
(31)

where V is the relative volume, C_{ij} is the right Cauchy-Green strain tensor, and v is Poisson's ratio which is set to .463 internally. The Cauchy stress s is then found from t using

$$s = \frac{1}{J}FtF^{T},$$
(32)

where F is the deformation gradient and J is the Jacobian of the deformation.

The Blatz-Ko hyperelastic model is often used to represent the behavior of rubber at moderately large strains. The Blatz-Ko formulation yields a slightly compressible material. Only one input parameter is required, and therefore this model may be useful when detailed test data is not available for the material of interest. For more sophisticated hyperelastic models for rubber, see the Compressible Mooney-Rivlin Rubber model (Material Type 27) or the Frazer-Nash Rubber model (Material Type 31).

INPUT FORMAT

DYNA3D User Manual

Material Type 8 (High Explosive Burn)

An equation of state must be used with this model.

Columns	<u> </u>	Format	
1-10	Card 3	Detonation velocity, D	E10.0
11-20		Chapman-Jouget pressure, P _{CJ}	E10.0
21-30		Lighting time option, <i>ILOPT</i> EQ.0.0: Programmed burn EQ.1.0: Beta burn	E10.0
	Card 4	Blank	
	•		
	•		•
	•		
	Card 8	Blank	

This model is based on work described in (Giroux, 1973), and is used to model the burning of explosives. The detonation velocity D is the velocity of a detonation or burn front. The Chapman-Jouguet pressure P_{CI} is the maximum pressure realizable in a constant volume adiabatic burn.

During DYNA3D initialization, the lighting time of each element is computed from specified detonation points using the programmed burn option. If detonation points are defined, then the lighting time t_L for an element is computed based on the distance from the center of the element to the nearest detonation point, divided by the detonation velocity D.

Burn fractions are computed to control the release of chemical energy for simulating high explosive detonations. If the "beta burn" option (ILOPT = 1) is selected, then the burn fraction F is computed from

$$F = \beta (1 - V) = \frac{1 - V}{1 - V_{CJ}},$$
(33)

where V is the current relative volume, V_{CJ} is the Chapman-Jouget relative volume, and

$$\beta = \frac{1}{1 - V_{CJ}}.\tag{34}$$

This model is useful to detect initiation due to shock compression of HE.

For the programmed burn option, the burn fraction is computed from

where

$$F = max(F_1, F_2),$$
 (35)

$$F_1 = ((t - t_L)D) / (1.5h)$$
(36)

if $t > t_L$, and $F_1 = 0$ if $t < t_L$, and h is a characteristic dimension of the element under consideration. F_2 is computed from

$$F_2 = \beta (1 - V) = \frac{1 - V}{1 - V_{CJ}}.$$
(37)

If the above equations produce a burn fraction that is greater than one, then it is reset to one.

The burn front propagates by multiplying the pressure computed from an equation-of-state by the current burn fraction,

$$p = F p_{EOS}(V, E), \qquad (38)$$

~

where $p_{EOS}(V, E)$ is the pressure computed from the equation-of-state at the current relative volume V and energy E. High explosives typically have large initial internal energies, E_0 , which yield large pressures as $F \rightarrow 1$.

DYNA3D User Manual

INPUT FORMAT

Material Type 9 (Fluid)

Columns		Quantity	Format
1-10	Card 3	Pressure cutoff, p_{cut} (positive in compression)	E10.0
11-20		Viscosity coefficient, μ	E10.0
	Card 4.	Blank	
	•		•
	•		•
	•		•
	Card 8	Blank	

An equation of state must be used with this model.

The fluid material has no stiffness, and must be used with an equation-of-state. A viscous stress is computed from

$$s_{ij} = \mu \dot{e}_{ij}, \tag{39}$$

where \dot{e}_{ij} is the deviatoric strain rate and s_{ij} is the deviatoric stress.

Materials with no viscosity may reach large distortions under very small shear loads, so a nonzero viscosity should always be used.

The pressure cutoff, p_{cut} , is negative in tension. If the pressure becomes more tensile than p_{cut} , then it is reset to that value. Thus, the pressure cutoff can be interpreted as an approximate model of cavitation. The deviatoric stresses arising from viscous effects are unaffected by the tensile pressure cutoff.

This model is most useful for analyzing structures with contained fluids. Large distortions in the fluid make some free-surface and fluid flow problems more amenable to analysis using other analysis codes employing an Eulerian formulation. Caution should therefore be used when applying this material model in situations where large distortions are expected.

Ĺ

ł.

Material Type 10 (Isotropic-Elastic-Plastic-Hydrodynamic)

An equation of state must be used with this model.

Columns		Quantity	Format
1-10	Card 3	Shear modulus, G	E10.0
11-20		Yield stress, σ_0	E10.0
21-30		Plastic modulus, E_p	E10.0
31-40		Pressure cutoff, p_{cut} (positive in compression) EQ.0.0: cutoff of $-\infty$ is assumed (no cutoff)	
41-50		Linear pressure hardening coefficient, a_1	E10.0
51-60		Quadratic pressure hardening coefficient, a_2	E10.0
61-70		Spall model, <i>ISPALL</i> EQ.0.0: default set to 1.0 EQ.1.0: Pressure limit model EQ.2.0: Maximum principal stress spall criterion EQ.3.0: Hydrostatic tension spall criterion	
1-10	Card 4	Effective plastic strain at failure, $\overline{\varepsilon}_{f}^{p}$ EQ.0.0: no failure	E10.0
1-10	Card 5	First tabulated effective plastic strain, $\overline{\varepsilon}_{1}^{p}$	E10.0
11-20		$\overline{\mathcal{E}}_2^p$	E10.0
•			•
•			•
•		•	•
71-80		$\overline{\mathcal{E}}_8^p$	E10.0
1-10	Card 6	$\overline{\mathcal{E}}_{9}^{p}$	E10.0
•		•	•
•			•
•			•
71-80		$\overline{\mathcal{E}}_{16}^{p}$	E10.0
1-10	Card 7	First tabulated yield stress, σ_{y_1}	E10.0

Columns		······································	Quantity	Format
71-80		σ_{y^8}		E10.0
1-10	Card 8	$\sigma_{_{y9}}$		E10.0
		•		•
•		•		•
•		•		•
71-80		σ_{y16}		E10.0

If a tabulated yield stress vs. effective plastic strain curve is not given (Cards 5-8 are blank), then the initial yield stress σ_0 , plastic hardening modulus E_p , and pressure hardening coefficients a_1 and a_2 are taken from Card 3. In this case, a pressure hardening bilinear stress-strain curve similar to that shown in Figure 3 is obtained with linear isotropic strain hardening ($\beta = 1.0$).

The yield condition can be written

$$\phi = \overline{\sigma} - \sigma_{\gamma} \left(\overline{\varepsilon}^{p}, p \right), \tag{40}$$

where $\overline{\sigma}$ is the effective stress and σ_y is the current yield stress, which may be a function of the effective plastic strain $\overline{\epsilon}^p$ and pressure p. The effective stress $\overline{\sigma}$ is given by

$$\overline{\sigma} = \left(\frac{3}{2}s_{ij}s_{ij}\right)^{\frac{1}{2}},\tag{41}$$

where s_{ii} is the deviatoric stress tensor.

The hardening law has the form

$$\sigma_{y} = \sigma_{0} + E_{p}\overline{\varepsilon}^{p} + (a_{1} + a_{2}p)\hat{p}, \qquad (42)$$

where p is the pressure (positive in compression), and \hat{p} is the tension-limited pressure found from

$$\hat{p} = max(p,0).$$
 (43)

The effective plastic strain $\overline{\varepsilon}^{P}$ is given by

$$\overline{\varepsilon}^{p} = \int_{0}^{t} d\overline{\varepsilon}^{p}, \qquad (44)$$

where the incremental effective plastic strain $d\overline{\epsilon}^{p}$ is found from the incremental plastic strain tensor $d\varepsilon_{ij}$ as



$$d\overline{\varepsilon}^{p} = \left(\frac{2}{3}d\overline{\varepsilon}^{p}_{ij}d\overline{\varepsilon}^{p}_{ij}\right)^{\frac{1}{2}}.$$
(45)

The plastic modulus can be related to Young's modulus E and the tangent modulus E_T using

$$E_p = \frac{EE_T}{E - E_T}.$$
(46)

The plastic modulus E_P is the slope of the inelastic portion of the effective stress $\overline{\sigma}$ vs. effective plastic strain $\overline{\epsilon}^P$ curve, and the tangent modulus E_T is the slope of the inelastic part of a uniaxial stress vs. strain curve (or equivalently, the effective stress vs. effective strain curve).

If tabulated values of yield stress vs. effective plastic strain are specified on Cards 5-8, a nonlinear strain hardening curve like that shown in Figure 7 may be defined. In this case the plastic hardening modulus and pressure hardening coefficients input on Card 3 are not used, and the yield stress is given as

$$\sigma_{y} = f(\overline{\varepsilon}^{\nu}), \qquad (47)$$

where $f(\overline{\epsilon}^p)$ is interpolated from the specified yield stress vs. effective plastic strain curve. Any number of points, from 2 to 16, may be used to define the hardening curve. This option permits additional detail to be included in the nonlinear strain hardening law, but pressure hardening is not modeled with this approach.

Sec. 2. 4

÷ 1

A choice of three spall models is offered to represent material splitting, cracking, and failure under tensile loads. The pressure limit model, (ISPALL = 1), limits the hydrostatic tension to the specified value, p_{cut} . If pressures more tensile than this limit are calculated, the pressure is reset to p_{cut} . This option is not strictly a spall model, since the deviatoric stresses are unaffected by the pressure reaching the tensile cutoff, and the pressure cutoff value p_{cut} remains unchanged throughout the analysis. The maximum principal stress spall model, (ISPALL = 2), detects spall if the maximum (most tensile) principal stress σ_{max} exceeds the limiting value $-p_{cut}$. Note that the negative sign is required because p_{cut} is measured positive in compression, while σ_{max} is positive in tension. Once spall is detected with this model, the deviatoric stresses are set to zero, and no hydrostatic tension (p < 0) is permitted. If tensile pressures are calculated, they are reset to 0 in the spalled material. Thus, the spalled material behaves as a rubble or incohesive material. The hydrostatic tension spall model, (ISPALL = 3), detects spall if the pressure becomes more tensile than the specified limit, p_{cut} . Once spall is detected the deviatoric stresses are set to zero, and the pressure is required to be compressive. If hydrostatic tension (p < 0) is subsequently calculated, the pressure is reset to 0 for that element.

An effective plastic strain failure criterion is provided in this model. If the SAND database is activated (see Control Card 3 in section 4.1 on page 41), then the element is deleted from the calculation when the effective plastic strain exceeds the prescribed value $\overline{\epsilon}_{f}^{p}$.

This model is applicable to a wide range of materials, including those with pressure-dependent yield behavior. The use of 16 points in the yield stress vs. effective plastic strain curve allows complex post-yield hardening behavior to be accurately represented. In addition, the incorporation of an equation of state permits accurate modeling of diverse volumetric behavior. The spall model options permit incorporation of material failure, fracture, and disintegration effects under tensile loads.

/

Material Type 11 (Steinberg-Guinan High Rate Elastic-Plastic)

An equation of state must be used with this model.

Columns		Quantity	<u>Format</u>
1-10	Card 3	Shear modulus constant, G_0	E10.0
11-20		Yield stress constant, σ_0	E10.0
21-30		Strain hardening law constant, β	E10.0
31-40		Strain hardening exponent, n	E10.0
41-50		Initial plastic strain, γ_i	E10.0
1-10	Card 4	Yield stress work hardening limit, σ_m	E10.0
11-20		Shear modulus pressure constant, b	E10.0
21-30		Yield stress pressure constant, b'	E10.0
31-40		Energy coefficient, h	E10.0
41-50		Energy exponential coefficient, f	E10.0
1-10	Card 5	Atomic weight, A (if $A = 0, R'$ must be defined)	E10.0
11-20		Melting temperature constant, T_{mo}	E10.0
21-30		Thermodynamic gamma, γ_0	E10.0
31-40		Thermodynamic constant, a	E10.0
41-50	,	Pressure cutoff, p_{cut}	E10.0
1-10	Card 6	Spall model, <i>ISPALL</i> EQ.0.0: default set to 2.0 EQ.1.0: Pressure limit model EQ.2.0: Maximum principal stress spall criterion EQ.3.0: Hydrostatic tension spall criterion	E10.0
11-20		R' (if $R' \neq 0$, the atomic weight A is not used)	E10.0
21-30		Effective plastic strain at failure, $\overline{\varepsilon}_{f}^{p}$ EQ.0.0: no failure	E10.0
41-50		Cold compression energy polynomial flag, <i>IVAR</i> EQ.0.0: Polynomial coefficients given or fit in term EQ.1.0: Polynomial coefficients given or fit in term	E10.0 s of η s of μ
51-60		Optional minimum limit for energy fit Input η_{min} if $IVAR = 0$ Input μ_{min} if $IVAR = 1$	E10.0

Columns		Quantity	Format
61-70		Optional maximum limit for energy fit Input η_{max} if $IVAR = 0$ Input μ_{max} if $IVAR = 1$	E10.0
1-16	Card 7	First cold compression polynomial coefficient, EC_0	E16.0
17-32		EC_1	E16.0
33-48		EC_2	E16.0
49-64		EC_3	E16.0
65-80		EC_4	E16.0
1-16	Card 8	ECs	E16.0
17-32		EC ₆	E16.0
33-48		EC_7	E16.0
49-64		EC_8	E16.0
65-80		EC,	E16.0

The formulation of this model is described by Steinberg and Guinan (1978), and some notes on the implementation are given in (Woodruff, 1973).

In terms of the foregoing input parameters, we define the shear modulus, G, before the material melts as:

$$G = G_0 \left[1 + bpV^{\frac{1}{3}} - h\left(\frac{E_i - E_c}{3R'} - 300\right) \right] e^{-\frac{JE_i}{E_m - E_i}}$$
(48)

where p is the pressure, V is the relative volume, E_i is the current energy, E_c is the cold compression energy, and E_m is the melting energy. The cold compression energy is calculated using

$$E_{c}(x) = \int_{0}^{x} p dx - \frac{900R'exp(ax)}{(1-x)^{2(\gamma_{0}-a-\frac{1}{2})}},$$
(49)

where x = 1 - V. The melting energy is found from the cold compression energy and the melting temperature using

$$E_{m}(x) = E_{c}(x) + 3R'T_{m}(x), \qquad (50)$$

where the melting temperature T_m is given by

$$T_{m}(x) = \frac{T_{mo}exp(2ax)}{V^{2(\gamma_{0}-a-\frac{1}{3})}}$$
(51)

and T_{mo} is the melting temperature at the initial density, ρ_0 .

In the above equations, R' is defined by

$$R' = \frac{R\rho}{A},\tag{52}$$

where R is the universal gas constant and A is the atomic weight. Note that if R' is not defined, DYNA3D computes it with R in the cm-gram-microsecond system of units. Thus, this option should not be used unless the entire model is defined in the cm-gram-microsecond second system of units.

If E_m exceeds E_i (i.e., the material has not melted), then the yield strength σ_y is given by:

$$\sigma_{y} = \sigma_{0}^{\prime} \left[1 + b^{\prime} p V^{\frac{1}{3}} - h \left(\frac{E_{i} - E_{c}}{3R^{\prime}} - 300 \right) \right] e^{\frac{j E_{i}}{E_{m} - E_{i}}}.$$
(53)

The work-hardened yield stress σ'_0 is found from the initial yield stress σ_0 and the accumulated effective plastic strain $\overline{\epsilon}^p$ using the hardening law

$$\sigma'_{0} = \sigma_{0} \left[1 + \beta \left(\gamma_{i} + \overline{\varepsilon}^{p} \right) \right], \tag{54}$$

where γ_i is the initial plastic strain. If the work-hardened yield stress σ_0 exceeds the limiting value σ_m , then σ'_0 is reset to σ_m . After the materials melts $(E_i > E_m)$, the yield stress σ_y and shear modulus G are reset to one half their initial value.

The evaluation of the cold compression energy $E_c(x)$ using (49) is too expensive to perform at each step of a calculation. As an approximation, many codes (including DYNA3D) use a polynomial to interpolate cold compression energy data during execution. If IVAR = 0, then the independent variable is chosen as η , and the polynomial takes the form

$$E_c = \sum_{i=0}^{9} E C_i \eta^i, \tag{55}$$

and if IVAR = 1, then the independent variable is μ and the polynomial takes the form

$$E_{c} = \sum_{i=0}^{9} E C_{i} \mu^{i}.$$
 (56)

Note that the density and compression variables are related by

$$x = 1 - V = \frac{\mu}{\mu + 1} = 1 - \frac{1}{\eta}.$$
 (57)

DYNA3D User Manual

For either form of the cold compression energy polynomial, the coefficients EC_0 through EC_9 may be defined in the input if they are known. If they are not specified in the input, DYNA3D will fit the cold compression energy to a ten term polynomial expansion using a least squares method. If DYNA3D performs this fit, the exact cold compression energy is compared with the cold compression energy found using the fitted polynomial at selected values of x, and the results printed in the "hsp" output file. These results should be examined closely to verify that a reasonably accurate polynomial fit has been obtained.

1.5 4 5.

A choice of three spall models is offered to represent material splitting, cracking, and failure under tensile loads. The pressure limit model, (ISPALL = 1), limits the hydrostatic tension to the specified value, p_{cut} . If pressures more tensile than this limit are calculated, the pressure is reset to p_{cut} . This option is not strictly a spall model, since the deviatoric stresses are unaffected by the pressure reaching the tensile cutoff, and the pressure cutoff value p_{cut} remains unchanged throughout the analysis. The maximum principal stress spall model, (ISPALL = 2), detects spall if the maximum (most tensile) principal stress σ_{max} exceeds the limiting value $-p_{cut}$. Note that the negative sign is required because p_{cut} is measured positive in compression, while σ_{max} is positive in tension. Once spall is detected with this model, the deviatoric stresses are set to zero, and no hydrostatic tension (p < 0) is permitted. If tensile pressures are calculated, they are reset to 0 in the spalled material. Thus, the spalled material behaves as a rubble or incohesive material. The hydrostatic tension spall model, (ISPALL = 3), detects spall if the pressure becomes more tensile than the specified limit, p_{cut} . Once spall is detected the deviatoric stresses are set to zero, and the pressure is required to be compressive. If hydrostatic tension (p < 0) is subsequently calculated, the pressure is reset to 0 for that element.

An effective plastic strain failure criterion is provided in this model. If the SAND database is activated by setting the appropriate flag (on Control Card 3 described in section 4.1 on page 41) then the element is deleted from the calculation when the effective plastic strain exceeds the prescribed value $\bar{\epsilon}_{f}^{p}$.

The Steinberg-Guinan model is applicable to metals at high strain rates (near $10^5 s^{-1}$), where the enhancement of the yield stress due to strain rate effects has reached a limiting value and compression heating effects are becoming important. Model predictions may become substantially less accurate at low strain rates (below $10^3 s^{-1}$). Examples of problems in this high strain rate regime include some explosive forming operations, metal penetration problems, and the simulation of high velocity ballistic impact.

Material Type 12 (Isotropic-Elastic-Plastic)

Columns		Quantity	Format
1-10	Card 3	Shear modulus, G	E10.0
11-20		Yield stress, σ_0 (see Figure 3)	E10.0
21-30		Tangent modulus, E_T	E10.0
1-10	Card 4	Bulk modulus, K	E10.0
	Card 5	Blank	
	Card 8	Blank	

This model produces bilinear elastoplastic behavior which is identical to Material Type 3 with $\beta = 1.0$, but is slightly faster and requires less storage. Note also that the input quantities vary slightly from those of Material Type 3.

The theoretical foundations of this model are similar to those described for Material Type 3. The numerical algorithms are based on those described in (Krieg and Key, 1976).

DYNA3D User Manual

Columns		Quantity	Format
1-10	Card 3	Shear modulus, G	E10.0
11-20		Yield stress, σ_0 (see Figure 3)	E10.0
21-30		Tangent modulus, E_T	E10.0
31-40		Effective plastic strain at failure, $\overline{\varepsilon}_{f}^{p}$	E10.0
41-50		Failure pressure, p_f (positive in compression)	E10.0
1-10	Card 4	Bulk modulus, K	E10.0
	Card 5	Blank	
	Card 8	Blank	

Material Type 13 (Elastic-Plastic with Failure)

This model produces bilinear elastoplastic behavior identical to Material Type 12, except that two failure criteria have been incorporated. Before failure occurs, this model will give exactly the same behavior as Material Type 12, or Material Type 3 with $\beta = 1.0$. The theoretical formulation for the elastoplastic models is described in detail for Material Type 3.

Two failure criteria have been implemented in this model: an effective plastic strain based criterion, and a hydrostatic tension based criterion. If the effective plastic strain reaches the failure value \overline{e}_{f}^{p} , then all shear stresses are set to zero, and the stress state becomes hydrostatic and remains hydrostatic for the duration of the analysis. This hydrostatic stress state may include tensile values which are admissible to the pressure criterion. The pressure criterion is checked even after the element fails due to the strain criterion, and if the failure pressure is subsequently exceeded due to hydrostatic tension, then the element fails completely and can support only hydrostatic compressive stresses. If the pressure in the element becomes more tensile (negative) than the failure pressure p_{f} , then all stresses are set to zero and the element can never again support any hydrostatic tension at all, and can never again support any shear stresses. The only possible stress state in such an element is pure hydrostatic compression. An effective plastic strain failure criterion is provided in this model. If the SAND database is activated by setting the appropriate flag (on Control Card 3 described in section 4.1 on page 41) then the element is deleted from the calculation when the effective plastic strain exceeds the prescribed value $\overline{\varepsilon}_{f}^{p}$.

This model is useful as an approximate representation of ductile (strain-based) or brittle (stressbased) failure in elastic or elastoplastic materials. After a material fails completely, its behavior is similar to that of a fluid in that it can only support purely hydrostatic compressive stresses.
DYNA3D User Manual

Material Type 14 (Soil and Crushable Foam with Failure)

÷.

Columns		Quantity	Format
1-10	Card 3	Shear modulus, G	E10.0
11-20		Bulk unloading modulus, K_u	E10.0
21-30		First yield function constant, a_0	E10.0
31-40		Second yield function constant, a_1	E10.0
41-50		Third yield function constant, a_2	E10.0
51-60		Failure pressure for hydrostatic tension criterion, p_f (used only if <i>IFLAG</i> = 0)	E10.0
61-70		Failure criterion flag, <i>IFLAG</i> EQ.0.0: Hydrostatic tension criterion EQ.1.0: Maximum principal stress criterion	E10.0
71-80		Maximum principal stress for failure criterion, σ_f , (used only if <i>IFLAG</i> = 1)	E10.0
1-10	Card 4	First (most tensile) tabulated volumetric strain, ε_{v1} (see Figure 4)	E10.0
11-20		First tabulated pressure, p_1	E10.0
21-30		Second tabulated volumetric strain, $\varepsilon_{\nu 2}$	E10.0
31-40		Second tabulated pressure, p_2	E10.0
1-10	Card 5	Third tabulated volumetric strain, $\mathcal{E}_{\nu 3}$	E10.0
11-20		Third tabulated pressure, p_3	E10.0
21-30		Fourth tabulated volumetric strain, ε_{v4}	E10.0
31-40		Fourth tabulated pressure, p_4	E10.0
1-10	Card 6	Fifth tabulated volumetric strain, \mathcal{E}_{v3}	E10.0
11-20		Fifth tabulated pressure, p_3	E10.0
21-30		Sixth tabulated volumetric strain, $\varepsilon_{\nu 4}$	E10.0
31-40		Sixth tabulated pressure, p_4	E10.0
1-10	Card 7	Seventh tabulated volumetric strain, ε_{ν_3}	E10.0
11-20		Seventh tabulated pressure, p_3	E10.0
21-30		Eighth tabulated volumetric strain, ε_{v4}	E10.0
31-40		Eighth tabulated pressure, p_4	E10.0

Columns	·	Quantity	Format
1-10	Card 8	Nineth tabulated volumetric strain, $\mathcal{E}_{\nu 7}$	E10.0
11-20		Nineth tabulated pressure, p_7	E10.0

Before failure, the behavior of this model is identical to Material Type 5.

Pressure is positive in compression, and volumetric strain is negative in compression. Volumetric strain is given by the natural log of the relative volume. The tabulated pressure-volumetric strain data may contain up to nine pairs of points, and must be given in order of *increasing* compression.

The deviatoric perfectly plastic yield function ϕ is defined as

$$\phi = J_2 - [a_0 + a_1 p + a_2 p^2], \qquad (58)$$

where a_0 , a_1 , and a_2 are constants, p is pressure, and J_2 is the second invariant of the deviatoric stress tensor s given by

$$J_2 = \frac{1}{2} s_{ij} s_{ij}.$$
 (59)

Plastic flow is nonassociative if a_1 or a_2 is nonzero.

On the yield surface $J_2 = \frac{1}{3}\sigma_y^2$, where σ_y is the uniaxial yield stress. Thus, the yield stress at any pressure p is given by

$$\sigma_{y} = \left[3\left(a_{0} + a_{1}p + a_{2}p^{2}\right)\right]^{\frac{1}{2}}.$$
(60)

There is no strain hardening in this model, so the yield stress is completely determined by the pressure.

To eliminate the pressure dependence of the yield strength, set $a_1 = a_2 = 0$ and $a_0 = \frac{1}{3}\sigma_y^2$. This approach is useful when a pressure-independent, elastic perfectly-plastic model with failure is desired for use with tabulated volumetric data.

This model allows one of two failure criteria to be used: a hydrostatic tension criterion (IFLAG = 0) or a maximum principal stress criterion (IFLAG = 1). The hydrostatic tension criterion detects failure if the pressure in the element is less than the specified failure pressure, p_f (recall that pressure is positive in compression). Once failure is detected by this criterion, all stresses in the element are set to zero, and the element can never again sustain any hydrostatic

tension or shear stresses. The element can only support purely compressive hydrostatic stresses, and therefore behaves like a fluid. The maximum principal stress criterion detects failure if the largest (most tensile) principal stress exceeds the specified failure stress, σ_f . Once failure is detected by the maximum principal stress criterion, the deviatoric stresses are gradually reduced to zero over a few time steps, and the hydrostatic pressure is left unchanged. This gradual reduction of the deviatoric stresses has been found to introduce less noise into some problems than an abrupt reduction of the deviatoric stresses to zero.

A failure criterion is provided in this model which activates when the effective plastic strain exceeds the prescribed value $\overline{\epsilon}_{f}^{p}$.



This material model is useful as a simple representation of pressure hardening, where the deviatoric yield strength of a material increases as the hydrostatic pressure increases. At any constant pressure, the deviatoric behavior is elastic perfectly-plastic. A failure criterion based on either hydrostatic tension or maximum principal stress may be used to approximately represent the degradation of material strength under extreme loads. This model permits a general representation of the volumetric behavior of the material via a tabulated pressure-volumetric strain curve. This flexi-

bility makes Material Type 14 useful for representing materials with a stiffening pressure-volume curve. Such stiffening behavior is frequently found in porous materials such as wood or foam. This model is also often used as a simplified model for concrete or soil.

DYNA3D User Manual

Material Type 15 (Johnson/Cook Elastic-Plastic)

· · ·

An equation of state must be used with this model for solid elements.

S. Carton

Columns		Quantity	Format
1-10	Card 3	Shear modulus, G	E10.0
11-20		Yield stress constant, A	E10.0
21-30		Strain hardening coefficient, B	E10.0
31-40		Strain hardening exponent, n	E10.0
41-50		Strain rate dependence coefficient, C	E10.0
51-60		Temperature dependence exponent, m	E10.0
61-70		Melt temperature, T_m , in degrees Kelvin	E10.0
71-80		Room temperature, T_r , in degrees Kelvin	E10.0
1-10	Card 4	Reference strain rate, $\dot{\varepsilon}_0$	E10.0
11-20		Specific heat, c_{v}	E10.0
21-30		Pressure cutoff, p_{cut} , or failure stress, σ_m	E10.0
31-40		Spall model, <i>ISPALL</i> EQ.1.0: Pressure limit model (default) EQ.2.0: Maximum principal stress spall criterion EQ.3.0: Hydrostatic tension spall criterion	
41-50	·	Plastic strain iteration flag, <i>ITER</i> EQ.0.0: fast approx. solution for plastic strain (defaul EQ.1.0: accurate iterative solution for plastic strain (More expensive than default.)	E10.0 t)
1-10	Card 5	First failure parameter, D_1	E10.0
11-20		D_2	E10.0
21-30		D_3	E10.0
31-40		D_4	E10.0
41-50		D_5	E10.0
•	Card 6	Blank	
•	Card 7	Blank	
1-10	Card 8	Young's modulus, E (shells only, blank for solid elements)	E10.0
11-20		Poisson's ratio, v (shells only, blank for solid elements)	E10.0
21-30		Time step size for element deletion, Δt_{crit} (shells only)	E10.0

The formulation of this model is described and constants for many materials are given in (Johnson and Cook, 1983).

The yield stress is written as

$$\sigma_{y} = [A + B(\bar{\varepsilon}^{p})^{n}] [1 + Cln(\dot{\varepsilon}^{*})] [1 - (T^{*})^{m}], \qquad (61)$$

where A, B, C, n and m are input constants, $\overline{\varepsilon}^{p}$ is the effective plastic strain, $\dot{\varepsilon}^{*}$ is the nondimensional strain rate, and T^{*} is the homologous temperature. The effective plastic strain $\overline{\varepsilon}^{p}$ is given by

$$\overline{\varepsilon}^{p} = \int_{0}^{t} d\overline{\varepsilon}^{p}, \tag{62}$$

where the incremental effective plastic strain $d\overline{\varepsilon}^{p}$ is found from the incremental plastic strain tensor $d\varepsilon_{ii}$ as

$$d\overline{\varepsilon}^{p} = \left(\frac{2}{3}d\overline{\varepsilon}^{p}_{ij}d\overline{\varepsilon}^{p}_{ij}\right)^{\frac{1}{2}}.$$
(63)

The nondimensional strain rate $\dot{\varepsilon}^*$ is calculated from

$$\dot{\varepsilon}^* = \frac{\dot{\overline{\varepsilon}}^p}{\dot{\varepsilon}_0} \tag{64}$$

where $\overline{\epsilon}^{p}$ is the effective plastic strain rate and $\dot{\epsilon}_{0}$ is the reference strain rate defined in the input. The homologous temperature T^{*} is the ratio of the current temperature to the melting temperature when both are expressed in degrees Kelvin. Temperature change in this model is computed assuming adiabatic conditions, i.e., no heat transfer between elements. This is usually a good assumption since transient dynamic problems typically occur over such a short time interval that the actual heat transfer is negligible. Heat is generated in an element by plastic work, and the resulting temperature rise is computed using the specific heat for the material.

Material constants for a variety of materials are provided in (Johnson and Cook, 1983).

Due to the nonlinearity in the dependence of the yield stress on plastic strain, an accurate value of the yield stress requires expensive iteration for calculation of the increment in plastic strain. However, by using a Taylor series expansion with linearization about the current state, σ_y can be approximated with sufficient accuracy to avoid iteration and achieve optimum execution speed.

This implementation of the Johnson-Cook model also contains a damage model. The strain at fracture ε_f is given by

$$\varepsilon_{f} = [D_{1} + D_{2}exp(D_{3}\sigma^{*})] [1 + D_{4}ln(\dot{\varepsilon}^{*})] [1 + D_{5}T^{*}]$$
(65)

where σ^* is the ratio of pressure divided by effective stress

$$\sigma^* = \frac{p}{\overline{\sigma}},\tag{66}$$

and effective stress $\overline{\sigma}$ is found from

$$\overline{\sigma} = \left(\frac{3}{2}s_{ij}s_{ij}\right)^{\frac{1}{2}}.$$
(67)

Note that this definition of σ^* may be reversed in sign from convention in the original publications of Johnson and Cook; the sign of D_3 should be chosen carefully.

Fracture occurs when the damage parameter D exceeds the value of 1. The evolution of the damage parameter is given by

$$D = \sum \frac{\Delta \overline{\varepsilon}^p}{\varepsilon_f},\tag{68}$$

where the summation is performed over all time steps in the analysis. When fracture occurs, all stresses are set to zero and remain zero for the rest of the calculation.

A choice of three spall models is offered to represent material splitting, cracking, and failure under tensile loads. The pressure limit model, (ISPALL = 1), limits the hydrostatic tension to the specified value, p_{cut} . If pressures more tensile than this limit are calculated, the pressure is reset to p_{cut} . This option is not strictly a spall model, since the deviatoric stresses are unaffected by the pressure reaching the tensile cutoff, and the pressure cutoff value p_{cut} remains unchanged throughout the analysis. The maximum principal stress spall model, (ISPALL = 2), detects spall if the maximum (most tensile) principal stress σ_{max} exceeds the limiting value σ_m . Once spall is detected with this model, the deviatoric stresses are set to zero, and no hydrostatic tension (p < 0) is permitted. If tensile pressures are calculated, they are reset to 0 in the spalled material. Thus, the spalled material behaves as a rubble or incohesive material. The hydrostatic tension spall model, (ISPALL = 3), detects spall if the pressure becomes more tensile than the specified limit, p_{cut} . Once spall is detected, the deviatoric stresses are set to zero and the pressure is required to be compressive. If hydrostatic tension (p < 0) is calculated, then the pressure is reset to 0 for that element.

In addition to the above failure criterion, this model also supports a shell element deletion criterion based on the maximum stable time step size for this element, Δt_{max} . Generally, Δt_{max} goes down as the element becomes more distorted. To assure stability of time integration, the global DYNA3D time step is the *minimum* of the Δt_{max} values calculated for all elements in the model. Using this option allows the selective deletion of elements whose time step Δt_{max} has fallen below the specified minimum time step, Δt_{crit} . Elements which are severely distorted often indicate that material has failed and supports little load, but these same elements may have very small time steps and therefore control the cost of the analysis. This option allows these highly distorted elements to be deleted from the calculation, and therefore the analysis can proceed at a larger time step, and thus at a reduced cost. Deleted elements do not carry any load, and are deleted from all applicable slide surface definitions. Clearly, this option must be judiciously used to obtain accurate results at a minimum cost.

Material Type 15 is applicable to the high rate deformation of many materials, including most metals. Unlike the Steinberg-Guinan model (Material Type 11), the Johnson-Cook model remains valid down to lower strain rates, and even into the quasistatic regime. Typical applications include explosive metal forming, ballistic penetration, and impact.

DYNA3D User Manual

INPUT FORMAT

Material Type 16 (Concrete/Geological Material)

-

• 2.55

This model must be used with equation of state type 8, 9, or 11.

<u>Columns</u>	. <u> </u>	Quantity	Format
1-10	Card 3	 First elastic constant Poisson's ratio, v, for constant v model Negative of shear modulus, -G, for constant G mode 	E10.0
11-20		Maximum principal ștress at failure, $\sigma_{\scriptscriptstyle cut}$	E10.0
21-30		Cohesion, a_0	E10.0
31-40		Pressure hardening coefficient, a_1	E10.0
41-50		Pressure hardening coefficient, a_2	E10.0
51-60		Damage scaling factor, b_1	E10.0
61-70		Cohesion for failed material, a_{0f}	E10.0
71-80		Pressure hardening coefficient for failed material, a_{if}	E10.0
1-10	Card 4	Percent reinforcement, $\hat{f}_r (0 \le \hat{f}_r \le 100\%)$	E10.0
11-20		Elastic modulus for reinforcement, E_r	E10.0
21-30		Poisson's ratio for reinforcement, v_r	E10.0
31-40		Initial yield stress, σ_0	E10.0
41-50		Tangent modulus, E_T	E10.0
51-60	•	Load curve giving rate sensitivity for principal material, N_1	E10.0
61-70		Load curve giving rate sensitivity for reinforcement, N_2	E10.0
1-10	Card 5	First tabulated effective plastic strain, $\overline{\varepsilon}_{1}^{p}$, or pressure, p_{1}	E10.0
			•
•			•
		•	•
71-80		$\overline{\varepsilon}_{s}^{p}$ or p_{s}	E10.0
1-10	Card 6	$\overline{\varepsilon}_{9}^{p}$ or p_{9}	E10.0
•			•
•			•
•			•
71-80		$\overline{\varepsilon}_{16}^{p}$ or p_{16}	E10.0

Columns		Quantity	Format
1-10	Card 7	First tabulated yield stress, σ_{y_1}	E10.0
•		•	
•		•	•
•		•	
71-80		σ_{y8}	E10.0
1-10	Card 8	σ_{y9}	E10.0
•		•	•
•			•
•		•	•
71-80		$\sigma_{_{y16}}$	E10.0

Material Type 16 was developed to give concrete and geological material modeling capabilities to DYNA3D. It can be used in two major modes - a fairly simple tabular pressure-dependent yield surface, and a potentially complex model featuring two yield versus pressure functions with various means of migrating from one curve to the other. For both modes, load curve N_1 is taken to be a strain rate multiplier for the yield strength.

Note that this model must be used with Equation-of-State type 8, 9 or 11.

Response Mode I. Tabulated Yield Stress Versus Pressure

This mode is well suited for implementing standard geologic models like the Mohr-Coulomb yield surface with a Tresca limit, as shown in Figure 9. Examples of converting conventional triaxial compression data to this type of model are found in (Desai and Siriwardane, 1984). Note that under conventional triaxial compression conditions, the DYNA3D input corresponds to an ordinate of

 $\sigma_1 - \sigma_3$ rather than the more widely used $\frac{\sigma_1 - \sigma_3}{2}$, where σ_1 is the maximum principal stress and σ_3 is the minimum principal stress.

Using Material Type 16 combined with Equation-of-State Type 9 (saturated) or Type 11 (air filled porosity), this approach has been very successfully used to model ground shocks and soil/structure interactions at pressures up to 100 kbar (approximately 1.5×10^6 psi).

To invoke Mode I of this model, set a_0 , a_1 , a_2 , b_1 , a_{0f} , and a_{1f} to zero. The tabulated values of pressure should then be specified on cards 5 and 6, and the corresponding values of yield stress should be specified on cards 7 and 8. The parameters relating to reinforcement properties, initial yield stress, and tangent modulus are not used in this response mode, and should be set to zero.

Call & Spint

Simple tensile failure

Note that a_{1f} is reset internally to 1/3 even though it is input as zero; this defines a failed material curve of slope 3p, where p denotes pressure (positive in compression). In this case the yield strength is taken from the tabulated yield vs. pressure curve until the maximum principal stress (σ_1) in the element exceeds the tensile cut-off (σ_{cut}) . For every time step that $\sigma_1 > \sigma_{cut}$ the yield strength is scaled back by a fraction of the distance between the two curves until after 20 time steps the yield strength is defined by the failed curve. The only way to inhibit this feature is to set σ_{cut} arbitrarily large.



Response Mode II. Two Curve Model with Damage and Failure

This approach uses two yield versus pressure curves of the form

$$\sigma_{y} = a_{0} + \frac{p}{a_{1} + a_{2}p}.$$
(69)

The upper curve is best described as the maximum yield strength curve and the lower curve is the failed material curve. There are a variety of ways of moving between the two curves, and each is discussed below.



Mode II.A: Simple tensile failure

Define a_0 , a_1 , a_2 , a_{0f} and a_{1f} , set b_1 to zero, and leave cards 5 through 8 blank. In this case the yield strength is taken from the maximum yield curve until the maximum principal stress (σ_1) in the element exceeds the tensile cut-off (σ_{cut}). For every time step that $\sigma_1 > \sigma_{cut}$ the yield strength is scaled back by a fraction of the distance between the two curves until after 20 time steps the yield strength is defined by the failed curve.

Mode II.B: Tensile failure plus plastic strain scaling

.

Define a_0 , a_1 , a_2 , a_{0f} and a_{1f} , set b_1 to zero, and use cards 5 through 8 to define a scale factor, η , versus effective plastic strain. DYNA3D evaluates η at the current effective plastic strain and then calculates the yield stress as

$$\sigma_{yield} = \sigma_{failed} + \eta \left(\sigma_{max} - \sigma_{failed} \right), \tag{70}$$

where σ_{max} and σ_{failed} are found as shown in Figure 10.

This yield strength is then subject to scaling for tensile failure as described above. This type of model allows the description of a strain hardening and/or softening material such as concrete.

Mode II.C: Tensile failure plus damage scaling

The change in yield stress as a function of plastic strain arises from physical mechanisms such as internal cracking, and the extent of this cracking is affected by the hydrostatic pressure when the cracking occurs. This mechanism gives rise to the "confinement" effect on concrete behavior. To account for this phenomenon a "damage" function was defined and incorporated into Material Type 16. The damage function is given the form

$$\lambda = \int_{0}^{\overline{\varepsilon}^{p}} \frac{d\overline{\varepsilon}^{p}}{\left(1 + \frac{p}{\sigma_{cut}}\right)^{b_{1}}}$$
(71)

Define a_0 , a_1 , a_2 , a_{0f} , a_{1f} , and b_1 . Cards 5 through 8 now give η as a function of λ and scale the yield stress as

$$\sigma_{yield} = \sigma_{failed} + \eta \left(\sigma_{max} - \sigma_{failed} \right) \tag{72}$$

and then apply any tensile failure criteria.

Mode II Concrete Model Options

Material Type 16 in Mode II provides for the automatic internal generation of a simple "generic" model for concrete. If a_0 is negative, then σ_{cut} is assumed to be the unconfined concrete compressive strength (f_c') and $-a_0$ is assumed to be a conversion factor from DYNA pressure units to psi. In this case the parameter values generated internally are:

$$\sigma_{cut} = 1.7 \left(\frac{(f_c^{\prime})^2}{-a_0} \right)^{1/3}$$
(73)

$$a_0 = \frac{f_c^\prime}{4} \tag{74}$$

$$a_1 = \frac{1}{3} \tag{75}$$

$$a_2 = \frac{1}{3f_c'} \tag{76}$$

$$a_{0f} = 0$$
 (77)

$$a_{1f} = 0.385$$
 (78)

Note that these a_{0f} and a_{1f} defaults will be overridden by nonzero entries on Card 3. If plastic strain or damage scaling is desired, Cards 5 through 8 and b_1 should be specified in the input. When a_0 is input as a negative quantity, the Equation-of-State can be given as 0 and a trilinear EOS Type 8 model will be automatically generated from the unconfined compressive strength and Poisson's ratio. The EOS 8 model is a simple pressure versus volumetric strain model with no internal energy terms, and should give reasonable results for pressures up to 5 kbar (approximately 75,000 psi).

Mixture model

A reinforcement fraction, \hat{f}_r , can be defined along with properties of the reinforcement material. The bulk modulus, shear modulus, and yield strength are then calculated from a simple mixture rule. This feature is *experimental* and should be used with caution. It gives an isotropic effect in the material instead of the true anisotropic material behavior. A reasonable approach would be to use mixture elements only where the reinforcing exists and plain elements elsewhere. When the mixture model is being used, the strain rate multiplier for the principal material is taken from load curve N_1 and the multiplier for the reinforcement is taken from load curve N_2 .

A Suggestion

Moor (1991) suggests using the damage function (Mode II.C.) in Material Type 16 with the following set of parameters:

10.01

$$a_0 = \frac{f_c'}{4} \tag{79}$$

$$a_1 = \frac{1}{3} \tag{80}$$

$$a_2 = \frac{1}{3f_c'} \tag{81}$$

$$a_{0f} = \frac{f_c^{\prime}}{10} \tag{82}$$

$$a_{1f} = 1.5$$
 (83)

$$b_1 = 1.25$$
 (84)

and a damage table of:

Card 5	0.0	8.62e-06	2.15e-05	3.14e-05	3.95e-05	5.17e-04	6.38e-04	7.98e-04
Card 6	9.67e-04	1.41e-03	1.97e-03	2.59e-03	3.27e-03	4.00e-03	4.79e-03	0.909
Card 7	0.309	0.543	0.840	0.975	1.00	0.790	0.630	0.469
Card 8	0.383	0.247	0.173	0.136	0.114	0.086	0.056	0.0

This set of parameters should give results consistent with (Dilger, Koch, and Kowalczyk, 1984) for plain concrete. It has been successfully used for reinforced structures where the reinforcing bars were modeled explicitly with embedded beam and shell elements. The model does not incorporate the major failure mechanism - separation of the concrete and reinforcement leading to catastrophic loss of confinement pressure. However, experience indicates that this physical behavior will occur when this model shows about 4% strain.

Material Type 17 (Isotropic Elastic-Plastic with Oriented Cracks)

An equation of state must be used with this model.

Columns		Quantity	Format
1-10	Card 3	Young's modulus, E	E10.0
1-10	Card 4	Poisson's ratio, v	E10.0
1-10	Card 5	Yield stress, σ_0	E10.0
1-10	Card 6	Tangent modulus, E_T	E10.0
1-10	Card 7	Fracture strength, σ_f	E10.0
1-10	Card 8	Pressure cutoff, p_{cut} (positive in compression)	E10.0

Material Type 17 is experimental, and should be used with caution.

This model provides isotropic elastic-plastic material response identical to Material Type 12, and additionally incorporates a fracture and failure model. In contrast to the fracture and failure models available in other elastic-plastic material models, this fracture model incorporates the effects of crack orientation.

The yield condition can be written

$$\phi = \overline{\sigma} - \sigma_{\gamma} (\overline{\varepsilon}^{p}), \tag{85}$$

where $\overline{\sigma}$ is the effective stress and σ_y is the current yield stress, which may be a function of the effective plastic strain $\overline{\epsilon}^p$ if strain hardening is included. The effective stress $\overline{\sigma}$ is given by

$$\overline{\sigma} = \left(\frac{3}{2}s_{ij}s_{ij}\right)^{\frac{1}{2}},\tag{86}$$

where s_{ij} is the deviatoric stress tensor.

The linear isotropic strain hardening law has the form

$$\sigma_{y} = \sigma_{0} + E_{p} \overline{\varepsilon}^{t}, \qquad (87)$$

where σ_0 is the initial yield stress and E_p is the plastic modulus. The effective plastic strain $\overline{\epsilon}^p$ is given by

$$\overline{\varepsilon}^{p} = \int_{0}^{t} d\overline{\varepsilon}^{p}, \qquad (88)$$

where the incremental effective plastic strain $d\overline{\varepsilon}^p$ is found from the incremental plastic strain tensor $d\varepsilon_{ij}$ as

$$d\overline{\varepsilon}^{p} = \left(\frac{2}{3}d\overline{\varepsilon}^{p}_{ij}d\overline{\varepsilon}^{p}_{ij}\right)^{\frac{1}{2}}.$$
(89)

The plastic modulus is found from Young's modulus E and the tangent modulus E_{τ} using

$$E_p = \frac{EE_T}{E - E_T}.$$
(90)

The plastic hardening modulus E_p is the slope of the effective stress $\overline{\sigma}$ vs. effective plastic strain $\overline{\epsilon}^p$ curve, and the tangent modulus E_T is the slope of the inelastic part of a uniaxial stress vs. strain curve (or equivalently, the effective stress vs. effective strain curve).

A pressure cutoff limits the hydrostatic tension to the specified value, p_{cut} . If pressures more tensile than this limit are calculated, the pressure is reset to p_{cut} .

The oriented crack fracture model is based on a maximum principal stress criterion. When the maximum principal stress exceeds the fracture stress, σ_f , the element fails on a plane perpendicular to the direction of the maximum principal stress. The normal stress and two shear stresses on that plane are then reduced to zero. This stress reduction is done according to a delay function that reduces the stresses gradually to zero over a small number of time steps. This delay function procedure is used to reduce the "ringing," or high frequency noise, introduced into the system caused by the sudden fracture.

After a tensile fracture, the element will not support any tensile stresses on the fracture plane, but in compression will support both normal and shear stresses. The orientation of this fracture surface is tracked throughout the deformation, and is updated to properly model finite deformation effects. If the maximum principal stress subsequently exceeds the fracture stress in another direction, the element fails isotropically. In this case the element completely loses its ability to support any shear stresses or hydrostatic tension, and only compressive hydrostatic stress states are possible. Thus, once isotropic failure has occurred, the material behaves like a fluid.

This model is applicable to elastic or elastoplastic materials under significant tensile or shear loadings when fracture is expected. Potential applications include brittle materials such as ceramics (with $\sigma_0 = \sigma_J$ in the yield function), as well as porous materials such as concrete in cases where pressure hardening effects are not significant.

<u>Columns</u>		Quantity	Format
1-10	Card 3	Young's modulus, E	E10.0
11-20		Poisson's ratio, v	E10.0
21-30		Yield stress coefficient, k	E10.0
31-40		Strain hardening exponent, n	E10.0
	Card 4	Blank	
	Card 5	Blank	
	•		
	•		
	•		
	Card 8	Blank	

Material Type 18 (Power Law Isotropic Elastic-Plastic)

The material behavior is elastoplastic with nonlinear isotropic strain hardening given by a power law expression. The yield condition can be written

$$\phi = \overline{\sigma} - \sigma_{y}(\overline{\varepsilon}^{p}), \tag{91}$$

where $\overline{\sigma}$ is the effective stress and σ_y is the current yield stress. The hardening law has the form

$$\sigma_{y} = k \left(\varepsilon_{0} + \overline{\varepsilon}^{p} \right)^{n}, \tag{92}$$

where ε_0 is the initial yield strain given by

$$\varepsilon_0 = \left(\frac{E}{k}\right)^{\frac{1}{n-1}}.$$
(93)

This model is generally applicable to ductile materials such as metals. Although similar to Material Types 3 and 12, this model provides additional flexibility in specifying the post-yield behavior with a nonlinear isotropic hardening law.

	INPUT	FORMAT
--	-------	--------

DYNA3D User Manual

*

Material Type 19 (Strain Rate Dependent Isotropic Elastic-Plastic)

Columns		Quantity	Format
1-10	Card 3	Young's modulus, E	E10.0
11-20		Poisson's ratio, v	E10.0
21-30		Load curve number for yield stress σ_0 as a function of strain rate $\dot{\overline{\epsilon}}$	E10.0
31-40		Tangent modulus, $E_{\hat{T}}$	E10.0
41-50		Optional load curve number defining Young's modulus as a function of strain rate	E10.0
51-60		Optional load curve number defining the tangent modulus as a function of strain rate	E10.0
61-70		Optional load curve number defining the effective stress at failure $\overline{\sigma}_f$ as a function of strain rate	E10.0
71-80		Time step size for automatic element deletion, Δt_{crit} (Shell elements only)	E10.0
	Card 4	Blank	
	Card 5	Blank	
	•		
	•		
	• ·		
	Card 8	Blank	

The material behavior is elastoplastic with strain rate dependent isotropic hardening. The yield condition can be written

$$\phi = \overline{\sigma} - \sigma_{y} \left(\overline{\varepsilon}^{p}, \dot{\overline{\varepsilon}} \right), \tag{94}$$

where $\overline{\sigma}$ is the effective stress and σ_y is the current yield stress, which may depend on the effective plastic strain $\overline{\epsilon}^p$ and the current effective strain rate, $\dot{\overline{\epsilon}}$. The effective stress is found from the deviatoric stress tensor s_{ii} as

$$\overline{\sigma} = \left(\frac{3}{2}s_{ij}s_{ij}\right)^{\frac{1}{2}}.$$
(95)

The isotropic strain hardening law has the form

$$\sigma_{y} = \sigma_{0} \left(\dot{\overline{\varepsilon}} \right) + E_{p} \overline{\varepsilon}^{p}, \tag{96}$$

where σ_0 is the initial yield stress, determined by the load curve specification of σ_0 vs. $\dot{\bar{E}}$, and E_p is the plastic hardening modulus. The effective strain rate $\dot{\bar{E}}$ is defined as

$$\dot{\bar{e}} = \left(\frac{2}{3}\dot{e}_{ij}\dot{e}_{ij}\right)^{\frac{1}{2}},$$
 (97)

where \dot{e}_{ij} is the deviatoric strain rate tensor. The plastic modulus E_p represents the slope of the effective stress vs. effective plastic strain curve, and the tangent modulus E_T represents the slope of the inelastic portion of the uniaxial stress vs. uniaxial strain curve. The plastic modulus can be written in terms of the Young's modulus E and the tangent modulus E_T as

$$E_p = \frac{EE_T}{E - E_T}.$$
(98)

Both Young's modulus and the tangent modulus may optionally be made functions of strain rate by specifying a load curve number giving their values as a function of strain rate. If these load curve numbers are input as 0, then the constant values specified in the input are used.

Note: All load curves used to define quantities as a function of strain rate must have the same number of points and the same strain rate values. This requirement is used to allow vectorized interpolation to enhance the execution speed of this model.

This model also contains a mechanism for modeling material failure. This option is activated by specifying a load curve defining the effective stress at failure $\overline{\sigma}_f$ as a function of strain rate. For solid elements, once the effective stress exceeds $\overline{\sigma}_f$, the element is deemed to have failed. For shell elements, the entire shell element is deemed to have failed if any integration point through the thickness has an effective stress greater than σ_f . Once failure is detected in an element, all stresses are set to zero, and the element cannot support any load for the remainder of the problem.

In addition to the above failure criterion, this material model also supports a shell element deletion criterion based on the maximum stable time step size for this element, Δt_{max} . Generally, Δt_{max} goes down as the element becomes more distorted. To assure stability of time integration, the global DYNA3D time step is the *minimum* of the Δt_{max} values calculated for all elements in the model. Using this option allows the selective deletion of elements whose time step Δt_{max} has fallen below the specified minimum time step, Δt_{crit} . Elements which are severely distorted often indicate that material has failed and supports little load, but these same elements may have very small time steps and therefore control the cost of the analysis. This option allows these highly distorted elements to be deleted from the calculation, and therefore the analysis can proceed at a larger time step, and

DYNA3D User Manual

.

thus at a reduced cost. Deleted elements do not carry any load, and are deleted from all applicable slide surface definitions. Clearly, this option must be judiciously used to obtain accurate results at a minimum cost.

Material Type 20 (Rigid)

Columns		Quantity	Format
1-10	Card 3	Young's modulus, E	E10.0
1-10	Card 4	Poisson's ratio, v	E10.0
1-10	Card 5	Rigid body constraint option, <i>ICOPT</i> EQ.0.0: Rigid body constraints found from nodal bc's EQ.1.0: Rigid body constraints input below	E10.0
1-10	Card 6	Rigid body translational constraint flag EQ.0: no constraints EQ.1: constrained x displacement EQ.2: constrained y displacement EQ.3: constrained z displacement EQ.4: constrained x and y displacements EQ.5: constrained y and z displacements EQ.6: constrained z and x displacements EQ.7: constrained x, y, and z displacements	E10.0
1-10	Card 7	 Rigid body rotational constraint flag EQ.0: no constraints EQ.1: constrained x rotation EQ.2: constrained y rotation EQ.3: constrained z rotation EQ.4: constrained x and y rotations EQ.5: constrained y and z rotations EQ.6: constrained z and x rotations EQ.7: constrained x, y, and z rotations 	E10.0

Card 8 Blank

This material model is used for modeling a rigid body, and may be used with all element classes (beams, shells, and solids). The rigid material provides an inexpensive method for modeling portions of a structure that are much stiffer than the regions of interest or which experience negligible deformations. For example, this approach may be used in many sheet metal forming problems, where the punch and die are often effectively rigid. In addition, the rigid material is very useful for debugging large models by making all materials rigid except the one of interest. This procedure can quickly isolate an incorrect material definition, even in a large model with many materials. The rigid material is considerably less expensive than any other material model, and may be used in parts of a mesh to minimize costs in the early stages of model development.

DYNA3D User Manual

-

By default, rigid body constraints are automatically derived from the nodal boundary conditions specified for nodes of a rigid body. If desired, constraints on the rigid body center-of-mass degrees of freedom may be directly specified by setting ICOPT = 1.0 on Card 5 and then giving the appropriate translational and rotational constraint flags on Cards 6 and 7. These constraints override constraints computed from nodal boundary condition data.

Two unique rigid materials may not be adjacent in the mesh unless a slide surface is defined between them (i.e., no one node may be used in the definition of two elements each of which use a different material if both materials are defined as rigid). Separate rigid material definitions should seldom be required for a single rigid body unless the mass density varies with position in the body. Adjacent rigid materials must be merged to form one rigid body using the Rigid Body Merge feature described in section 4.29 on page 278.

All elements which reference a given material number corresponding to the Rigid material must be contiguous. Thus, if two disjoint groups of elements on opposite sides of a model are modeled as rigid, separate material numbers should be created for each of the contiguous element groups. This requirement arises from the fact that DYNA3D internally computes the six degrees-offreedom for each rigid body (rigid material or set of merged materials), and if disjoint groups of rigid elements use the same material number, the resulting rigid body properties internally calculated by DYNA3D may be inaccurate. This restriction should not pose an inconvenience is most models.

Inertial properties for rigid materials may be defined in either of two ways. By default, the inertial properties are calculated from the geometry of the elements of the rigid material and the density specified on the material control card. Alternatively, the inertial properties and initial velocity for a rigid body may be directly specified, and this overrides data calculated from the material property definition and nodal initial velocity definitions.

The material constants E and v are used for determining sliding interface parameters if the rigid body interacts along sliding interfaces. Realistic values for these constants should be defined.

í

Material Type 21 (Thermal Orthotropic Elastic)

Columns		Quantity	Format
1-10	Card 3	Elastic modulus, E_a (see Figure 1 and Figure 2)	E10.0
11-20		Elastic modulus, E_b	E10.0
21-30		Elastic modulus, E_c	E10.0
1-10	Card 4	Poisson's ratio, v_{ba}	E10.0
11-20		Poisson's ratio, V_{ca}	E10.0
21-30		Poisson's ratio, V_{cb}	E10.0
31-40		Coefficient of thermal expansion, α_a	E10.0
41-50		Coefficient of thermal expansion, α_b	E10.0
51-60		Coefficient of thermal expansion, α_c	E10.0
1-10	Card 5	Shear modulus, G_{ab}	E10.0
11-20		Shear modulus, G_{bc}	E10.0
21-30		Shear modulus, G_{ca}	E10.0
1-10	Card 6	Material axes definition option, AOPT	E10.0
	EQ.0.0:	locally orthotropic with material axes determined by element nodes n_1 , n_2 , and n_4 as shown in Figure 1. Cards 7 and 8 must be blank with this option.	
	EQ.1.0:	locally orthotropic with material axes determined by a point in space P and the global location of the element ce as shown in Figure 1. Card 8 below is blank.	nter,
	EQ.2.0:	globally orthotropic with material axes determined by vectors defined on Cards 7 and 8. (See Figure 2).	
	EQ.3.0:	applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axis a by an angle β from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector (See Figure 2). The angle β is defined on Card 8, and may be overridden by specifying a value on the element card.	V

DYNA3D User Manual

1

Columns	<u> </u>	Quantity	Format
1-10	Card 7	x_p , define only if AOPT = 1.0	E10.0
11-20		y_p , define only if AOPT = 1.0	E10.0
21-30		z_p , define only if AOPT = 1.0	E10.0
1-10	Card 7	a_x , define only if AOPT = 2.0	E10.0
11-20		a_y , define only if AOPT = 2.0	E10.0
21-30		a_z , define only if AOPT = 2.0	E10.0
1-10	Card 7	v_x , define only if AOPT = 3.0	E10.0
11-20		v_y , define only if AOPT = 3.0	E10.0
21-30		v_z , define only if AOPT = 3.0	E10.0
1-10	Card 8	d_x , define only if AOPT = 2.0	E10.0
11-20		d_y , define only if AOPT = 2.0	E10.0
21-30		d_z , define only if AOPT = 2.0	E10.0
1-10	Card 8	Material angle β , define only if AOPT=3.0 (may be overridden on the element card)	E10.0

The constitutive matrix C that relates increments in stress to increments in strain (in the global coordinate system) is defined as:

$$\mathbf{C} = \mathbf{T}^T \mathbf{C}_L \mathbf{T},\tag{99}$$

where T is the transformation matrix between the local material coordinate system and the global coordinate system, and C_L is the constitutive matrix defined in terms of the material constants of the local orthogonal material axes a, b, and c.

Note that symmetry of the elastic compliance \mathbf{C}_{L}^{-1} implies

$$\frac{v_{ab}}{E_a} = \frac{v_{ba}}{E_b}, \frac{v_{ca}}{E_c} = \frac{v_{ac}}{E_a}, \text{ and } \frac{v_{cb}}{E_c} = \frac{v_{bc}}{E_b}.$$
(100)

Further, positive definiteness of C_L yields the following restrictions on the elastic constants:

$$v_{ba} < \left(\frac{E_b}{E_a}\right)^{\frac{1}{2}}, v_{ca} < \left(\frac{E_c}{E_a}\right)^{\frac{1}{2}}, \text{ and } v_{cb} < \left(\frac{E_c}{E_b}\right)^{\frac{1}{2}}.$$
 (101)

Nonphysical energy growth may occur in the solution if these restrictions on the elastic constants are not observed.

In the chosen local coordinate system, the components of the elastic compliance matrix are given by

$$C_{L}^{-1} = \begin{bmatrix} \frac{1}{E_{a}} - \frac{v_{ba}}{E_{b}} - \frac{v_{ca}}{E_{c}} & 0 & 0 & 0 \\ -\frac{v_{ab}}{E_{a}} & \frac{1}{E_{b}} - \frac{v_{cb}}{E_{c}} & 0 & 0 & 0 \\ -\frac{v_{ac}}{E_{a}} - \frac{v_{bc}}{E_{b}} & \frac{1}{E_{c}} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{ab}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{bc}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{ca}} \end{bmatrix}.$$
(102)

Note that the values of Poisson's ratio are defined as

$$v_{ij} = \frac{-\varepsilon_j}{\varepsilon_i} \tag{103}$$

where ε_i is the strain in the j^{th} direction and ε_i is the strain in the i^{th} direction in a uniaxial stress test in the i^{th} direction.

This model is similar to Material Type 2, but includes thermal strains. It is applicable to a variety of orthotropic elastic materials such as fiber composites or textured materials.

Note that since temperatures are required to compute the thermal strains, the thermal effects option on Control Card 6 must be nonzero.

INPUT FORMAT

DYNA3D User Manual

*

Material Type 22 (Fiber Composite with Damage)

Columns	<u></u>	Quantity	Format
1-10	Card 3	Elastic modulus in longitudinal direction, E_a	E10.0
11-20		Elastic modulus in transverse direction, E_b	E10.0
21-30		Elastic modulus in normal direction, E_c	E10.0
31-40		Bulk modulus of failed material, K_f (solid elements only)	E10.0
41-50		Normal tensile strength, S_n (solid elements only)	E10.0
51-60		Transverse shear strength, S_{bc} (solid elements only)	E10.0
61-70		Transverse shear strength, S_{ca} (solid elements only)	E10.0
1-10	Card 4	Poisson's ratio, V_{ba}	E10.0
11-20		Poisson's ratio, V_{ca}	E10.0
21-30		Poisson's ratio, V_{cb}	E10.0
1-10	Card 5	Shear modulus, G_{ab}	E10.0
11-20		Shear modulus, G_{bc}	E10.0
21-30		Shear modulus, G_{ca}	E10.0
1-10	Card 6	Material axes definition option, AOPT	E10.0
	EQ.0.0:	locally orthotropic with material axes determined by element nodes n_1 , n_2 , and n_4 as shown in Figure 1. Cards 7 and 8 are blank with this option.	
	EQ.1.0:	locally orthotropic with material axes determined by a point in space P and the global location of the element cer as shown in Figure 1. Card 8 below is blank.	iter,
	EQ.2.0:	globally orthotropic with material axes determined by vectors defined on Cards 7 and 8. (See Figure 2).	
	EQ.3.0:	applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axis a by an angle β from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector (See Figure 2). The angle β is defined as described below for e through-thickness integration point. In addition, a material may be specified on each element card, and these values a	v ach angle re added.

Columns		Quantity	Format
11-20	Material ax EQ.1.0 EQ.2.0 EQ.3.0	E10.0	
1-10	Card 7	x_p , define only if AOPT = 1.0	E10.0
11-20		y_p , define only if AOPT = 1.0	E10.0
21-30		z_p , define only if AOPT = 1.0	E10.0
1-10	Card 7	a_x , define only if AOPT = 2.0	E10.0
11-20		a_y , define only if AOPT = 2.0	E10.0
21-30		a_z , define only if AOPT = 2.0	E10.0
1-10	Card 7	v_x , define only if AOPT = 3.0	E10.0
11-20		v_y , define only if AOPT = 3.0	E10.0
21-30		v_z , define only if AOPT = 3.0	E10.0
1-10	Card 8	d_x , define only if AOPT = 2.0	E10.0
11-20		d_y , define only if AOPT = 2.0	E10.0
21-30		d_z , define only if AOPT = 2.0	E10.0
31-40		Shear strength on $a-b$ plane, S_c	E10.0
41-50		Longitudinal tensile strength along a axis, X_t	E10.0
51-60		Longitudinal tensile strength along b axis, Y_t	E10.0
61-70		Transverse compressive strength, Y_c	E10.0
71-80		Nonlinear shear stress parameter, α	E10.0

This material model allows a different fiber orientation to be specified at each through-thickness integration point for shell elements. This capability may be combined with the User-Defined Integration Rules option to accurately represent a general composite laminate.

21.5

:

÷., •

Additional Material Type 22 Input For Shell Elements Only:

Define the material angle β for each of the through-the-thickness integration points for 4-node shell elements. This data must follow the Cross Section Properties data on Card 11.

Columns	Quantity	Format
1-10	Material angle at first integration point, β_1	E10.0
11-20	Material angle at second integration point, β_2	E10.0
21-30	Material angle at third integration point, β_3	E10.0
•	•	
•	• •	•
•	•	•
71-80	Material angle at eighth integration point, eta_{8}	E10.0

Continue on additional cards until NIP points have been defined, where NIP is the number of thickness integration points specified on Card 10 for shell elements. Material angle values may also be specified on the element cards. The material angle used in the calculation is the sum of the value specified above for each integration point and the value specified on the element card.

The formulation of this model is described by Chang and Chang (1987a, 1987b).

The material behavior produced by this model is approximately orthotropic elastic, with the addition of a nonlinear shear stress term. A stress-based failure model is included to incorporate the effects of fiber breakage, matrix cracking, matrix crushing, and delamination. The failure criterion for fiber breakage is

$$e_{f} = \frac{max(0, \sigma_{aa}^{2})}{X_{t}^{2}} + \frac{\sigma_{ab}^{2} + \frac{3}{4}\alpha\sigma_{ab}^{4}}{\frac{S_{c}^{2}}{2G_{ab}} + \frac{3}{4}\alpha S_{c}^{4}} \le 1.0.$$
(104)

Analogously, the criterion for matrix cracking is

$$e_{m} = \frac{max(0, \sigma_{bb}^{2})}{Y_{t}^{2}} + \frac{\sigma_{ab}^{2} + \frac{5}{4}\alpha\sigma_{ab}^{4}}{\frac{S_{c}^{2}}{2G_{ab}} + \frac{3}{4}\alpha S_{c}^{4}} \le 1.0.$$
(105)

The criterion to detect matrix crushing is

$$e_{d} = \frac{\min(0, \sigma_{bb}^{2})}{4S_{c}^{2}} + \left(\frac{Y_{c}^{2}}{4S_{c}^{2}} - 1\right)\frac{\sigma_{bb}}{Y_{c}} + \frac{\sigma_{ab}^{2} + \frac{3}{4}\alpha\sigma_{ab}^{4}}{\frac{S_{c}^{2}}{2G_{ab}} + \frac{3}{4}\alpha S_{c}^{4}} \le 1.0,$$
(106)

and, for solid elements only, the criterion to detect delamination is

$$e_{l} = \frac{max(0, \sigma_{cc}^{2})}{S_{a}^{2}} + \frac{\sigma_{bc}^{2}}{S_{bc}^{2}} + \frac{\sigma_{ca}^{2}}{S_{ca}^{2}} \le 1.0.$$
(107)

When failure is detected, the appropriate stress components are reduced to zero linearly over the next 100 time steps. This stress reduction function has been found useful to minimize spurious oscillations, or ringing, caused by the abrupt release of energy at failure.

This model is applicable to laminated fiber composites. It includes the effects of directionality in the material stress-strain response, and includes criteria to detect the major failure mechanisms in laminated fiber composites.

DYNA3D User Manual

:

Material Type 23 (Thermal Orthotropic Elastic with Variable Properties)

<u>Columns</u>	•••••	Quantity	Format
1-10	Card 3	Number of points in material constant vs. temperature curves, N_{pts} , $(2 \le N_{pts} \le 48)$	E10.0
1-10	Card 4	Blank	E10.0
1-10	Card 5	Blank	E10.0
1-10	Card 6	Material axes definition option, AOPT	E10.0
	EQ.0.0:	locally orthotropic with material axes determined by element nodes n_1 , n_2 , and n_4 as shown in Figure 1. Cards 7 and 8 are blank with this option.	
	EQ.1.0:	locally orthotropic with material axes determined by a point in space P and the global location of the element ce as shown in Figure 1. Card 8 below is blank.	nter,
	EQ.2.0:	globally orthotropic with material axes determined by vectors defined on Cards 7 and 8. (See Figure 2).	
	EQ.3.0:	applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axis a by an angle β from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector (See Figure 2). The angle β is defined for each through-thickne integration point as described below. In addition, a materia angle may be specified on the element cards, and the sum of these angles is used for computation.	v ess al
1-10	Card 7	x_p , define only if AOPT = 1.0	E10.0
11-20		y_p , define only if AOPT = 1.0	E10.0
21-30		z_p , define only if AOPT = 1.0	E10.0
1-10	Card 7	a_x , define only if AOPT = 2.0	E10.0
11-20		a_y , define only if AOPT = 2.0	E10.0
21-30		a_z , define only if AOPT = 2.0	E10.0
1-10	Card 7	v_x , define only if AOPT = 3.0	E10.0
11-20		v_y , define only if AOPT = 3.0	E10.0
21-30		v_z , define only if AOPT = 3.0	E10.0

Columns		Format	
1-10	Card 8	d_x , define only if AOPT = 2.0	E10.0
11-20		d_y , define only if AOPT = 2.0	E10.0
21-30		d_z , define only if AOPT = 2.0	E10.0

Define the following input card set for each of the 12 orthotropic material constants and the list of corresponding temperatures using the format (8E10.0). For shell elements, this data must follow the Cross Section Properties specified on cards 10 and 11. For solid elements, this data follows the eight Material Cards described above.

Columns	Quantity	Format
1-10	Elastic modulus E_a at temperature T_1	E10.0
11-20	Elastic modulus E_a at temperature T_2	E10.0
•	•	•
•	•	•
•	•	•
71-80	Elastic modulus E_a at temperature T_8	E10.0

Continue with additional cards in this format until N_{pts} points have been defined for E_a . Then, input definitions for the elastic moduli E_b and E_c , the Poisson's ratios v_{ba} , v_{ca} , and v_{cb} , the coefficients of thermal expansion $\overline{\alpha}_a$, $\overline{\alpha}_b$, and $\overline{\alpha}_c$, the shear moduli G_{ab} , G_{bc} , and G_{ca} , and the list of temperatures T following the same format. In the above, subscripts a, b, and c denote the local material axes.

This model allows a different orientation to be specified at each through-thickness integration point for shell elements. This capability may be combined with the User-Defined Integration Rules option to accurately represent a general orthotropic elastic laminate.

The additional input required to specify fiber orientation for each integration point of a shell element is described on the following page.

DYNA3D User Manual

t

Additional Material Type 23 Input For Shell Elements Only:

For shell elements only, define the material angle for each through-thickness integration point.

Columns	Quantity	Format
1-10	Material angle at first integration point, β_1	E10.0
11-20	Material angle at second integration point, β_2	E10.0
21-30	Material angle at third integration point, β_3	E10.0
•		
•	•	•
•		
71-80	Material angle at eighth integration point, β_8	E10.0

For shell elements, continue on additional cards as necessary until NIP points have been defined, where NIP is the number of thickness integration points specified on Card 10. Material angles may also be specified on the element cards. The value used for computation is the sum of the material angle specified by integration point above and the value specified on the element card.

For shell elements, plane stress conditions are assumed on each laminae, but transverse shear stresses are included.

Since temperature dependent material properties are used, the thermal effects option on Control Card 6 must be nonzero.

This material model is useful for studying the thermoelastic behavior of orthotropic materials, including both homogeneous compositions and laminated materials such as fiber composites. It allows the material orientation angle to be specified by integration point and by element for shell elements, and by element for solid elements. This approach allows an accurate representation of laminates where the orientation of the fibers varies between laminae. This capability is frequently combined with user-defined integration rules for shell elements to define one integration point in each lamina or set of lamina in a multi-layer composite structure.

DYNA3D User Manual

Material Type 24 (F	Rate-Dependent '	Tabular)	Isotropic	Elastic-Plastic)
---------------------	------------------	-----------	-----------	--------------------------

<u>Columns</u>		Quantity	Format
1-10	Card 3	Young's modulus, E	E10.0
1-10	Card 4	Poisson's ratio, v	E10.0
1-10	Card 5	Yield stress, σ_0	
1-10	Card 6	Tangent modulus, E_T (not used if the stress-strain curve is defined below)	E10.0
11-20		Effective plastic strain at failure, $\overline{\varepsilon}_{f}^{p}$	E10.0
21-30		Time step size for automatic element deletion, Δt_{crit} (shell elements only)	E10.0
31-40		Load curve number to scale yield stress for strain rate effects	E10.0
1-80	Card 7	Tabulated effective plastic strain values, $\overline{\epsilon}^{p}$ (define up to 8 values)	E10.0
1-80	Card 8	Corresponding tabulated yield stress values, $\overline{\sigma}$	E10.0

This model yields rate-dependent elastoplastic material behavior. It is similar to Material Types 3, 12, and 19, but allows a general strain hardening law to be specified via tabulated data. In addition, arbitrary rate dependence of the yield stress is incorporated via a load curve.

The yield condition can be written

$$\phi = \overline{\sigma} - \sigma_{y} \left(\overline{\varepsilon}^{p}, \dot{\overline{\varepsilon}} \right), \qquad (108)$$

where $\overline{\sigma}$ is the effective stress and σ_y is the current yield stress, which may depend on the effective plastic strain $\overline{\epsilon}^p$ and the current effective strain rate, $\overline{\epsilon}$. The effective stress is found from the deviatoric stress tensor s_{ij} as

$$\overline{\sigma} = \left(\frac{3}{2}s_{ij}s_{ij}\right)^{\frac{1}{2}}.$$
(109)

If a tabulated yield stress vs. effective plastic strain curve is not defined, then a linear hardening law is used,

$$\sigma_{y} = \sigma_{0} + E_{p} \overline{\varepsilon}^{p}, \qquad (110)$$

where σ_0 is the initial yield stress. The hardening modulus E_p represents the slope of the effective stress vs. effective plastic strain curve, and the tangent modulus E_T represents the slope of the inelastic portion of the uniaxial stress vs. uniaxial strain curve. The hardening modulus can be written in terms of the Young's modulus E and the tangent modulus E_T as

$$E_p = \frac{EE_T}{E - E_T} \tag{111}$$

If a tabulated yield stress vs. effective plastic strain curve is defined, then the input value of the tangent modulus E_T is ignored and the current value of the effective plastic strain determines the yield stress from the tabulated function.

If a nonzero load curve number is specified, this yield stress is multiplied by a scale factor from the load curve based on the current effective strain rate, $\dot{\bar{\epsilon}}$. The load curve should define a strain rate scale factor f as a function of strain rate $\dot{\bar{\epsilon}}$. Thus, load curve points should be input as pairs of effective strain rate and the corresponding yield scale factor, $(\dot{\bar{\epsilon}}, f(\dot{\bar{\epsilon}}))$. A strain rate scale factor of 1.0 would give no strain rate effect, and values greater than one give material strengthening effects. The effective strain rate is defined as

$$\dot{\overline{\varepsilon}} = (\frac{2}{3}\dot{e}_{ij}\dot{e}_{ij})^{\frac{1}{2}},$$
 (112)

where \dot{e}_{ij} is the deviatoric strain rate tensor. If the load curve number is input as 0, then the constant value specified in the input is used with no modification for strain rate effects.

This model also contains a failure criterion based on the total accumulated effective plastic strain, $\overline{\varepsilon}^p$. When the effective plastic strain exceeds the specified value for the effective plastic strain at failure, $\overline{\varepsilon}_{f}^{p}$, failure is detected. For shell elements, failure is assumed to occur when the effective plastic strain at *any* through-thickness integration point reaches $\overline{\varepsilon}_{f}^{p}$. Once any integration point in a shell element fails, the effective plastic strain is set to the failure value $\overline{\varepsilon}_{f}^{p}$ for all through-thickness integration points in that element, and all stresses in that element are set to zero. The element remains failed throughout the rest of the calculation. For solid elements failure is detected when the effective plastic strain at the center of the element exceeds the critical value. When a solid element fails, the yield stress σ_0 and the hardening modulus E_p are cut by a factor of 1000. Thus, the solid element retains its hydrostatic strength even after failure has occurred. The effective plastic strain may reach large values using this formulation, and provides a useful index on the degree of failure predicted in the element.

In addition to the above failure criterion, this model also supports a shell element deletion criterion based on the maximum stable time step size for this element, Δt_{max} . Generally, Δt_{max} goes down as the element becomes more distorted. To assure stability of the global time integration, the DYNA3D time step is the *minimum* of the Δt_{max} values calculated for all elements in the model. Using this option allows the selective deletion of elements whose time step Δt_{max} has fallen below the specified minimum time step, Δt_{crit} .

Elements which are severely distorted often indicate that material has failed and supports little load, but these same elements may have very small time steps and therefore control the cost of the analysis. This option allows these highly distorted elements to be deleted from the calculation, and therefore the analysis can proceed at a larger time step, and thus at a reduced cost. Deleted elements do not carry any load, and are deleted from all applicable slide surface definitions. Clearly, this option must be judiciously used to obtain accurate results at a minimum cost.

This model is available for SAND failure based on Δt_{crit} or on $\overline{\varepsilon}_f^p$ by setting the appropriate flag (on Control Card 3 described in section 4.1 on page 41).
- 1-

Material Type 25 (Extended Two Invariant Geologic Cap)

, •

· · · · · ·

Columns		Quantity	Format
1-10	Card 3	Initial bulk modulus, K	E10.0
11-20		Initial shear modulus, G	E10.0
1-10	Card 4	Failure envelope parameter, α	E10.0
11-20		Failure envelope linear coefficient, θ	E10.0
21-30		Failure envelope exponential coefficient, γ	E10.0
31-40		Failure envelope exponent, β	E10.0
41-50		Cap surface axis ratio, R	E10.0
1-10	Card 5	Hardening law exponent, D	E10.0
11-20		Hardening law coefficient, W	E10.0
21-30		Hardening law parameter, X_0	E10.0
31-40		Kinematic hardening coefficient, \overline{c}	E10.0
41-50		Kinematic hardening parameter, N	E10.0
1-10	Card 6	Plot database flag, <i>IPLOT</i> EQ.1.0: Hardening variable, κ EQ.2.0: Cap - J_1 axis intercept, $X(\kappa)$ EQ.3.0: Volumetric plastic strain, ε_{ν}^{P} EQ.4.0: First stress invariant, J_1 EQ.5.0: Second stress invariant, $\sqrt{J_{2D}}$ EQ.6.0: Not used EQ.7.0: Not used EQ.8.0: Response mode number EQ.9.0: Number of iterations	E10.0
11-20		Vectorization flag, <i>IVEC</i> EQ.0.0: Vectorized (fixed number of iterations) EQ.1.0: Fully iterative	E10.0
1-10	Card 8	Tension cutoff, $T < 0$ (positive in compression)	E10.0

DYNA3D User Manual

This implementation of an extended two invariant cap model was developed by Whirley and Engelmann (1992) based on the formulations of Simo, Ju, Pister, and Taylor (1988), Simo, Ju, and Taylor (1990), and Sandler and Rubin, (1979). In this model, the two invariant cap theory is extended to include nonlinear kinematic hardening as suggested in (Isenberg, Vaughn, and Sandler, 1978). A brief discussion of the extended cap model and its parameters is given below.



The cap model is formulated in terms of the invariants of the stress tensor. The square root of the second invariant of the deviatoric stress tensor, $\sqrt{J_{2D}}$, is found from the deviatoric stresses s as

$$\sqrt{J_{2D}} \equiv \sqrt{\frac{1}{2}s_{ij}s_{ij}} \tag{113}$$

and is an objective scalar measure of the distortional or shearing stress. The first invariant of the stress, J_1 , is simply the sum of the normal stresses, or equivalently, three times the pressure.

The cap model consists of three surfaces in $\sqrt{J_{2D}}$ - J_1 space, as shown in Figure 11. First, there is a failure envelope surface, denoted f_1 in the figure. The functional form of f_1 is

$$f_1 = \sqrt{J_{2D}} - \min(F_e(J_1), T_{mises})), \qquad (114)$$

where F_{e} is given by

$$F_{e}(J_{1}) \equiv \alpha - \gamma \exp\left(-\beta J_{1}\right) + \theta J_{1}$$
(115)

and $T_{mises} \equiv |X(\kappa_n) - L(\kappa_n)|$. This failure envelope surface is fixed in $\sqrt{J_{2D}} - J_1$ space, and therefore does not harden, unless kinematic hardening is present. Next, there is a cap surface, denoted f_2 in the figure, with f_2 given by

$$f_2 = \sqrt{J_{2D}} - F_c (J_1, \kappa) , \qquad (116)$$

where F_c is defined by

$$F_{c}(J_{1},\kappa) = \frac{1}{R} \sqrt{[X(\kappa) - L(\kappa)]^{2} - [J_{1} - L(\kappa)]^{2}},$$
(117)

 $X(\kappa)$ is the intersection of the cap surface with the J_1 axis,

$$X(\kappa) = \kappa + RF_{e}(\kappa), \qquad (118)$$

and $L(\kappa)$ is defined by

$$L(\kappa) \equiv \begin{cases} \kappa & \text{if } \kappa > 0 \\ 0 & \text{if } \kappa \le 0 \end{cases}.$$
(119)

The hardening parameter κ is related to the plastic volume change ε_{ν}^{p} through the hardening law

$$\varepsilon_{\nu}^{p} = W \{ 1 - exp \left[-D \left(X \left(\kappa \right) - X_{0} \right) \right] \}.$$
(120)

Geometrically, κ is seen in the figure as the J_1 coordinate of the intersection of the cap surface and the failure surface. Finally, there is the tension cutoff surface, denoted f_3 in the figure. The function f_3 is given by

$$f_3 \equiv T - J_1, \tag{121}$$

where T is an input material parameter which specifies the maximum hydrostatic tension sustainable by the material. The elastic domain in $\sqrt{J_{2D}}$ - J_1 space is then bounded by the failure envelope surface above, the tension cutoff surface on the left, and the cap surface on the right.

An additive decomposition of the strain into elastic and plastic parts is assumed:

$$e = e^e + e^p, \tag{122}$$

where $\boldsymbol{\varepsilon}^{e}$ is the elastic strain and e^{p} is the plastic strain. Stress is found from the elastic strain using Hooke's law,

$$S = C \left(e - e^p \right), \tag{123}$$

where s is the stress and C is the elastic constitutive tensor.

The yield condition may be written

$$f_1(s) \le 0$$

$$f_2(s, \kappa) \le 0$$

$$f_3(s) \le 0$$
(124)

and the plastic consistency condition requires that

$$\begin{aligned} \lambda_k f_k &= 0\\ \dot{\lambda}_k &\ge 0 \end{aligned} \qquad (125)$$

where λ_k is the plastic consistency parameter for surface k. If $f_k < 0$, then $\dot{\lambda}_k = 0$ and the response is elastic. If $f_k > 0$, then surface k is active and $\dot{\lambda}_k$ is found from the requirement that $\dot{f}_k = 0$.

Associated plastic flow is assumed, so using Koiter's flow rule the plastic strain rate is given as the sum of contributions from all of the active surfaces,

$$\dot{e}^{p} = \sum_{k=1}^{3} \dot{\lambda}_{k} \frac{\partial f_{k}}{\partial s}.$$
 (126)

One of the major advantages of the cap model over other classical pressure-dependent plasticity models is the ability to control the amount of dilatency produced under shear loading. Dilatency is produced under shear loading as a result of the yield surface having a positive slope in $\sqrt{J_{2D}} - J_1$ space, so the assumption of plastic flow in the direction normal to the yield surface produces a plastic strain rate vector that has a component in the volumetric (hydrostatic) direction (see Figure 11). In models such as the Drucker-Prager and Mohr-Coulomb, this dilatency continues as long as shear loads are applied, and in many cases produces far more dilatency than is experimentally observed in material tests. In the cap model, when the failure surface is active, dilatency is produced just as with the Drucker-Prager and Mohr-Coulomb models. However, the hardening law permits the cap surface to contract until the cap intersects the failure envelope at the stress point, and the cap remains at that point. The local normal to the yield surface is now vertical, and therefore the normality rule assures that no further plastic volumetric strain (dilatency) is created. Adjustment of the parameters that control the rate of cap contraction permits experimentally observed amounts of dilatency to be incorporated into the cap model, thus producing a constitutive law which better represents the physics to be modeled.

Another advantage of the cap model over other models such as the Drucker-Prager and Mohr-Coulomb is the ability to model plastic compaction. In these models all purely volumetric response is elastic. In the cap model, volumetric response is elastic until the stress point hits the cap surface. Thereafter, plastic volumetric strain (compaction) is generated at a rate controlled by the hardening law. Thus, in addition to controlling the amount of dilatency, the introduction of the cap surface adds another experimentally observed response characteristic of geological materials into the model.

The inclusion of kinematic hardening will result in hysteretic energy dissipation under cyclic loading conditions. Following the approach of (Isenberg, Vaughn, and Sandler, 1978), a nonlinear kinematic hardening law is used for the failure envelope surface when nonzero values of \overline{c} and Nare specified. In this case, the failure envelope surface is replaced by a family of yield surfaces

DYNA3D User Manual

bounded by an initial yield surface and a limiting failure envelope surface. Thus, the shape of the yield surfaces described above remains unchanged, but they may translate in a plane orthogonal to the J_1 axis.

Translation of the yield surfaces is permitted through the introduction of a "back stress" tensor, a. The formulation including kinematic hardening is obtained by replacing the stress s with the translated stress tensor $h \equiv s - a$ in all of the above equations. The history tensor a is assumed deviatoric, and therefore has only 5 unique components. The evolution of the back stress tensor is governed by the nonlinear hardening law

$$\dot{\boldsymbol{\alpha}} = \bar{c}\bar{F}(s,\boldsymbol{\alpha})\,\dot{e}^{p},\tag{127}$$

where \bar{c} is a constant, \bar{F} is a scalar function of s and a, and \dot{e}^{p} is the rate of deviatoric plastic strain. The constant \bar{c} may be estimated from the slope of the shear stress - plastic shear strain curve at low levels of shear stress.

The function \overline{F} is defined as

$$\overline{F} \equiv max\left(0, 1 - \frac{(\boldsymbol{\sigma} - \boldsymbol{\alpha}) \cdot \boldsymbol{\alpha}}{2NF_{\epsilon}(J_{1})}\right), \qquad (128)$$

where N is a constant defining the size of the yield surface. The value of N may be interpreted as the radial distance between the outside of the initial yield surface and the inside of the limit surface. In order for the limit surface of the kinematic hardening cap model to correspond with the failure envelope surface of the standard cap model, the scalar parameter α must be replaced with $\alpha - N$ in the definition of F_{ϵ} .

The cap model contains a number of parameters which must be chosen to represent a particular material, and are generally based on experimental data. The parameters α , β , θ , and γ are usually evaluated by fitting a curve through failure data taken from a set of triaxial compression tests. The parameters W, D, and X_0 define the cap hardening law. The value of W represents the void fraction of the uncompressed sample and D governs the slope of the initial loading curve in hydrostatic compression. The value of R is the ratio of major to minor axes of the quarter ellipse defining the cap surface. Additional details and guidelines for fitting the cap model to experimental data are found in (Chen and Baladi, 1985).

This model represents a new implementation of the two-invariant cap model into DYNA3D. This version is highly vectorized and more reliable than the experimental implementation available in previous releases. In addition, this new implementation incorporates nonlinear kinematic hardening to model cyclic energy dissipation due to shear hysteresis.

Material Type 26	(Metallic Honeycomb)
------------------	----------------------

Columns		Quantity	Format
1-10	Card 3	Young's modulus for fully compacted material, E	E10.0
11-20		Poisson's ratio for fully compacted material, v	E10.0
21-30		Yield stress for fully compacted material, σ_0	E10.0
31-40		Load curve number for $\sigma_{aa,max}$ vs. relative volume V	E10.0
41-50		Load curve number for $\sigma_{bb,max}$ vs. relative volume V	E10.0
51-60		Load curve number for $\sigma_{cc,max}$ vs. relative volume V	E10.0
61-70		Load curve number for $\sigma_{ab,max}$, $\sigma_{bc,max}$, and $\sigma_{ca,max}$ vs. relative volume V	E10.0
71-80		Relative volume at full compaction, V_{min}	E10.0
1-10	Card 4	Initial elastic modulus, E_a	E10.0
11-20		Initial elastic modulus, E_b	E10.0
21-30		Initial elastic modulus, E_c	E10.0
31-40		Initial shear modulus, G_{ab}	E10.0
41-50		Initial shear modulus, G_{bc}	E10.0
51-60		Initial shear modulus, G_{ca}	E10.0
	Card 5	Blank	
	Card 6	Blank	
	Card 7	Blank	
	Card 8	Blank	

This model is an empirically based formulation for the representation of crushable metallic honeycomb materials, but it may be applicable to other materials as well. The initial behavior is orthotropic elastic with the components of stress treated in an uncoupled way (i.e., an *a*-direction component of strain generates only an *a*-direction component of stress, and the other stress components are unaffected). As volumetric compaction occurs, the directional elastic moduli vary linearly from their initial values to the fully compacted value, E. In the partially compacted states, the stress components are limited by the maximum values at the current relative volume V as

specified on load curves. When full compaction is reached, the material behavior becomes isotropically elastic-plastic with no strain hardening. Additional details on the implementation of this model are given below.

The metallic honeycomb model is predicated on the observation that honeycomb materials markedly change their behavior as a function of volumetric crushing (compaction). Compaction is measured analytically by the relative volume V defined by

$$V \equiv \frac{V_{current}}{V_{initial}} = det(F), \qquad (129)$$

where F is the deformation gradient. Note that V > 1 for an expanded material and V < 1 for a compacted material. The relative volume at which full compaction is achieved, V_{min} , is specified as an input parameter. In this model, compaction is measured as the minimum relative volume ever achieved by a given element over the history of the deformation. Thus, once an element begins compacting, it never loses its compaction. This is in accordance with physical observations.

For partially compacted states, a maximum stress magnitude may be specified for each component $(\sigma_{aa,max}, \sigma_{bb,max}, \sigma_{cc,max}, and \sigma_{ab,max})$ as a function of relative volume, using a load curve. In order to allow a vectorized interpolation scheme for good performance, *it is required that each load curve used for this model must have the same number of points and exactly the same abscissa (relative volume) points*. The first point in each load curve should correspond to a relative volume slightly less than the minimum relative volume for fully compacted material, V_{min} .

The constitutive equation used for materials in the initial or expanded configurations is directionally elastic with the components of stress uncoupled. The local a, b, and c axes are found from the local node numbers defining the solid element, as shown in Figure 12. First, the local c axis is found from the cross product of the two diagonals of the lower surface of the solid element (i.e., the surface bounded by nodes n_1 , n_2 , n_3 , and n_4). Next, a temporary local \hat{a} axis is constructed as a vector from node n_1 to node n_2 , and the local b axis is found by crossing the c axis with the \hat{a} axis. Finally, the local a axis is constructed by modifying the \hat{a} axis to be mutually perpendicular to the other two local axes. The resulting local coordinate system is also shown in Figure 12.

For partially compacted states ($V_{min} < V < 1$), a trial elastic stress is first computed using directionally elastic moduli. These moduli are linearly interpolated based on the current relative volume between the input initial directional values and the isotropic values of Young's modulus E and Poisson's ratio v at $V = V_{min}$. Next, the limiting stress magnitudes are found for each component at the current relative volume by interpolation from the specified load curves. If the trial stress

component is smaller in magnitude than this limiting value, then the trial stress component becomes the final stress component. If the magnitude of the trial stress component exceeds the limiting value, then the stress component is scaled back to this limiting value.



Once the material becomes fully compacted ($V = V_{min}$), it is modeled as isotropically elastic perfectly-plastic with a yield stress of σ_0 . The behavior remains elastic-plastic for the remainder of the calculation, even if the element subsequently goes into tension.

*

Columns		Quantity	Format
1-10	Card 3	First invariant coefficient, A	E10.0
11-20		Second invariant coefficient, B	E10.0
21-30		Poisson's ratio, $v (0.48 \le v < 0.50)$	E10.0
	Card 4	Blank	
	•	•	
	•	•	
	•		
	Card 8	Blank	

Material Type 27 (Compressible Mooney-Rivlin Hyperelastic Rubber)

The DYNA3D implementation of this model was developed by Brad Maker.

The strain energy density function is defined as:

$$W = A (I_1 - 3) + B (I_2 - 3) + C (I_3^{-2} - 1) + D (I_3 - 1)^2,$$
(130)

where

$$C = \frac{1}{2}A + B,$$
 (131)

$$D = \frac{A(5v-2) + B(11v-5)}{2(1-2v)},$$
(132)

v is Poisson's ratio, and I_1 , I_2 , and I_3 are the invariants of the right Cauchy-Green tensor, C. The right Cauchy-Green tensor is found from the deformation gradient F using

$$C = F^T F, (133)$$

or from the Green-Lagrange strain E using

$$C = 2E + 1,$$
 (134)

where 1 is the identity two-tensor.

For small deformations, the shear modulus of linear elasticity G is given by

ł

$$G = 2(A+B).$$
 (135)

Poisson's ratio should be chosen such that $0.48 \le v < 0.50$, yielding a slightly compressible hyperelastic material. This is a hyperelastic model and is often used to represent the behavior of rubber-like materials at moderate to large strains. Other rubber models include Material Type 7, the Blatz-Ko model, and Material Type 31, the Frazer-Nash rubber model.

DYNA3D User Manual

<u>Columns</u>		Quantity	Format
1-10	Card 3	Young's modulus, E	E10.0
1-10	Card 4	Poisson's ratio, v	· E10.0
1-10	Card 5	Yield stress, σ_0	E10.0
1-10	Card 6	Hardening modulus, E_T (shells only)	E10.0
1-10	Card 7	Blank	
1-10	Card 8	Blank	

Material Type 28 (Resultant Plasticity)

This model is formulated in terms of resultant quantities for structural elements, and is available for the Belytschko-Schwer beam and the Belytschko-Tsay shell elements only. All other DYNA3D material models for structural elements relate stresses directly to strains, and numerical integration is performed over the beam or shell element cross section to calculate the resultant forces and moments. Formulating the material model directly in terms of resultants yields some improvements in speed, but sacrifices accuracy when the material behavior becomes plastic (Whirley, 1990). Therefore, this model is primarily useful for scoping calculations where precise results are not required. For more accurate results, the use of Material Type 3 or 12 is recommended.

The Illyushin yield criterion is used as described in (Kennedy, Belytschko, and Lin, 1986). For beams, the behavior is elastic-perfectly plastic. For shell elements, an approximation to isotropic hardening has been incorporated.

Material Type 29 (Not Currently Available)

Columns	Quantity	Format
	Quantity	Tomat

This model is not currently available.

DYNA3D User Manual

Columns		Quantity	 Format
1-10	Card 3	Young's modulus, E	E10.0
1-10	Card 4	Poisson's ratio, v	E10.0
1-10	Card 5	Yield stress, σ_0	E10.0
1-10	Card 6	Tangent modulus, E_T	E10.0
	Card 7	Blank	
	Card 8	Blank	

Material Type 30 (Closed-Form Update Elastic-Plastic for Shells)

This model implements a closed form solution for the plane stress constitutive update under conditions of perfect plasticity or kinematic hardening for a bilinear von Mises model. Thus, this model yields exactly the same behavior as Material Type 3 with $\beta = 0.0$ (i.e., purely kinematic hardening). The only difference between Material Type 3 and Material Type 30 lies in the numerical algorithms used to integrate the rate equations of plasticity under conditions of plane stress, as found in shell elements. The numerical implementation of Material Type 30 is described in (Whirley, Hallquist, and Goudreau, 1988).

The details of the constitutive equations and the resulting stress-strain behavior are discussed in the definition of Material Type 3.

Material Type 31 (Frazer-Nash Hyperelastic Rubber)

Columns		Quantity	Format
1-10	Card 3	First strain energy density coefficient, C_{001}	E10.0
11-20		Second strain energy density coefficient, C_{010}	E10.0
21-30		Third strain energy density coefficient, C_{020}	E10.0
31-40		Fourth strain energy density coefficient, C_{100}	E10.0
41-50		Fifth strain energy density coefficient, C_{101}	E10.0
1-10	Card 4	Sixth strain energy density coefficient, C_{110}	E10.0
11-20		Seventh strain energy density coefficient, C_{200}	E10.0
21-30		Eighth strain energy density coefficient, C_{210}	E10.0
31-40		Ninth strain energy density coefficient, C_{300}	E10.0
41-50		Tenth strain energy density coefficient, C_{400}	E10.0
1-10	Card 5	Strain limit option, <i>ILIMIT</i> EQ.0.0: stop if strain limits are exceeded EQ.1.0: continue if strain limits are exceeded	E10.0
1-10	Card 6	Maximum strain limit, E_{max}	E10.0
11-20		Minimum strain limit, E_{min}	E10.0
	Card 7	Blank	
	Card 8	Blank	

This model implements a hyperelastic constitutive law described in (Kenchington, 1988). It is useful for representing the behavior of rubber-like materials at moderate to large strains. Other hyperelastic models include Material Type 7 (Blatz-Ko) and Material Type 27 (Compressible Mooney Rivlin).

The strain energy density function is of the form

$$W = C_{100}I_1 + C_{200}I_1^2 + C_{300}I_1^3 + C_{400}I_1^4 + C_{010}I_2 + C_{020}I_2^2 + C_{110}I_1I_2 + C_{210}I_1^2I_2,$$
(136)
+ $C_{001}I_3 + C_{101}I_1I_3$

where I_1 , I_2 , and I_3 are the strain invariants defined in terms of engineering components of the Green-Lagrange strain tensor E by

$$I_1 = E_{11} + E_{22} + E_{33}, \tag{137}$$

$$I_{2} = (E_{11}E_{22} + E_{11}E_{33} + E_{22}E_{33}) - \frac{1}{4}(E_{12}^{2} + E_{23}^{2} + E_{31}^{2}), \qquad (138)$$

and

$$I_{3} = (E_{11}E_{22}E_{33} + \frac{1}{4}E_{12}E_{23}E_{31}) - \frac{1}{4}(E_{11}E_{23}^{2} + E_{22}E_{31}^{2} + E_{33}E_{12}^{2}).$$
(139)

The second Piola-Kirchhoff stress t is found by differentiating the strain energy density function W with respect to the Green-Lagrange strain,

$$t = \frac{\partial W}{\partial E}.$$
 (140)

Cauchy stress s is then found from the second Piola-Kirchhoff stress using

$$s = \frac{1}{J}FtF^{T}, (141)$$

where F is the deformation gradient and J is its determinant.

The values of the ten C coefficients in the strain energy density function must be determined from experimental data on the material. A procedure is described in (Kenchington, 1988) to determine these parameters using data from a uniaxial stress test, a biaxial stress or strain test, and a shear test.

The model input includes a maximum strain limit E_{max} , a minimum strain limit E_{min} , and a strain limit option flag, *ILIMIT*. This feature is particularly useful if the model has been fitted to data only over a limited range, and therefore caution should be exercised if strains outside this range are encountered. At each step, the maximum and minimum normal strains are tested against the limit criteria. If the maximum normal strain is greater than E_{max} or the minimum normal strain is less than E_{min} , then a message is written to the screen and hsp printout file, and execution terminates if *ILIMIT* = 0 or continues if *ILIMIT* = 1.

Material Type 32	(Ramberg-Osgood	Elastic-Plastic)
------------------	-----------------	------------------

Columns	<u> </u>	Quantity	Format
1-10	Card 3	Reference shear strain, γ_y	E10.0
1-10	Card 4	Reference shear stress, τ_y	E10.0
1-10	Card 5	Stress coefficient, α	E10.0
1-10	Card 6	Stress exponent, r	E10.0
1-10	Card 7	Bulk modulus, K	
	Card 8	Blank	

The Ramberg-Osgood equation is an empirical constitutive relation to represent the one-dimensional elastic-plastic behavior of many materials. This implementation of the model was developed by Whirley and Engelmann, and allows a simple rate-independent representation of the hysteretic energy dissipation observed in materials subjected to cyclic shear deformation. For monotonic loading, the stress-strain relationship is given by

$$\frac{\gamma}{\gamma_{y}} = \frac{\tau}{\tau_{y}} + \alpha \left| \frac{\tau}{\tau_{y}} \right|^{r} \quad \text{if} \quad \gamma \ge 0, \tag{142}$$

$$\frac{\gamma}{\gamma_{y}} = \frac{\tau}{\tau_{y}} - \alpha \left| \frac{\tau}{\tau_{y}} \right|^{r} \quad \text{if} \quad \gamma < 0, \tag{143}$$

where γ is the shear strain and τ is the stress. The model approaches perfect plasticity as the stress exponent $r \to \infty$. These equations must be augmented to correctly model unloading and reloading material behavior. The first load reversal is detected by $\gamma \dot{\gamma} < 0$. After the first reversal, the stressstrain relationship is modified to

$$\frac{\gamma - \gamma_0}{2\gamma_y} = \frac{\tau - \tau_0}{2\tau_y} + \alpha \left(\frac{\tau - \tau_0}{2\tau_y}\right)^r \quad \text{if} \quad \gamma > 0, \tag{144}$$

$$\frac{\gamma - \gamma_0}{2\gamma_y} = \frac{\tau - \tau_0}{2\tau_y} - \alpha \left(\frac{\tau_0 - \tau}{2\tau_y}\right)^r \quad \text{if} \quad \gamma < 0, \tag{145}$$

where γ_0 and τ_0 represent the values of strain and stress at the point of load reversal. Subsequent load reversals are detected by $(\gamma - \gamma_0) \dot{\gamma} < 0$.

The Ramberg-Osgood equations are inherently one-dimensional, and are assumed to apply to shear components. To generalize this theory to the multidimensional case, it is assumed that each component of the deviatoric stress and deviatoric tensorial strain is independently related by the one-dimensional stress-strain equations. A projection is used to map the result back into deviatoric stress space, if required. The volumetric behavior is elastic, and therefore the pressure p is found by

$$p = -K\varepsilon_{\nu},\tag{146}$$

where ε_V is the volumetric strain.

This model is primarily intended as a simple model for shear behavior, as is often used in seismic analysis.

í –

Material Type 33 (General Anisotropic Elastic-Plastic)

Columns		Quantity	Format
1-10	Card 3	Elastic modulus, E_a	E10.0
11-20		Elastic modulus, E_b	E10.0
21-30		Elastic modulus, E_c	E10.0
31-40		Anisotropy coefficient, R (1948 Hill theory)	E10.0
41-50		Anisotropy coefficient, P	E10.0
51-60		Anisotropy coefficient, Q_{bc}	E10.0
61-70		Anisotropy coefficient, Q_{ab}	E10.0
71-80		Anisotropy coefficient, Q_{ac}	E10.0
1-10	Card 4	Poisson's ratio, V_{ba}	E10.0
11-20		Poisson's ratio, V_{ca}	E10.0
21-30		Poisson's ratio, V_{cb}	E10.0
31-40		Material axes option, AOPT	E10.0
1-10	Card 5	Yield stress in <i>a</i> -direction, σ_{ya}	E10.0
11-20		Material angle, β	E10.0
21-30		x_p for $AOPT = 1$ or a_x for $AOPT = 2$	E10.0
31-40		y_p for $AOPT = 1$ or a_y for $AOPT = 2$	E10.0
41-50		z_p for $AOPT = 1$ or a_z for $AOPT = 2$	E10.0
51-60		d_x for $AOPT = 2$ or v_x for $AOPT = 3$	E10.0
61-70		d_y for $AOPT = 2$ or v_y for $AOPT = 3$	E10.0
71-80		d_z for $AOPT = 2$ or v_z for $AOPT = 3$	E10.0
1-10	Card 6	Hardening modulus in <i>a</i> -direction, E_p^a	E10.0
11-20		Shear modulus, G_{bc}	E10.0
21-30		Shear modulus, G_{ab}	E10.0
31-40		Shear modulus, G_{ac}	i.
41-50		blank	
51-60		blank	

DYNA3D User Manual

Columns		Quantity	Format
61-70		Number of plasticity substeps, ISS	E10.0
1-10	Card 7	First tabulated plastic strain in <i>a</i> -direction, ε_a^p	E10.0
11-20		Second tabulated plastic strain in <i>a</i> -direction, ε_a^p	E10.0
•			•
•		•	•
•			•
71-80		Eighth tabulated plastic strain in <i>a</i> -direction, ε_a^p	E10.0
1-10	Card 8	First tabulated yield stress in <i>a</i> -direction, σ_{ya}	E10.0
•			•
•			•
•			•
71-80		Eighth tabulated yield stress in <i>a</i> -direction, σ_{xa}	E10.0

This model combines the orthotropic elasticity of Material Type 2 with the 1948 Hill orthotropic plasticity model described in (Hill, 1950). The numerical implementation at present is described in (Logan, 1988). This approach is based on a straightforward substepping method described in (Bathe, 1982) and elsewhere. The number of elastic-plastic substeps, *ISS*, defaults to *ISS* = 1, and can be overridden by user input. Additional substeps may be required for certain loading rates, strain paths, and degrees of anisotropy, so some caution is required.

It is most convenient to chose a reference direction when using the 1948 Hill theory, and in this model the *a*-direction is chosen. Thus, the tabulated plastic strain vs. yield stress values, if specified on cards 7 and 8, should refer to *uniaxial* material tests in the *a*-direction.

The elastic parameters and material axes definition option *AOPT* are specified as described for the Orthotropic Elastic model, Material Type 2.

This model may include linear strain hardening, or a tabulated nonlinear strain hardening law may optionally be specified. The linear strain hardening law has the form

$$\sigma_{y} = \sigma_{0} + E_{p}\overline{\varepsilon}^{p}, \qquad (147)$$

where the effective plastic strain $\overline{\varepsilon}^{p}$ is given by

$$\overline{\varepsilon}^{p} = \int_{0}^{t} d\overline{\varepsilon}^{p}, \qquad (148)$$

and the increment in effective plastic strain $d\vec{\varepsilon}^p$ is found from the incremental plastic strain tensor $d\varepsilon_{ij}^p$ as

$$(d\overline{\varepsilon}^{p})^{2} = \frac{R+1}{(1+R+P)^{2}R} \left[A+B+C\right] + \frac{R+1}{2} \left[\frac{(d\gamma_{bc}^{p})^{2}}{L} + \frac{(d\gamma_{ab}^{p})^{2}}{M} + \frac{(d\gamma_{ac}^{p})^{2}}{N}\right] (149)$$

where

$$A = P \left((1+R) d\varepsilon_{c}^{p} + R d\varepsilon_{a}^{p} \right)^{2}$$

$$B = R \left((1+P) d\varepsilon_{a}^{p} + P d\varepsilon_{c}^{p} \right)^{2}$$

$$C = \left(R d\varepsilon_{a}^{p} - P d\varepsilon_{c}^{p} \right)^{2}$$
(150)

This expression evolves directly from the use of the effective stress in the 1948 Hill theory. Using the conventional variables F, G, H, L, M, and N for clarity, the effective stress may be written as

$$\overline{\sigma}^2 = \frac{F(\sigma_c - \sigma_b)^2 + G(\sigma_b - \sigma_a)^2 + H(\sigma_a - \sigma_c)^2 + D}{R+1},$$
(151)

where

$$D = 2L\sigma_{ac}^{2} + 2M\sigma_{ab}^{2} + 2N\sigma_{bc}^{2}.$$
 (152)

In the above equations, F, G, and H are found in terms of the anisotropy parameters R and P as

$$F = \frac{R}{P},\tag{153}$$

$$G = 1, \tag{154}$$

and

 $H = R. \tag{155}$

The shear terms L, M, and N are found using

$$L = (Q_{bc} + \frac{1}{2}) (R + Z)$$
(156)

$$M = (Q_{ba} + \frac{1}{2}) (R + Z), \qquad (157)$$

$$N = (Q_{ca} + \frac{1}{2}) (Z + 1).$$
(158)

The anisotropy parameters R, P, Q_{bc} , Q_{ba} , and Q_{ca} can be determined from simple uniaxial material tension tests in orthogonal directions. Using the local coordinate system defined above,

DYNA3D User Manual

(159)

INPUT FORMAT

where ε_b and ε_c are the transverse strains measured in a uniaxial tension test in the *a*-direction. Similarly,

 $R=\frac{\varepsilon_c}{\varepsilon_b},$

S.

$$P = \frac{\varepsilon_a}{\varepsilon_b} \tag{160}$$

in a uniaxial tension test in the c-direction. In the b-c plane, the shear term Q_{bc} is found from

$$Q_{bc} = \frac{\varepsilon_{b^{-}}}{\varepsilon_{a}} \tag{161}$$

in a uniaxial tension test in the c^{""} direction. The b^{""} and c^{""} axes are defined by rotating the b and c axes by 45 degrees counterclockwise in the b-c plane. Similarly, in the a-b plane,

$$Q_{ba} = \frac{\varepsilon_{b^*}}{\varepsilon_c} \tag{162}$$

for a uniaxial test in the $a^{"}$ direction, where the $a^{"}$ and $b^{"}$ axes are rotated 45 degrees counterclockwise in the a-b plane. In the a-c plane, the shear term Q_{ca} is found from

$$Q_{ca} = \frac{\varepsilon_{c'}}{\varepsilon_b} \tag{163}$$

in a uniaxial tension test in the a' direction, where the a' and c' axes are defined by rotating the a and c axes counterclockwise by 45 degrees in the a-c plane.

In the present model, only isotropic strain hardening is included. The *a*-direction hardening modulus E_p^a can be written in terms of the *a*-direction tangent modulus E_T^a as

$$E_{p}^{a} = \frac{E_{a}E_{T}^{a}}{E_{a} - E_{T}^{a}},$$
(164)

where E_a is the elastic modulus in the *a*-direction. The plastic hardening modulus E_p^a is the slope of the stress vs. *plastic* strain curve in a uniaxial stress test, and the tangent modulus E_T^a is the slope of the inelastic portion of a uniaxial stress vs. strain curve.

A nonlinear strain hardening law may be specified by defining the σ_{ya} vs. ε_a^p relationship on cards 7 and 8. This data should be based on a uniaxial tension test in the *a*-direction. If this curve is defined, then ε_{a1}^p must be zero, and the yield and linear hardening data on cards 5 and 6 is not used.

Material Type 34	(Normal Anisotr	opic Elastic-Plastic	for Shells)
------------------	-----------------	----------------------	-------------

Columns	uns Quantity		Format
1-10	Card 3	Young's modulus, E	E10.0
1-10	Card 4	Poisson's ratio, v	E10.0
1-10	Card 5	Yield stress, σ_0	E10.0
1-10	Card 6	Tangent modulus, E_T	E10.0
1-10	Card 7	Anisotropy parameter, R	
	Card 8	Blank	

This model incorporates a version of Hill's model of anisotropic plasticity adapted for shell elements. The numerical algorithms used in this implementation are described in Whirley and Engelmann (1991). The yield criterion used in this model is specialized to normal anisotropy, where the properties do not vary with direction in the plane of the shell, but the transverse direction may have different properties. The anisotropy parameter R is the ratio of the in-plane plastic strain rate to the out-of-plane plastic strain rate. It can also be shown that

$$R = \frac{2}{K^2} - 1,$$
 (165)

where $K = \frac{\sigma_0}{\sigma_{03}}$ is the ratio of the in-plane initial yield stress to the initial yield stress in the transverse (through-thickness) direction. Note that the model reverts to isotropically elastic-plastic when R = 1.

Using the assumption of plane stress on each lamina of the shell, the yield function may be written

$$F(\sigma_{ij}) = \left[\sigma_{11}^2 + \sigma_{22}^2 - \frac{2R}{R+1}\sigma_{11}\sigma_{22} + 2\frac{2R+1}{R+1}\sigma_{12}^2\right]^{\overline{2}},$$
 (166)

where σ_{ij} denotes Cauchy stress in the local coordinates of the shell. Isotropic strain hardening is included, with the in-plane yield stress σ_{j} found from

$$\sigma_{y} = \sigma_{0} + E_{p} \overline{\varepsilon}^{p}, \qquad (167)$$

where E_p is the plastic hardening modulus and $\overline{\epsilon}^p$ is the effective plastic strain. The plastic hardening modulus is found from the input values of Young's modulus E and the tangent modulus E_T using

$$E_p = \frac{EE_T}{E - E_T}.$$
(168)

The plastic hardening modulus E_p is the slope of the inelastic portion of the effective stress $\overline{\sigma}$ vs. effective plastic strain $\overline{\varepsilon}^p$ curve, and the tangent modulus E_T is the slope of the inelastic part of a uniaxial stress vs. strain curve (or equivalently, the effective stress vs. effective strain curve).

Normal anisotropic plasticity behavior arises in many metals used in sheet-forming operations.

1 :

Material Type 35 (Elastic-Plastic with Forming Limit Diagram)

<u>Columns</u>		Quantity	Format
1-10	Card 3	Young's modulus, E	E10.0
11-20		Forming Limit Failure Criterion Option, IFLD	E10.0
1-10	Card 4	Poisson's ratio, v	E10.0
11-20		Load curve for left side of diagram, LCLH	E10.0
21-30		Load curve for right side of diagram, LCRH	E10.0
31-40		Load curve giving pressure dependence, LCPX	E10.0
41-50		Load curve giving rate dependence of FLD, LCEDF	E10.0
51-60		Load curve for rate dependent yield stress, LCEDM	E10.0
1-10	Card 5	Yield stress, σ_0	E10.0
11-20		Failure scale factor for deviatoric stresses, SCLDEV	E10.0
1-10	Card 6	Tangent modulus, E_T	E10.0
11-20		Hardening parameter, β	E10.0
1-10	Card 7	First tabulated effective plastic strain, $\overline{\varepsilon}_1^p$	E10.0
11-20		Second tabulated effective plastic strain, $\overline{\varepsilon}_2^p$	E10.0
•			•
•		•	•
•			•
71-80		Eighth tabulated effective plastic strain, $\overline{\varepsilon}_8^p$	E10.0
1-10	Card 8	First tabulated yield stress, σ_{y1}	E10.0
		•	•
•			•
			•
71-80		Eighth tabulated yield stress, σ_{y8}	E10.0

This model combines the elastic-plastic Material Type 3 with a Forming Limit Diagram (FLD) failure model developed by Logan and described in (Logan, 1991). The constitutive behavior of the material is exactly as in Material Type 3, with one exception (rate-dependent yield) which is

DYNA3D User Manual



described below. The FLD method (Keeler, 1968) involves plotting the major (largest principal) . strain versus the minor (next largest) principal strain. When the major strain reaches the FLD limiting value, which is a function of the minor strain, material failure is predicted. Once failure is detected in this model, the yield stress σ_y and the tangent modulus E_T are reduced by the factor *SCLDEV*. In addition, a failure-fraction parameter F is set to one to indicate that failure has occurred at a material point.

The shape of the forming limit diagram is determined by the left-hand and right-hand load curves, *LCLH* and *LCRH*. The first column of each these load curves gives the ratio of minor true strain to major true strain, R_{mm} , and the second column gives the major true strain, ε_{maj} . The left-hand load curve should cover the minor/major strain ratio range of -0.5 to 0.0, and the right-hand curve should cover the strain ratio range from 0.0 to 1.0, as shown in Figure 13.



The pressure dependence of the forming limit diagram is determined by the pressure load curve, *LCPX*. Strictly speaking, the forming limit diagram is a plane-stress sheet forming concept. Thus, for any monotonic path on the forming limit diagram, there is an associated value of normalized pressure \hat{p} , where \hat{p} is given by

$$\hat{p} = \frac{p}{\sigma_y},\tag{169}$$

where p is the pressure and σ_y is the current yield stress. For uniaxial tension, $\hat{p} = -\frac{1}{3}$, and for biaxial tension $\hat{p} = -\frac{2}{3}$. Any additional normalized pressure, \hat{p} , beyond these standard values for plane stress conditions may cause the forming limit diagram to grow or shrink by an increment $d\varepsilon^{\hat{p}}_{maj}$ on each strain path R_{mm} . Load curve *LCPX* should contain pairs giving additional normalized pressure in the first column and the corresponding increment $d\varepsilon^{\hat{p}}_{maj}$ in the second column. Figure 14 illustrates the operation of the pressure-dependence load curve.

. .

DYNA3D User Manual



The size of the forming limit diagram (i.e., the limiting major strain for any given minor-to-major strain ratio) may be dependent on the total strain rate. The load curve *LCEDF* specifies this strain rate dependence of the forming limit diagram. The load curve should contain strain rates in the first column, and corresponding $d\varepsilon^{\dot{\varepsilon}_{maj}}$ values in the second column. These $d\varepsilon^{\dot{\varepsilon}_{maj}}$ are applied along all R_{mm} paths given in the left-hand and right-hand FLD load curves to derive the curves actually used in the calculation. The operation of the rate-dependence load curves is illustrated in Figure 15.



The material constitutive behavior differs from the elastic-plastic model, Material Type 3, in that the yield stress may be multiplied by a strain rate dependent factor found from load curve *LCEDM*. The first column of this load curve should contain total strain rate values, and the second column should give the scale factor to be multiplied times the current yield stress, σ_y . Note that this strain rate scaling is applied after strain hardening is taken into account.

The load curves for pressure (LCPX), strain rate dependence of the FLD (LCEDF), and strain rate dependence of the yield stress (LCEDM), are optional; if these are input as zero, the option is ignored and the appropriate scale factor becomes unity. Load curves for the left-hand FLD (LCLH) and the right-hand FLD (LCRH) are required and must contain at least two points each.

DYNA3D User Manual



The determination of failure or proximity to failure is made in one of three ways, depending on the value of the forming limit failure criterion option, *IFLD*. For the TOTAL FLD approach (*IFLD* = 1; see Figure 16), the total major and minor strains are compared to the limiting value given by the forming limit diagram, and the failure fraction parameter F is determined. The pressure used in pressure dependence option with this criterion is a "damped" pressure, which is a running average of the instantaneous pressures. The strain rate is similarly damped. This procedure approximates the "total" forming limit diagram method used in press shop practice. In the INCRE-MENTAL FLD approach (*IFLD* = 2; see Figure 17), the total value of the major strain is used to get the vertical position on the FLD. The current incremental value of minor-to-major strain ratio, dR_{mm} , is used to establish the R_{mm} line used. This R_{mm} line is traversed from the origin up to the current ε_{maj} value to establish the position on the FLD. The current pressure and strain rate are used for the pressure dependence and rate dependence options with this criterion, and this may lead to



sudden failure when the loading path changes, as is sometimes observed in practice. For the DAMAGE FLD approach (*IFLD* = 3; see Figure 18), the increment in major strain is considered along with the current pressure and strain-rate. An increment in failure fraction ΔF is found from

$$\Delta F = \frac{\Delta \varepsilon_{major}}{\Delta \varepsilon_{major}^{path}},\tag{170}$$

where $\varepsilon_{major}^{path}$ is the major strain to failure in the direction of $\Delta \varepsilon_{major}$ on the current forming limit diagram (after pressure and strain rate effects are incorporated). The failure fraction then accumulates according to

$$F_{n+1} = F_n + \Delta F, \tag{171}$$

where the subscripts denote time step number. Failure occurs when $F \ge 1.0$. This failure criterion option accounts for strain path changes in a way that is less prone to spurious failures due to numerical noise than is the INCREMENTAL FLD criterion.

.

10.00

· ;

Material Type 36 (Brittle Damage Model)

<u>Columns</u>		Quantity	<u> </u>
1-10	Card 3	Young's modulus, E	E10.0
1-10	Card 4	Poisson's ratio, v	E10.0
1-10	Card 5	Tensile limit, f_{t0}	E10.0
11-20		Shear limit, f_{s0}	E10.0
1-10	Card 6	Fracture Toughness, g_c	E10.0
1-10	Card 7	Shear retention, β	E10.0
1-10	Card 8	Viscosity, η	E10.0

This model was developed by Sanjay Govindjee of University of California, Berkeley.

166

INPUT FORMAT

-

*

Material Type 37 (Three-Invariant Viscoplastic Cap Model)

Columns		Quantity	Format
1-10	Card 3	Shear modulus, G	E10.0
11-20		Bulk modulus, K	E10.0
21-30		Gruneisen ratio (optional), Γ	E10.0
31-40		Shock parameter (optional), S_l	E10.0
41-50		Pore compression flag, <i>IPRES</i> EQ.0.0: Explicit pore compression EQ.1.0: Constant bulk modulus	E10.0
1-10	Card 4	Shear failure surface constant, α	E10.0
11-20		Shear failure surface linear coefficient, θ	E10.0
21-30		Shear failure surface exponential coefficient, γ	E10.0
31-40		Shear failure surface exponent, β	E10.0
41-50		Tensile pressure cutoff (negative in tension)	E10.0
51-60		Tensile return mapping mode	E10.0
61-70		Kinematic hardening parameter, N^{α}	E10.0
71-80		Kinematic hardening coefficient, c^{α}	E10.0
1-10	Card 5	Initial ellipticity, R_0	E10.0
11-20		Initial J_1 -axis intercept, X_0	E10.0
21-30		Cap contraction option, <i>IROCK</i> EQ.0.0: Contraction allowed (soils) EQ.1.0: Contraction omitted (rocks) EQ.2.0: Contraction w/ hardening (rocks)	E10.0
31-40		Shear-enhanced compaction parameter	E10.0
1-10	Card 6	Maximum plastic volume strain, W	E10.0
11-20		Exponent D_1	E10.0
21-30		Exponent D_2	E10.0
31-40		Plot variable output option (see Table 1)	E10.0
41-50		Maximum strain increment	E10.0
1-10	Card 7	Three-invariant parameter Q_1	E10.0

DYNA3D User Manual

()

Columns		Quantity	
11-20		Three-invariant parameter Q_2 GE.0.0: Formulation parameter LE.0.0: Friction angle, φ (degrees)	E10.0
21-30		Rounded vertices parameter, $\Delta\beta_0$ (degrees)	E10.0
31-40		Rounded vertices parameter, δ	E10.0
41-50		Viscoplasticity fluidity parameter	E10.0
51-60		Viscoplastic flow function form, NFORM LT.0.0: $\phi(f) = (f/f_0)^N$, where $N = NFORM $ GT.0.0: $\phi(f) = exp(f/f_0)^N - 1$	E10.0
	Card 8	Blank	

This model was developed by Len Schwer and Yvonne Murray and is described in (Schwer and Murray, 1993).

NPLOT	Function	Description
1	L(K)	J_1 value at cap-shear surface intersection
2	Χ(κ)	J_1 intercept of cap surface
3	<i>R</i> (<i>κ</i>)	Cap surface ellipticity
4	$\overline{\mathcal{E}}_{v}^{p}$	Plastic volume strain
5	J_1	First stress invariant
6	J_2^{i}	Second invariant of deviatoric stress
7	J' ₃	Third invariant of deviatoric stress
8	\hat{J}'_3	$(3\sqrt{1.5}J_3') / (J_2')^{3/2} = -\sin(3\beta)$
9	β	Lode angle (degrees)
10	R	Octahedral plane radius
11	J	Relative volume
12	φ	Porosity
13	\$\$ \$\$	Relative change in volume of solid phase

Table 1: Output variables for NPLOT plotting option.

1

NPLOT	Function	Description
14	P _{hs}	Pressure in the solid phase
15	E _{hs}	Energy in the solid phase
16	nsubs	Number of strain subincrements
17	$1 - (\mathcal{R}^2 F_f^2 F_c) / J_2'$	Deviation from failure surface
18	G^{lpha}	Kinematic hardening limiting function
19*	J_2^{α}	Kinematic hardening backstress

Table 1: Output variables for NPLOT plotting option.

2

7

1.1.1

• 7

/ t

Material Type 38 (Bammann Plasticity Model)

<u>Columns</u>	·	Quantity	Format
1-10	Card 3	Young's modulus, E	E10.0
11-20		Poisson's ratio, v	E10.0
21-30		Initial temperature, T_0	E10.0
31-40		Heat generation coefficient, HC	E10.0
1-10	Card 4	C_1	E10.0
11-20		<i>C</i> ₂	E10.0
21-30		C_3	E10.0
31-40		C_4	E10.0
1-10	Card 5	C_5	E10.0
11-20		C_6	E10.0
21-30		C_7	E10.0
31-40		C ₈ .	E10.0
41-50		<i>C</i> ₁₃	E10.0
51-60		<i>C</i> ₁₄	E10.0
1-10	Card 6	C ₉	E10.0
11-20		<i>C</i> ₁₀	E10.0
21-30		<i>C</i> ₁₅	E10.0
31-40		C ₁₆	E10.0
41-50		<i>C</i> ₁₁	E10.0
51-60		<i>C</i> ₁₂	E10.0
61-70		C ₁₇	E10.0
71-80		C ₁₈	E10.0
1-10	Card 7	Initial tensor internal variable, α_{xx}	E10.0
11-20		Initial α_{yy}	E10.0
21-30		Initial α_{xy}	E10.0
31-40		Initial α_{yz}	E10.0
41-50		Initial α_{zx}	E10.0
Columns		Quantity	Format
---------	--------	--	--------
51-60		Initial scalar internal variable, κ_0	E10.0
	Card 8	Blank	

This model is described in (Bammann, 1990) and (Bammann, Johnson, and Chiesa, 1990). It is a phenomenological plasticity model using a set of internal state variables whose evolution is based on micromechanics. The model includes rate and temperature dependence, and heat generation due to plastic work. Since internal state variables are used to track the deformation, the history effects of strain rate and temperature are correctly captured.

The number of material parameters may seem prohibitive, but rarely are all of the constants used. The model reduces to linear strain hardening with only two required parameters. For rate insensitive materials the number of parameters is reduced by four. If temperature dependence is not required (i.e. when heat generation is not important), then the number of parameters is reduced by a factor of two. All of the parameters can be determined using simple tension and compression data.

The evolution of the Cauchy stress σ is governed by an equation of the form

$$\dot{\boldsymbol{\sigma}} = \lambda Tr\left(d^{e}\right) \boldsymbol{I} + 2Gd^{e}, \qquad (172)$$

where d^{e} is the elastic part of the rate of deformation, λ is the elastic Lame parameter given by

$$\lambda = \frac{Ev}{(1+v)(1-2v)},$$
(173)

and G is the elastic shear modulus. The rate of deformation d (symmetric part of the velocity gradient) is decomposed as

$$d = d^{e} + d^{p} + d^{th}, (174)$$

where d^{p} is the deviatoric plastic part, and d^{th} is the thermal expansion part. The deviatoric plastic part of the rate of deformation is given by

$$d^{p} = f(T) \sinh\left[\frac{|\boldsymbol{\xi}| - \kappa - Y(T)}{V(T)}\right] \frac{\boldsymbol{\xi}}{|\boldsymbol{\xi}|} \text{ for } |\boldsymbol{\xi}| - \kappa - Y(T) \ge 0$$
(175)

and

$$d^{p} = 0 \text{ for } (|\xi| - \kappa - Y(T)) < 0, \tag{176}$$

where T is temperature, κ is a scalar hardening variable, ξ is the translated stress found from the deviatoric Cauchy stress s and the tensor hardening variable α as

$$\boldsymbol{\xi} \equiv \boldsymbol{s} - \frac{2}{3}\boldsymbol{\alpha}, \tag{177}$$

and f(T), Y(T), and V(T) are scalar functions. Assuming isotropic thermal expansion with coefficient \hat{a} , the thermal part of the rate of deformation can be written

$$d^{th} = \hat{a}\dot{T}\mathbf{1}.\tag{178}$$

The evolution of the internal plasticity variables α and κ is found from

$$\dot{\boldsymbol{\alpha}} = h(T) d^{p} - [r_{d}(T) \,\overline{d} + r_{s}(T)] \,\overline{\boldsymbol{\alpha}} \,\boldsymbol{\alpha}$$
(179)

$$\dot{\kappa} = H(T) \left| d^{\vec{p}} - \left[R_d(T) \, \overline{d} + R_s(T) \right] \, \kappa^2, \tag{180}$$

where h(T) and H(T) are hardening moduli (which may be functions of temperature),

$$\bar{d} = \sqrt{\frac{2}{3}} |d^{p}|, \qquad (181)$$

$$\overline{\alpha} = \sqrt{\frac{2}{3}} |\alpha|, \qquad (182)$$

and $r_s(T)$, $R_s(T)$, $r_d(T)$, and $R_d(T)$ are scalar functions.

To compute temperature change, it is assumed that no heat is conducted out of an element and 90% of the plastic work is dissipated as heat, so it follows that

$$\dot{T} = \frac{0.9}{\rho c_{\nu}} \left(\boldsymbol{\sigma} \bullet d^{P} \right), \qquad (183)$$

where ρ is the material density and c_{v} is the specific heat. To include this effect the heat generation coefficient, HC, should be defined in the input:

$$HC = \frac{0.9}{\rho c_v}.$$
 (184)

Nine functions are used to describe the inelastic response. They can be grouped into three classes: those associated with the initial yield stress, the hardening functions, and the recovery functions. The temperature dependence of the yield functions are given by

$$V(T) = C_1 exp(-C_2/T)$$
(185)

$$Y(T) = C_3 exp(C_4/T)$$
 (186)

$$F(T) = C_5 exp(-C_6/T).$$
(187)

The function Y(T) describes the rate independent yield strength as a function of temperature. The function F(T) determines the rate at which the material transitions from rate-insensitive to rate-dependent, and V(T) describes the amount of rate dependence.

DYNA3D User Manual

INPUT FORMAT

Two internal state variables are used to model hardening. The tensor variable α is used to describe the translation of the yield surface and the scalar variable κ is used to track growth of the yield surface. These two history variables evolve independently, and their evolution is characterized by a hardening contribution minus a recovery contribution. The hardening functions h(T) and H(T)are given by

$$h(T) = C_9 exp(C_{10}/T)$$
(188)

$$H(T) = C_{15} exp(C_{16}/T).$$
(189)

Without recovery terms the model reduces to linear hardening with a tangent modulus of

$$E_T = (E(h+H)) / (E+h+H).$$
(190)

There are two recovery functions associated with each of the state variables α and κ . Larger values of recovery result in faster deviation from linear hardening and lower saturation stresses. The dynamic recovery function results in rate-independent hardening while the static (or thermal) recovery results in rate-dependent hardening.

The recovery functions are strongly temperature-dependent, and their form is given by

$$r_d(T) = C_7 exp(-C_8/T)$$
(191)

$$r_s(T) = C_{11} exp\left(-C_{12}/T\right)$$
(192)

$$R_d(T) = C_{13} exp(-C_{14}/T)$$
(193)

$$R_s(T) = C_{17} exp(-C_{18}/T).$$
(194)

At higher strain rates and lower temperatures the dynamic recovery is dominant while at lower strain rates and higher temperatures the static recovery is dominant.

For high rate problems there can be a significant temperature increase due to plastic work. This allows the model to calculate thermal softening and thermal instabilities. Note that the heat generation coefficient *HC* will have no effect unless the functions are temperature-dependent. Typically, for strain rates less than 1.0 $\frac{1}{\sec}$ the problem is not adiabatic and therefore the heat generation coefficient should *not* be included.

The parameters that give initial values to the components of the internal variable α may often be defined as zero. Nonzero values may be used to describe a material that is not initially isotropic, such as material deformed by a rolling process.

1

ı

Material Type 39 (Sandia Damage Model)

<u>Columns</u>		Quantity	Format
1-10	Card 3	Young's modulus, E	E10.0
11-20		Poisson's ratio, v	E10.0
21-30		Initial temperature, T_0	E10.0
31-40		Heat generation coefficient, HC	E10.0
1-10	Card 4	C_1	E10.0
11-20		C_2	E10.0
21-30		C_3	E10.0
31-40		C_4	E10.0
1-10	Card 5	C_5	E10.0
11-20		C_6	E10.0
21-30		C_7	E10.0
31-40		C_8	E10.0
41-50		C ₁₃	E10.0
51-60		C_{14}	E10.0
1-10	Card 6	С,	E10.0
11-20		C_{10}	E10.0
21-30		C ₁₅	E10.0
31-40		C_{16}	E10.0
41-50		<i>C</i> ₁₁	E10.0
51-60		<i>C</i> ₁₂	E10.0
61-70		C ₁₇	E10.0
71-80		C ₁₈	E10.0
1-10	Card 7	Initial tensor internal variable, α_{xx}	E10.0
11-20		Initial α_{yy}	E10.0
21-30		Initial α_{xy}	E10.0
31-40		Initial α_{yz}	E10.0
41-50		Initial α_{zz}	E10.0

DYNA3D User Manual

Columns	<u> </u>	Quantity	Format
51-60		Initial scalar internal variable, κ_0	E10.0
61-70		Damage exponent, \overline{m}	E10.0
71-80		Initial void volume fraction (porosity), \mathcal{D}_0	E10.0
	Card 8	Blank	

The constitutive equations of this model are described in (Bammann, 1990) and (Bammann, Johnson, and Chiesa, 1990). The damage model is described in (Bammann, Chiesa, McDonald, Kawahara, Dike, and Revelli, 1990) and Bammann, Chiesa, Horstemeyer, and Weingarten, 1993). This is a phenomenological plasticity model using a set of internal state variables whose evolutionis based on micromechanics. The model includes rate and temperature dependence, and heat generation due to plastic work. Since internal state variables are used to track the deformation, the history effects of strain rate and temperature are correctly captured. Ductile failure in materials is predicted by the model using a void growth evolution law.

The number of material parameters may seem prohibitive, but rarely are all of the constants used. The model reduces to a linear strain hardening with only two required parameters. For rate insensitive materials the number of parameters is reduced by four. If temperature dependence is not required (i.e. when heat generation is not important), then the number of parameters is reduced by a factor of two. All of the parameters (except for the two associated with damage) can be determined using simple tension and compression data.

The evolution of the Cauchy stress σ is governed by an equation of the form

$$\dot{\boldsymbol{\sigma}} = \lambda \left(1 - \mathcal{D}\right) Tr\left(d^{e}\right) I + 2G \left(1 - \mathcal{D}\right) d^{e} - \frac{\mathcal{D}}{1 - \mathcal{D}} \boldsymbol{\sigma},$$
(195)

where d^{ϵ} is the elastic part of the rate of deformation, D is a scalar damage variable, λ is the elastic Lame parameter given by

$$\lambda = \frac{Ev}{(1+v)(1-2v)},$$
(196)

and G is the elastic shear modulus. The rate of deformation d (symmetric part of the velocity gradient) is decomposed as

$$d = d^{e} + d^{p} + d^{v} + d^{th}, (197)$$

where d^{p} is the deviatoric plastic part, d^{v} is the dilitational plastic part, and d^{th} is the thermal expansion part. The deviatoric plastic part of the rate of deformation is given by

$$d^{p} = f(T) \sinh\left[\frac{|\boldsymbol{\xi}| - \kappa - Y(T)(1-\mathcal{D})}{V(T)(1-\mathcal{D})}\right] \frac{\boldsymbol{\xi}}{|\boldsymbol{\xi}|} \text{ for } |\boldsymbol{\xi}| - \kappa - Y(T)(1-\mathcal{D}) \ge 0 \quad (198)$$

and

$$d^{p} = 0 \text{ for } (|\boldsymbol{\xi}| - \kappa - Y(T) (1 - \mathcal{D})) < 0,$$
(199)

where T is temperature, κ is a scalar hardening variable, ξ is the translated stress found from the deviatoric Cauchy stress s and the tensor hardening variable α as

$$\boldsymbol{\xi} \equiv \boldsymbol{s} - \frac{2}{3}\boldsymbol{\alpha}, \qquad (200)$$

and f(T), Y(T), and V(T) are scalar functions. The diliational plastic part of the rate of deformation depends only on the damage variable and is given by

$$d^{\nu} = \frac{\mathcal{D}}{1 - \mathcal{D}} \mathbf{1}.$$
 (201)

Assuming isotropic thermal expansion with coefficient \hat{a} , the thermal part of the rate of deformation can be written

$$d^{th} = \hat{a}\dot{T}\mathbf{1}.\tag{202}$$

The evolution of the internal plasticity variables $\boldsymbol{\alpha}$ and $\boldsymbol{\kappa}$ is found from

$$\dot{\boldsymbol{\alpha}} = h(T) d^{p} - [r_{d}(T) \,\overline{d} + r_{s}(T)] \,\overline{\boldsymbol{\alpha}} \,\boldsymbol{\alpha}$$
(203)

$$\dot{\kappa} = H(T) \left| d^{p} \right| - \left[R_{d}(T) \, \overline{d} + R_{s}(T) \, \right] \, \kappa^{2}, \tag{204}$$

where h(T) and H(T) are hardening moduli (which may be functions of temperature),

$$\overline{d} = \sqrt{\frac{2}{3}} |d^{p}|, \qquad (205)$$

$$\overline{\alpha} = \sqrt{\frac{2}{3}} |\boldsymbol{\alpha}|, \qquad (206)$$

and $r_s(T)$, $R_s(T)$, $r_d(T)$, and $R_d(T)$ are scalar functions.

The evolution of the damage parameter \mathcal{D} is given by

$$\dot{\mathcal{D}} = \chi \left[\frac{1}{\left(1 - \mathcal{D}\right)^{\overline{m}}} - \left(1 - \mathcal{D}\right) \right] \left| d^{t} \right|, \tag{207}$$

where χ is a stress triaxiality factor given by

$$\chi = \sinh\left[\frac{2\left(2\overline{m}-1\right)p}{\left(2\overline{m}+1\right)\overline{\sigma}}\right],\tag{208}$$

 \overline{m} is a void growth constant, p is pressure, and $\overline{\sigma}$ is effective stress.

•

To compute temperature change, it is assumed that no heat is conducted out of an element and 90% of the plastic work is dissipated as heat, so it follows that

$$\dot{T} = \frac{0.9}{\rho c_{\nu}} \left(\boldsymbol{\sigma} \bullet \boldsymbol{d}^{P} \right), \qquad (209)$$

where ρ is the material density and c_v is the specific heat. To include this effect the heat generation coefficient, *HC*, should be defined in the input:

$$HC = \frac{0.9}{\rho c_{\nu}}.$$
 (210)

Nine functions are used to describe the inelastic response. They can be grouped into three classes: those associated with the initial yield stress, the hardening functions, and the recovery functions. The temperature dependence of the yield functions are given by

$$V(T) = C_1 exp(-C_2/T)$$
(211)

$$Y(T) = C_3 exp(C_4/T)$$
 (212)

$$F(T) = C_5 exp(-C_6/T).$$
(213)

The function Y(T) describes the rate independent yield strength as a function of temperature. The function F(T) determines the rate at which the material transitions from rate-insensitive to rate-dependent, and V(T) describes the amount of rate dependence.

Two internal state variables are used to model hardening. A tensor variable α is used to describe the translation of the yield surface and a scalar variable κ is used to track growth of the yield surface. These two history variables evolve independently, and their evolution is characterized by a hardening contribution minus a recovery contribution. The hardening functions h(T) and H(T)are given by

$$h(T) = C_9 exp(C_{10}/T)$$
(214)

$$H(T) = C_{15} exp(C_{16}/T).$$
(215)

Without recovery terms the model reduces to linear hardening with a tangent modulus of

$$E_T = (E(h+H)) / (E+h+H).$$
(216)

There are two recovery functions associated with each of the state variables α and κ . Larger values of recovery result in faster deviation from linear hardening and lower saturation stresses. The dynamic recovery function results in rate-independent hardening while the static (or thermal) recovery results in rate-dependent hardening. The recovery functions are strongly temperature-dependent, and their form is given by

$$r_d(T) = C_7 exp(-C_8/T)$$
(217)

$$r_s(T) = C_{11} exp\left(-C_{12}/T\right)$$
(218)

$$R_d(T) = C_{13} exp(-C_{14}/T)$$
(219)

$$R_s(T) = C_{17} exp\left(-C_{18}/T\right).$$
(220)

At higher strain rates and lower temperatures the dynamic recovery is dominant while at lower strain rates and higher temperatures the static recovery is dominant.

For high rate problems there can be a significant temperature increase due to plastic work. This allows the model to calculate thermal softening and thermal instabilities. Note that the heat generation coefficient *HC* will have no effect unless the functions are temperature-dependent. Typically, for strain rates less than $1.0 \frac{1}{\text{sec}}$ the problem is not adiabatic and therefore the heat generation coefficient should *not* be included.

The parameters that give initial values to the components of the internal variable α may often be defined as zero. Nonzero values may be used to describe a material that is not initially isotropic, such as material deformed by a rolling process.

The evolution of the damage internal state variable \mathcal{D} is motivated by the Cocks-Ashby solution for the growth of a spherical void in a rate-dependent plastic material. Note that there is strong dependence on the ratio of mean stress to effective stress. Based on microscopic measurements of initial void volume fractions for metals, a value of 0.0001 is typically used for \mathcal{D}_0 . The value for \overline{m} must be determined from test data, but unfortunately a simple procedure has not been found. Typically a notch tensile test is modeled and \overline{m} is varied through a trial-and-error process until the correct strain to failure is produced. Other notch tests can then be used as validation. If no notch data exists, then a standard tensile test may be modeled and the strain at failure can be used (although this is less accurate). Most metals will fail at void fractions of several percent. However, at five percent damage, the growth rate is usually so large that a damage fraction of one (total material failure) is usually reached in a few time steps. For this reason the damage is limited to 0.99, at which point failure is assumed. The element stress and stiffness are inversely proportional to the damage, and at a damage level of 0.99 the element is essentially removed from the calculation.

DYNA3D User Manual

-

ţ

Ì.

0

Material Type 40 (Fahrenthold Brittle Damage)

~

<u>Columns</u>	s Quantity		Format
1-10	Card 3	Elastic modulus, E (see Figure 1 and Figure 2)	E10.0
11-20		Damage evolution coefficient, A	E10.0
21-30		Damage evolution exponent, k	E10.0
1-10	Card 4	Poisson's ratio, v	E10.0
	Card 5	Blank	
1-10	Card 6	Material axes definition option, AOPT	E10.0
	EQ.0.0:	locally orthotropic with material axes determined by element nodes n_1 , n_2 , and n_4 as shown in Figure 1. Cards 7 and 8 must be blank with this option.	
	EQ.1.0:	locally orthotropic with material axes determined by a point in space P and the global location of the element cer as shown in Figure 1. Note that d is parallel to the global z Card 8 below is blank.	iter, z-axis.
	EQ.2.0:	globally orthotropic with material axes determined by vectors defined on Cards 7 and 8. (See Figure 2).	
	EQ.3.0:	applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axis a by an angle β from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector (See Figure 2). The angle β is defined on Card 8, and may be overridden by specifying a value on the element card.	v
1-10	Card 7	x_p , define only if AOPT = 1.0	E10.0
11-20		y_p , define only if AOPT = 1.0	E10.0
21-30		z_p , define only if AOPT = 1.0	E10.0
1-10	Card 7	a_x , define only if AOPT = 2.0	E10.0
11-20		a_{y} , define only if AOPT = 2.0	E10.0
21-30		a_z , define only if AOPT = 2.0	E10.0
1-10	Card 7	v_x , define only if AOPT = 3.0	E10.0
11-20 ·		v_y , define only if AOPT = 3.0	E10.0

<u>Columns</u>	·	Quantity	Format
21-30		v_z , define only if AOPT = 3.0	E10.0
1-10	Card 8	d_x , define only if AOPT = 2.0	E10.0
11-20		d_y , define only if AOPT = 2.0	E10.0
21-30		d_z , define only if AOPT = 2.0	E10.0
1-10	Card 8	Material angle β , define only if AOPT=3.0 (may be overridden on the element card)	E10.0

This is an experimental model for damage of brittle materials, such as ceramics. The theory and initial implementation was developed by Fahrenthold (1991), and the details of the formulation are described therein. This model is currently experimental in DYNA3D, and care should be exercised in its use.

DYNA3D User Manual

. . .

Material Type 41 (Fabric with Damage)

Columns	Quantity		Format
1-10	Card 3	Elastic modulus in longitudinal direction, E_a	E10.0
11-20		Elastic modulus in transverse direction, E_b	E10.0
21-30		Elastic modulus in normal direction, E_c	E10.0
31-40		Bulk modulus of failed material, K_f (solid elements only)	E10.0
41-50		Normal tensile strength, S_n (solid elements only)	E10.0
51-60		Transverse shear strength, S_{bc} (solid elements only)	E10.0
61-70		Transverse shear strength, S_{ca} (solid elements only)	E10.0
1-10	Card 4	Poisson's ratio, V_{ba}	E10.0
11-20		Poisson's ratio, V_{ca}	E10.0
21-30		Poisson's ratio, V_{cb}	E10.0
31-40		Ratio of soft E to stiff E in any direction, δ	E10.0
41-50		Stress value at transition, σ_{sl}	E10.0
51-60		Fillet radius for transition, R_{fil}	E10.0
61-70		Minimum time step for element deletion, Δt_f	E10.0
1-10	Card 5	Shear modulus, G_{ab}	E10.0
11-20		Shear modulus, G_{bc}	E10.0
21-30		Shear modulus, G_{ca}	E10.0
1-10	Card 6	Material axes definition option, AOPT	E10.0
	EQ.0.0:	locally orthotropic with material axes determined by element nodes n_1 , n_2 , and n_4 as shown in Figure 1. Cards 7 and 8 are blank with this option.	
	EQ.1.0:	locally orthotropic with material axes determined by a point in space P and the global location of the element cer as shown in Figure 1. Card 8 below is blank.	nter,
	EQ.2.0:	globally orthotropic with material axes determined by vectors defined on Cards 7 and 8. (See Figure 2).	
		(options continued next page)	

<u>Columns</u>		Quantity	Format
	EQ.3.0:	applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the materia axis a by an angle β from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell permet vector (See	1 : v
		Figure 2). The angle β is defined as described below for a through-thickness integration point. In addition, a materia may be specified on each element card, and these values a	each al angle are added.
1-10	Card 7	x_p , define only if AOPT = 1.0	E10.0
11-20		y_p , define only if AOPT = 1.0	E10.0
21-30		z_p , define only if AOPT = 1.0	E10.0
1-10	Card 7	a_x , define only if AOPT = 2.0	E10.0
11-20		a_y , define only if AOPT = 2.0	E10.0
21-30		a_z , define only if AOPT = 2.0	E10.0
1-10	Card 7	v_x , define only if AOPT = 3.0	E10.0
11-20		v_y , define only if AOPT = 3.0	E10.0
21-30		v_z , define only if AOPT = 3.0	E10.0
1-10	Card 8	d_x , define only if AOPT = 2.0	E10.0
11-20		d_y , define only if AOPT = 2.0	E10.0
21-30		d_z , define only if AOPT = 2.0	E10.0
31-40		Shear strength on $a-b$ plane, S_c	E10.0
41-50		Longitudinal tensile strength along a axis, X_t	E10.0
51-60		Longitudinal tensile strength along b axis, Y_t	E10.0
61-70		Transverse compressive strength, Y_c	E10.0
71-80		Nonlinear shear stress parameter, α	E10.0

This experimental material model allows for the initial slack effect on stiffness in a woven fabric material under tension. It is experimental, and further refinement is expected to improve its general utility. A different fiber orientation may be specified at each through-thickness integration point for shell elements to represent complex weaves. This capability may be combined with the User-Defined Integration Rules option.

Additional Material Type 41 Input:

Define the material angle β for each of the through-the-thickness integration points for 4-node shell elements. This data must follow the Cross Section Properties data on Card 11.

Columns	Quantity	Format
1-10	Material angle at first integration point, $m{eta}_1$	E10.0
11-20	Material angle at second integration point, β_2	E10.0
21-30	Material angle at third integration point, β_3	E10.0
•	•	

71-80 Material angle at eighth integration point, β_8 E10.0

Continue on additional cards until NIP points have been defined, where NIP is the number of thickness integration points specified on Card 10 for shell elements. Material angle values may also be specified on the element cards. The material angle used in the calculation is the sum of the value specified above for each integration point and the value specified on the element card.

This model is based on an extension of the Composite Damage model, (material type 22) which accounts for the effects of loose weave in fabrics. The elastic portion of this model is bilinear in each direction, with the soft branch of the curve ($\sigma < \sigma_{sl}$) having elastic modulus $E_{sl} = \delta E$. A smooth transition with radius of curvature R_{fil} is used between the two moduli. The moduli are evaluated independently in each direction based on the normal stress in that direction.

The material behavior produced by this model is approximately orthotropic elastic, with the addition of a nonlinear shear stress term. A stress-based failure model is included to incorporate the effects of fiber breakage, matrix cracking, matrix crushing, and delamination. The failure criterion for fiber breakage is

$$e_{f} = \frac{max(0, \sigma_{aa}^{2})}{X_{t}^{2}} + \frac{\sigma_{ab}^{2} + \frac{3}{4}\alpha\sigma_{ab}^{4}}{\frac{S_{c}^{2}}{2G_{ab}} + \frac{3}{4}\alpha S_{c}^{4}} \le 1.0.$$
(221)

Analogously, the criterion for matrix cracking is

$$e_{m} = \frac{max\left(0, \sigma_{bb}^{2}\right)}{Y_{t}^{2}} + \frac{\sigma_{ab}^{2} + \frac{3}{4}\alpha\sigma_{ab}^{4}}{\frac{S_{c}^{2}}{2G_{ab}} + \frac{3}{4}\alpha S_{c}^{4}} \le 1.0.$$
(222)

The criterion to detect matrix crushing is

$$e_{d} = \frac{\min(0, \sigma_{bb}^{2})}{4S_{c}^{2}} + \left(\frac{Y_{c}^{2}}{4S_{c}^{2}} - 1\right)\frac{\sigma_{bb}}{Y_{c}} + \frac{\sigma_{ab}^{2} + \frac{5}{4}\alpha\sigma_{ab}^{4}}{\frac{S_{c}^{2}}{2G_{ab}} + \frac{3}{4}\alpha S_{c}^{4}} \le 1.0,$$
(223)

and, for solid elements only, the criterion to detect delamination is

$$e_{l} = \frac{max\left(0, \sigma_{cc}^{2}\right)}{S_{n}^{2}} + \frac{\sigma_{bc}^{2}}{S_{bc}^{2}} + \frac{\sigma_{ca}^{2}}{S_{ca}^{2}} \le 1.0.$$
(224)

When failure is detected, the appropriate stress components are reduced to zero linearly over the next 100 time steps. This stress reduction function has been found useful to minimize spurious oscillations, or ringing, caused by the abrupt release of energy at failure.

12.1

EQUATION OF STATE DEFINITION

An equation-of-state defines the volumetric behavior of a material, and must be used *only* in combination with a hydrodynamic material model for solid elements.

Define equation-of-state cards only for Material Types 8, 9, 10, 11, 15, 16, and 17.



This equation-of-state title will appear in the printed output. It is often helpful to use this heading to identify the physical material for which the DYNA3D equation of state model was constructed.

Cards 10, . . .

Equation-of-State Form 1 (Linear Polynomial)

Columns		Quantity	Format
1-10	Card 10	Pressure constant, C ₀	E10.0
11-20		Linear compression coefficient, C1	E10.0
21-30		Quadratic compression coefficient, C2	E10.0
31-40		Cubic compression coefficient, C3	E10.0
41-50		First energy coefficient, C ₄	E10.0
51-60		Second energy coefficient, C5	E10.0
61-70		Third energy coefficient, C ₆	E10.0
71-80		Initial internal energy, E_0	E10.0
1-10	Card 11	Initial relative volume, V_0	E10.0

The linear polynomial equation-of-state is linear in internal energy. The pressure is given by:

$$p = C_0 + C_1 \mu + C_2 \overline{\mu}^2 + C_3 \mu^3 + (C_4 + C_5 \mu + C_6 \overline{\mu}^2) E$$
(225)

where the excess compression μ is given by

$$\mu \equiv \frac{\rho}{\rho_0} - 1, \qquad (226)$$

E is the internal energy, ρ is the current density, and ρ_0 is the initial density. The tension-limited excess compression $\overline{\mu}$ is given by

$$\overline{\mu} = max(\mu, 0). \tag{227}$$

Relative volume is related to excess compression and density by

$$V = \frac{1}{1+\mu} = \frac{\rho_0}{\rho}.$$
 (228)

This equation-of-state is linear in internal energy, and may be used to fit experimental data for many materials. If $C_i = K$ (the elastic bulk modulus) and all other $C_i = 0$, then linear elastic volumetric response is obtained.

This form of equation-of-state was adapted from that discussed in (Woodruff, 1973).

DYNA3D User Manual

INPUT FORMAT

Equation-of-State Form 2 (JWL)

. . ·

Columns		Quantity	Format
1-10	Card	Α	E10.0
11-20		B	E10.0
21-30		R_1	E10.0
31-40		R_2	E10.0
41-50		ω	E10.0
51-60		Initial internal energy, E_0	E10.0
61-70		Initial relative volume, V_0	E10.0

The JWL equation-of-state defines the pressure as

$$p = A \left(1 - \frac{\omega}{R_1 V}\right) e^{-R_1 V} + B \left(1 - \frac{\omega}{R_2 V}\right) e^{-R_2 V} + \frac{\omega E}{V},$$
(229)

where V is relative volume and E is internal energy.

The JWL equation of state is often used for detonation products of high explosives. Additional information in given in (Dobratz, 1981).

Columns		Quantity	Format
1-10	Card 10	A_1	E10.0
11-20		A_2	E10.0
21-30		A ₃	E10.0
31-40		B_1	E10.0
41-50		B ₂	E10.0
51-60		Initial internal energy, E_0	E10.0
61-70		Initial relative volume, V_0	E10.0

Equation-of-State Form 3 (Sack)

The Sack equation-of-state defines pressure p as

$$p = \frac{A_3}{V^{A_1}} e^{-A_2 V} \left(1 - \frac{B_1}{V}\right) + \frac{B_2}{V} E,$$
(230)

where V is relative volume and E is the internal energy.

This equation-of-state form is often used for detonation products of high explosives, and was adapted from (Woodruff, 1973).

DYNA3D User Manual

Columns	. <u> </u>	Quantity	Format
1-10	Card 10	Velocity curve intercept, C	E10.0
11-20		First slope coefficient, S_1	E10.0
21-30		Second slope coefficient, S_2	E10.0
31-40		Third slope coefficient, S_3	E10.0
41-50		Gruneisen coefficient, γ_0	E10.0
51-60		First order volume correction coefficient, a	E10.0
61-70		Initial internal energy, E_0	E10.0
71-80		Initial relative volume, V_0	E10.0

Equation-of-State Form 4 (Gruneisen)

The Gruneisen equation-of-state with cubic shock velocity-particle velocity defines pressure for compressed materials ($\mu > 0$) as

$$p = \frac{\rho_0 C^2 \mu \left[1 + (1 - \frac{\gamma_0}{2}) \mu - \frac{a}{2} \mu^2 \right]}{\left[1 - (S_1 - 1) \mu - S_2 \frac{\mu^2}{\mu + 1} - S_3 \frac{\mu^3}{(\mu + 1)^2} \right]^2} + (\gamma_0 + a\mu) E,$$
(231)

and for expanded materials ($\mu < 0$) as

$$p = \rho_0 C^2 \mu + (\gamma_0 + a\mu) E, \qquad (232)$$

where C is the intercept of the shock velocity vs. particle velocity $(v_s - v_p)$ curve, S_1 , S_2 , and S_3 are the coefficients of the slope of the $v_s - v_p$ curve, γ_0 is the Gruneisen gamma, and a is the first order volume correction to γ_0 . The excess compression μ is defined by

$$\mu \equiv \frac{\rho}{\rho_0} - 1, \tag{233}$$

where ρ is the current density and ρ_0 is the initial density.

The implementation of the Gruneisen equation-of-state is adapted from (Woodruff, 1973).

Equation-of-State Form 5 (Ratio of Polynomials)

<u>Columns</u>	. <u> </u>	Quantity	Format
1-16	Card 10	A ₁₀	E16.0
17-32		A ₁₁	E16.0
33-48		A ₁₂	E16.0
49-64		A ₁₃	E16.0
1-16	Card 11	A ₂₀	E16.0
17-32			E16.0
33-48		A ₂₂	E16.0
49-64		A ₂₃	E16.0
1-16	Card 12	A ₃₀	E16.0
17-32		A ₃₁	E16.0
33-48		A ₃₂	E16.0
49-64		A ₃₃	E16.0
1-16	Card 13	A ₄₀	E16.0
17-32		A ₄₁	E16.0
33-48		A ₄₂	E16.0
49-64		A ₄₃	E16.0
1-16	Card 14	A ₅₀	E16.0
17-32		A ₅₁	E16.0
33-48		A ₅₂	E16.0
49-64		A ₅₃	E16.0
1-16	Card 15	A ₆₀	E16.0
17-32		A ₆₁	E16.0
33-48		A ₆₂	E16.0
49-64		A ₆₃	E16.0

INPUT FORMAT

DYNA3D User Manual

Columns		Quantity	Format
1-16	Card 16	A ₇₀	E16.0
17-32		A ₇₁	E16.0
33-48		A ₇₂	E16.0
49-64		A ₇₃	E16.0
1-16	Card 17	α	E16.0
17-32		β	E16.0
33-48		A ₁₄	E16.0
49-64		A ₂₄	E16.0
1-16	Card 18	Initial internal energy, E_0	E16.0
17-32		Initial relative volume, V_0	E16.0

The ratio of polynomials equation-of-state defines the pressure as

$$p = \frac{F_1 + F_2 E + F_3 E^2 + F_4 E^3}{F_5 + F_6 E + F_7 E^2} (1 + \alpha \mu)$$
(234)

where the each of the F_i are polynomials in terms of the excess compression μ of the form

$$F_{i} = \sum_{k=0}^{n} A_{ik} \mu^{k}$$
(235)

with n = 4 for F_1 and F_2 , and n = 3 for F_3 through F_7 . The excess compression μ is defined by

$$\mu \equiv \frac{\rho}{\rho_0} - 1, \tag{236}$$

where ρ is the current density and ρ_0 is the initial density.

In expanded elements ($\mu < 0$), F_1 in (234) is replaced by \overline{F}_1 , where \overline{F}_1 is defined by

$$\overline{F}_1 = F_1 + \beta \mu^2. \tag{237}$$

This equation-of-state was adapted from (Woodruff, 1973).

Equation-of-State Form 6 (Linear Polynomial with Energy Deposition)

Columns		Quantity	<u>Format</u>
1-10	Card 10	Pressure constant, C_0	E10.0
11-20		Linear compression coefficient, C_1	E10.0
21-30		Quadratic compression coefficient, C_2	E10.0
31-40		Cubic compression coefficient, C_3	E10.0
41-50		First energy coefficient, C_4	E10.0
51-60		Second energy coefficient, C_5	E10.0
61-70		Third energy coefficient, C_6	E10.0
71-80		Initial internal energy, E_0	E10.0
1-10	Card 11	Initial relative volume, V_0	E10.0
11-20		Load curve number giving energy deposition rate, LC	E10.0

This equation-of-state form is similar to equation-of-state form 1 except that this form allows internal energy to be deposited into the material at a specified rate. The pressure is given by:

$$p = C_0 + C_1 \mu + C_2 \overline{\mu}^2 + C_3 \mu^3 + (C_4 + C_5 \mu + C_6 \overline{\mu}^2) E$$
(238)

where the excess compression μ is given by

$$\mu \equiv \frac{\rho}{\rho_0} - 1, \tag{239}$$

E is the internal energy, ρ is the current density, and ρ_0 is the initial density. The tension-limited excess compression $\overline{\mu}$ is given by

$$\overline{\mu} = max(\mu, 0). \tag{240}$$

Relative volume is related to excess compression and density by

$$V = \frac{1}{1+\mu} = \frac{\rho_0}{\rho}.$$
 (241)

Internal energy is added into the material at a rate specified by load curve LC. This feature allows energy transfer mechanisms to be included which are not considered in detail in the analysis model.

This equation-of-state is linear in internal energy, and may be used to fit experimental data for many materials.

INPUT FORMAT

DYNA3D User Manual

÷

Columns		Quantity	Format
1-10	Card 10	A_p	E10.0
11-20		B_p	E10.0
21-30		R_{1p}	E10.0
31-40		R_{2p}	E10.0
41-50		Second ignition coefficient, G	E10.0
1-10	Card 11	$\omega_p c_p$	E10.0
11-20		A _e	E10.0
21-30		B _e	E10.0
31-40		$\omega_{\epsilon}c_{\epsilon}$	E10.0
41-50		R_{1e}	E10.0
1-10	Card 12	R_{2e}	E10.0
11-20		Critical fraction reacted, F_{crit} , (usually = 1.0)	E10.0
21-30		First ignition coefficient, I	E10.0
31-40		Growth coefficient, H	E10.0
41-50		Pressure exponent, z	E10.0
1-10	Card 13	Unreacted fraction exponent, x	E10.0
11-20		Reacted fraction exponent, y	E10.0
21-30		Heat capacity of reaction products, c_p	E10.0
31-40		Heat capacity of unreacted HE, c_e	E10.0
1-10	Card 14	Product compression exponent, m (often $m = 0$)	E10.0
11-20		Initial energy of HE per unit volume, E_0	E10.0
21-30		Initial temperature, T_0 (in $^{\circ}K$)	E10.0
	Card 15	Blank	

Equation-of-State Form 7 (Ignition and Growth of Reaction in HE)

The pressure in the unreacted HE is given by a JWL equation of state,

$$p_{e} = A_{e} \left(1 - \frac{\omega_{e}}{R_{1e}V_{e}}\right) e^{-R_{1e}V_{e}} + B_{e} \left(1 - \frac{\omega_{e}}{R_{2e}V_{e}}\right) e^{-R_{2e}V_{e}} + \frac{\omega_{e}E_{e}}{V_{e}}, \qquad (242)$$

.

and the pressure in the reaction products is defined by another JWL equation of state,

$$p_{p} = A_{p} \left(1 - \frac{\omega_{p}}{R_{1p}V_{p}}\right) e^{-R_{1p}V_{p}} + B_{p} \left(1 - \frac{\omega_{p}}{R_{2p}V_{p}}\right) e^{-R_{2p}V_{p}} + \frac{\omega_{p}E_{p}}{V_{p}}, \qquad (243)$$

where p is the pressure, V is the relative volume, and E is the internal energy. Subscript e denotes quantities applicable to the unreacted HE, and subscript p indicates quantities applicable to reacted HE products.

The mixture of unreacted explosive and reaction products is defined by the fraction reacted, F (F = 0 indicates no reaction, while F = 1 indicates complete conversion from explosive to products). The pressures and temperature are assumed to be in equilibrium and the volumes are assumed to be additive,

$$V = (1 - F) V_e + F V_p.$$
(244)

The reaction rate is given by

$$\dot{F} = I \left(F_{crit} - F \right)^{y} \left(V_{e}^{-1} - 1 \right)^{3} \left[1 + G \left(V_{e}^{-1} - 1 \right) \right] + H \left(1 - F \right)^{y} F^{x} P^{z} \left(V_{p}^{-1} - 1 \right)^{m}.$$
 (245)

This equation-of-state is based in part on work reported in (Woodruff, 1973). The JWL equations of state and the reaction rates have been fitted to one- and two-dimensional shock initiation and detonation data for four explosives: PBX-9404, RX-03-BB, PETN, and cast TNT. The details of the calculational method are described in (Cochran and Chan, 1979). The detailed one-dimensional calculations and parameters for the four explosives are given in (Lee and Tarver, 1980).

INPUT FORMAT

DYNA3D User Manual

· ·

Equation-of-State Form 8 (Tabulated with Compaction)

<u>Columns</u>		Quantity	Format
1-16	Card 10	${\cal E}_{\nu 1}$	E16.0
17-32		Ev2	E16.0
33-48		$\mathcal{E}_{\nu 3}$	E16.0
49-64		${\cal E}_{\nu 4}$	E16.0
65-80		E _{v5}	E16.0
1-16	Card 11	εν6	E16.0
17-32		εν7	E16.0
33-48		${\cal E}_{\nu 8}$	E16.0
49-64		E _{v9}	E16.0
65-80		\mathcal{E}_{v10}	E16.0
1-16	Card 12	C_1	E16.0
17-32		C_2	E16.0
33-48		C_3	E16.0
49-64		C_4	E16.0
65-80		C_5	E16.0
1-16	Card 13	C_6	E16.0
17-32	•	<i>C</i> ₇	E16.0
33-48		C_8	E16.0
49-64		С,	E16.0
65-80		<i>C</i> ₁₀	E16.0
1-16	Card 14	First temperature, T_1	E16.0
17-32		Second temperature, T_2	E16.0
33-48		T_3	E16.0
49-64		T_4	E16.0
65-80		T_5	E16.0
1-16	Card 15	T ₆	E16.0

<u>Columns</u>		Quantity		Format
17-32		T_7		E16.0
33-48		T_8		E16.0
49-64		T_{9}		E16.0
65-80		Tenth temperature, T_{10}		E16.0
1-16	Card 16	First unloading bulk modulus, K_1	,	E16.0
17-32		Second unloading bulk modulus, K_2		E16.0
33-48		K ₃		E16.0
49-64		K_4		E16.0
65-80		K _s		E16.0
1-16	Card 17	K ₆		E16.0
17-32		K_7		E16.0
33-48		K_8		E16.0
49-64		K ₉		E16.0
65-80		Tenth unloading bulk modulus, K_{10}		E16.0
1-16	Card 18	γ	ž.	E16.0
17-32		Initial internal energy, E_0	-	E16.0
33-48		Initial relative volume, V_0		E16.0

Pressure is positive in compression, and volumetric strain ε_{ν} is positive in tension. The tabulated compaction model is linear in internal energy. Pressure is defined by

$$p = C(\varepsilon_{\nu}) + \gamma T(\varepsilon_{\nu}) E \tag{246}$$

during loading (compression). Note that in (246) $C(\varepsilon_v)$ denotes a function evaluation, not a multiplication. Thus, the input coefficients C_i have the units of pressure. Unloading occurs at a slope corresponding to the bulk modulus at the peak (most compressive) volumetric strain, as shown in Figure 19. Reloading follows the unloading path to the point where unloading began, and then continues on the loading path described by (246).

The volumetric strain is found from the relative volume V as

$$\varepsilon_{v} = \ln\left(V\right). \tag{247}$$

The tabulated functions may contain from 2 to 10 points, and the model will extrapolate (using the last two points) to find the pressure if required.



Ľ

Equation-of-State Form 9 (Tabulated)

<u>Columns</u>		Quantity	Format
1-16	Card 10	$\mathcal{E}_{\mathbf{v}1}$	E16.0
17-32		\mathcal{E}_{v2}	E16.0
33-48		E _{v3}	E16.0
49-64		${\cal E}_{\nu 4}$	E16.0
65-80		\mathcal{E}_{v5}	E16.0
1-16	Card 11	${\cal E}_{ m v6}$	E16.0
17-32		${\cal E}_{\nu 7}$	E16.0
33-48		${\cal E}_{\nu 8}$	E16.0
49-64		E _v 9	E16.0
65-80		\mathcal{E}_{v10}	E16.0
1-16	Card 12	C_1	E16.0
17-32		C_2	E16.0
33-48		C_3	E16.0
49-64		C_4	E16.0
65-80		C_5	E16.0
1-16	Card 13	C_6	E16.0
17-32		C_{7}	E16.0
33-48		C_8	E16.0
49-64		C ₉	E16.0
65-80		<i>C</i> ₁₀	E16.0
1-16	Card 14	First temperature, T_1	E16.0
17-32		Second temperature, T_2	E16.0
33-48		T_3	E16.0
49-64		T_4	E16.0
65-80		T ₅	E16.0

DYNA3D User Manual

INPUT FORMAT

Columns		Quantity	Format
1-16	Card 15	T ₆	E16.0
17-32		T_7	E16.0
33-48		T ₈	E16.0
49-64		T ₉	E16.0
65-80		Tenth temperature, T_{10}	E16.0
1-16	Card 16	γ	E16.0
17-32		Initial internal energy, E_0	E16.0
33-48		Initial relative volume, V_0	E16.0

Pressure p is positive in compression, and volumetric strain ε_{ν} is positive in tension. The tabulated compaction model is linear in internal energy. Pressure is defined by

$$p = C(\varepsilon_{\nu}) + \gamma T(\varepsilon_{\nu}) E, \qquad (248)$$

where E is internal energy. Note that in (246) $C(\varepsilon_v)$ denotes a function evaluation, not a multiplication. Thus, the input coefficients C_i have the units of pressure.

The volumetric strain is found from the relative volume V as

$$\varepsilon_{v} = \ln\left(V\right). \tag{249}$$

The tabulated functions may contain from 2 to 10 points, and the model will extrapolate to find the pressure if required.

.*

Į

Equation-of-State Form 10 (Not Available)

Columns Quantity Format

Equation-of-state form 10 is not used in this version.

DYNA3D User Manual

Equation-of-State Form 11 (Pore Collapse)

Card 10

Columns	Quantity	Format
1-5	Number of Virgin Loading Curve points, NLD	15
6-10	Number of Completely Crushed Curve points, NCR	15
11-20	Excess Compression required before any pores can collapse, μ_1	E10.0
21-30	Excess Compression point where the Virgin Loading Curve and the Completely Crushed Curve intersect, μ_2	E10.0
31-40	Initial Internal Energy, E_0	E10.0
41-50	Initial Excess Compression, μ_0	E10.0

Virgin Loading Curve Definition

Cards 11 thru NLD+10

Columns	Quantity	Format
1-15	Excess Compression, μ	E15.0
16-30	Corresponding pressure, p	E15.0

Continue on additional Virgin Loading Curve Definition Cards until all *NLD* pairs have been defined.

Completely Crushed Curve Definition

	Cards NLD+11 thru NLD+NCR+10	
Columns	Quantity	Format
1-15	Excess Compression, μ	E15.0
16-30	Corresponding pressure, p	E15.0

Continue on additional Completely Crushed Curve Definition cards until all NCR points have been defined.

The Pore Collapse equation-of-state is based on (Burton, 1982) and uses two curves: the virgin loading curve and the completely crushed curve, as shown in Figure 20. Two critical points are defined: the excess compression point required for pore collapse to begin (μ_1), and the excess compression point required to completely crush the material (μ_2). From this data and the maximum compression the material has attained (μ_{max}), the pressure for any excess compression μ can be determined. Unloading occurs along the virgin loading curve until the excess compression surpasses μ_1 . After that, the unloading follows a path between the completely crushed curve and the virgin loading curve. Reloading will follow this curve back up to the virgin loading curve. Once the excess compression exceeds μ_2 , then all unloading will follow the completely crushed curve.



For unloading between μ_1 and μ_2 a partially crushed curve is determined by the relationship:

$$p_{pc}(\mu) = p_{cc} \left(\frac{(1+\mu_B) (1+\mu)}{1+\mu_{max}} - 1 \right), \qquad (250)$$

where

DYNA3D User Manual

INPUT FORMAT

$$\mu_B = p_{cc}^{-1} (p_{max}) \tag{251}$$

is the excess compression corresponding to a pressure of p_{max} on the completely crushed curve. In the above, subscript pc refers to the partially crushed state and subscript cc refers to the completely crushed state.

In terms of the relative volume V,

$$V = \frac{1}{1+\mu},\tag{252}$$

$$p_{pc}(V) = p_{cc}\left(\frac{V_B}{V_{min}}V\right).$$
 (253)

Thus, for a fixed $V_{min} = \frac{1}{\mu_{max} + 1}$, the partially crushed curve will separate linearly from the completely crushed curve as V increases to account for pore recovery in the material.

The bulk modulus K is determined as one plus the excess compression times the slope of the current curve,

$$K = (1+\mu)\frac{\partial p}{\partial \mu}.$$
 (254)

It then follows that the slope for the partially crushed curve is

$$\frac{\partial p}{\partial \mu} = \frac{(1+\mu_B)}{1+\mu_{max}} \frac{\partial p_{cc}}{\partial \mu} \left(\frac{(1+\mu_B)(1+\mu)}{(1+\mu_{max})} \right).$$
(255)

The bulk sound speed is determined from the slope of the completely crushed curve at the current pressure to avoid instabilities. The virgin loading and completely crushed curves are modeled with monotonic cubic splines. An optimal vectorized interpolation scheme is then used to evaluate the cubic splines as required during the solution. The bulk modulus and sound speed are derived from a linear interpolation on the derivatives of the cubic splines.

Cross Section Property Definition for Structural Elements

Define structural element cross section parameters if this material applies to truss, beam, shell, or thick shell elements.

	Card 9	
Columns	Quantity	Format
1-10	Cross section identification	12A6

This title will be printed with the Cross Section Properties in the hsp printout file. It is useful for labeling the physical cross section for which this property set is applicable.

DYNA3D User Manual



{

Truss Element Cross Section Properties

<u>Columns</u>	Quantity		Format
	Card 10	Blank	
1-10	Card 11	Cross-sectional area, A	E10.0

The geometry of DYNA3D truss elements is shown in Figure 21. Note that the position of the orientation node, node n_3 , is not significant for truss elements. Node n_3 must be defined in the input, however.

This is a simple truss element, which transmits only axial force.
_ ^.

ž

••••

<u>,</u>

.

1.12

Hughes-Liu Beam Element Cross Section Properties

,

Columns		Quantity	Format
1-10	Card 10	Shear factor, κ (default is $\kappa = 1.0$)	E10.0
11-20		 Quadrature rule for beam element cross section LT.0: absolute value is User-Defined Integration Rule number EQ.1.0: degenerate beam (truss) EQ.2.0: 2 × 2 Gauss quadrature (default) EQ.3.0: 3 × 3 Gauss quadrature EQ.4.0: 3 × 3 Lobotto quadrature EQ.5.0: 4 × 4 Gauss quadrature 	E10.0
21-30		Cross section type, <i>ICROSS</i> EQ.0.0: rectangular EQ.1.0: tubular EQ.2.0: arbitrary (User-Defined Integration Rule)	E10.0
1-10	Card 11	Geometric dimension at node n_1 ICROSS.EQ.0: beam thickness in s direction ICROSS.EQ.1: beam outer diameter ICROSS.EQ.2: beam thickness in s direction	E10.0
11-20		Geometric dimension at node n_2 ICROSS.EQ.0: beam thickness in s direction ICROSS.EQ.1: beam outer diameter ICROSS.EQ.2: beam thickness in s direction	E10.0
21-30		Geometric dimension at node n_1 ICROSS.EQ.0: beam thickness in t direction ICROSS.EQ.1: beam inner diameter ICROSS.EQ.2: beam thickness in t direction	E10.0
31-40		Geometric dimension at node n_2 ICROSS.EQ.0: beam thickness in t direction ICROSS.EQ.1: beam inner diameter ICROSS.EQ.2: beam thickness in t direction	E10.0
41-50		Location of reference surface normal to s axis .EQ. 1.0: on beam outer surface at $s = 1$.EQ. 0.0: at beam centerline (default) .EQ1.0: on beam outer surface at $s = -1$	E10.0
51-60		Location of reference surface normal to t axis .EQ. 1.0: on beam outer surface at $t = 1$.EQ. 0.0: at beam centerline (default) .EQ1.0: on beam outer surface at $t = -1$	E10.0

The geometry of DYNA3D beam elements is shown in Figure 21. Note that the position of the orientation node, node n_3 , determines the *initial* element coordinate system. The coordinate system remains fixed in the element, and thus rotates with the element during the deformation.

The Hughes-Liu beam element was adapted from the degenerated continuum shell element described in (Hughes and Liu, 1981a) and (Hughes and Liu, 1981b). All constitutive computations are performed at the stress-strain level, and numerical integration is performed over the cross section to evaluate forces and moments.

Two reference surfaces may be defined to include effects of loads applied to the beam on its outer surface rather than on its centerline. This situation typically occurs when beam elements are used as stiffeners for thin shell structures. In that case, the load is transferred to the beam along the beam surface which contacts the shell. These effects may be incorporated into the analysis model by defining the Hughes-Liu beam element reference surface as the surface in contact with another body. Note that if the either reference surface is displaced from the center of the beam, a purely axial deformation will induce moments into the beam.

Hughes-Liu Beam Cross Section Property Notes

10:2 Choose the desired numerical integration rule for the integration over element cross section. For rectangular or tubular cross sections, either C	r the
or Lobotto quadrature may be used. Gauss quadrature gives increased accuracy for a fixed number of points, but generally does not include integration point on the outer element surface. Lobotto integration is somewhat less accurate than Gauss for the same number of points, bu always has points located on the outermost fibers of the beam. For con cross sections, User-Defined Integration rules must be used to accurate evaluate the element forces and moments.	d an ut mplex itely
10:3 Choose the appropriate cross section geometry. Solid cylindrical beams be modeled by specifying tubular cross section and giving the inner diameter as 0.0. If the User-Defined Integration rule is selected, it mu defined as described in section 4.3 on page 214. Note that if a negative number is input for the Quadrature Rule option, then the cross-section must be selected as 2.0 (User-Defined Integration Rule).	is may 1st be ve n type
11:1-4 Beam dimensions may be defined independently at each of the two er nodes. Linear variation is assumed along the element length.	nd

11:5-6 Choose the location of the reference surface to correspond with the point of application of the load. The reference surface should be the contact surface if beam elements are used as shell stiffeners. The default reference location is at the element center, which is appropriate for most applications.

INPUT FORMAT

<u>Columns</u>		Quantity	Format
1-10	Card 10	Shear factor, κ (default is $\kappa = 1.0$)	E10.0
1-10	Card 11	Cross-sectional area, A	E10.0
11-20		Area moment of inertia about s axis, I_{ss}	E10.0
21-30		Area moment of inertia about t axis, I_{tt}	E10.0
31-40		Polar moment of inertia about r axis, I_{rr}	E10.0
41-50		Shear area, A_s	E10.0

Belytschko-Schwer Beam Element Cross Section Properties

The Belytschko-Schwer beam element is based on a classical resultant beam formulation, and is described in (Belytschko, Schwer, and Klein, 1977). Constitutive evaluations are performed at the resultant level, where forces and moments are related to displacements and curvatures. This element is somewhat faster in computation since no numerical integration is performed over the cross section, but may be less accurate for elastoplastic analysis when partial section yield is important. Since the Belytschko-Schwer beam is a resultant-based formulation, any type of cross-section may be easily modeled by specifying its cross-sectional area and area moments of inertia. Experience has shown that this element performs well in a wide range of analysis situations.

Belytschko-Schwer Beam Cross Section Property Notes

card:field	Comments
11:1	The cross-sectional area A is used in computing axial forces and bending moments, and additionally to compute the element mass.
11:2-4	It is assumed that the r , s , and t axes are principal axes of the cross section, so that all cross-terms in the moment of inertia matrix vanish.
11:5	The shear area A_s is used in computing shear forces.

209

INPUT FORMAT

DYNA3D User Manual



Ŀ

Shell Element Cross Section Properties

<u>Columns</u>	-	Quantity	Format
1-10	Card 10	Shear factor, κ (default is $\kappa = 1.0$)	E10.0
11-20		Number of through-thickness integration points EQ.1: 1 point (membrane element) EQ.2: 2 points EQ.3: 3 points EQ.4: 4 points EQ.5: 5 points GT.5: trapezoidal rule or User-Defined Integration Ru	E10.0
21-30		Shell element printed output option EQ.1: average resultants and fiber lengths EQ.2: resultants at plan points and fiber lengths EQ.3: resultants, stresses at all points, fiber lengths (d	E10.0 efault)
31-40		Integration rule LT.0: absolute value is User-Defined Integration Rule number EQ.0: Gauss quadrature (maximum of 5 points) EQ.1: trapezoidal rule (any number of pts may be use	E10.0
1-10	Card 11	Shell thickness at node n_1	E10.0
11-20		Shell thickness at node n_2	E10.0
21-30		Shell thickness at node n_3	E10.0
31-40		Shell thickness at node n_4	E10.0
41-50	·	Reference surface location (Hughes-Liu formulation only) EQ. 1.0: top surface EQ. 0.0: midsurface (default) EQ1.0: bottom surface	E10.0

The above Cross Section Properties should be defined for all 4-node quadrilateral and 3-node triangular shell elements. These shell thickness values can be overridden on the element cards. The above values are used if the thickness values are zero on the element cards.

211

DYNA3D User Manual

 $(x_{i}) \in \mathcal{K}_{i}$

Shell Element Cross Section Property Notes

.....

· · · · · ·

1. 1.

2

- î.

card:field	Comments
10:1	The elastic moduli are scaled by the shear correction factor κ for calculating the transverse shear stresses.
10:2	Numerical integration is used to compute element resultant forces and moments from stresses calculated at discrete points through the shell element thickness. Several different integration methods (rules) are supported, and each may be used with a selected number of points. Using more integration points through the shell thickness increases the cost, but also increases the accuracy of the element, particularly when partial section yielding is expected in predominantly bending deformations. Two integration points will exactly capture linearly elastic material behavior.
10:3	This option governs the amount of printout written to the "hsp" file when shell elements are specified in Element Time History Blocks, as described in section 4.15 on page 237.
10:4	Three numerical integration algorithms are available. Gauss quadrature gives the best accuracy for a fixed number of points, but may not have points exactly on the upper and lower shell surface. Up to five points are allowed with Gauss quadrature. The trapezoidal rule is slightly less accurate than Gauss quadrature for a fixed number of points, but always has points on the endpoints of the interval (i.e., on the upper and lower surface of the shell). User-Defined Integration Rules allow a general integration scheme to be defined with an arbitrary number of points through the thickness. This feature is often used to construct accurate integration algorithms for laminated composite shells by putting one integration point in each lamina or group of lamina.
	Shell results are output to the TAURUS database at three planes through the thickness: the bottom-most integration point, the midsurface, and the top-most integration point. The desired plane is selected in TAURUS using the shell inner, shell middle , and shell outer commands, respectively. Note that if Gauss integration is used, the bottom-most integration point may not be physically on the shell bottom surface, and similarly with the top-most integration point. If surface stresses are desired, trapezoidal rule integration should be used. See the TAURUS manual (Spelce and Hallquist, 1991) for further details.
11-5	The location of the reference surface can be important whenever beam elements, shell elements, and solid elements are interconnected. It is often desirable to have beam elements and shell elements share a common reference surface when beams are used as shell stiffeners. In these cases, the reference surface for the shell element and beam element should be on the interface surface.

i

Thick Shell Element Cross Section Properties

Columns		Quantity	Format
1-10	Card 10	Shear factor, κ (default is $\kappa = 1.0$)	E10.0
11-20		Number of through-thickness integration points EQ.1: 1 point (membrane element) EQ.2: 2 points EQ.3: 3 points EQ.4: 4 points EQ.5: 5 points GT.5: trapezoidal rule or User-Defined Integration H	E10.0 Rule
31-40		Integration rule LT.0: absolute value is User-Defined Integration Rule number EQ.0: Gauss quadrature (maximum of 5 points) EQ.1: trapezoidal rule (any number of pts may be us	E10.0 sed)
	Card 11	Blank	

Thick Shell Element Cross Section Property Notes

card:field	Comments
10:1	The elastic moduli are scaled by the shear correction factor κ for calculating the transverse shear stresses.
10:2	Numerical integration is used to compute element resultant forces and moments from stresses calculated at discrete points through the shell element thickness. Several different integration methods (rules) are supported, and each may be used with a selected number of points. Using more integration points through the shell thickness increases the cost, but also increases the accuracy of the element, particularly when partial section yielding is expected in predominantly bending deformations. Two integration points will exactly capture linearly elastic material behavior.
10:4	Three numerical integration algorithms are available. Gauss quadrature gives the best accuracy for a fixed number of points, but may not have points exactly on the upper and lower shell surface. Up to five points are allowed with Gauss quadrature. The trapezoidal rule is slightly less accurate than Gauss quadrature for a fixed number of points, but always has points on the endpoints of the interval (i.e., on the upper and lower surface of the shell). User-Defined Integration Rules allow a general integration scheme to be defined with an arbitrary number of points through the thickness.

4.3 HUGHES-LIU BEAM USER DEFINED INTEGRATION RULES

.

Define NUSBIR Card sets in this section.

	Card 1	
Columns	Quantity	Format
1-5	Number of integration points, NIP	15
6-15	Relative area of cross section, A_{rel} . $A_{rel} = \frac{A_{cs}}{t_s t_t}$, where A_{cs} is the actual cross-sectional area, t_s is the specified thickness in the s direction, and t_t is the specified thickness in the t direction.	E10.0
	Cards 2, 3,, NIP+1	
Columns	Quantity	Format
1-10	s coordinate of integration point, ξ_k (-1.0 $\leq \xi_k \leq$ 1.0)	E10.0
11-20	t coordinate of integration point, η_k (-1.0 $\leq \eta_k \leq 1.0$)	E10.0
21-30	Weighing factor, W_k . This is typically the area associated with integration point divided by actual cross sectional area	E10.0

User-Defined Integration Rules are based on the general numerical integration formula

$$\int f(\xi, \eta) dx \approx \sum_{k=1}^{NIP} f(\xi_k, \eta_k) W_k, \qquad (256)$$

where $f(\xi, \eta)$ is the function to be integrated, ξ_k , η_k , k = 1,...,NIP, are the integration point coordinates, and W_k is the integration weight for point k.

DYNA3D User Manual

.

4.4 SHELL USER DEFINED INTEGRATION RULES

Define NUSSIR card sets in this section.

	Card 1	
Columns	Quantity	Format
1-5	Number of integration points, NIP	15
6-10	Equal spacing of integration points option, <i>ISPACE</i> EQ.0: integration point locations are defined below EQ.1: integration points are equally spaced through the shell thickness such that the shell is subdivided into NIP layers of equal thickness.	15

Cards 2, 3, . . . , NIP+1

Define these cards only if the equal-spacing option is *not* active, (ISPACE = 0).

Columns	Quantity	Format
1-10	Coordinate of integration point, ζ_k (-1.0 $\leq \zeta_k \leq$ 1.0)	E10.0
11-20	Weight factor, W_k . This is typically the thickness associated with the integration point divided by actual shell thickness.	E10.0
21-25	Optional material number for this integration point if different than the number specified on the element card. The material type must be the same as for the material number given on the element card.	15

User-Defined Integration Rules are based on the general numerical integration formula

$$\int f(\zeta) dx \approx \sum_{k=1}^{NIP} f(\zeta_k) W_k,$$
(257)

where $f(\zeta)$ is the function to be integrated, ζ_k , k = 1,...,NIP, are the integration point coordinates, and W_k is the integration weight for point k.

DYNA3D User Manual

1

°

Shell User-Defined Integration Rule Notes

• >

- -

۰,

٦,

card:field	Comments
1:2	This option is most useful for quickly defining a large number of points for even-thickness layers in a shell.
2:3	This option allows a different set of material constants to be specified at each thickness integration point to model a shell element with properties which vary through the thickness. If only the orientation angle varies for an orthotropic elastic material, this may be more easily specified as described for Material Types 2, 21, 22, and 23. NOTE: The density is always taken from the data for the material number specified on the element card.

4.5 NODES

Define NUMNP nodes in this section. Column numbers in parenthesis apply for the "large" input option which allows more than 99,999 nodes.

Columns		Quantity F	Format	
1-5	(1-8)	Node number I5	(I8)	
6-10	(9-13)	Displacement boundary condition code15EQ.0: no constraintsEQ.1: constrained x displacementEQ.2: constrained y displacementEQ.3: constrained z displacementEQ.4: constrained x and y displacementsEQ.5: constrained y and z displacementsEQ.6: constrained z and x displacementsEQ.7: constrained x, y, and z displacementEQ.8: constrained in direction specified on the firstSliding Boundary Plane card.EQ.9: constrained in direction specified on the secondSliding Boundary Plane card.EQ.7 + n: constrained in direction specified on the n th Sliding Boundary Plane card.	(15)	
11-30	(14-33)	x-coordinate E20.0*	(E20.0)	
31-50	(34-53)	y-coordinate E20.0	(E20.0)	
51-70	(54-73)	z-coordinate E20.0	(E20.0)	
71-75	(-)	Nodal increment for generation, k I5	(omit)	
76-80	(74-78)	Rotational boundary condition codeI5EQ.0: no constraintsEQ.1: constrained x rotationEQ.2: constrained x rotationEQ.3: constrained y rotationEQ.4: constrained x and y rotationsEQ.5: constrained y and z rotationsEQ.5: constrained z and x rotationsEQ.6: constrained x and y rotationsEQ.7: constrained x, y, and z rotations	(I5)	

* or as defined on Control Card 3.

۶. #

Node cards do not need to be in order, but the first node card must define node 1, and the last node card must define node *NUMNP*. Gaps are not permitted in node numbering, and nodes may be generated internally in DYNA3D. Whenever node definitions are missing, node numbers are automatically generated according to the sequence

$$n_i, n_i + k, n_i + 2k, \dots, n_i$$
 (258)

where n_i and n_j are the node numbers defined on two consecutive cards, and k is taken from the card defining n_i . Linear interpolation is used to obtain the coordinates of the generated nodes. The boundary condition code for generated nodes is set to zero if the boundary condition code of n_i differs from that of n_j , otherwise it is set to the boundary condition code of node n_i . Unconstrained nodes can be generated between constrained nodes that have the same boundary condition code by making the code negative on one of the two input node cards. After the nodal data is generated, all negative boundary condition codes are reset to positive values.

Boundary conditions which are not parallel to a global axis may be defined using Sliding Boundary Planes, as described in section 4.12 on page 232, or Nodal Single Point Constraints, as described in section 4.11 on page 230. Boundary conditions parallel to the global axes should be defined on the node cards as described above for fastest computation.

DYNA3D User Manual

4.6 SOLID ELEMENTS

Define the following card for each of the *NUMELH* solid elements. All solid elements are numbered sequentially starting from 1.

Columns		Quantity	Format	
1-5	(1-8)	Solid element number	15	(I8)
6-10	(9-13)	Material number	15	(I5)
11-15	(-)	Generation increment, k	15	(omit)
16-20	(14-21)	Node n_1	15	(I8)
21-25	(22-29)	Node n ₂	15	(I8)
26-30	(30-37)	Node n ₃	15	(I8)
		•	•	
•			•	
			•	
51-55	(70-77)	Node n ₈	I5	(I8)

Nodes $n_1 - n_8$ define the corner nodes of the 8-node solid elements. Elements having fewer than 8 nodes are defined by repeating one or more nodes to "collapse" the appropriate side of the brick element into the desired shape. Four, six, and eight node elements are shown in Figure 23. Input of nodes on the element cards for the two degenerate elements is:

4-node	n_1	n_2	<i>n</i> ₃	<i>n</i> ₄	n_4	<i>n</i> ₄	$n_4 n_4$	
<u>6-node</u>	n_1	n_2	n ₃	n_4	n ₅	n_5	$n_6 n_6$	

Note: In all cases the first four node numbers must be unique.

Element cards must be in element number order. The first card must contain data for the first solid element, and the last card must contain data for the last solid element. Cards containing intermediate element data may be omitted and these element definitions will be generated internally. Connecting node numbers n_j are automatically generated with respect to the first card prior to the omitted data by

$$n_j^{i+1} = n_j^i + k, \quad j = 1, ..., 8$$
 (259)

where superscripts denote the element number to be defined and subscripts denote the local node number (in the connectivity list) which is to be incremented. The generation increment k and the element attributes, such as the material number, for the generated elements are taken from the last element card before the omitted data. The default value of the generation increment k is l.





220

4.7 BEAM AND TRUSS ELEMENTS

Define the one of the following cards for each of the *NUMELB* truss or beam elements. All truss and beam elements are numbered sequentially, starting from 1, regardless of the formulation used. The element formulation is specified on the Material Control Card as described in section 4.2 on page 59.

Columns		Quantity	Format
1-5	(1-8)	Beam element number I	5 (I8)
6-10	(9-13)	Material number I	5 (I5)
11-15	(-)	Generation increment, k I	5 (omit)
16-20	(14-21)	Node n_1 If	5 (I8)
21-25	(22-29)	Node n_2 If	5 (I8)
26-30	(30-37)	Node n_3 (see Figure 21) If	5 (I8)
31-40	(38-45)	Beam thickness in s direction at node n_1 E10.0) (E8.0)
41-50	(46-53)	Beam thickness in s direction at node n_2 E10.0) (E8.0)
51-60	(54-61)	Beam thickness in t direction at node n_1 E10.0) (E8.0)
61-70	(62-69)	Beam thickness in t direction at node n_2 E10.0) (E8.0)

Hughes-Liu Beam Element

Belytschko-Schwer Beam Element

Columns		Quantity	Fo	ormat
1-5	(1-8)	Beam element number	15	(18)
6-10	(9-13)	Material number	15	(I5)
11-15	(-)	Generation increment, k	15	(omit)
16-20	(14-21)	Node n_1	15	(18)
21-25	(22-29)	Node n ₂	15	(I8)

DYNA3D User Manual

Columns		Quantity	Format		
26-30	(30-37)	Node n_3 (see Figure 21)	15	(I8)	
31-40	(38-45)	Cross-sectional area, A	E10.0	(E8.0)	
41-50	(46-53)	Moment of inertia about s axis, I_{ss}	E10.0	(E8.0)	
51-60	(54-61)	Moment of inertia about t axis, I_{tt}	E10.0	(E8.0)	
61-70	(62-69)	Polar moment of inertia about r axis, I_{rr}	E10.0	(E8.0)	
71-78	(70-77)	Shear area, A _s	E10.0	(E8.0)	

Truss Element

Columns		Quantity	Format	
1-5	(1-8)	Beam element number	15	(I8)
6-10	(9-13)	Material number	15	(I5)
11-15	(-)	Generation increment, k	15	(omit)
16-20	(14-21)	Node n_1	I5	(I8)
21-25	(22-29)	Node n ₂	15	(I8)
26-30	(30-37)	Node n_3 (see Figure 21)	15	(I8)
31-40	(38-45)	Cross-sectional Area, A	E10.0	(E8.0)

Element cards must be in element number order. The first card must contain data for the first beam \cdot or truss element, and the last card must contain data for the last beam or truss element. Cards containing intermediate element data may be omitted and these element definitions will be generated internally. Connecting node numbers n_j are automatically generated with respect to the first card prior to the omitted data by

$$n_j^{i+1} = n_j^i + k, \quad j = 1, 2$$
 (260)

where superscripts denote the element number to be defined and subscripts denote the local node number (in the connectivity list) which is to be incremented. Note that the orientation node n_3 is not incremented. The generation increment k and the element attributes, such as the material number, cross section properties, and orientation node, for the generated elements are taken from the last element card before the omitted data. The default value of the generation increment k is l.

If any cross-sectional properties are undefined (or are defined as 0.0) on the element card, these values are taken from the data specified in the Material Cards. Thus, if many elements share the same properties with only a few elements differing, the common properties may be specified in the Material Cards, and only the unique element properties need be defined on the element cards.

224

76-80 (41-50)

2

1

4.8 SHELL AND MEMBRANE ELEMENTS

Define the following card set for each of the NUMELS shell or membrane elements. All shell and membrane elements are numbered sequentially, starting from 1, regardless of the formulation used. Note that two cards per element are required if the LARGE input option is active.

Columns		Quantity	Format	
1-5	(1-8)	Shell element number 1	.5	(I8)
6-10	(9-13)	Material number 1	.5	(I5)
11-15	(-)	Generation increment k	[5	(omit)
16-20	(14-21)	Node n_1 (see Figure 22)	[5	(I8)
21-25	(22-29)	Node n_2	[5	(I8)
26-30	(30-37)	Node n_3	[5	(I8)
31-35	(38-45)	Node n_4	[5	(I8)
		second card here if LARGE input option is active		
36-45	(1-10)	Shell thickness at node n_1 (optional) E10.	.0	(E10.0)
46-55	(11-20)	Shell thickness at node n_2 (optional) E10.	.0	(E10.0)
56-65	(21-30)	Shell thickness at node n_3 (optional) E10.	.0	(E10.0)
66-75	(31-40)	Shell thickness at node n_4 (optional) E10.	.0	(E10.0)

Triangular elements are defined by repeating the third node (i.e., by setting $n_4 = n_3$). It should be noted that quadrilateral shell elements perform very poorly in bending when degenerated into triangles. In this case, the use of one of the triangular shell element formulations is recommended. Shell and membrane element formulations are discussed in section 2.6 on page 12.

Orthotropic material angle, β , in degrees (optional) E5.00 (E10.0)

Element cards must be in element number order. The first card must contain data for the first shell or membrane element, and the last card must contain data for the last shell or membrane element. Cards containing intermediate element data may be omitted and these element definitions will be generated internally. Connecting node numbers n_j are automatically generated with respect to the last element card prior to the omitted data by

$$n_j^{i+1} = n_j^i + k, \quad j = 1, 2, 3, 4$$
 (261)

where superscripts denote the element number to be defined and subscripts denote the local node number (in the connectivity list) which is to be incremented. The generation increment k and the element attributes, such as the material number and cross section properties, for the generated elements are taken from the last element card before the omitted data. The default value of the generation increment k is l.

If any element thicknesses are undefined (or are defined as 0.0) on the element card, these values are taken from the data specified in the Material Cards. Thus, if many elements share the same properties with only a few elements differing, the common properties may be specified in the Material Cards, and only the unique element properties need be defined on the element cards.

4.9 THICK SHELL ELEMENTS

Define the following card for each of the *NUMELT* 8-node thick shell elements. All thick shell elements are numbered sequentially starting from 1.

Columns		Quantity	F	ormat
1-5	(1-8)	Thick shell element number	15	(I8)
6-10	(9-13)	Material number	I5	(I5)
11-15	(-)	Generation increment, k	15	(omit)
15-20	(14-21)	Node n_1	15	(I8)
21-25	(22-29)	Node n ₂	15	I8)
26-30	(30-37)	Node n_3	I5	(I8)
•				•
•		• •		•
•		• •		•
•		• •		•
51-55	(70-77)	Node n_8	15	(I8)

Element cards must be in element number order. The first card must contain data for the first thick shell element, and the last card must contain data for the last thick shell element. Cards containing intermediate element data may be omitted and these element definitions will be generated internally. Connecting node numbers n_j are automatically generated with respect to the first card prior to the omitted data by

$$n_j^{i+1} = n_j^i + k, j = 1,...,8$$
 (262)

where superscripts denote the element number to be defined and subscripts denote the local node number (in the connectivity list) which is to be incremented. The generation increment k and the element attributes, such as the material number, for the generated elements are taken from the last element card prior to the omitted data. The default value of the generation increment k is l.

Nodes $n_1 - n_8$ define the corner nodes of the 8-node thick shell elements. Nodes n_1 to n_4 define the lower surface, and nodes n_5 to n_8 define the upper surface. The integration points lie along the *t* axis as depicted in Figure 24. Care should be used in defining the connectivity to insure the proper element orientation.



DYNA3D User Manual

7

4.10 INTERFACE SAVE SEGMENT DEFINITION

Columns		Quantity	Format	
1-5	(1-8)	Interface segment number	15	(I8)
6-10	(-)	Generation increment, k	15	(omit)
11-15	(9-16)	Node n_1	15	(I8)
16-20	(17-24)	Node n ₂	15	(I8)
21-25	(25-32)	Node n ₃	I5	(I8)
26-30	(33-40)	Node n_4	15	(I8)

Define NUMIFS Interface Save Segments.

Omitted segment definitions are automatically generated with respect to the last card prior to the omitted data using the generation increment k to compute connecting nodes for the missing segment definitions. The generation parameter k is taken from the first card. Node numbering may be either clockwise or counterclockwise. Nodes $n_1 - n_4$ define the corner nodes of the segments as shown in Figure 30. Triangular segments are defined by repeating a node.

Interface save segments are used to define a surface for which the displacement and velocity time history will be saved during an analysis. This surface is then used as the master surface of a tied slide surface in a second analysis, where the discretization may be arbitrarily modified. This feature is especially useful for studying the detailed response of a small member in a large structure. For the first analysis, the member of interest need only be discretized sufficiently that the displacements and velocities on its boundaries are reasonable approximations to the actual solution. After the first analysis is completed, the member is finely discretized in another model, and the interface save segments are defined to correspond with the first analysis. Finally, the second analysis is performed to obtain highly detailed information in the local region of interest.

An analysis using interface save segments consists of two phases (DYNA3D runs). In the input file for the first analysis, specify the number of interface segments for linking (*NUMIFS*) and the output interval for the interface save segment file on Control Card 2 (as described in section 4.1 on page 41). Then define the interface save segments as described above. Next, execute the first phase analysis, and specify a name for the interface save segment file using the "u=isf1" parameter on the execution line (as described in section 3.2 on page 33). This will save the displacement and velocity at every interface save segment defined in the first model into the specified file. Now, construct the

228

finite element model for the second phase analysis, and in this input file the number of interface segments and the time interval for interface data output should be set to zero. The second phase model typically contains a more detailed model of the subregion of interest than was present in the first phase model. Define the slave surface of a type 2 (tied) sliding interface over the portion of the new model which corresponds to the interface save segments from the first phase model. It is *not* necessary that the node and element positions in this area be the same in the two models. Define the master surface as 0 in this type 2 slide surface definition. Finally, when starting the second phase analysis using this new model, specify the name of the interface save segment file (created by the first run) using the "v=isf2" parameter on the DYNA3D execution line. The displacements and velocities for the interface region will then be read from this file and applied as external boundary conditions onto the phase two model.

4.11 NODAL SINGLE POINT CONSTRAINTS

Define NODSPC cards to specify single point constraints on nodes.

Columns		Quantity	Format	
1-5	(1-8)	First node number, n _{first}	I5	(I8)
6-10	(9-13)	Local coordinate system number ($\leq NSPCOR$)	15	(I5)
11-14	(14-17)	Blank		
15	(18)	Translation code in local x direction	I1	(I1)
16	(19)	Translation code in local y direction	I1	(I1)
17	(20)	Translation code in local z direction	I1	(I1)
18	(21)	Rotation code about local x axis	I1	(I1)
19	(22)	Rotation code about local y axis	I1	(I1)
20	(23)	Rotation code about local z axis	I1	(I1)
21-25	(24-31)	Last node number, n_{last} EQ.O: $n_{last} = n_{first}$ (no generation)	15	(I8)
26-30	(32-39)	Increment for generation, k EQ.O: default set to 1	15	(18)

Single point constraints allow the definition of a completely arbitrary local coordinate system, and the specification of nodal boundary conditions (constraints) in this local system. One local system may be used for many nodes, or a different local system may be defined for each node. A boundary code of 1 specifies a constrained degree-of-freedom, while a boundary condition code of 0 indicates an unconstrained degree-of-freedom. This option was originally developed by Dr. C. S. Tsay.

A set of nodal single point constraints may be generated by specifying the first node number in the sequence, n_{first} , the last node number in the sequence, n_{last} , and the node number increment for generation, k. Nodal single point constraints will be generated for nodes between n_{first} and n_{last} using the set of boundary condition codes as specified for n_{first} .

Nodal single point constraints require the definition of a local coordinate system, as described on the following page.

Columns	Quantity	Format
1-5	Local coordinate system number ($\leq NSPCOR$)	15
6-15	x-coordinate of point P	E10.0
16-25	y-coordinate of point P	E10.0
26-35	z-coordinate of point P	E10.0
36-45	x-coordinate of point Q	E10.0
46-55	y-coordinate of point Q	E10.0
56-65	z-coordinate of point Q	E10.0

Define NSPCOR local coordinate systems for single point constraints.

The local coordinate system is constructed at the global origin by specifying two points, P and Q, as shown in Figure 25. First, the local x-axis is constructed by defining a vector x_L from the global origin through point P. Next, a temporary vector t is constructed from the global origin through point Q. The local z-axis is then found by computing a vector which is mutually perpendicular to x_L and t,

$$z_L = x_L \times t. \tag{263}$$

Finally, the local y-axis is found as the vector which is mutually perpendicular to x_L and z_L ,

$$y_L = z_L \times x_L. \tag{264}$$



DYNA3D User Manual

*

4.12 SLIDING BOUNDARY PLANES

Define the number of Sliding Boundary Plane Cards specified on Control Card 4 of section 4.1 on page 41.

Columns	Quantity	Format
1-10	x-coordinate of vector from global origin	E10.0
11-20	y-coordinate of vector from global origin	E10.0
21-30	z-coordinate of vector from global origin	E10.0
31-40	Constraint option, <i>IOPT</i> EQ.0.0: node is constrained to move on normal plane EQ.1.0: node is constrained to translate in vector direction	E10.0

Sliding Boundary Planes allow any node to be constrained to move on an arbitrarily oriented plane or line. Each Sliding Boundary Plane card defines a vector originating at the global origin and terminating at the coordinates defined above. This vector can be of arbitrary length; DYNA3D internally converts it to a unit vector for computation.

If IOPT = 0, then the vector is interpreted as defining a plane which has the given vector as its normal, and nodes are constrained to move only on this plane. Nodes may not separate from this plane, but may move freely in any direction in the plane. An arbitrary oblique boundary condition may be defined using this option. One common application is the definition of a "pie-slice" three-dimensional model of an axisymmetric problem, where all nodes on the side faces of the model must be constrained from translating in the circumferential direction. The Sliding Boundary Plane vector is defined as the normal to the side face, and all nodes on that face are given appropriate boundary condition codes to reference the Sliding Boundary Plane.

If IOPT = 1, then the vector is interpreted as defining a line, and nodes are allowed to translate only along that line. Since nodes can move in either direction along this line, the orientation of the given vector along the desired line is not significant. This type of constraint can also be equivalently defined using Nodal Single Point Constraints, as described in section 4.11 on page 230.

To constrain a node to move on a Sliding Boundary Plane, set its displacement boundary condition code (described on the Node Card, section 4.5 on page 217) to 7 + n, where n is the number of the Sliding Boundary Plane to be used. Any number of Sliding Boundary Planes may be defined.

232

4.13 SYMMETRY PLANES WITH FAILURE

Define the number of cards specified on Control Card 4 of section 4.1 on page 41.

	Card 1	
Columns	Quantity	Format
1-5	Number of segments in Symmetry Plane with Failure, NSGNDS	15
6-15	x-coordinate of tail point P	E10.0
16-25	y-coordinate of tail point P	E10.0
26-35	z-coordinate of tail point P	E10.0
36-45	x-coordinate of head point Q	E10.0
46-55	y-coordinate of head point Q	E10.0
56-65	z-coordinate of head point Q	E10.0
66-75	Tensile failure stress, σ_f (positive in tension)	E10.0

As shown in Figure 26, the outward normal vector n is defined from point P and passes through point Q. This vector should originate on the symmetry plane and point toward the body.

Cards 2, 3, ..., NSGNDS+1

Columns		Quantity	Format	
1-5	(1-8)	Segment number	15	(I8)
6-10	(9-16)	Node n_1	15	(I8)
11-15	(17-24)	Node n ₂	15	(I8)
16-20	(25-32)	Node n_3	15	(I8)
21-25	(33-40)	Node n_4	15	(I8)
26-35	(41-50)	Failure stress, σ_f , if different from default value	E10.0	(E10.0)

Symmetry Planes with Failure are specified by listing quadrilateral or triangular segments which initially lie on the symmetry plane. Segments are defined by their corner nodes, and normally each segment represents one element face. Triangular segments may be defined by repeating the last node $(n_4 = n_3)$. The symmetry plane is defined by its normal vector, which is assumed to be drawn from point P to point Q, as shown in Figure 26.



As long as the normal stress (in the direction of n) remains less than the failure stress σ_f , the Symmetry Plane with Failure acts just like a Sliding Boundary Plane. Nodes initially on the symmetry plane may move freely in any direction in the plane, but may not move normal to the plane. When the normal stress exceeds σ_f , the nodes are released and may move freely throughout the remainder of the calculation. Symmetry Planes with Failure are frequently used to model oblique penetration. One half-space of the target material is modeled, and the vertical symmetry plane is defined as a Symmetry Plane with Failure. The projectile may be given an initial velocity into the target material at any angle, and nodes on the Symmetry Plane with Failure will separate when the tensile criterion is exceeded, thus simulating a penetration process. In this application, the segments on the symmetry plane must also be defined as one side of a sliding interface, with the other side defined as the external part of the penetrating object. This approach has proven effective in modeling 3-D penetration problems.

4.14 NODE TIME HISTORY BLOCKS

Skip this section if the number of node time history blocks is zero on Control Card 3 of section 4.1 on page 41. Otherwise, define up to 2000 history blocks that may contain a total of 2000 nodes. Use only the number of cards required to define all blocks.

<u>Columns</u> Quantity		Format		
1-5	(1-8)	First node of first time history block	15	(I8)
6-10	(9-16)	Last node of first time history block	15	(I8)
11-15	(17-24)	First node of second time history block	I5	(I8)
16-20	(25-32)	Last node of second time history block	I5	(I8)
•	•		•	
•	•		•	
•	•	•	•	
•	•	•		

Node time history blocks are used to define sets of nodes for inclusion in the time history plot database. Typically, the time interval between writes to this time history plot database is much smaller than the time interval between writes to the state plot database, which contains information for the complete analysis model at each plot time point (state). Thus, for the nodes included in the time history plot database, much higher frequency resolution is possible in post-processing time history plots than is available in time history plots made from the state plot database. The plot databases are described in more detail in section 3.5 on page 38.

The time history plot database may be post-processed using TAURUS or other codes to generate time-history plots of nodal quantities such as displacement, velocity, or acceleration.

4.15 ELEMENT TIME HISTORY BLOCKS

(Solid Elements)

Skip this section if the number of solid element time history block is zero on Control Card 3 of section 4.1 on page 41. Otherwise, define up to 2000 time history blocks that may contain a total of 2000 solid elements. Use as many cards as required to define the specified number of blocks. All cards are given in the format shown below.

Columns		Quantity	Format	
1-5	(1-8)	First element of first time history block	15	(I8)
6-10	(9-16)	Last element of first time history block	15	(I8)
11-15	(17-24)	First element of second time history block	15	(I8)
16-20	(25-32)	Last element of second time history block	15	(I8)
•		•		
•		•		

(Beam Elements)

Skip these cards if the number of beam element time history blocks is zero on Control Card 3 of section 4.1 on page 41. Up to 2000 time history blocks may be defined containing a total of 2000 beam and truss elements. The input format is as shown above.

(Shell Elements)

Skip these cards if the number of shell element time history blocks is zero on Control Card 3 of section 4.1 on page 41. Up to 2000 time history blocks may be defined containing a total of 2000 4-node shell elements. The input format is as shown above.

(Thick Shell Elements)

Skip these cards if the number of solid shell element time history blocks is zero on Control Card 3 of section 4.1 on page 41. Up to 2000 time history blocks may be defined containing a total of 2000 8-node thick shell elements. The input format is as shown above.

237

7

3

Element time history blocks are used to define sets of elements for inclusion in the time history plot database. The time history plot database is typically written at a much smaller time increment than the state database, and therefore contains more high frequency response information.

The time history plot database may be post-processed using TAURUS or other codes to generate time-history plots of element quantities such as stress or strain.

~

4.16 GRAVITY STRESS INITIALIZATION

Skip this section if the number of points in the density versus depth curve is zero on Control Card 4 of section 4.1 on page 41. Otherwise, supply NUMDP + 1 cards.

Card 1

Columns	Quantity	Format
1-10	Gravitational acceleration, g	E10.0
15	Direction of gravitational loading EQ.1: global x-direction EQ.2: global y-direction EQ.3: global z-direction	11
16-20	Material number of materials to be initialized (maximum of 12) EQ.0: all EQ.n: define list of n materials below (n ≤ 12)	15
21-25	Material number of first material to be initialized	15
26-30	Material number of second material to be initialized	15
•	•	••
•		••
•		••
		••

Define NUMDP cards giving the mass density as a function of depth.

Cards 2, 3, ..., NUMDP+1

Columns		Quantity	Format
1-10	Mass density, $ ho$		E10.0
11-20	Depth, d		E10.0

Density vs. depth curves are often used to initialize hydrostatic stresses arising in a material due to gravity acting on an overburden material. The hydrostatic pressure acting on a material point at a depth d_0 is given by

$$p = -\int_{d_0}^{d_{top}} \rho(z) g dz,$$
 (265)

where p is pressure, d_{top} is depth at the top of the material to be initialized (usually $d_{top} = 0$), $\rho(z)$ is the mass density at depth z, and g is the acceleration of gravity. This integral is evaluated numerically for each material to be initialized.

Depth may be measured along any of the global coordinate axes, and the sign convention of the global coordinate system should be respected. The sign convention of gravity also follows that of the global coordinate system. For example, if the positive z axis points "up," then gravitational acceleration should be input as a negative number.

4.17 BRODE FUNCTIONS

Skip this section if columns 21-25 are blank on Control Card 4 in section 4.1 on page 41. Otherwise, enter two cards for the pertinent Brode function data.

	Card 1	
Columns	Quantity	Format
1-10	Yield (Kt)	E10.0
11-20	Height of burst	E10.0
21-30	Model x coordinate of Brode origin, x_{BO}	E10.0
31-40	Model y coordinate of Brode origin, y_{BO}	E10.0
41-50	Model z coordinate of Brode origin, z_{BO}	E10.0
51-60	Model time at Brode time origin, t_{BO}	E10.0
61-65	*Load curve number giving time of arrival versus range relative to Brode origin	• E10.0
66-70	Load curve giving yield scaling versus scaled time (time relative to Brode origin divided by [yield $(1/3)$])	E10.0
	Card 2	

Columns	Quantity	Format
1-10	Conversion factor - kft to DYNA length units	E10.0
11-20	Conversion factor - milliseconds to DYNA time units	E10.0
21-30	Conversion factor - psi to DYNA pressure units	E10.0

*Both load curves must be specified for the variable yield option. If this option is used, the shock time of arrival is found from the time of arrival curve. The yield used in the Brode formulas is computed by taking the value from the yield scaling curve at the current time/[yield^(1/3)] and multiplying that value by yield.

The development and limitations of the equations underlying this option are given in (Speicher and Brode, 1987).
ž

3, °C.

4.18 CROSS SECTION DEFINITIONS FOR FORCE OUTPUT

Define only if *NUMCSD* is nonzero on Control Card 4 in section 4.1 on page 41. For each cross section where force output is desired, define the following card sets.

Define NUMCSD Cross Section Definition Control Cards:

-

· , ·

Cards 1,2, ..., NUMCSD

Columns	Quantity	Format
1-5	Number of nodes in this cross section definition	15
6-10	Number of beam elements in this cross section definition	15
11-15	Number of shell elements in this cross section definition	I5

Define *NUMCSD* Cross Section Definition data card sets (nodes, beam elements, and shell elements) as described below:

NODES

Repeat the following card until all nodes have been defined for this Cross Section Definition.

Cards NUMCSD+1,...

Columns	Quantity	Format
. 1-10	First cross section node	I10
11-20	Second cross section node	110
•	•	•
•		•
•		•
71-80	Eighth cross section node	I10

2

BEAM ELEMENTS

Repeat the following card until all beam elements have been defined for this Cross Section Definition.

Columns	Quantity	Format
1-10	First beam element	I10
11-20	Second beam element	I10
•	•	۰.
•		•
•		•
71-80	Eighth beam element	I10

SHELL ELEMENTS

Repeat the following card until all shell elements have been defined for this Cross Section Definition.

Columns	Quantity	Format
1-10	First shell element	I10
11-20	Second shell element	I10
•	•	•
•		•
•	•	•
71-80	Eighth shell element	I10

Repeat the last 3 card groups for each Cross Section Definition until all *NUMCSD* cross sections have been defined.

DYNA3D User Manual

INPUT FORMAT

DYNA3D allows a cross section to be defined through a structural model, and at selected intervals will output the total resultant force on this cross section. This feature is often used to quickly evaluate a global response quantity, such as the total force transmitted from one part of a model to another. The current implementation only computes forces arising from structural elements (truss, beam, and shell elements), but is fully supported for all element formulation options.

The first step in defining a cross section for force output is to select a group of nodes in the model to define the cross section cutting plane. These nodes need not all lie in the same plane, but should not overlap along the load path, or misleading results may be obtained. Typically, a group of nodes is chosen which defines a cut or slice through the analysis model. Once the group of nodes is selected, the next step is to define all elements which connect to this plane of nodes *on one side only*. Elements may be defined on either side of the cross section, but not both sides. The global components of the total resultant force on the cross sectional plane will be written to the "hsp" printout file at the time interval specified on Control Card 4 in section 4.1 on page 41.

4.19 LOAD CURVES

Define the number of load curves specified on Control Card 5 in section 4.1 on page 41. Repeat the following card set for each load curve.

	Card 1	
Columns	Quantity	Format
1-5	Load curve number	15
6-10	Number of points in this load curve, NPTS	15
11-15	Load curve option, <i>LCOPT</i> EQ.0: load curve used in transient dynamic analysis only EQ.1: load curve used in dynamic relaxation solution but not in transient dynamic analysis EQ.2: load curve used in both dynamic relaxation solution and transient analysis	15

	Cards 2,, NPTS+1	
<u>Columns</u>	Quantity	Format
1-10	Time, t , or independent variable, x	E10.0
11-20	Load value, $f(t)$, or function value, $f(x)$	E10.0

Various options in DYNA3D use load curves to define arbitrary functional relationships. Many loads and boundary conditions are functions of time, and for these options, the specified load curve should relate load (or boundary condition) value f(t) to time t. Some material models require the specification of load curves defining the variation of material parameters f(x) with temperature x. Other models require the specification of load curves defining effective stress f(x) as a function of effective plastic strain or other material variable, x.

There is no limit on the number of load curves, or on the number of points in a load curve definition.

DYNAMIC RELAXATION STATIC ANALYSIS

DYNA3D contains a limited capability for performing quasistatic analysis using a dynamic relaxation algorithm. This feature is primarily intended to be used to generate a static stress solution as an initial condition for a transient dynamic analysis, but it has been applied with some success to the solution of more general static problems.

During then DR static solution process, "time" is really just a parameter to describe the solution process, and does not correspond to physical time. The current implementation uses a DR time step equal to the standard dynamic time step. Thus, if it is desired to slowly apply the static loads to minimize overshoot in the solution, then a short trial dynamic run can be made to determine the time step size. The static loads to be applied during the DR solution can then be applied over some number of time steps (typically 5000-10,000 but problem dependent), and this determines the time points to be used on the load curve controlling the static loads.

The current implementation of DR in DYNA3D is susceptable to dynamic overshoot if static loads are applied too quickly. If only history-independent material models (such as elasticity) are used, then the resulting solution will still be correct and this overshoot behavior is of little consequence. If history-dependent material models (such as plasticity) are used, however, this dynamic overshoot can cause yielding which is erroneous, and therefore an incorrect static solution is obtained. Thus, the DR static solution capability can be used with confidence for elastic initialization, but must be carefully used with slowly applied loads to prevent overshoot and inaccuracy in history-dependent static problems.

A DR static solution is activated by specifying the "load curve option" LCOPT = 1 for a load curve to control the application of the static loads, as discussed in section 4.19 on page 245. The dynamic relaxation factor, the DR time step, and the DR convergence tolerance can be modified from default values on Control Card 8 as described in section 4.1 on page 41.

246

4.20 NODAL FORCES AND FOLLOWER FORCES

Define the number of concentrated nodal loads specified on Control Card 5 in section 4.1 on page 41.

Colu	umns	Quantity	F	ormat
1-5	(1-8)	Node number, m_1 , on which this load acts	15	(I8)
6-10	(9-13)	Direction in which this load acts, <i>IDIR</i> EQ.1: x-direction EQ.2: y-direction EQ.3: z-direction EQ.4: follower force EQ.5: moment about the global x-axis EQ.6: moment about the global y-axis EQ.7: moment about the global z-axis	15	(15)
11-15	(14-18)	Load curve number giving time variation	I5	(I5)
16-25	(19-28)	Scale factor (default =1.0) E1	0.0	(E10.0)
26-30	(29-36)	Node m_1 (define only for follower force, $IDIR = 4$)	I5	(I8)
31-35	(37-44)	Node m_2 (define only for follower force, $IDIR = 4$)	15	(I8)
36-40	(45-52)	Node m_3 (define only for follower force, $IDIR = 4$)	I5	(I8)

Nodal loads and follower forces may be imposed as a prescribed function of time. The time variation is taken from a load curve, and may be arbitrarily scaled in the nodal load or follower force definition.

A follower force F acts normal to the plane defined by nodes m_1 , m_2 , and m_3 as shown in Figure 27. The positive direction is determined by a counterclockwise orientation ("right-hand-rule") of the nodes. The follower force *remains normal* to the plane throughout the deformation.

INPUT FORMAT

DYNA3D User Manual

- .-~u *



4.21 PRESSURE LOADS

Define the number of element face pressure cards specified on Control Card 5 in section 4.1 on page 41.

Col	umns	Quantity	Quantity Format	
1-5	(-)	Pressure card number I	5 (omit)	
6-10	(1-5)	Load curve number, N_L , giving time variation I LT.O: Brode function is used to determine pressure	5 (I5)	
11-15	(6-13)	Node n_1 (see Figure 28) I	5 (18)	
16-20	(14-21)	Node n ₂ I	5 (I8)	
21-25	(22-29)	Node n ₃ I	5 (I8)	
26-30	(30-37)	Node n_4	5 (I8)	
31-40	(38-47)	Load curve multiplier at node n_1 E10. EQ.0.0: default set to 1.0	0 (E10.0)	
41-50	(48-57)	Load curve multiplier at node n_2 E10. EQ.0.0: default set to 1.0	0 (E10.0)	
51-60	(58-67)	Load curve multiplier at node n_3 E10. EQ.0.0: default set to 1.0	0 (E10.0)	
61-70	(68-77)	Load curve multiplier at node n_4 E10. EQ.0.0: default set to 1.0	.0 (E10.0)	
71-80	(-)	Time pressure begins acting on surface, t_0 E10.	.0 (omit)	

Element face pressures always act normal to the element face in the current configuration, and positive pressure acts toward the element face, as shown in Figure 28. Pressure loads may be applied to solid elements, shell elements, or thick shell elements. The load curve multipliers may be used to increase or decrease the pressure given in the load curve definition. Triangular element faces may be defined by repeating the last node, $(n_4 = n_3)$.

If the load curve number, N_L , is set to a value less than zero, then the pressure time history to be applied is computed from the Brode function data given in section 4.17 on page 241. The time-of-arrival of the pressure is then computed based on the distance from the Brode origin to the centroid of the element face receiving the pressure loading.

249

DYNA3D User Manual



. 4

·····

DYNA3D User Manual

÷

4.22 PRESCRIBED VELOCITIES AND ACCELERATIONS

Define the number of cards specified on Control Card 5 in section 4.1 on page 41.

Col	umns	Quantity	F	ormat
1-5	(1-8)	Node number or rigid body material number GT.0: node number LT.0: absolute value is material number of the rigi	I5 id bod	(I8) ly
6-10	(9-13)	Load curve number giving time variation	15	(I5)
11-15	(14-18)	 Applicable degrees-of-freedom, <i>IDOF</i> EQ.1: x-translational degree-of-freedom EQ.2: y-translational degree-of-freedom EQ.3: z-translational degree-of-freedom EQ.4: translational velocity in direction of vector defined below EQ.5: x-rotational degree-of-freedom EQ.6: y-rotational degree-of-freedom EQ.7: z-rotational degree-of-freedom EQ.8: rotational velocity about vector v defined be EQ.9: tangential component for rotation about ge EQ.11: tangential component for rotation about ge 	I5 v v bal x lobal lobal	(I5) axis y axis z axis
16-25	(19-28)	Scale factor E	10.0	(E10.0)
26-35	(29-38)	x-coordinate of vector v E	10.0	(E10.0)
36-45	(39-48)	y-coordinate of vector v E	10.0	(E10.0)
46-55	(49-58)	z-coordinate of vector v E	10.0	(E10.0)
56-60	(59-63)	velocity/acceleration flag, <i>IACCEL</i> EQ.0: velocity (rigid bodies and nodes) EQ.1: acceleration (nodes only)	15	(I5)

Nodal velocities or accelerations may be imposed as a prescribed function of time. The time variation is taken from a load curve, and may be arbitrarily scaled in the prescribed velocity/acceleration definition. This is convenient when many nodes have the same velocity or acceleration time history but different amplitudes, since only one load curve needs to be defined in that case. Nodal velocities or accelerations may be specified with respect to the global axes, or with respect to a vector v originating at the specified node and pointing in the direction given in the input. Further,

251

•

velocities and accelerations may be specified as either translational or rotational. Note that solid elements do not have rotational degrees-of-freedom, and therefore rotational velocities (IDIR = 5-8) should not be prescribed for nodes connected only to solid elements.

It is not required that velocities be initially zero. However, if zero velocity initial conditions are specified in section 4.25 on page 259 and prescribed velocity load curves are not 0 at t = 0, then a discontinuity in initial conditions is created, and this may excite very high frequency response in the model.

Nodal velocities/accelerations may be prescribed *only* for nodes of deformable bodies. Prescribed translational and rotational velocities may be specified for an entire rigid body by giving the negative of its material number in the node number field of the input. The acceleration flag *IACCEL* pertains to nodes of deformable bodies only; acceleration may not be directly prescribed for a rigid body.

Angular velocity may be specified for an entire rigid body by choosing IDOF = 8. Velocity is then specified in the units of radians per unit time. The rigid body is assumed to rotate about a vector \mathbf{v} passing through its center of mass if constraints permit, otherwise it will rotate as permitted by the constraints to achieve the prescribed angular velocity at its center of mass. Rotation of a rigid body about an axis other than its natural center of mass may be achieved by prescribing Rigid Body Inertial Properties (section 4.38 on page 295) to relocate the center of mass to the desired axis of rotation.

For nodes of a deformable material, choosing $IDOF \ge 9$ interprets the prescribed quantity (velocity or acceleration) as a tangential nodal component for rotation about the global axis. This option is useful for prescribing a spin velocity about a global axis, such as for the outer nodes of a cylinder spinning about the global z-axis. Using this approach, only the magnitude of the tangential velocity of the node needs to be prescribed, and DYNA3D internally accounts for the change in velocity components due to the rotation. Since $IDOF \ge 9$ generates nodal velocity boundary conditions, it is applicable *only* to non-rigid materials.

4.23 RIGID WALLS

Define the number of rigid walls specified on Control Card 5 in section 4.1 on page 41. Repeat the following set of cards for each rigid wall definition.

		Card 1	
Col	umns	Quantity F	ormat
1-5	(1-8)	Number of slave nodes I5	(I8)
6-15	(9-18)	x-coordinate of point P (must be on rigid wall) E10.0	(E10.0)
16-25	(19-28)	y-coordinate of point P E10.0	(E10.0)
26-35	(29-38)	z-coordinate of point P E10.0	(E10.0)
36-45	(39-48)	x-coordinate of point Q (must be in space) E10.0	(E10.0)
46-55	(49-58)	y-coordinate of point Q E10.0	(E10.0)
56-65	(59-68)	z-coordinate of point Q E10:0	(E10.0)
66-70	(69-73)	Moving rigid wall option, $IMSWF$ I5 EQ.0: rigid wall is fixed in space EQ.1: rigid wall has mass m_{wall} and initial velocity v_0	(I5)
71-75	(74-78)	Stick condition option, ISTICKI5EQ.0: frictionless sliding after contactEQ.1: no sliding after contact	(I5)
76-80	(79-80)	Finite size rigid wall option, <i>LIMIT</i> I5 EQ.0: rigid wall extends to infinity EO.1: size and orientation is defined	(I2)

The geometry of rigid wall definitions is shown in Figure 29. Point P is at the origin of the rigid wall local coordinate system, and therefore must lie on the rigid wall surface. Point Q is a point in space, and the rigid wall normal vector n is defined as a vector from P to Q. This normal vector must point towards the impacting body. If LIMIT = 1, the dimensions of the rigid wall plane l_x and l_y (as specified on the next card) are measured from the origin at point P.

253

After impact, two friction options are available for rigid walls. If ISTICK = 0, then the wall is frictionless, and no resistance is provided to transverse motion in the plane of the wall. If ISTICK = 1, then sufficient friction is provided to prevent any transverse motion of the contacting body in the plane of the wall.



Card 1

Define this card only if LIMIT = 1.

Columns	Quantity	Format
1-10	x-coordinate of point R	E10.0
11-20	y-coordinate of point R	E10.0
21-30	z-coordinate of point R	E10.0
31-40	Length of x_l edge, l_x (see Figure 29) EQ.0.0: extends from negative to positive infinity	E10.0
41-50	Length of y_l edge, l_y EQ.0.0: extends from negative to positive infinity	E10.0

Note that l_x and l_y cannot both be zero if the finite rigid wall option is active (LIMIT = 1).

Card 1+LIMIT+IMSWF

Define this card only if IMSWF = 1.

Columns	Quantity	Format
1-10	Total mass of rigid wall, m_{wall}	E10.0
11-20	Initial velocity of rigid wall, v_0 , in direction of normal vector, n	E10.0

Card 2+LIMIT+IMSWF,...

Define these cards for all rigid walls.

.*

Columns		Quantity		Format	
1-10	(1-8)	Slave number	15	(I8)	
6-10	(9-16)	Node number	15	(I8)	

DYNA3D User Manual

•

Omitted slave nodes are automatically generated using node numbers found from the formula

$$n_{i+1} = n_i + k$$
 (266)

where the node number increment k is given by

$$\frac{n_i - n_j}{s_i - s_j}.$$
(267)

A rigid wall is a flat surface defined by a normal vector, n. It may be of infinite extent in each direction (*LIMIT* = 0), or may be finite in either or both in-plane directions (*LIMIT* = 1). The rigid wall may be fixed in space (*IMSWF* = 0), or may have a defined mass and initial velocity in the direction of the normal vector n (*IMSWF* = 1). The surface of the wall may be frictionless (*ISTICK* = 0) or frictional (*ISTICK* = 1). The impacting body is defined by listing nodes which are not permitted to penetrate the rigid wall. Nodes which are *not* listed may penetrate the wall without resistance. Defining a rigid wall of finite extent allows the possibility that the impacting body may pass outside the area of the rigid wall, and in that case no impact will occur.

Rigid walls are an inexpensive method for modeling unilateral contact (i.e., contact between a deforming body and a rigid body) when the target surface is planar. Moving rigid walls can be useful in modeling pendulum impact tests.

4.24 NODAL CONSTRAINTS

Define the number of nodal constraint sets specified on Control Card 5 in section 4.1 on page 41.

Card 1

<u>Columns</u> Quantity		For	<u>rmat</u>	
1-5	(1-8)	Number of nodes that share degrees-of-freedom, N_c	15	(I8)
6-10	(9-13)	Degrees-of-freedom in common, <i>IDOF</i> EQ.0: special "spotwelded nodes" option EQ.1: <i>x</i> -translational degree-of-freedom EQ.2: <i>y</i> -translational degree-of-freedom EQ.3: <i>z</i> -translational degree-of-freedom EQ.4: <i>x</i> and <i>y</i> -translational degrees-of-freedom EQ.5: <i>y</i> and <i>z</i> -translational degrees-of-freedom EQ.6: <i>z</i> and <i>x</i> -translational degrees-of-freedom EQ.7: <i>x</i> , <i>y</i> , and <i>z</i> -translational degrees-of-freedom	15 n	(I5)

Cards 2, . . .

Columns	Quantity		Format	
1-5 (1-8)	Node number of first node to be tied	15	(I8)	
6-10 (9-16)	Node number of second node to be tied	15	(I8)	
11-15 (17-24)	Node number of third node to be tied	15	(I8)	
•		•		
•	•	•		

Continue on additional cards in the same format until N_c nodes have been specified.

Nodal constraints allow sets of nodes to share a common degree-of-freedom. These degrees-of-freedom are in the global coordinate system if $IDOF \ge 1$. Any number of nodes may be included in a nodal constraint set.

The "spotwelded nodes option," (*IDOF* = 0), allows two nodes to be constrained to remain a constant distance apart while rotating into arbitrary configurations during the deformation. Only two nodes may be specified if the spotwelded nodes option is used. With this option, a vector is internally defined from node n_1 to node n_2 , and the nodes are rigidly constrained along this direction. This could equivalently be modeled using a rigid beam or truss element between nodes n_1 and n_2 , but the present option is less expensive and simpler to define.

4.25 INITIAL CONDITIONS

Cards 1, . . .

Define these cards if the initial condition flag is nonzero on Control Card 5 in section 4.1 on page 41.

Columns		Quantity	Format	
1-5	(1-8)	Node number I5	(18)	
6-15	(9-18)	Initial velocity in x-direction E10.0	(E10.0)	
16-25	(19-28)	Initial velocity in y-direction E10.0	(E10.0)	
26-35	(29-38)	Initial velocity in z-direction E10.0	(E10.0)	
36-40	(39-43)	Generation increment, k I5	(I5)	

Nodal velocity initial conditions are defined in this section. Initial velocities must be input for the first node, node 1, and the last node, NUMNP, if the initial condition flag on Control Card 5 is nonzero. Initial velocities for intermediate nodes may be input directly, or may be generated internally using a node number increment k. Linear interpolation between specified values is used to define initial velocities for omitted nodes.

Initial velocities for rotating bodies may be easily generated in DYNA3D using the Material Initialization for Rotational Motion option described in section 4.43 on page 303.

If prescribed nodal velocities (section 4.22 on page 251) are also present in the model, care should be taken to ensure that the given initial velocity agrees with the prescribed nodal velocity at t = 0.

2

.-

÷.,

4

*.*с.

DYNA3D User Manual

,

:

.

1. 8 14

. .

(n. a

4.26 SLIDING INTERFACE DEFINITIONS

Define *NUMSI* sliding interfaces as specified on Control Card 5 in section 4.1 on page 41. Define all *NUMSI* slide surface control card sets first, then define slave and master segment/node cards as needed. Note that type 11 (SAND) slide surface requires several control cards, and all other slide surface types require only one control card per slide surface.

Card Sets1, . . ., NUMSI

Col	umns	Quantity	F	ormat
1-5	(1-8)	Number of slave segments (types 1-4, 9, 10 & 11) or slave nodes (types 5-8), NSS	15	(I8)
6-10	(9-16)	Number of master segments, NMS	15	(I8)
11-15	(17-21)	Slide surface type, <i>ITYPE</i> : 1-sliding only 2-tied 3-sliding with separation and friction 4-single surface contact 5-discrete nodes impacting surface 6-discrete nodes tied to surface 7-shell edge tied to shell surface 8-nodes spotwelded to surface 9-tied with failure 10-one way algorithm for sliding with separation a 11-SAND (adaptive) contact for material failure 12-automatic contact	I5 and f	(I5)
16-25	(22-31)	Static coefficient of friction, μ_s E	10.0	(E10.0)
26-35	(32-41)	Dynamic coefficient of friction, μ_k EI	10.0	(E10.0)
36-45	(42-51)	Exponential friction decay coefficient, β E	10.0	(E10.0)
46-50	(52-56)	If <i>ITYPE</i> .ne.12, small penetration search flag, <i>IPEN</i> EQ.0: default EQ.1: penetration > element thickness ignored If <i>ITYPE</i> .eq.12, search robustness option EQ.1: fastest search EQ.3: most robust search	15 15	(I5) (I5)
51-55	(57-61)	Output slave side data in printed interface force file EQ.0: don't print EQ.1: print	15	(I5)

.....

DYNA3D User Manual

:

Columns		Quantity	Fo	rmat
56-60	(62-66)	Output master side data in printed interface force file EQ.0: don't print EQ.1: print	15	(I5)
61-70	(67-73)	Slave constraint factor, S_s E1 GT.0.0: Scale factor on default slave penalty stiffn EQ.0.0: Default set to 1.0 LT.0.0: Use Lagrangian constraint enforcement me (slidesurface types 3 and 12 only)	.0.0 ess ethod	(E7.0)
71-80	(74-80)	Scale factor on default master penalty stiffness, $S_m E1$	0.0	(E7.0)

. . .

ADDITIONAL CONTROL CARDS FOR SAND (TYPE 11) CONTACT SURFACES

Define the following additional Slide Surface Control Cards for SAND (type 11) contact:

Columns		Quantity		Format	
1-5	(1-5)	No. of materials in slave SAND volume, NSVM	15	(I5)	
6-10	(6-10)	No. of materials in master SAND volume, NMRM	I5	(I5)	

Next, define NSVM cards listing the material numbers which comprise the slave side SAND volume. Each card should have the following format:

Columns	Quantity		Format	
1-5 (1-5)	Material number to be included in the slave SAND volume	15	(I5)	

Now, define NMRM cards listing the material numbers which comprise the master side SAND volume. Each card should have the following format:

Colu	mns	Quantity		Format	
1-5	(1-5)	Material number to be included in the master SAND volume	15	(I5)	

Note that the same material number should never appear in both a slave SAND volume definition and a master SAND volume definition. It is acceptable to have no SAND volumes on either the slave or master sides, but in this case the slide surface will not be adaptively redefined, and slide surface type 3 may yield a more efficient solution to the problem. Other than the definition of SAND volumes as described above, type 11 (SAND) contact is defined in exactly the same way as type 3 slide surface (sliding with separation and friction).

For type 12 (automatic) contact, no additional input is necessary. For all other contact types, define *NUMSI* sets of slave segments or nodes and master segments as specified on the Slide Surface Control Cards described above. For each sliding interface, define first the slave segments or nodes, then the master segments. Repeat this pattern until slave segments or nodes and master segments are defined for all *NUMSI* sliding interfaces.

SLAVE SEGMENTS

Cards NUMSI+1, ..., NUMSI+NSS

Define the slave surface using the following *slave segment* cards for sliding interface types 1-4, 9, 10 and 11. For slide surface types 5-8, omit this slave surface definition and instead use the slave node definitions described later in this section.

Columns		Quantity	Format	
1-5	(1-8)	Slave segment number	15	(I8)
6-10	(-)	Generation increment, k	15	(omit)
11-15	(9-16)	Node n_1	15	(I8)
16-20	(17-24)	Node n_2	15	(I8)

DYNA3D User Manual

è

2

Columns		Quantity	<u>F</u>	Format	
21-25	(25-32)	Node n_3	15	(I8)	
26-30	(33-40)	Node n_4	I5	(I8)	
31-40	(41-50)	Normal failure stress, σ_{nf} (Type 9 only)	E10.0	(E10.0)	
41-50	(51-60)	Shear failure stress, σ_{sf} (Type 9 only)	E10.0	(E10.0)	

Omitted slave segment definitions are automatically generated by incrementing the node numbers by the generation increment, k.

Define the discrete *slave nodes* for sliding interface types 5-8. For sliding interface types 1-4, 9, 10 and 11, omit this slave node definition and instead use the slave surface definition described previously.

SLAVE NODES

Columns		Quantity	Format	
1-5	(1-8)	Slave number	15	(I8)
6-10	(9-16)	Node number	I5	(I8)
11-20	(17-26)	Normal force at failure, F_{fn} (type 8 only) E	10.0	(E10.0)
21-30	(27-36)	Shear force at failure, F_{fs} (type 8 only) E	10.0	(E10.0)
31-40	(37-46)	Failure criterion exponent for normal force, <i>n</i> E (type 8 only)	10.0	(E10.0)
41-50	(47-56)	Failure criterion exponent for shear force, m E (type 8 only)	10.0	(E10.0)

Define the master surface using the following master segment cards for sliding interface types 1-3 and 5-11 only. A master surface is not defined for single surface contact (sliding interface type 4).

MASTER SEGMENTS

Cards NUMSI+NSS+1, ..., NUMSI+NSS+NMS

Columns		Quantity	Format	
1-5	(1-8)	Master segment number	15	(18)
6-10	(-)	Generation increment, k	15	(omit)
11-15	(9-16)	Node n_1	15	(I8)
16-20	(17-24)	Node n ₂	I5	(I8)
21-25	(25-32)	Node n_3	15	(I8)
26-30	(33-40)	Node n_4	I5	(I8)

Node numbers for omitted master segments are generated internally using a node number increment of k.

SEGMENT DEFINITION

Sliding interfaces are described by defining one slave surface and one master surface. Each surface is then described as a collection of quadrilateral or triangular segments. Master and slave segments are described by listing four corner nodes, $n_1 - n_4$; triangular segments may be defined by repeating the last node ($n_4 = n_3$). Sliding interface segments correspond to element faces on the contact surface. For faces of solid elements, the ordering of nodes defining the slide surface segment may be either clockwise or counterclockwise; DYNA3D internally computes the correct segment outward normal vector. For faces of shell elements, the nodes defining the slide surface segment must be counterclockwise such that the resulting segment normal vector, computed using the "right-hand rule" as seen in Figure 30, points toward the potentially contacting surface. That is, normal vectors for slave segments should point toward the slave surface (see Figure 31). Segments may be given in any order to define a master or slave surface.





The contact formulations used in single surface (type 4) and SAND (type 11) account for the thickness of a shell element in contact calculations. All other contact options operate using the midplane definition of shell elements for the surface definition. Situations arise in modeling complex objects when both capabilities are needed, and the slide surface type can be chosen to obtain the desired behavior.

SLIDING INTERFACE TYPES

The sliding only (type 1) slide surface is a two-surface method based on a nodal constraint formulation. The two surfaces are allowed to slide arbitrarily large distances without friction, but are not permitted to separate or interpenetrate. This option performs well when extremely high interface pressures are present. The more coarsely meshed surface should be chosen as the master surface for best performance.

The tied (type 2) slide surface is not really a slide surface at all, but is a feature for joining two parts of a mesh with differing mesh refinement. This is also a nodal constraint formulation, so the more coarsely meshed surface should be chosen as the master surface. Tied interfaces may only be defined between surfaces of bodies discretized with solid elements.

.

The sliding with separation and friction (type 3) slide surface is the most generally applicable option. This implementation is a penalty formulation, and allows two bodies to be either initially separate or in contact. Large relative motions are permitted, and Coulomb friction is included. Surfaces may separate and come together in a completely arbitrary fashion. Contact between shell surfaces and solid body surfaces is supported. The choice of master and slave surfaces is not important in this formulation.

A rate-dependent Coulomb friction model is used in DYNA3D. The coefficient of friction is given by

$$\mu = \mu_k + (\mu_s - \mu_k) e^{-\beta v_{rel}},$$
(268)

where μ_s and μ_k are the static and kinetic friction coefficients, β is a transition coefficient governing the rate of change from static friction to kinetic friction, and v_{rel} is the relative velocity between the two sliding surfaces. Note that if $\mu_k = 0$ and $\beta = 0$, then a rate-independent friction model is recovered with $\mu = \mu_s$.

The single surface contact (type 4) slide surface is a penalty formulation used for modeling two portions of the same body which may come into contact. This situation often arises in buckling problems, where one surface develops folds and comes into contact with itself. This option may be used with either solid or shell element surfaces. Frictional sliding is permitted between surfaces in contact. When used with shell element surfaces, it is only necessary to define one side of the surface in the slide surface definition; internal logic accounts for the possibility that contact may occur on either side. Due to the complexity of this algorithm, it is slightly more expensive than the other slide surface options, and therefore should only be used where necessary. Note that no master surface is defined for single surface contact.

The discrete nodes impacting surface (type 5) slide surface is a penalty formulation which allows separate definition of the slave *nodes* and master *surface*. This option is necessarily not symmetric, since no slave segments are defined. This feature can be used to model the impact of beams or shell edges onto a solid or shell surface, for example. The target surface is specified as the master surface, and the nodes of the beams or the shell edge nodes are defined as the slave nodes. Coulomb friction may be defined between the nodes and the master surface.

The **discrete nodes tied to surface** (type 6) slide surface is analogous to the "discrete nodes impacting surface" option described above except that the slave nodes are required to remain rigidly attached to the master surface at their initial location. This feature is sometimes useful in joining two parts of a difficult mesh.

The shell edge tied to shell surface (type 7) slide surface is another specialization of the tied slide surface idea, although the implementation is somewhat different. This option allows the easy definition of intersections between a shell edge and a shell surface when the mesh spacing prevents merging nodes. Nodes should be merged in lieu of using this slide surface when possible, however, to obtain the least expensive solution.

The **node spotwelded to surface** (type 8) slide surface is a combination of the type 5 and type 6 slide surfaces. Slave nodes are constrained to remain on the master surface until a prescribed failure criterion is satisfied. Once the spotweld has "failed," the node is released and then the slide surface functions as a type 5 slide surface.

The failure criterion for spotwelds is

$$\left(\frac{F_n}{F_{fn}}\right)^n + \left(\frac{F_s}{F_{fs}}\right)^m \ge 1,$$
(269)

where F_{fn} and F_{fs} are the specified normal and shear failure strengths, F_n and F_s are the current normal and shear forces in the spotweld, and n and m are the specified failure criterion exponents.

The tied with failure (type 9) slide surface is an *experimental* penalty method formulation. This option functions as a tied slide surface until a prescribed failure criterion is satisfied, thereafter the slide surface functions as a type 3 slide surface. This option may only be used between surfaces defined from solid elements, and the designation of the master and slave surfaces does not affect the results with this formulation.

The failure criterion for the type 9 "tie-break" interface is

$$\left(\frac{F_n}{F_{nf}}\right)^2 + \left(\frac{F_s}{F_{sf}}\right)^2 \ge 1,$$
(270)

where F_n and F_s are the total normal and shear forces acting on a segment, and F_{nf} and F_{sf} are the normal and shear failure forces for the segment; these values are internally computed based on the segment area and the specified normal and shear failure stresses, σ_{nf} and σ_{sf} .

The one-way algorithm for sliding with separation and friction (type 10) slide surface is an unsymmetric implementation of the algorithm described for type 3 above. This option may effectively be used when one surface is defined as part of a rigid body, as often occurs in metal forming problems. This algorithm is somewhat less expensive than the symmetric type 3 algorithm, but may yield poor results if used between two deformable bodies. The master surface should correspond to the rigid surface, and the slave surface should correspond to the deformable surface.

The SAND (Slidesurfaces with Adaptive New Definitions) (type 11) slide surface is an algorithm for modeling material failure along contact surfaces. A failure criterion is defined for a volume of material adjacent to a SAND contact surface; these volumes of potentially failing material are called SAND volumes (see Figure 33). As material within an element on the contact surface fails, the failed element is removed from the calculation and the slidesurface definition adapts to the new exterior boundary of the unfailed material. SAND contact may include arbitrary mixtures of solid elements and shell elements, as shown in Figure 32. A SAND volume may be composed entirely of solid elements or shell elements for simple structures, or may include an external layer of shell elements followed by layers of solid elements and shell elements as needed to model a honeycomb or composite structure. Only the outside surface region of potential contact needs to be included in the initial sliding interface definition, so this definition is exactly the same as it would be for the usual (type 3) slide surface. SAND volumes are defined by listing the DYNA3D material numbers (not material types) for all materials which comprise the SAND volume. A SAND contact interface may have a SAND volume on only the master side, on only the slave side, or on both the master and slave sides. This capability allows efficient modeling of problems where failure is expected to occur only on one side and problems where failure may occur on both sides of a sliding interface. SAND contact interfaces allow improved modeling of many penetration and failure problems with DYNA3D.



definition to model composite and "built-up" structures.

Notes for Using SAND: SAND volume materials are currently restricted to be-of Material Type 24 for solid or shell elements, or Material Types 10, 11, 13, or 35 for solid elements. Also, SAND requires that the results be processed using an appropriate postprocessor, as the plot database must contain new information. To signal DYNA3D to write this new format database, the SAND database flag must be set on Control Card 3 (section 4.1 on page 41). The list of SAND materials with failure criteria will be greatly expanded in the next release of the code, and in addition some new kinds of failure criteria will soon be available.



Definition of SAND volumes containing material which can potentially fail. SAND volumes are adjacent to a SAND contact surface on one or more sides.

Automatic contact (type 12) is a new experimental feature in this release of DYNA3D. Although the underlying algorithms differ greatly from the other DYNA3D contact options, from a user's perspective the main difference is that automatic contact does *not* require the specification of contact surfaces - they are "automatically" defined. This greatly simplifies the development of complex DYNA3D models, but may lead to a slight increase in computation time. As this capability matures, with constructive feedback from our Collaborators, this option will improve rapidly and may soon become the standard technique for modeling contact.

271

SMALL PENETRATION FLAG

States Area A

If this option is activated, then DYNA3D ignores any apparent penetration found by the contact algorithm when the penetration depth is greater than one element thickness. Actual contact penetrations are usually much less than an element thickness, so it is likely that a large penetration is found due to a convex contact surface and a node getting "behind" the opposite surface. This behavior is usually indicated when a calculation is interrupted by nodes on a contact surface "shooting" off in random directions. The small penetration flag can almost always be used without harm, except in hypervelocity impact problems. Many analysts activate this option in all analysis models.

PRINTED INTERFACE FORCE FILE

If this option is activated on the slidesurface control card, a file called "FORCES" will be generated at run time containing the contact force at each node of the contact surface. This file is written at the interval prescribed as the time history plot interval on Control Card 6 in section 4.1 on page 41. For each node on the contact surface, the total interface force at that node is printed in both global and local coordinates. The total force across the interface is also written to this file. Results from this file may be processed to yield a variety of interface force results.

Interface force data may also be written to a binary plot file as discussed in section 2.15 on page 24 and section 3.6 on page 39.

CONTACT CONSTRAINT ENFORCEMENT OPTIONS

DYNA3D uses the penalty method by default to enforce the no-interpenetration contact constraint. As an alternative, a Lagrangian formulation has also been implemented in this version of DYNA3D for slidesurface types 3 and 12.

When the penalty formulation is chosen to enforce the contact constraint, a restoring force is placed on a penetrating node to return it to the surface. This force is proportional to the depth of penetration, the bulk modulus of the penetrated material, the dimensions of the penetrated element, and a scale factor specified by the user. The amount of penetration observed is inversely proportional to the chosen scale factor. Using a larger penalty scale factor reduces the observed penetration, but may make the problem more difficult to solve. Using a smaller penalty scale factor makes the problem easier to solve, but may lead to unacceptably large interpenetrations. One global penalty scale factor is defined in the Control Section of the DYNA3D input file, and another

272

penalty scale factor may be defined separately for each surface of a slide surface pair. The default value of the global penalty scale factor is 0.10, and the default for the local penalty scale factors is 1.0. These two factors are combined multiplicatively to arrive at the actual penalty scale factor used for computations.

When the Lagrangian formulation is chosen to enforce the contact constraint, a restoring force is again placed on a penetrating node to return it to the surface, but is computed differently. This option computes the force by determining the predicted location of a node on the interface at the end of the time step, and computing the contact force required to place the node on the surface at the end of the time step. This method yields exact enforcement of the contact constraint for node-on-node contact, and good approximations for most reasonable mesh refinements. Although this is a new option in this release of DYNA3D, early use at LLNL has shown considerable promise. This option works particularly well for contact between very thin shells, such as sometimes occurs in sheet forming problems. It addition, it is also advantageous in modeling contact betweem materials of widely varying stiffness.

.

4.27 TIE-BREAKING SHELL SLIDELINES

Define the number of tie-breaking shell slidelines, *NTBSL*, as specified on Control Card 7 in section 4.1 on page 41. First, input the following control card for all *NTBSL* tie-breaking shell slidelines.

		Cards 1,, NTBSL		
Columns		Quantity	Format	
1-5	(1-8)	Number of slave nodes in this slideline, N_s	15	(I8)
6-10	(9-16)	Number of master nodes in this slideline, N_m	15	(I8)

Following the definitions of all *NTBSL* control cards, define the following slideline card set for each tie-breaking shell slideline.



Cards NSN+2, \dots , NSN+NMN+1

Columns		Quantity	Format	
1-5	(1-8)	Master number EQ.0: the preceding slave number is incremented	I5 by 1	(I8)
6-10	(9-16)	Node number	15	(I8)

Slidelines are defined by listing a set of nodes along the edge of a shell structure. One line of nodes is designated as the slave side, and the other line is designated as the master side. Nodes should be given in the order in which they appear along the surface. Nodes on the slave slideline may be given in order moving down the slideline in either direction, but nodes on the master slideline must be given in order such that the slave slideline lies to the left. *Tie-breaking shell slidelines must not cross*.

Tie-breaking shell slidelines may be used to tie shell edges together with a failure criterion on the joint. If the average volume-weighted effective plastic strain $\overline{\epsilon}^p$ in the shell elements adjacent to a node exceeds the specified plastic strain at failure, $\overline{\epsilon}_f^p$, then the tied slideline is released at that node. One default failure plastic strain is defined for the entire tie-breaking shell slideline, but this may be overridden on the slave node cards to define a unique failure plastic strain for each node.

Tie-Breaking Shell Slidelines may be used to simulate the effect of failure along a predetermined line, such as a seam or structural joint. When the failure criterion is reached in the adjoining elements, nodes along the slideline will begin to separate. As this effect propagates, the tied slideline will appear to "unzip," thus simulating failure of the connection.

A more general (albeit more complex) fracture and failure modeling approach uses Tied Node Sets with Failure, as described in section 4.28 on page 276.

DYNA3D User Manual

2

4.28 TIED NODE SETS WITH FAILURE

Define control cards for *NTNWF* Tied Node Sets with Failure as specified on Control Card 7 in section 4.1 on page 41.



Repeat the following group of cards to define NTNWF tied node data sets:

Define the following Constraint Set Default Card as the first card in each constraint set.



Repeat the following Constraint Definition Card(s) as necessary to define NTNC constraints in this set.

Cards NTNWF+2, ..., NTNWF+NTNC

Columns		Quantity	<u>F</u>	Format	
1-10	(1-10)	Plastic strain failure, $\overline{\varepsilon}_{f}^{p}$ (optional)	E10.0	(E10.0)	
11-15	(17-24)	First node to be tied in this constraint	15	(I8)	
16-20	(25-32)	Second node to be tied in this constraint	15	(I8)	
•	•	•	•	•	
•	•	•	•	•	
•	•	•	•	•	
75-80	(57-64)	Twelfth (sixth) node to be tied in this constraint	15	(I8)	

Use additional cards as necessary, continuing with a format of 1415 (or 1018 for the "large" input option) on subsequent cards, until all nodes for this constraint have been defined.

A problem may contain several tied node sets with failure, as specified by *NTNWF*. One tied node set has *NTNC* constraints, and each constraint may consist of from two to *NCMAX* nodes. All constraints in one tied node set share a common default plastic strain at failure, $\overline{\varepsilon}_{f}^{p}$, but this may be overridden by giving a value for $\overline{\varepsilon}_{f}^{p}$ in the constraint definition.

Tied Node Sets with Failure apply only to shell elements. The specified nodes are tied together and are coincident until the volume-weighted effective plastic strain, averaged over all elements connected to nodes in a given constraint, exceeds the specified value. Once this failure criterion is exceeded, all nodes at that constraint are released to simulate the initiation of a crack or fracture.

To use this feature to simulate fracture and fragmentation of a plate or shell, all elements in the potential failure region should be generated to have unique nodes (i.e., no elements share nodes in common with other elements). These free elements are then tied together using Tied Node Sets with Failure at every corner, where nodes would be merged in a conventional model. Using this approach, a fracture may propagate through the plate as dictated by the analysis, and some estimate of the crack direction may be obtained. The fracture is required to move along element boundaries with this approach, however.

Tied Node Sets with Failure are similar to Tie-Breaking Shell Slidelines (described in section 4.27 on page 274), except that entire regions of individual shell elements may be tied together with this option and the fracture direction is not pre-defined. Tie-Breaking Shell Slidelines are more suited to a pre-defined potential fracture surface, and are somewhat less expensive and easier to define.

277
2

4.29 RIGID BODY MERGES

Define NRBC rigid body merge cards as specified on Control Card 4 in section 4.1 on page 41.

Columns	Quantity	Format
1-5	Master rigid body material number	15
6-10	Slave rigid body material number	15

The slave rigid body is merged into the master rigid body.

Two rigid materials are not permitted to have any nodes in common. If common nodes exist, define the two rigid materials as above to be merged into one rigid body for the analysis. This situation occurs infrequently, but may arise when rigid materials are used for solid elements, shell elements, and beam elements in close proximity in a model.

It is not necessary (or reasonable) to merge rigid materials which are completely separate and have no nodes in common.

•

4.30 EXTRA NODES FOR RIGID BODIES

Define one card set for each rigid body for which extra nodes are defined until *NRBEN* sets have been defined. *NRBEN* is specified on Control Card 7 in section 4.1 on page 41.

Columns	Quantity	Format
1-5	Rigid body material number	15
6-10	Number of extra nodes for this rigid material, NEN	15

Cards 2, ..., NEN+1

Columns		Quantity	<u> </u>	Format	
1-5	(1-8)	First extra node for this rigid material	15	(I8)	
6-10	(9-16)	Second extra node for this rigid material	15	(I8)	
11-15	(17-24)	Third extra node for this rigid material	15	(I8)	
	•			•	
•	•		•	•	
•	•			•	
45-50	(73-80)	Tenth extra node for this rigid material	15	(I8)	

Continue in this format until NEN extra nodes have been defined for this rigid material. Then, repeat both cards above until extra nodes have been defined for NRBEN rigid materials.

Extra nodes may be needed for rigid materials to facilitate the definition of a rigid body joint between two rigid materials, or to allow attachment of another structural member. Rigid body joints are described in section 4.31 on page 280.

DYNA3D User Manual

<u>ج</u>،

4.31 RIGID BODY JOINTS

Define the number of joint definitions specified on Control Card 7 in section 4.31 on page 280.

Columns		Quantity	<u>F</u>	Format	
1-10	(1-10)	Relative penalty stiffness (default is 1.0)	E10.0	(E10.0)	
11-15	(11-15)	Joint type EQ.1: Spherical EQ.2: Revolute EQ.3: Cylindrical EQ.4: Planar EQ.5: Universal EQ.6: Translational	15	(15)	
16-20	(16-23)	Node n_1	15	(I8)	
21-25	(24-31)	Node n ₂	15	(I8)	
26-30	(32-39)	Node n ₃	15	(I8)	
31-35	(40-47)	Node n_4	15	(I8)	
36-40	(48-55)	Node n ₅	15	(I8)	
41-45	(56-63)	Node n ₆	I5	(I8)	

The geometry of rigid body joints is shown in Figure 34. At each timestep, the relative penalty stiffness is multiplied by a function dependent on the time step size to give the maximum stiffness that will not destroy the stability of the solution. If the errors in the joint constraints are too large, the timestep must be reduced; simply increasing the relative stiffness will have no effect.

(

ĩ

:

4 1



,

4.32 PRESCRIBED BASE ACCELERATIONS

Define nonzero "base acceleration" loads as specified on Control Card 5 in section 4.1 on page 41.

X-Direction Base Acceleration

Define this card only if a nonzero x-direction base acceleration is specified on Control Card 5.

Columns	Quantity	Format
1-5	Load curve number giving time variation	15
6-15	Scale factor on x-acceleration EQ.0.0: default set to "1.0"	E10.0

Y-Direction Base Acceleration

Define this card only if a nonzero y-direction base acceleration is specified on Control Card 5.

<u>Columns</u>	Quantity	Format
1-5	Load curve number giving time variation	15
6-15	Scale factor on y-acceleration EQ.0.0: default set to "1.0"	E10.0

Z-Direction Base Acceleration

Define this card only if a nonzero z-direction base acceleration is specified on Control Card 5.

<u>Columns</u>	Quantity	Format
1-5	Load curve number giving time variation	15
6-15	Scale factor on z-acceleration EQ.0.0: default set to "1.0"	E10.0

Translational base accelerations allow body force loads, such as gravity, to be imposed on a structure. Conceptually, base acceleration may be thought of as accelerating the coordinate system in the direction specified, and thus the inertial loads acting on the model are of opposite sign. For example, if a cylinder were fixed to the x-y plane and extended in the positive z direction, then a positive z-direction base acceleration would tend to shorten the cylinder.

4.33 PRESCRIBED ANGULAR VELOCITIES

Define nonzero angular velocity components about the global axes as specified on Control Card 5 in section 4.1 on page 41.

ANGULAR VELOCITY ABOUT GLOBAL X-AXIS

Define this card only if a nonzero angular velocity component about the global x-axis is specified on Control Card 5.

Columns	Quantity	Format
1-5	Load curve number giving time variation	15
6-15	Scale factor on angular velocity EQ.0.0: default set to "1.0"	E10.0

ANGULAR VELOCITY ABOUT GLOBAL Y-AXIS

Define this card only if a nonzero angular velocity component about the global y-axis is specified on Control Card 5.

Columns	Quantity	Format
1-5	Load curve number giving time variation	15
6-15	Scale factor on angular velocity EQ.0.0: default set to "1.0"	E10.0

ANGULAR VELOCITY ABOUT GLOBAL Z-AXIS

Define this card only if a nonzero angular velocity component about the global z-axis is specified on Control Card 5.

<u>Columns</u>	Quantity	Format
1-5	Load curve number giving time variation	15
6-15	Scale factor on angular velocity (default = 1.0)	E10.0

Body force loads due to the angular velocity are always calculated with respect to the deformed configuration, and act radially outward from the axis of rotation. Torsional effects arising from changes in angular velocity are not included. Angular velocity is assumed to have the units of radians per unit time.

The body force density b at a point P in the body is calculated from

$\boldsymbol{b} = \rho \left(\boldsymbol{\omega} \times \boldsymbol{\omega} \times \boldsymbol{r} \right), \tag{271}$

where ρ is the mass density, ω the angular velocity, and r is a position vector from the origin to point P. Note that although the angular velocity may vary with time, the effects of angular acceleration are not included in this formulation.

This feature is useful for studying transient deformations of three-dimensional objects which are spinning about any axis of rotation through the global origin. Typical applications of this feature could include modeling the deformations of a tire spinning about an axle, or the impact of a foreign object on a satellite spinning rapidly about its own axis.

4.34 MOMENTUM DEPOSITION IN SOLID ELEMENTS

Card 1

Define momentum deposition data for the number of solid elements specified on Control Card 5 in section 4.1 on page 41.

Columns		Quantity	<u> </u>	Format	
1-5	(1-8)	Solid element number to receive momentum	(I5)	(I8)	
6-15	(9-18)	x-direction momentum	E10.0	(E10.0)	
16-25	(19-28)	y-direction momentum	E10.0	(E10.0)	
26-35	(29-38)	z-direction momentum	E10.0	(E10.0)	
36-45	(39-48)	Deposition time	E10.0	(E10.0)	

Momentum may be deposited in selected solid elements at a specified time during an analysis. This feature is useful for modeling impacts of a large body with many smaller bodies where only the momentum transfer is significant, and where detailed modeling of the collisions of the smaller bodies may lead to excessive cost. An example application of this feature would be the effect of dense blowing sand on a fast moving body, such as an automobile or low-flying aircraft.

4.35 DETONATION POINTS

Define one card for each high explosive detonation point specified on Control Card 5 in section 4.1 on page 41.

Columns	Quantity	Format
1-10	Lighting time for detonation point	E10.0
11-15	Material to be lit EQ.0: all H.E. materials are considered	15
16-25	x-coordinate of detonation point	E10.0
26-35	y-coordinate of detonation point	E10.0
36-45	z-coordinate of detonation point	E10.0

During DYNA3D initialization, the lighting time of each element is computed using a simple programmed burn algorithm. The resulting point detonation is illustrated in Figure 35. The lighting time t_L for an element is computed based on the distance from the center of the element to the nearest detonation point l_d , the detonation velocity D, and the lighting time of that detonation point t_{det} , using

$$t_L = t_{det} + \frac{l_d}{D}.$$
 (272)



4.36 SHELL-SOLID INTERFACES

Define the number of shell-solid interfaces specified on Control Card 7 in section 4.1 on page 41.

Card 1

<u>Columns</u>	Quantity	Format
1-5	Number of shell element nodes, NSHLN	I5
6-10	Number of solid element nodes tied to each shell node (maximum of eight)	15

Card 2, ..., NSHLN

Columns	Quantity	For	mat
1-5 (1-8)	Shell element node, n_s	15	(I8)
6-10 (9-16)	First solid element node, n_{b1}	15	(I8)
11-15 (17-24)	Second solid element node, n_{b2}	15	(I8)
		•	
•	•	•	
•	•	•	
•	•	•	
41-45 (65-72)	Eighth solid element node, n_{b8}	15	(I8)

The shell-solid interface is an *experimental* capability for coupling regions of 4-node shell elements to regions of solid elements. Since shell elements have 6 degrees-of-freedom per node (three translations and three rotations) and solid elements have only 3 degrees-of-freedom per node (three translations), a mismatch in degrees-of-freedom occurs along a shell-solid boundary. The approach followed in this option is to constrain nodes of the solid elements to lie along the shell fiber direction at the interface, thus transferring moments in the shell elements into spatially

varying stresses in the solid elements. The nodes of the solid elements are allowed to move along the tangent nodal fiber, and are constrained to move only along this fiber direction, as shown in Figure 36.

The solid element nodes must be specified in the order in which they occur along the shell fiber direction, but the fiber may be traversed in either the positive or negative direction.



4.37 DISCRETE SPRINGS, DAMPERS, AND MASSES

Define discrete springs, dampers, and masses in this section if INPSD = 1 on Control Card 7 in section 4.1 on page 41. Discrete element input follows the general organization of the entire DYNA3D input file: first a control card defines the number of discrete element materials, discrete elements, and lumped nodal masses. Next, each discrete element material is defined by a Discrete Element Material Control Card and a Discrete Element Material Data Card. Then, discrete elements are defined on Discrete Element Data Cards, and discrete masses are specified on Lumped Mass Data Cards.

First, define the Discrete Element Control Card:



Next, for each of the *NDMAT* discrete element materials, define the Discrete Element Material Control Card and the Discrete Element Material Data Card as described below.

The Discrete Element Material Control Card is defined as:

	Cards 2, 4, 6,	
Columns	Quantity	Format
1-5	Discrete element material number ($\leq NDMAT$)	15
6-10	Discrete element material type: EQ.1: linear elastic EQ.2: linear viscous EQ.3: isotropic elastoplastic EQ.4: nonlinear elastic EQ.5: nonlinear viscous EQ.6: general tabulated nonlinear	15

•

:

· · · · ·

DYNA3D User Manual

.

~ .r

The Discrete Element Material Data Card is defined as described below for each discrete element material type.

Cards 3, 5, 7, ...

	Discrete Element Material Type 1: (Linear Elastic)	
Columns	Quantity	Format
1-10	Elastic stiffness (spring constant), k (force/displ.)	E10.0
	Discrete Element Material Type 2: (Linear Viscous)	
Columns	Quantity	Format
1-10	Damping constant (viscosity), c (force/velocity)	E10.0
	Discrete Element Material Type 3: (Isotropic Elastoplastic)	
<u>Columns</u>	Quantity	Format
1-10	Elastic stiffness, k (force/displ.)	E10.0
11-20	Tangent stiffness, k_T (force/displ.)	E10.0
21-30	Yield, F_{γ} (force)	E10.0
	Discrete Element Material Type 4: (Nonlinear Elastic)	
<u>Columns</u>	Quantity	Format
1-10	Load curve number giving force vs. displacement curve	E10.0
	Element Material Type 5: (Nonlinear Viscous)	
Columns	Quantity	Format
1-10	Load curve number giving force vs. velocity curve	E10.0

Discrete Element Material Type 6: (General Tabulated Nonlinear)

Columns	Quantity	Format
1-10	Load curve number giving force vs. displacement for loading, LC_L	E10.0
11-20	Load curve number giving force vs. displacement for unloading, LC_U	E10.0
21-30	Plastic hardening parameter, β ($0 \le \beta \le 1$)	E10.0
31-40	Initial yield force in tension $(F_{\gamma T} > 0)$	E10.0
41-50	Initial yield force in compression ($F_{YC} < 0$)	E10.0

Load curves define force as a function of displacement. The load curve points must be in order, starting with the most negative (compressive) displacement and ending with the most positive (tensile). The curves need not be symmetrical. The curve must not have a zero slope at any point; however, the slope may be made a small nonzero value if desired. The behavior of Discrete Element Material Type 6 is shown in Figure 37.

The displacement origin of the "unloading" curve is arbitrary, since it will be shifted as necessary as the element extends and contracts. On reverse yielding the "loading" curve will also be shifted along the displacement axis. The initial tensile and compressive yield forces (F_{YT} and F_{YC}) define a range within which the element remains elastic (i.e. the "loading" curve is used for both loading and unloading). If at any time the force in the element exceeds this range, the element is deemed to have yielded, and at all subsequent times the "unloading" curve is used for unloading.

DYNA3D User Manual



Define *NUMELD* Discrete Element Data Cards as specified on the Discrete Element Control Card.

Cards 2+2*NDMAT, ..., 1+2*NDMAT+NUMELD

Col	umns	Quantity	Format
1-5	(1-8)	Discrete element number I5	(I8)
6-10	(9-16)	First node, n_1 I5	(I8)
11-15	(17-24)	Second node, n_2 I5	(I8)
16-20	(25-32)	Discrete element material number I5	(I8)
21-30	(33-42)	Scale factor on computed force (default = 1.0) E10.0	(E10.0)

Define NUMMAS Lumped Mass Data Cards as specified on the Discrete Element Control Card.

Cards 2+2*NDMAT+NUMELD, ..., 1+2*NDMAT+NUMELD+NUMMAS

Colu	imns	Quantity	<u>F</u>	format
1-5	(1-8)	Node number,	15	(I8)
6-15	(9-18)	Mass	E10.0	(E10.0)

Lumped nodal masses are added to any existing mass at a node.

Discrete elements (springs and dampers) and lumped nodal masses provide a convenient method for imposing prescribed force-displacement relationships between two nodes. These features may be used alone to allow DYNA3D to solve lumped parameter spring-mass-damper models, or may be used in combination with a finite element model to represent a complex system.

DYNA3D User Manual

Note that if discrete springs are connected to a rigid material for which rigid body inertial properties (section 4.38 on page 295) have been defined, it is possible in some circumstances to experience difficulties with the time step automatically chosen by DYNA3D. In most cases, simply altering the specified material density for the rigid material up to increase the time step or down to decrease the time step will provide the necessary user control without affecting anything else in the problem.

. . . S.

Internal forces in discrete elements may now be output to an ASCII file called FORDIS. This option is activated using the flag on Control Card 3 in section 4.1 on page 41.

-

۴

4.38 RIGID BODY INERTIAL PROPERTIES

Define rigid body inertial properties for rigid materials as specified on Control Card 7 in section 4.1 on page 41. All requested data must be provided, and this data overrides other input data (such as nodal initial velocities) for nodes of rigid bodies.

Card 1

Columns	Quantity	Format
1-5	Material number of rigid body	15
6-15	x-coordinate of center of mass	E10.0
16-25	y-coordinate of center of mass	E10.0
26-35	z-coordinate of center of mass	E10.0
36-45	Translational mass	E10.0

Rigid materials are defined using Material Type 20 as described in section 4.2 on page 59. Normally, each rigid material is considered a separate rigid body unless the Rigid Body Merge option (section 4.29 on page 278) has been used.

Card 2

Columns	Quantity	Format
1-10	Moment of inertia in the global x direction, I_{xx}	E10.0
11-20	Product of inertia in x-y direction, I_{xy}	E10.0
21-30	Product of inertia in x-z direction, I_{xz}	E10.0
31-40	Moment of inertia in the global y direction, I_{yy}	E10.0
41-50	Product of inertia in y-z direction, I_{yz}	E10.0
51-60	Moment of inertia in the global z direction, I_{zz}	E10.0

The mass moments of inertia are defined in the global coordinate system.

DYNA3D User Manual

Card 3

Columns	Quantity	Format
1-10	Initial rigid body translational velocity in the x-direction	E10.0
11-20	Initial rigid body translational velocity in the y-direction	E10.0
21-30	Initial rigid body translational velocity in the z-direction	E10.0
31-40	Initial rigid body angular velocity about the x-axis (radians/time unit)	E10.0
41-50	Initial rigid body angular velocity about the y-axis	E10.0
51-60	Initial rigid body angular velocity about the z-axis	E10.0

Note that these initial velocities must be specified for any rigid body with inertial properties defined in this section. This data overrides nodal initial velocities specified in section 4.25 on page 259 for nodes of the rigid body. Initial velocity data for other nodes in the model is not affected.

Rigid body inertial properties are calculated by default from the geometry of the elements and the mass density ρ specified for the material. The Rigid Body Inertial Properties option allows the complete set of inertial properties to be specified directly for a rigid body. This approach is useful if the body has a spatially varying mass density which is not conveniently included in the material definitions, or if geometric approximations are made in the model which render the inertial properties calculated using the default procedure inaccurate.

Note: kinetic energy reported by DYNA3D may not properly account for the modified mass and moment of inertia of the rigid body, but actual computations use the modified values. Also, time step calculations for discrete springs and dampers connected to rigid bodies are based upon the original (as-input) material density. If undesirable time steps are selected by the code, modify the density specified for the rigid material until a suitable time step is achieved.

Useful tricks include defining the rigid material with a density such that the total mass is computed correctly, or with a very small density so that no significant contribution to kinetic energy is calculated from the rigid material.

4.39 NONREFLECTING BOUNDARY SEGMENTS

Define NNRBS nonreflecting boundary segments as specified on Control Card 7 in section 4.1 on page 41.

Col	umns	Quantity	Fe	ormat
1-5	(1-8)	Nonreflecting boundary segment number	15	(I8)
6-10	(-)	Generation increment, k	15	(omit)
11-15	(9-16)	Node n_1	15	(I8)
16-20	(17-24)	Node n_2	15	(I8)
21-25	(25-32)	Node n_3	I5	(I8)
26-30	(33-40)	Node n_4	15	(I8)

Nonreflecting boundaries are only used with *solid* elements. Boundaries are defined as a collection of segments, and segments are equivalent to element faces on the boundary. Segments are defined by listing the corner nodes in either a clockwise or counterclockwise order. The first and last nonreflecting boundary segments must be explicitly defined. Gaps in intermediate segment numbers are filled by automatically generating segment definitions by adding the generation increment k to each node number of the previous segment.

Nonreflecting boundaries are used on the exterior boundaries of an analysis model of an infinite domain, such as a half-space, to prevent artificial stress wave reflections generated at the model boundaries from reentering the model and contaminating the results. Internally, DYNA3D computes an impedance matching function, in both normal and shear directions, for all nonreflecting boundary segments based on an assumption of *linear material behavior*. Thus, the finite element mesh should be constructed so that all significant nonlinear behavior is contained within the discrete analysis model.

4.40 TEMPERATURE INPUT OPTION I

Define one temperature card for each of the *NUMNP* nodes if the thermal effects option is set to -9999 on Control Card 6 in section 4.1 on page 41.

Col	umns	Quantity F	'ormat
1-5	(1-8)	Node number, n_i I5	(I8)
6-15	(9-18)	Temperature scale factor at this node, S_i E10.0	(E10.0)
16-25	(19-28)	Base temperature, T_i^{base} E10.0	(E10.0)
26-30	(29-36)	Load curve number giving time variation I5	(I8)
31-35	(37-44)	Generation increment, k I5	(I8)

Temperature data for missing nodes is generated using the specified node number increment, k. The temperature scale factor S_i and the base temperature T_i^{base} are linearly interpolated between the starting and ending values.

At any time t, the temperature at a node n_i is given by

$$T(t) = T_i^{base} + S_i f(t),$$
 (273)

where T_i^{base} is the specified base temperature, S_i is the specified temperature scale factor, and f(t) is the value of the specified load curve at the current time.

4.41 TEMPERATURE INPUT OPTION II

Define one temperature card for each of the NUMNP nodes if the thermal effects option is set to -2 on Control Card 6 in section 4.1 on page 41.

Col	umns	Quantity	ormat
1-5	(1-8)	Node number I5	(I8)
16-25	(19-28)	Temperature E10.0	(E10.0)

Data for missing nodes is automatically generated by linearly interpolating from the endpoint values.

The reference temperature state is assumed to be a zero state with this option. One set of nodal temperatures is defined and held constant throughout the analysis.

· .

4.42 ONE DIMENSIONAL SLIDELINES

1 25

Define NUMSL one-dimensional slidelines as specified on Control Card 9 in section 4.1 on page 41. Repeat the following set of three cards for each one-dimensional slideline.

Define the One-Dimensional Slideline Control Card:

Columns	Quantity	Format
1-5	Number of slave nodes, NSN	15
6-10	Number of master nodes, NMN	15
11-20	External radius of rebar, R_o	E10.0
21-30	Compressive strength of concrete, f'_c	E10.0
31-40	Bond shear modulus, G	E10.0
41-50	Maximum shear displacement, u_{max}	E10.0
51-60	Exponent in damage curve, h_{dmg}	E10.0
61-70	Bond type flag, <i>IBOND</i> EQ.0: elastic perfectly plastic GT.0: elastic-plastic with hardening	E10.0

Next, define the slave nodes as described below:

Cards 2, ..., NSN+1

Colu	imns	Quantity Fo	rmat
1-5	(1-8)	Slave number I5 EQ.0: the preceding slave number is incremented by 1	(I8)
6-10	(9-16)	Node number I5	(I8)

Omitted slave node definitions are automatically generated by incrementing the node numbers by

$$\frac{n_i - n_j}{s_i - s_j} \tag{274}$$

where s_i and s_j are the slave numbers on two successive cards, and n_i and n_j are their corresponding node numbers.

The master nodes are now defined as described below:

Cards NSN+2, \dots , NSN+NMN+1

Colu	imns	Quantity	For	mat
1-5	(1-8)	Master number EQ.0: the preceding slave number is incremented	I5 l by 1	(I8)
6-10	(9-16)	Node number	15	(I8)

Omitted master node definitions are automatically generated by incrementing the node numbers by

$$\frac{n_i - n_j}{s_i - s_i} \tag{275}$$

where s_i and s_j are the master numbers on two successive cards, and n_i and n_j are their corresponding node numbers. The master nodes must be given in the order in which they appear as one moves along the slideline.

One-dimensional slidelines are useful for modeling the pullout of reinforcing bars from concrete structures. The present formulation was developed by Pelesson (1989). The concrete body is modeled using solid elements, and the reinforcing bars are modeled using truss or beam elements. The solid elements and beam elements should not have any nodes in common. A line of nodes along the intersection of solid elements is defined as the master line, and a line of nodes (connected by beam or truss elements) is defined as the slave line.

The bond between the reinforcing bars (beams) and concrete (solids) may be either elastic perfectly plastic (*IBOND* = 0) or elastic-plastic with strain hardening (*IBOND* > 0). In either case, a maximum allowable slip d_{max} is found from

$$d_{max} = u_{max} e^{-h_{dmg}D}, (276)$$

where u_{max} and h_{dmg} are input parameters and D is a damage parameter to be discussed below. For the perfectly plastic bond model, the shear force at step n + 1 is found from

$$f_{n+1} = min(f_n - GA_s d, GA_s d_{max}),$$
 (277)

4

INPUT FORMAT

where d is the nondimensional increment in slideline displacement (i.e., bond shear strain), A_s is the shear area of the concrete-bar bond, G is the bond shear modulus, and f_n is the bond shear force at the previous time step. For the elastic-plastic bond with hardening, the updated bond shear force is given by

$$f_{n+1} = f_n + \Delta f_{max} \left(1 - e^{-d/|d|} \right), \tag{278}$$

where

$$\Delta f_{max} = GA_s d_{max} - f_n. \tag{279}$$

The evolution of damage is related to the increment in shear force $\Delta f = f_{n+1} - f_n$ by

$$D_{n+1} = D_n + |d| - \frac{|f_{n+1} - f_n|}{GA_s}.$$
 (280)

4.43 GENERALIZED RAYLEIGH DAMPING

Define stiffness proportional Rayleigh damping for *NDAMP* materials as specified in section 4.1 on page 41. Use one card for each material Rayleigh damping definition.

Columns	Quantity	Format
1-5	Material number, n	15
6-15	Stiffness proportional damping constant for this material, β_n	E10.0
16-25	Effective Young's modulus for damping computations, E_n	E10.0
26-35	Effective Poisson's ratio for damping computations, v_n	E10.0

Stiffness proportional damping constants and effective elastic material parameters should be specified for materials where stiffness proportional damping is desired. Materials for which damping parameters are not defined will have zero stiffness proportional damping.

This global damping capability is based on concepts from Rayleigh damping in linear analysis. In traditional Rayleigh damping, a global damping matrix C is constructed as a linear combination of the mass matrix M and stiffness matrix K as

$$\mathbf{C} = \alpha \mathbf{M} + \beta \mathbf{K},\tag{281}$$

where α and β are damping coefficients. This approach can be used to introduce a specified fraction of critical damping at two frequencies, since two parameters are available. The mass proportional damping coefficient α is specified on Control Card 9 in section 4.1 on page 41. The stiffness proportional constant β may be specified by material to give additional flexibility in the introduction of viscous dissipation into the model.

When used with nonlinear material models, the stiffness proportional damping is found using the isotropic elastic part of the material stiffness constructed using the elastic constants specified above. This approach prevents the amount of damping from varying with the material behavior (such as plastic yielding), and allows accurate computation of the degree of damping introduced into the model.

Mass proportional damping yields a fraction of critical damping which decreases with increasing frequency, whereas stiffness proportional damping produces a fraction of critical damping which increases with frequency.

4.44 MATERIAL INITIALIZATION FOR ROTATIONAL MOTION

Define *NMROT* materials with nodes to have initial velocities computed for rotational motion as specified on Control Card 9 in section 4.1 on page 41.



Continue on additional cards, sixteen materials per card, until all *NMROT* materials have been listed.

Materials are initialized for rotational motion by computing initial velocities from $\mathbf{v} = \boldsymbol{\omega} \times \mathbf{r}$ for all nodes of all elements using the listed materials. If static initialization is used (via either input from NIKE3D or dynamic relaxation in DYNA3D), \mathbf{r} is evaluated using the *post-initialization* geometry. This feature is useful when using DYNA3D for the transient dynamic analysis of rotating machinery, such as the impact of a foreign object into a spinning fan.

4.45 BODY FORCE LOADS BY MATERIAL

Define *NMBDF* materials to have body force loads applied as specified on Control Card 9 in section 4.1 on page 41. The first six cards define the desired body force load components and the corresponding load curves, and the remaining cards list the materials affected.





DYNA3D User Manual

BODY FORCE LOADS BY MATERIAL

Cards 7, . . .

Columns	Quantity	Format
1-5	First material to be initialized	15
6-10	Second material to be initialized	15
•	•	•
•	•	•
•	•	•
76-80	Sixteenth material to be initialized	15

Continue on additional cards, sixteen materials per card, until all *NMBDF* materials have been listed.

Specify a load curve number of zero for body force load components which are not active.

Body force loads arising from base acceleration (gravity) or from rotational motion may be specified for a selected set of materials using this option. This capability is especially useful for static initialization of centrifical stresses in rotating bodies or gravity stresses in soil prior to a transient dynamic analysis.

This option may *not* be used in combination with the global body force options Prescribed Base Acceleration and Prescribed Angular Velocity. Those two options apply body force to the entire model instead of selectively by material.

This option is intended to work on disjoint parts, and slight inaccuracies may result if body forces are applied to only part of a monolithic structure. If body force loads by material are specified for a rigid body composed of multiple materials using the rigid body merge feature, then all materials composing that rigid body should be listed to receive body force loads.

4.46 CVS (MADYMO/ATB) COUPLING DATA

Define ellipsoids for coupling as defined on Control Card 4 in section 4.1 on page 41.

Specify NMADEL cards to define the ellipsoids for coupling.

Columns	Quantity	Format
1-5	Ordinal position of the ellipsoid in the MADYMO/ATB input	15
6-10	DYNA3D material number of the corresponding rigid ellipsoid	15

When MADYMO is used as the CVS code, the ordinal position refers to the position of the ellipsoid definition in the COUPLING input section. When ATB is used as the CVS code, modifications must be made to the ATB source code to define a list of coupling ellipsoids and to implement the PVM interface as defined below.

This coupling option allows the CVS representation of the occupant to impact interior surfaces in the passenger compartment modeled in DYNA3D, and thus yields a more realistic simulation. In the current implementation of this coupling, during a time step the CVS program gives to DYNA3D the positions and orientations of each ellipsoid. DYNA3D then uses its contact algorithms to evaluate the total force and moment arising from impact between the ellipsoid and the occupant environment, and transmits this information back to the CVS code. To use this full coupling, the DYNA3D model should contain a geometric representation of the ellipsoids in the CVS model, and each ellipsoid should be defined as a separate DYNA3D rigid material.

5.0 RESTART INPUT FORMAT

The following sections describe the format of the DYNA3D restart input file. The procedure for initiating a DYNA3D restart analysis is described in section 3.4 on page 36.

The specification of a restart input file is optional when restarting an analysis which was terminated before the termination time specified in the initial input file. If no changes to the model or analysis parameters are desired, the restart may be initiated by omitting the restart input file and specifying only the restart dump file (or the running restart file) which contains the desired starting point data. DYNA3D will then resume execution and continue until the termination time specified in the original input file.

A restart input file is required when restarting a run which completed normally (to, for example, increase the termination time), or when changes to the model or analysis parameters are desired. A restart input file may be used to:

- change the termination time,
- change the time interval between writes of time history plot data,
- · change the time interval between writes of state plot data,
- delete sliding interfaces,
- delete materials,
- delete solid, beam, shell, or thick shell elements,
- reset the time step scale factor,
- · reset the interval between restart or running restart dumps,
- change translational or rotational boundary conditions on nodes or rigid bodies.

If a restart input file is used, it should be specified along with the appropriate restart dump file (or running restart file) on the DYNA3D command line used to initiate the restart.

Changes made at a restart will be reflected in all subsequent restart dumps.

After a restart, the familied files for the state plot database, time history database, and restart dump files will retain their original names (by default) and will continue with numbering according to the original sequence. This allows convenient plotting of the entire analysis results using only one family of plot database files.

DYNA3D User Manual

į

. . .

5.1 TITLE CARD

	Card 1	
<u>Columns</u>	Quantity	Format
1-72	Heading or problem title	12A6
76-80	Format option for node numbers Input "large" if node numbers exceed 99999.	A5

. . . .

- , ,

Note that in the following input description, column numbers and format descriptors in parenthesis apply to the "large" input format option.

...

ĺ

5.2 RESTART CONTROL

Card 2

<u>Columns</u>	Quantity	Format
1-10	New termination time EQ.0.0: termination time remains unchanged	E10.0
11-20	New time interval between writes of time history plot data EQ.0.0: interval remains unchanged	E10.0
21-30	New time interval between writes of state plot data EQ.0.0: interval remains unchanged	E10.0
31-35	Number of sliding interfaces to be deleted (maximum of 48)	15
36-40	Number of materials to be deleted	15
41-45	Number of solid element blocks to be deleted	15
46-50	Number of beam element blocks to be deleted	15
51-55	Number of 4-node shell element blocks to be deleted	15
56-60	Number of 8-node thick shell element blocks to be deleted	15

Note that sliding interfaces and materials to be deleted are simply listed, but elements to be deleted are specified in blocks, or groups of elements. Groups are defined by giving the first and last element numbers of a consecutive sequence of elements to be deleted.

RESTART INPUT FORMAT

-1

2

Card 3

<u>Columns</u>	Quantity	Format
1-10	New time step scale factor EQ.0.0: time step scale factor remains unchanged	E10.0
11-15	Number of changed translational boundary condition codes	15
16-20	Number of materials for stress initialization. With this option an entire DYNA3D input deck is required as part of the restart input. Materials and sliding interfaces can be added to or deleted from the calculation by using this option. EQ1: all materials will be initialized.	15
21-25	Number of time steps between restart dumps. EQ.0: no change	15
26-30	Number of time steps between running restart dumps EQ.0: no change	15

(

5.3 DELETED SLIDING INTERFACES

Skip this section if there are no sliding interfaces to be deleted at restart.

Columns		Quantity	Format	
1-5	(1-8)	Number of first sliding interface to be deleted	15	(I8)
6-10	(9-16)	Number of second sliding interface to be deleted	15	(I8)
10-15	(17-24)	Number of third sliding interface to be deleted	15	(I8)
•			•	
٠		•	•	
•	•		•	
•	•		•	

Currently, a maximum of 48 sliding interfaces may be deleted at restart. This should not be a serious limitation in most situations.
5.4 DELETED MATERIALS

Skip this section if there are no materials to be deleted at restart.

Columns Quantity		For	Format	
1-5	1-5 (1-8) Material number of first material to be deleted		I5	(I8)
6-10	(9-16)	Material number of second material to be deleted	15	(I8)
10-15	(17-24)	Material number of third material to be deleted	I5	(I8)
•	•	•	•	
•	•	•	•	
•	•			
•	•			

When a material is deleted, all elements associated with that material are also deleted. There is no limit to the number of materials which may be deleted. Deleting materials is a way of deleting a group of elements which all share the same material number, and may be more convenient than listing the elements individually if the element numbers are not contiguous.

5.5 DELETED SOLID ELEMENT BLOCKS

Skip this section if there are no solid elements to be deleted at restart.

Columns Quantity		For	Format	
1-5	(1-8)	First solid element of first block to be deleted	15	(I8)
6-10	(9-16)	Last solid element of first block to be deleted	15	(I8)
10-15	(17-24)	First solid element of second block to be deleted	15	(I8)
16-20	(25-32)	Last solid element of second block to be deleted	15	(I8)
•	•		•	
•	•	•	•	
•	•		•	
•	•	•	•	

All solid elements with numbers between the first and last numbers of each block are deleted.

5.6 DELETED BEAM ELEMENT BLOCKS

Skip this section if there are no beam elements to be deleted at restart.

Columns		Quantity	For	mat
1-5	(1-8)	First beam element of first block to be deleted	15	(I8)
6-10	(9-16)	Last beam element of first block to be deleted	15	(I8)
10-15	(17-24)	First beam element of second block to be deleted	15	(I8)
16-20	(25-32)	Last beam element of second block to be deleted	15	(I8)
•	•		•	
•	•		•	
•	•		•	

All beam elements with numbers between the first and last numbers of each block are deleted.

1

2

5.7 DELETED SHELL ELEMENT BLOCKS

Skip this section if there are no shell elements to be deleted at restart.

Columns		Quantity	For	Format	
1-5	(1-8)	First shell element of first block to be deleted	15	(I8)	
6-10	(9-16)	Last shell element of first block to be deleted	15	(18)	
10-15	(17-24)	First shell element of second block to be deleted	15	(I8)	
16-20	(25-32)	Last shell element of second block to be deleted	15	(18)	
•	•	•	•		
•	•	•	•		
	•		•		
	•	•			

All shell elements with numbers between the first and last numbers of each block are deleted.

RESTART INPUT FORMAT

and the second second

-

;

5.8 DELETED THICK SHELL ELEMENT BLOCKS

Skip this section if there are no deleted thick shell elements.

Columns Quantity		Format		
1-5	1-5 (1-8) First thick shell element of first block to be deleted		15	(I8)
6-10	(9-16)	Last thick shell element of first block to be deleted	15	(I8)
10-15	(17-24)	First thick shell element of second block to be deleted	15	(I8)
16-20	(25-32)	Last thick shell element of second block to be deleted	15	(I8)
•	•	•	•	
•	•	•	•	
•	•	•	•	
	•			

All thick shell elements with numbers between the first and last numbers of each block are deleted.

5.9 CHANGED BOUNDARY CONDITIONS

Skip this section if there are no nodal boundary condition codes to be changed at restart.

Columns		Quantity	<u> </u>	Format	
1-5	(1-8)	Node number	15	(I8)	
6-10	(9-13)	New translational boundary condition code	F5.0	(F5.0)	
11-15	(14-18)	New rotational boundary condition code	F5.0	(F5.0)	

Nodal boundary condition codes are described in section 4.5 on page 217. Changed boundary condition codes offer flexibility in modeling. For example, one might apply forces to a panel to establish a tensile preload using the dynamic relaxation initialization option, and then restart and change the boundary conditions on boundary nodes to fixed to simulate a clamped prestressed plate. Changed boundary conditions can also be used to simulate some aspects of structural failure.

Boundary condition codes may be changed for nodes on rigid or deformable bodies.

RESTART INPUT FORMAT

DYNA3D User Manual

4

- -1

6.0 MATERIAL MODEL DRIVER

6.1 OVERVIEW

This section describes a Material Model Driver which is incorporated into DYNA3D. The Material Model Driver allows plotting of the stress-strain response predicted by a material model under a given load path. This feature is particularly useful when fitting complex material models to experimental data, or when using a complex model for the first time. The interactive graphics plotting capability of the Material Model Driver allows the simulated material stress-strain behavior to be easily compared with actual material test results or theoretical predictions.

The material model driver works with any DYNA3D material model, including both solid and shell element material models, and computes the stress history corresponding to a given strain history without including the effects of dynamic response. The dynamic equations of motion are not used by the material model driver, and therefore the material behavior is simulated independent of inertial effects. These inertial effects would be present and complicate the interpretation of the results if a "one-element" problem were used to demonstrate the material model behavior. Rate dependence in the constitutive model is included, so realistic strain rates must be used with the Material Model Driver for rate-dependent materials.

6.2 INPUT DEFINITION

The Material Model Driver is invoked by setting the total number of beam, shell, and solid elements to zero in a standard DYNA3D input file (described in Chapter 4 of this manual). The number of load curves should be set to nine, and the termination time should be set to the desired length of the simulation. The complete state dump interval is interpreted as the time step to be used in the Material Model Driver run. This value should be chosen in conjunction with the strain vs. time description in the load curves to yield realistic strain increments at each step of the simulation. Plotting information is saved at every step of a Material Model Driver simulation.

The remainder of the input file should be as specified in Chapter 4 up through the material model definition. Immediately after the material definition, nine load curves describing the strain path should be defined. These nine curves describe the time history of the displacement gradient components shown in Table 1, where x, y, and z represent three orthogonal coordinate directions and u,

MATERIAL MODEL DRIVER

- . E .

v, and w represent displacement fields in those directions, respectively. Note that if a ratedependent model is used, then the "time" in the Material Model Driver corresponds to physical time and must yield realistic strain rates. If a rate-independent material model is used, then "time" is really a nonphysical quasi-time used to parameterize the strain history, and any convenient scale may be used (such as Driver time step size of 1.0 and a termination time equal to the number of steps desired).

The Material Model Driver supports material models for both solid and shell elements. The choice of solid element vs. shell element material model is indicated on the Material Control Card. For shell element material models, the input value of ε_{zz} is not used since it is not an independent variable in the plane stress constitutive formulation used in shell elements.

Load Curve Number	Component Definition
1	$\frac{\partial u}{\partial x}$
2	$\frac{\partial v}{\partial y}$
3	$\frac{\partial w}{\partial z}$
4	<u>ди</u> ду
5	$\frac{\partial v}{\partial x}$
б	$\frac{\partial u}{\partial z}$
7	$\frac{\partial w}{\partial x}$
8	$\frac{\partial v}{\partial z}$
9	$\frac{\partial w}{\partial y}$

Table 1 - Load Curve Definitions

The strain rate, or rate of deformation in a finite strain context, is found by taking the symmetric part of a finite difference time derivative of the components specified by the load curves. For example, if the above components are considered to form a tensor S, then a tensor L at time t_k (corresponding to step k) is calculated from

$$L(t_k) = \frac{S(t_k) - S(t_{k-1})}{(t_k - t_{k-1})},$$
(282)

and then the rate of deformation d is found from

$$d = \frac{1}{2} (L + L^{T})$$
(283)

and the spin is found from:

$$\boldsymbol{\omega} = \frac{1}{2} \left(\boldsymbol{L} - \boldsymbol{L}^T \right). \tag{284}$$

In a small strain context, load curves 1-6 may be used to specify strain time histories, and load curves 7-9 simply defined as zero throughout the duration of the simulation. All nine load curves must be defined in all cases, and must be defined over the entire time interval of the simulation. The small strain interpretation of the load curve quantities is given in Table 2.

Load Curve Number	Small Strain Interpretation
1	ε _{xx}
2	$\boldsymbol{arepsilon}_{yy}$
3	$\boldsymbol{\varepsilon}_{zz}$
4	E _{xy}
5	$\boldsymbol{\varepsilon}_{yz}$
6	ε_{xz}
7	E _{zx}
8	ε_{yz}
9	ε _{zy}

Table 2: Small Strain Load Curve Definitions

6.3 INTERACTIVE COMMANDS

The Material Model Driver contains an integral plotting package to allow immediate display of simulation results. After reading the input file and completing the calculations, DYNA3D gives a command prompt to the terminal. A summary of the available interactive commands is given in this section. An on-line help package is available by typing help at the prompt. Refer to the notational conventions described in section 2.1 on page 8.

- **ASCL** scale all abscissa data by f. default is f = 1.
- **OSCL** scale all ordinate data by f. default is f = 1.
- ASET *amin omax* set minimum and maximum values on abscissa to *amin* and *amax*, respectively. If *amin=amax=*0, scaling is automatic.
- **OSET** *omin omax* set minimum and maximum values on ordinate to *omin* and *omax*, respectively. If *omin=omax=*0, scaling is automatic.
- GRID tmds or graphics display will be overlayed by a grid of orthogonal lines
- NOGRID tmds or graphics displays will not be overlayed by a grid of orthogonal lines
- LOGO puts LLNL logo on all plots (default). Retyping this command removes the logo.
- TV *n* use tmds *n* or graphics device *n*, where *n* is the monitor or device number
- TV n1 n2 n3 (LLNL only) use color trnds with monitor numbers n1, n2, and n3 for red, green, and blue channels, respectively.
- PLOTS box *ann* (LLNL only) create a plotfile for box number *ann* that contains a record of the tmds display
- CLASS *lev* (LLNL only) reset classification level of hardcopy output from default unclassified to: proglev, pard, adp, confidnt, srd, or system. This command must precede the plots command if used.
- **RJET** *n* (LLNL only) send a copy of the fr80 file to rjet *n*
- TIME c plot component c versus time
- CROSS c1 c2 plot component c1 versus c2
- **PRINT** print plotted time history data in file "pamper." Only data plotted after this command is printed. File name can be changed with the "FILE" command. The "pampers" file contains 2-column ASCII data suitable for plotting with other software.

• FILE *name* - change pampers filename to *name*.

- RDLC m n r1 z1 ... rn zn redefine load curve m using n coordinate pairs (r1,z1),(r2,z2),...,(rn,zn)
- CONTINUE reanalyze material model
- ECOMP display component numbers on the tmds or graphics display. The component numbers are also shown in Table 3.
- CHGL *n* change label for component *n*. Program will prompt for new label.
- QUIT, END, T exit the material model driver

Componer	t Quantity	Compone	ent Quantity
1	x-stress	12	3rd principal deviatoric stress
2	y-stress	13	maximum shear stress
3	z-stress	14	1st principal stress
4	xy shear stress	15	2nd principal stress
5	yz shear stress	16	3rd principal stress
6	zx shear stress	17	$ln(v/v_0)$
7	effective plastic strain	18	relative volume
8	pressure	19	$v_0/v-1$
9	effective stress	20	1st history variable
10	1st principal deviatoric stress	21	2nd history variable
11	2nd principal deviatoric stress	22	3rd history variable

Table 3: Element Components for interactive graphics display.

MATERIAL MODEL DRIVER

DYNA3D User Manual

÷.

ł

7.0 EXAMPLES

This chapter presents several large-scale DYNA3D simulations of actual engineering problems, and gives two brief examples illustrating the use of the Material Model Driver described in Chapter 6. The DYNA3D Examples Manual (Lovejoy and Whirley, 1990) describes ten tutorial examples in more detail, and includes the INGRID mesh generation files and TAURUS post-processing commands used in each analysis.

7.1 NOSE CONE IMPACT

This example shows an analysis of a nose cone designed by Sandia National Laboratories in Livermore to limit the impact force transmitted to the aft section. An oblique impact event was simulated as described in (Chiesa and Callabresi, 1981).

The initial mesh, containing 6074 nodes and 4356 solid elements, is shown in Figure 38. The nose cone is made of steel, with a yield strength of 0.0048 Mbar and a tangent modulus of 0.0138 Mbar. The mass of the aft section was included by using a high density material in the top rows of elements. This is an interesting problem due to the importance of the slide surfaces and contact, which are essential in computing a realistic deformed geometry. The tied interfaces were used for convenience in the design of the mesh, as they allowed regions of different mesh refinement to be tied together. The sliding interfaces permit the collapse of the voids between ribs in the nose piece. The locations of the slide surfaces are shown in Figure 39.

Comparisons with experimental data reported in (Chiesa and Callabresi, 1981) showed excellent agreement with the calculation. A sequence of deformed shapes at 3000 μ s intervals is shown in Figure 40. At 15,000 μ s, the peak deformation is reached and the nose cone begins to rebound. The final shape obtained in the experiment was very close to the final computed shape. The computed force deflection curve plotted by TAURUS is compared to that obtained from the experiment in Figure 41. Only minor discrepancies exist, and the importance of the slide surfaces in modeling force spikes due to void closure is evident.



٠.

.

1.1

. . . .

Figure 38 Initial mesh of steel node cone.

NOSE CONE IMPACT

+



Figure 39 Location of tied and sliding interfaces.

, . . · ·

A. 199

DYNA3D User Manual





ĺ



.



7.2 OBLIQUE IMPACT OF ROD

This example presents the simulation of a long aluminum rod impacting a rigid surface. The rod is 30.5 cm long, 0.658 cm in diameter, and is made of aluminum. The rod impacts the target at an angle of 10 degrees and at a velocity of 20,170 cm/sec.

The computational model of the rod uses only solid elements. The material behavior is simulated using the Steinberg-Guinan model (Material Type 11) with the properties given in (Steinberg and Guinan, 1978). The target is modeled using the Rigid Wall option in DYNA3D to save the cost of discretizing the rigid target. The initial mesh is shown in Figure 42.

A sequence of deformed shapes is shown in Figure 43, and a close-up view of the mesh at 300 μs is shown in Figure 44. The final shape from the experiment is compared to that predicted by DYNA3D in Figure 45. The computed shapes agree well with experimental profiles, supplied by J. Zukas at the Ballistics Research Laboratory, up to approximately 600 μs . At later times the agreement is fair, but the experimental results tended to show more curvature in the rod than did the computational predictions. Some factors which may have led to these discrepancies include:

- coarse zoning,
- inaccurate material properties,
- rigid wall approximation to armor plate,
- lack of interface friction.

Even considering these approximations, the overall qualitative prediction of the rod behavior is quite good.

2

OBLIQUE IMPACT OF ROD

i



Figure 42 Calculational mesh for oblique rod impact problem.

DYNA3D User Manual

ž



· · · · · · ·

•/ •

`.

÷,

Deformed shapes of rod impacting oblique rigid wall.

ĺ

ž



Figure 44 Plan view of rod at 300 µs.

DYNA3D User Manual

2



Figure 45 Final profiles: (a) experiment and (b) computed.

7.3 FRAME MEMBER IMPACT

This example illustrates the deformation and collapse of a mild steel rail under impact loading. The geometry definitions of a mild steel frame member dynamically crash tested by Suzuki were provided in (Igarashi, 1985) and discussed in (Benson, et. al, 1986). Frame members are attached to a sled that accelerates to 30 km/hr before impact into a fixed barrier, as shown in Figure 46. Experimental output ceases at 35ms when the sled impacts a bumper that stops its motion. Force time histories are obtained by load cells at the frame-barrier interface, and movies record the deformation history.

Preliminary calculations have been performed on coarsely meshed members to study the feasibility of this type of explicit simulation using DYNA3D. Figure 47 shows a close-up view of the initial mesh with 1600 shell elements. Figure 48 shows the cross section of the 1000mm beam, which is fabricated by spot welding along the flange.

Although the spot welds were not modeled in detail, such modeling may be important in capturing the details of the force histories. Deformed shapes are plotted at 10 μ s intervals in Figure 49. The overall deformed shape is predicted quite well, but a more refined mesh must be used to resolve the highly localized buckling. Although the peak force was accurately predicted, the width of the force pulse was under-predicted.

Since the peak force occurs almost immediately and decays rapidly, the interesting part of the force time history occurs early in the event (well before 5 ms). Thus, simulations of 5 ms duration can be done very cheaply, and parameter studies are quite feasible with only modest computer requirements.

DYNA3D User Manual

9 28



71





Figure 47 Close-up view of the finite element mesh. The mesh contains 1600 shell elements.



Figure 48 Cross section of frame member. All dimensions are in millimeters.



Figure 49 Deformed shapes at 10 ms output intervals.

7.4 Material Model Driver Example 1

The first example is a simple linear elastic material (Material Type 1) subjected to uniaxial strain. The plots were interactively generated using the **CROSS** command (as described in section 6.3 on page 324), and show the Cauchy stress vs. infinitesimal strain curves generated by the Material Model Driver. For this example, a Young's modulus of 30×10^6 psi and a Poisson's ratio of 0.25 were used. From Figure 50, it is easily verified that the slope of the line is $\lambda + 2G = 3.6 \times 10^7$, where λ is the Lame parameter and G is the shear modulus. Figure 51 shows the transverse Cauchy stresses plotted against axial strain, and the slope is seen to be λ , as expected. Thus, this simple example has illustrated the agreement between elasticity theory and the DYNA3D elastic material model for the case of uniaxial strain.



DYNA3D User Manual

· . ·

•



1. . . . V

7.5 Material Model Driver Example 2

The second example considers a geologic material represented by the Soil and Crushable Foam model (Material Type 5) in DYNA3D. The parameter values and pressure-volume are taken from an actual soil model used for production calculations, and are given below.

The prescribed strain history is a shear deformation in the x - y direction, increasing linearly from zero to a strain of 5% at a time of 100. Superimposed on this shear deformation is a volumetric compaction starting at a time of 50.0, and increasing linearly to a volumetric strain of -0.03 at a time of 100. Thus, the first half of the simulation illustrates the material response at zero pressure, while the second half of the simulation shows the effect of increasing pressure on the shear behavior. Figure 52 shows the resulting plot of shear stress vs. pressure, and Figure 53 shows the pressure-volume behavior predicted by the model. Figure 54 directly shows the influence of pressure on the shear stress-shear strain curve. Clearly, plots such as these can be easily correlated with experimental data to determine the appropriate parameters for a DYNA3D material model.

Parameter	Value	
shear modulus	6.70×10^{2}	
bulk modulus	4.20×10^{6}	
a_0	9.90×10^{6}	
a_1	3.50×10^{3}	
a_2	0.30	
pressure cutoff	-5.70×10^{2}	•

.

EXAMPLES

•

DYNA3D User Manual

/

•

.

1

2

:: #

η.

Relative Volume	Pressure	
0.0	0.0	
-4.0×10^{-3}	2.9×10^3	
-1.0×10^{-2}	8.7×10^3	
-1.6×10^{-2}	1.8×10^{4}	
-2.0×10^{-2}	2.5×10^{4}	
-2.6×10^{-2}	3.5×10^{4}	
-3.0×10^{-2}	4.1×10^{4}	
-3.7×10^{-2}	5.3×10^{4}	
-4.0×10^{-2}	5.8×10^4	



2 2

1 1

345

. .

<u>____</u>

DYNA3D User Manual



5



1 1

8 2





347

12

2.8

2

ACKNOWLEDGEMENTS

Ale se el

1.1

The authors first wish to acknowledge the many suggestions and requests from the DYNA3D user community. Collaborative feedback from analysts and researchers, both inside LLNL and in industry, continues to be invaluable in contributing to this manual and to the continuing evolution of DYNA3D.

In particular, thanks to Mr. Mike Burger (LLNL), Mr. Doug Faux (LLNL), and Dr. Len Schwer (SE&CS) for their day-to-day feedback and suggestions for the improvement of DYNA3D. Thanks also to all of the LLNL engineering analysts, whose discussions, experiences, and frustrations have shaped the continuing development of DYNA3D.

The authors also acknowledge the following who contributed directly to the development of new features in this version of DYNA3D: Dr. Sanjay Govindjee (material type 36), Dr. Len Schwer and Ms. Yvonne Murray (material 37), Dr. Doug Bammann, Dr. Mike Chiesa and SNLL coworkers (materials 38, 39).

The following collaborators are also recognized for sharing feedback and suggestions on their applications of DYNA3D: Mr. Javaid Nazir, Mr. Carl Berner, and Mr. Stuart Kari (Boeing); Dr. Tom Moyer (NKF Engineering); Dr. Tom Vasko (UTRC) and Dr. Ken Brown (CASA); Dr. Mike Kleinburger (NHTSA); Mr. Nigel Barry and Mr. Alan Tanner (BAe); Dr. Tom Khalil (GM); Mr. Bill DePorter (FMC Corporation); Dr. Alan Pifko (Grumman); Dr. Brett Lewis (APTEK); Mr. Stan Challener and Mr. Ian Bush (Frazer-Nash).

The authors also wish to acknowledge their colleagues in the Methods Development Group at LLNL. Dr. Jerry Goudreau, Dr. Mark Christion, Dr. Brad Maker, and Mr. Don Dovey all provided ideas, technical insight, and continuous encouragement throughout our work.

Finally, the authors wish to again acknowledge Dr. John Hallquist for his initial work on the DYNA3D code and his contributions to the field of computational mechanics.

2

REFERENCES

- 1. Bammann, D. J., "Modeling Temperature and Strain Rate Dependent Large Deformation of Metals," *Applied Mechanics Reviews*, Vol. 43, pp. 312-319, **1990**.
- 2. Bammann, D. J., Johnson, J. C., and Chiesa, M. L., "A Strain Rate Dependent Flow Surface Model of Plasticity," Sandia National Laboratories, Report SAND90-8227, **1990**.
- Bammann, D. J., Chiesa, M. L., McDonald, A., Kawahara, W. K., Dike, J. J., and Revelli, V. D., "Prediction of Ductile Failure in Metals," in *Failure Criteria and Analysis in Dynamic Response*, H. E. Lindberg, ed., ASME AMD Vol. 107, pp. 7-12, November, 1990.
- Bammann, D. J., Chiesa, M. L., Horstemeyer, M. F., and Weingarten, L, I., "Failure in Ductile Materials Using Finite Element Methods," in Structural Crashworthiness and Failure, N. Jones and T. Wierzbicki, eds, Elsevier Applied Science Publishers, 1993.
- Bazeley, G.P., W.K. Cheung, R.M. Irons, and O.C. Zienkiewicz, "Triangular Elements in Plate Bending-Conforming and Nonconforming Solutions in Matrix Methods and Structural Mechanics," *Proc. Conf. on Matrix Methods in Structural Analysis*, Rept. AFFDL-R-66-80, Wright Patterson AFB, pp. 547-576, 1965.
- 6. Belytschko, T.B. and A.H. Marchertas, "Nonlinear Finite Element Method for Plates and its Application to the Dynamic Response of Reactor Fuel Subassemblies," *ASME J. Pressure Vessel Tech.*, pp. 251-257, **1974**.
- 7. Belytschko, T.B., L. Schwer, and M.J. Klein, "Large Displacement Transient Analysis of Space Frames," *Int. J. Num. Meth. Eng.*, 11, pp. 65-84, **1977**.
- 8. Belytschko, T.B. and C.S. Tsay, "Explicit Algorithms for Nonlinear Dynamics of Shells," AMD-Vol.48, ASME, pp. 209-231, **1981**.
- 9. Belytschko, T.B., and C. S. Tsay, "A Stabilization Procedure for the Quadrilateral Plate Element with One-Point Quadrature," *Int. J. Num. Meth. Eng.*, 19, pp. 405-419, **1983**.
- 10. Belytschko, T.B. and C.S. Tsay, "Explicit Algorithms for Nonlinear Dynamics of Shells," *Comp. Meth. Appl. Mech. Eng.*, 43, pp. 251-276, **1984**.
- 11. Belytschko, T.B., H. Stolarski, and N. Carpenter, "A C⁰ Triangular Plate Element with One-Point Quadrature," Int. J. Num. Meth. Eng., 20, pp. 787-802, **1984**.
- 12. Benson, D. J. and J.O. Hallquist, "A Simple Rigid Body Algorithm for Structural Dynamics Program," Int. J. Num. Meth. Eng., 22, 1986.
- 13. Benson, D.J. and J.O. Hallquist, "A Single Surface Contact Algorithm for the Postbuckling Analysis of Shell Structures," *Computer Methods in Applied Mechanics and Engineering*, Vol. 78, pp. 141-163, **1990**.
- 14. Benson, D. J., Hallquist, J. O., Igarashi, M., Shimomaki, K., and Mizuno, M., "The Application of DYNA3D in Large Scale Crashworthiness Calculations," Proc. of the ASME Int. Computers in Engineering Conference, Vol. 1, pp. 311-317, **1986**.
15. Burton, D.E. et al., "Physics and Numerics of the TENSOR Code," University of California, Lawrence Livermore National Laboratory, Report UCID-19428, July, **1982**.

مين بر مراجع المين الم

- 16. Chang, F.K. and K.Y. Chang, "Post-Failure Analysis of Bolted Composite Joints in Tension or Shear-Out Mode Failure," J. of Composite Materials, 21, pp. 809-833, **1987a**.
- 17. Chang, F.K. and K.Y. Chang, "A Progressive Damage Model for Laminated Composites Containing Stress Concentration," J. of Composite Materials, 21, pp. 834-855, **1987b**.
- 18. Chen, W. F., and Baladi, G. Y., *Soil Plasticity: Theory and Implementation*, Elsevier, New York, **1985**.
- 19. Chiesa, M. and M. Callabresi, "Nonlinear Analysis of a Mitigating Steel Nose Cone," *Computers and Structures*, Vol. 13, p. 295, **1981**.
- Cochran, S.G. and J. Chan, "Shock Initiation and Detonation Models in One and Two Dimensions," University of California, Lawrence Livermore National Laboratory, Rept. UCID-18024, 1979.
- 21. Desai, C. S., and Siriwardane, H. J., Constitutive Laws For Engineering Materials With Emphasis On Geologic Materials, Prentice-Hall, Chapter 10, 1984.
- 22. Dilger, W. H., Koch, R., and Kowalczyk, R., "Ductility of Plain and Confined Concrete Under Different Strain Rates," ACI Journal, January-February, **1984**.
- 23. Dobratz, B.M., "LLNL Explosives Handbook, Properties of Chemical Explosives and Explosive Simulants," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52997, **1981**.
- Dovey, D. J., and Spelce, T. E., "GRIZ: Finite Element Analysis Results Visualization for Unstructured Grids - User Manual," University of California, Lawrence Livermore National Laboratory, UCRL Report, October, 1993.
- 25. Engelmann, B. E., and Hallquist, J. O., "NIKE2D: A Nonlinear, Implicit, Two-Dimensional Finite Element Code for Solid Mechanics User Manual," University of California, Lawrence Livermore National Laboratory, UCID Report, **1991**.
- 26. Engelmann, B. E., and Whirley, R. G., "A New Explicit Shell Element Formulation for Impact Analysis," in Computational Aspects of Impact and Penetration, Elme Press, **1991**.
- 27. Engelmann, B. E., and Whirley, R. G., "A New Elastoplastic Shell Element Formulation for DYNA3D," Proceedings of the 1990 DYNA3D User Group Conference, Bournemouth, England, September, **1990**.
- 28. Engelmann, B. E., Whirley, R. G., and Goudreau, G. L., "A Simple Shell Element Formulation for Large-Scale Elastoplastic Analysis," in *Analytical and Computational Models of Shells*, ASME CED Vol. 3, December, **1989**.
- 29. Flanagan, D.P. and T. Belytschko, "A Uniform Strain Hexahedron and Quadrilateral and Orthogonal Hourglass Control," *Int. J. Numer. Meths. Eng.*, 17, pp. 679-706, **1981**

c,

- 30. Fahrenthold, E., "A Continuum Damage Model for Fracture of Brittle Solids Under Dynamic Loading," ASME Journal of Applied Mechanics, Vol. 58, pp. 904-909, December, **1991**.
- 31. Ginsberg, M. and J. Johnson, "Benchmarking the Performance of Physical Impact Simulation Software on Vector and Parallel Computers," *Applications Track of Supercomputing*, IEEE monograph, Computer Society Press, March, **1989**.
- 32. Giroux, E.D. "HEMP User's Manual," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-51079, **1971**.
- 33. Goudreau, G.L. and J.O. Hallquist, "Recent Developments in Large Scale Finite Element Lagrangian Hydrocode Technology," J. Comp. Meths. Appl. Mechs. Eng., 30, 1982.
- 34. Hallquist, J.O. "A Procedure for the Solution of Finite Deformation Contact-Impact Problems by the Finite Element Method," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52066, **1976**.
- 35. Hallquist, J.O. "A Numerical Procedure for Three-Dimensional Impact Problems," American Society of Civil Engineers, Preprint 2956, 1977.
- 36. Hallquist, J.O. "A Numerical Treatment of Sliding Interfaces and Impact," in: K.C. Park and D.K. Gartling (eds.), *Computational Techniques for Interface Problems*, AMD Vol. 30, ASME, New York, **1978**.
- 37. Hallquist, J.O. "Theoretical Manual for DYNA3D," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19401, **1982**.
- 38. Hallquist, J.O., D.J. Benson, and G.L. Goudreau, "Implementation of a Modified Hughes-Liu Shell into a Fully Vectorized Explicit Finite Element Code," *Proceedings of the International Symposium on Finite Element Methods for Nonlinear Problems*, University of Trondheim, Trondheim, Norway, **1985**.
- Hallquist, J.O., G.L. Goudreau, and D.J. Benson, "Sliding Interfaces with Contact-Impact in Large Scale Lagrangian Computations," *Comp. Meths. Appl. Mech. Eng.*, 51, pp. 107-137, 1985.
- 40. Hallquist, J.O. and D.J. Benson, "A Comparison of an Implicit and Explicit Implementation of the Hughes-Liu Shell," *Finite Element Methods for Plate and Shell Structures*, T.J.R. Hughes and E. Hinton, Editors, pp. 394-431, Pineridge Press Int., Swanea, U.K., **1986**.
- 41. Hill, R., The Mathematical Theory of Plasticity, Clarendon Press, Oxford, 1950.
- 42. Hughes, T.J.R. and W.K. Liu, "Nonlinear Finite Element Analysis of Shells: Part I. Three-Dimensional Shells," Comp. Meths. Appl. Mechs., 27, 331-362, 1981a.
- 43. Hughes, T.J.R. and W.K. Liu, "Nonlinear Finite Element Analysis of Shells: Part II. Two-Dimensional Shells," Comp. Meths. Appl. Mechs., 27, 167-181, 1981b.
- 44. Hughes, T.J.R. W.K. Liu and I. Levit, "Nonlinear Dynamics Finite Element Analysis of Shells," *Nonlinear Finite Element Analysis in Struct. Mech.*, Eds. W. Wunderlich, E. Stein, and K.J. Bathe, Springer-Verlag, Berlin, pp. 151-168, **1981**.

REFERENCES

а 3

50

- 45. Igarashi, M., Private Communication, Suzuki Motor Co., Ltd., Hamamatsu, Japan, May, 1985.
- 46. Isenberg, J., Vaughn, D. K., and Sandler, I., "Nonlinear Soil-Structure Interaction," EPRI Report MP-945, Weidlinger Associates, December, **1978**.
- 47. Johnson, G.R. and W. H. Cook, "A Constitutive Model and Data for Metals Subjected to Large Strains, High Strain Rates and High Temperatures," Presented at the Seventh International Symposium on Ballistics, the Hague, The Netherlands, April, 1983.
- 48. Keeler, S. P., "Understanding Sheet Metal Formability," Parts IV and VI, *Machinery Magazine*, May and July, **1968**.
- 49. Kenchington, G.J., "A Non-Linear Elastic material Model for DYNA3D," Proceedings of the DYNA3D Users Group Conference, September, **1988**, published by Boeing Computer Services (Europe) Limited.
- 50. Kennedy, J. M., Belytschko, T., and Lin, J. I., "Recent Developments in Explicit Finite Element Techniques and Their Application to Reactor Structures," *Nuclear Engineering and Design*, Vol. 97, **1986**.
- Key, S.W. "HONDO A Finite Element Computer Program for the Large Deformation Dynamic Response of Axisymmetric Solids," Sandia National Laboratories, Albuquerque, N.M., Rept. 74-0039, 1974.
- 52. Krieg, R.D. and S.W. Key, "Implementation of a Time Dependent Plasticity Theory into Structural Computer Programs," Vol. 20 of *Constitutive Equations in Viscoplasticity: Computational and Engineering Aspects*, American Society of Mechanical Engineers, New York, N.Y., pp. 125-137, **1976**.
- 53. Lee, E.L. and C.M. Tarver, "Phenomenological Model of Shock Initiation in Heterogenous Explosives,: PHYS. Fluids, Vol. 23, p. 2362, **1980**.
- Logan, R. W., "Incorporating Non-quadratic and Crystallographic Yield Surfaces into Finite Element Codes," University of California, Los Alamos National Laboratory, Report LA-CP-88-186, 1988.
- 55. Logan, R. W., "Implementation of a Pressure and Rate Dependent Forming Limit Diagram Model into NIKE and DYNA," University of California, Lawrence Livermore National Laboratory, Report UCRL-ID-105760, **1991**.
- 56. Lovejoy, S. C., and Whirley, R. G., "DYNA3D Example Problem Manual," University of California, Lawrence Livermore National Laboratory, Report UCRL-MA-105259, October, 1990.
- 57. Maker, B. N., Ferencz, R. M., and Hallquist, J. O., "NIKE3D: A Nonlinear, Implicit, Three-Dimensional Finite Element Code for Solid and Structural Mechanics - User's Manual, University of California, Lawrence Livermore National Laboratory, Report UCRL-MA-105268, January, 1991.
- 58. Moor, E., University of California, Lawrence Livermore National Laboratory, Private Communication, April, **1991**.

- Obergefell, L. A., Gardner, T. R., Kaleps, I., and Fleck, J. T., "Articulated Total Body Model Enhancements - Volume 2: User's Guide," Harry G. Armstrong Aerospace Medical Research Laboratory, Wright-Patterson Air Force Base, Ohio, Report AAMRL-TR-88-043, 1988.
- 60. Pelesson, D., Private communication, GA Technologies, P. O. Box 85608, San Diego, CA., 1989.
- 61. Sackett, S.J., "Geological/Concrete Model Development," Private Communication, 1987.
- 62. Sandler, I.S. and D. Rubin, "An Algorithm and a modular subroutine for the cap model," *Int. J. Numer. Analy. Meth. Geomech.*, 3, pp. 173-186, **1979**.
- 63. Schwer, L. E., and Murray, Y., "A Single Surface Three-Invariant Cap Model with Mixed Hardening," *International Journal for Numerical and Analytical Methods in Geomechanics*, submitted for publication, **1993**.
- Shapiro, A. B., "TOPAZ3D A Three Dimensional Finite Element Heat Transfer Code," University of California, Lawrence Livermore National Laboratory, Report UCID-20484, 1985.
- 65. Shapiro, A. B., and Edwards, A. L., "TOPAZ2D Heat Transfer Code Users Manual and Thermal Property Data Base," University of California, Lawrence Livermore National Laboratory, Report UCRL-ID-104558, May, **1990**.
- Simo, J. C., J.W. Ju, K.S. Pister and R.L. Taylor, "An assessment of the cap model: Consistent return algorithms and rate-dependent extension,: J. Eng. Mech., Vol. 114, No. 2, pp. 191-218, 1988.
- Simo, J. C., J.W. Ju, K.S. Pister and R.L. Taylor, "Softening Response, Completeness Condition, and Numerical Algorithms for the Cap Model," Int. J. Numer. Analy. Meth. Eng., 1990.
- 68. Speicher, S. J., and Brode, H. L., Pacific-Sierra Research Corporation, Report PSR-1630, April, 1987.
- Steinberg, D.J. and M.W. Guinan, "A High-Strain-Rate Constitutive Model for Metals," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-80465, 1978.
- Spelce, T., and Hallquist, J. O., "TAURUS: An Interactive Post-Processor for the Analysis Codes NIKE3D, DYNA3D, TOPAZ3D, and GEMINI, University of California, Lawrence Livermore National Laboratory, Report UCRL-MA-105401, May, 1991.
- Stillman, D. W. and J. O. Hallquist, "INGRID: A Three-Dimensional Mesh Generator for Modeling Nonlinear Systems," University of California, Lawerence Livermore National Laboratory, Rept. UCID-20506, 1985.
- 72. TNO Road-Vehicles Research Institute, "MADYMO User's Manual 3D, Version 5.0," July, 1992.

REFERENCES

8

73. Whirley, R. G., J.O. Hallquist and G.L. Goudreau, "An Assessment of Numerical Algorithms for Plane Stress and Shell Elastoplacity on Supercomputers," Engineering Computations, Vol. 6, No. 2, June, **1989**.

- 74. Whirley, R. G., "Elastoplastic Shell Analysis in DYNA3D," *Engineering Research and Development Report FY89*, University of California, Lawrence Livermore National Laboratory, Report UCRL-53868-89, June, **1990**.
- 75. Whirley, R. G., and Engelmann, B. E., "An Implementation of Hill's Theory of Normal Anisotropic Plasticity for Explicit Shell Analysis", University of California, Lawrence Livermore National Laboratory, UCID Report, **1991**.
- 76. Whirley, R. G., and Engelmann, B. E., "Large-Scale Nonlinear Finite Element Analysis -Explicit Aspects", *Control and System Dynamics*, C. T. Leondes, ed., **1991a**
- 77. Whirley, R. G., and Engelmann, B. E., "Recent Developments in DYNA3D for Impact Problems," *Nuclear Engineering and Design*, Vol. 138, pp. 11-22, **1992**.
- 78. Whirley, R. G., and Engelmann, B. E., "Automatic Contact in DYNA3D for Vehicle Crashworthiness Analysis," *Proceedings of the ASME Winter Annual Meeting, Symposium on Crashworthiness and Occupant Protection*, **1993**.
- 79. Whirley, R. G., and Engelmann, B. E., "An Automatic Contact Algorithm in DYNA3D for Impact Analysis," *Nuclear Engineering and Design*, accepted for publication. **1993a**.
- Wilkins, M.L., R.E. Blum, E. Cronshagen, and P. Grantham, "A Method for Computer Simulation of Problems in Solid Mechanics and Gas Dynamics in Three Dimensions and Time," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-51574, 1974.
- 81. Woodruff, J.P. "KOVEC User's Manual," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-51079, **1973**.