

# **NIKE3D, A Nonlinear, Implicit, Three- Dimensional Finite Element Code for Solid and Structural Mechanics; User's Manual Update Summary**

*M. Puso, B. N. Maker, R. M. Ferencz, J. O. Hallquist*

**U.S. Department of Energy**

Lawrence  
Livermore  
National  
Laboratory

**March 24, 2000**

## **DISCLAIMER**

This document was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor the University of California nor any of their employees, makes 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 the University of California. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or the University of California, and shall not be used for advertising or product endorsement purposes.

This work was performed under the auspices of the U. S. Department of Energy by the University of California, Lawrence Livermore National Laboratory under Contract No. W-7405-Eng-48.

# **NIKE3D**

**A Nonlinear, Implicit,  
Three-Dimensional Finite Element Code  
For Solid And Structural Mechanics**

**User's Manual Update Summary**

**Version 3.0.0 April 24, 1995**

**to**

**Version 3.3.6 March 24, 2000**

**Michael A. Puso  
Methods Development Group  
Mechanical Engineering Department**

**Bradley N. Maker, contributor**

**Robert M. Ferencz, consultant**

**John O. Hallquist, originator**

**March 24, 2000**

**NIKE3D:**  
**A Nonlinear, Implicit, Three-Dimensional Finite Element Code**  
**For Solid And Structural Mechanics - User's Manual**

**ABSTRACT**

This report provides the NIKE3D user's manual update summary for changes made from version 3.0.0 April 24, 1995 to version 3.3.6 March 24, 2000. The updates are excerpted directly from the code printed output file (hence the Courier font and formatting), are presented in chronological order and delineated by NIKE3D version number.

NIKE3D is a fully implicit three-dimensional finite element code for analyzing the finite strain static and dynamic response of inelastic solids, shells, and beams. Spatial discretization is achieved by the use of 8-node solid elements, 2-node truss and beam elements, and 4-node membrane and shell elements. Thirty constitutive models are available for representing a wide range of elastic, plastic, viscous, and thermally dependent material behavior. Contact-impact algorithms permit gaps, frictional sliding, and mesh discontinuities along material interfaces. Several nonlinear solution strategies are available, including Full-, Modified-, and Quasi-Newton methods. The resulting system of simultaneous linear equations is either solved iteratively by an element-by-element method, or directly by a direct factorization method.

NIKE3D - VERSION 3.3.6      VERSION DATE 3/23/00

\*\*\*\*\* NIKE3D HEADLINES (hotline: x28198) \*\*\*\*\*

--- 3/23/00 VERSION 3.3.6 ---

1. An unsymmetric stiffness matrix can now be assembled and solved for an analysis. For certain material models with unsymmetric tangent stiffness matrices, this can help convergence substantially. New unsymmetric stiffness contributions are currently being added e.g. follower forces. To activate, enter a 2 in the first field on card #8.

COL.	CARD	QUANTITY	FORMAT
1-5	8	Unsymmetric stiffness flag	I5
2. Some new features that improve behavior (i.e. convergence, accuracy) for problems with contact have been added:
  - i. A new contact algorithm that uses better linearizations for the contact matrix has been implemented. This algorithm is run for all surfaces by putting a 2 on field 70 on card #8. This algorithm can also do symmetric augmented Lagrange which sometimes works very well for problems with friction. This can be specified by using the keyword input "symmaug 1" for the desired slide surface.
  - ii. A new feature that updates contact segments at the beginning of each time step has been implemented. This is activated by using the keyword input "iseg 1" for the desired slide surface.
  - iii. The type 3 contact algorithm, by default, uses a procedure that only searches over connected segments when searching for the nearest master node to a slave node. This search can fail when a node jumps from one surface to another and there is no connected path to the new surface. A more robust search can be specified by using the keyword input "inode 1". This may be slightly slower but will give more reliable results in many cases.
3. Three new sense switch flags have been added:

graf.	Plots all uncovered states during equilibrium iterations for given time step. This is valuable for debugging purposes. The first time step can be plotted if sw2 is on in input file.
sw8	Outputs a large quantity of diagnostics. Useful for debugging purposes.
sw9	Plots state when element inverts and program crashes. Useful for debugging purposes.

4. Convergence can be forced after a fixed number of reforms by setting the maximum number of reforms per step time step ILIMIT to be a negative number:  

COL.	CARD	QUANTITY	FORMAT
31-35	6	Max # of reforms/step (ILIMIT)	I5

e.g. ilimit = -3 will force convergence after 3 reforms for every time step. This is good if a quick and dirty solution is necessary and for debugging purposes.
  
5. A new setting for geometric (brick) stiffness is available so that a geometric stiffness is used only after a specified number of iterations. Sometimes the geometric stiffness will cause divergence if used on the first iteration but will ultimately help convergence after the initial large amount of motion. By setting the geometric stiffness flag to -n, the geometric stiffness will only be used after n iterations, other wise it will be ignored.  

COL.	CARD	QUANTITY	FORMAT
31-35	8	Brick element geometric stiffness	I5
  
6. Brick elements can be forced to use linear kinematics by putting a 1 on card 3 in column 68.  

COL.	CARD	QUANTITY	FORMAT
66-68	3	Linear brick kinematics (LINEARB)	I5

This can be very useful for debugging purposes
  
7. An ABAQUS like user material interface has now been added. This feature will allow users to easily add temporary material models to NIKE3D.
  
8. By default NIKE3D ties nodes at current (displaced) locations for tied slide surfaces when handing off from stress initialization (dyna-nike link) restarts. By specifying keywords "no\_rel 2" for the desired tied surface, initial coordinates will be used to initialize the tied nodes and nodes will be constrained to the projection made in the reference state (undisplaced).
  
9. The NIKE3D-DYNA3D link is now available. This allows analysis to be run in nike and handed off to dyna or run in dyna and handed off to nike. It is good for stress initialization in nike using implicit analysis and then subsequent explicit analysis in dyna. It is also good for explicit dynamic analysis in dyna and then implicit springback analysis in nike. For example, a stress initialization type problem would be run as such:  
    nike3d i=inputn s=linkf  
    dyna3d i=inputd m=linkf  
Dyna3d specifies the output file using the nike3d keyword specification "nike3d linkf" in the input file. To run a springback problem for example:

```
dyna3d i=inputn
nike3d i=inputn m=linkf
```

The link can also be used for nike to nike handoffs.

The user can change boundary conditions, material models slide surfaces, load curves etc. from one phase of the analysis to the next. The only thing that cant be changed is the mesh. The nike to nike link is run:

```
nike3d i=inputn1 n=nike3d s=linkf
nike3d i=inputn2 n=nike3d m=linkf
```

The n=nike3d flag is required for the nike to nike link.

10. Contact surface smoothing is now available. This feature will interpolate a smooth surface over the facets so that nodes will slide smoothly over edges of adjacent facets. The feature is still experimental and there is still a lot of work to needed to make it production ready. The free format input "smooth 1" will apply smoothing to the given slide surface definition.

--- 5/11/99 VERSION 3.3.5 ---

1. Parallel direct solvers are available for three platforms: SGI, HP and DEC. Using the default solver, the number of processors to be used are set as follows:  
SGI setenv OMP\_NUM\_THREADS n  
DEC setenv THREAD\_COUNT n  
HP setenv OMP\_NUM\_THREADS n  
where n is the desired number of processors.
2. NIKE3D now uses the norm of the displacement increment to calculate the required convergence displacement tolerance. The old default was to use the norm of the total displacement to calculate the required displacement tolerance. It is no longer necessary to put a negative value for the displacement tolerance on card #6 to active the displacement increment tolerance. To use the old default (total displacement method), enter a negative value for the energy tolerance on card #6.
3. Dynamic contact algorithms have been added to provide energy conserving contact for implicit dynamics. Often the energy grows or dissipates when contact is made. This new algorithm can eliminate change in energy due to normal contact. Friction will dissipate energy.  
There are three different versions of this algorithm. Use the keywords "idyn n" to activate where n=1,2 or 3. For example:  
432 36 3 0.100E+02 1.000E-01  
idyn 1  
endfree

NOTES:

- idyn 1 is only available for penalty method (no augmented Lagrange) but seems to be the most robust for flexible contact at this point in time
- idyn 2 can be used with augmented Lagrange. This algorithm will accept an initial penetration. After the penetration has been detected, the algorithm applies forces to keep the gap from increasing.
- idyn 3 can be used with augmented Lagrange. This algorithm applies forces to keep any the gap from increasing so no initial gap is accepted. Any energy dissipated during contact will be recovered upon release of node from facet.

To use dynamic contact, the generalized mid-step rule must be used. We suggest you run with the following Newmark parameters on card 7:

First Newmark parameter (gamma) = 1.0  
Second Newmark parameter (beta) = 0.5  
Third mid-step parameter (alpha) = 0.5

4. Two new plasticity models have been added. Both do linear and nonlinear hardening and viscoplasticity. The major difference is that they are hyperelastic-plastic. That is, they have nonlinear elasticity based on a strain energy potential. Because they are not hypo-elastic (i.e. material 3) they can be better linearized and can potentially converge better. The material model 26 uses a principle stretch formulation while material model 27 uses the left Cauchy Green strain for the elasticity.
5. An energy conserving material model 28 has been added for implicit dynamics. This material is the same as model 2 except that it samples strain in such a way that the work done by the nodal forces is the same as the increment in strain energy. This allows the material to conserve energy and should be used with dynamic contact (above) when contact is involved. Furthermore, the Newmark parameters shown above need to be used. NOTE: material is not available for incompatible modes elements at this time.
6. Mass, energy and momentum by material are now output to the print file for brick elements when dynamics is specified.
7. Rotational inertias can be activated for beam and shell elements now. In the past no rotational inertia effects were included. This will have the most impact for deep beam and shell sections. The new feature was not made the default so that old results could be reproduced. To activate the new feature, enter 1 on card 8 in col 70-75.

COL.	CARD	QUANTITY	FORMAT
71-75	8	Rotational inertia flag	I5



8. Translational and rotational spring elements are now available. Unlike discrete elements, these elements can have zero length and be given translational AND rotational properties. The element is used by defining a beam element with the foundation boundary material 17 for the response matrix i.e.:

```

|Fx|  | K11 K12 K13 K14 K15 K16 |  |dtx|
|Fy|  |      K22 K23 K24 K25 K26 |  |dty|
|Fz| = |      K23 K24 K25 K26 |  |dtz|
|Mx|  |      K24 K25 K26 |  |drx|
|My|  |      K25 K26 |  |dry|
|Mz|  |      K26 |  |drz|

```

where dt is the translational displacement and dr is the rotational displacement at the two nodes. Inertial properties are set on cards 3 and 4 i.e.

COL.	CARD	QUANTITY	FORMAT
61-70	3	Translational mass	I5
61-70	4	Rotary inertia	I5

--- 3/30/99 VERSION 3.3.4 ---

1. A bug which caused friction forces to jump when nodes passed from one facet to the next was fixed.
2. The consistent (material) tangent has been implemented for material model 3 for optional use. Most problems converge much faster (quadratically for linear geometric analysis) with this tangent. Some thin membrane problem don't work as well since the tangent causes the elements to invert. To use, enter a one next to the Poisson's ration in the material cards:

COL.	CARD	QUANTITY	FORMAT
11-20	4	Consistent tangent flag	I5

3. A single point brick element has been added and 6 different hourglass controls have been implemented. The single point brick is element type 10 and the default hourglass control is the perturbation stiffness variety (the dyna3d default viscous is not available since this behaves poorly for statics problems).

MATERIAL			
COL.	CARD	QUANTITY	FORMAT
21-25	1	Element type (ITYPE)	I5
71-75	1	Hourglass coefficient	E10.0
76-80	1	Element type version	I5
eq.1		stiffness based hourglass	
eq.2		stiffness based hourglass (mean strain)	
eq.3		physical stabilization form	

- (only available for certain materials)
- eq.4 total displacement physical stabilization  
(for model 2 only)
- eq.5 physical stabilization form  
(mean strain, only available for certain materials)
- eq.6 total displacement physical stabilization  
(mean strain, for model 2 only)

4. The version of the type 0 brick can use either full integration or B-Bar (mean volumetric strain). Most materials use the B-Bar formulation. To determine which formulation is being used for a given brick, look at the material model output in the hsp file. To override the default, specify a 1 for b-bar or a 2 for full integration on the first material card i.e.

MATERIAL			
COL.	CARD	QUANTITY	FORMAT
76-80	1	Element type version	I5
eq.0		default for material type	
eq.1		B-Bar mean volumetric strain	
eq.2		full integration	

5. A new implementation of incompatible modes elements can be activated on a material basis. The element type is 20. Unlike the old incompatible modes, this element is nonlinear and can be used for both hypoelastic & hyperelastic materials. The new element is an enhanced element formulation. It is more accurate (particularly for eigenanalysis and provides much better bending behavior than the default element. Unlike the incompatible modes element, it experiences no volumetric locking.
6. A load curve can now be used to scale contact penalties for turning contact on and off on the fly during an analysis. With free format input "icurve n" will use load curve #n to scale penalty. By default this will not relocate nodes at initialization. By setting load curve to a negative #, nodes will be relocated at initialization.
7. Slide surface penalties can now be initialized without scaling for master facet size. Typically the area of the master facet is used along with the material bulk modulus for selecting penalties. This may cause problems when the mesh density changes abruptly and a slave node is sliding over this portion of the mesh. The slave node will see a big jump in stiffness and this can cause bad convergence problems. To ignore master facet areas in penalty calculation use the keyword "no\_size" for the desired slide surface.

8. A new parameter has been added that will modify the equilibrium iterations scheme using mthsol=1 (bfgs). By setting INREF to  $n > 0$ ,  $n$  consecutive reforms will be done at the beginning of a time step. This is handy for problems where contact is known to change at each time step and that reforms will account for the change better than the bfgs iterations. After the  $n$  reforms, the regular scheme of say 10 bfgs, 1 reform is performed. The feature is activated by placing the desired number of reforms "n" on C.C. 6 in the 13th field i.e.

COL.	QUANTITY	FORMAT
60-71	Residual force tolerance	E10.0
71-80	Line search tolerance	E10.0
81-85	Initial # of reforms (INREF)	I5

9. Auto contact has been implemented as contact type 13. Free format input has also been added for specifying slide surface input parameters. The following input parameters are used in free format form:
  - ibk\_step 0, bucket sort within each step
  - 1, bucket sort when needed
  - mat\_in (list), this material list will determine which element facets will be considered for the given contact definition.
  - mat\_ex (list), this material list will determine which element facets will be excluded for the given contact definition.
  - sym\_plane x-normal y-normal z-normal x-point y-point z-point  
this feature will alert auto contact that this plane will be treated as a symmetry plane. Auto contact will then give special treatment to these nodes. (Auto contact does not enforce the symmetry plane)
  - xmin x-value minimum x bounds
  - ymin y-value minimum y bounds
  - zmin z-value minimum z bounds
  - xmax x-value maximum x bounds
  - ymax y-value maximum y bounds
  - zmax z-value maximum z bounds

--- 10/9/98 VERSION 3.3.3 ---

1. Slide surface initialization can now be made such that nodal relocation due to initial penetration is not done. This is good for simulation of press fit parts etc. To deactivate nodal relocation use the key word input "no\_rel 1" after input card for the desired slide surface #. All free format input on these cards must be preceded by the endfree keyword.
  2. The stiffness insertion can now be modified. The stiffness insertion includes a scaled stiffness matrix contribution
-

in the global stiffness matrix for node/facet pairs that are almost in penetration. The scale goes from zero to one and is based on a characteristic length "dx" of the elements in incipient contact. The stiffness insertion is useful for slave facet pairs that are nearly always in contact. It can cause problems when slide surfaces are detaching. The modification scales the characteristic length "dx = iscale\*dx" and is implemented using the keyword "dx\_tol iscale" where  $0 < \text{iscale} < 1$ . By default, dx\_tol = 1 when BFGS iterations are used and dx\_tol = 0 when no BFGS iterations are used (i.e. full Newton).

3. A cyclic plasticity model using the Chaboche memory surfaces has been implemented as material 25. The model has nonlinear isotropic and kinematic hardening and is available for bricks, shells (H-L and YASE) and beams (H-L and C1 cont.) A detailed description is given in the new online manual.
4. A single crystal plasticity model has been added.
5. Material model 5 can now be prestrained by volumetric or uniaxial strain for different orientations i.e. spherical symmetry, globally orthotropic or cylindrical symmetry.
6. Sliding with gaps now works with the augmented Lagrange method.

--- 5/5/98 VERSION 3.3.2 ---

1. The default binary scratch file is now 20 mb.
  2. New parallel iterative solvers are available for the DEC platform by specifying 10-12 in the first field of C.C. #10. 10 use diagonal preconditioning, 11 uses incomplete Cholesky decomposition and 12 uses polynomial preconditioning. You can also specify the order of polynomial used for polynomial preconditioning.
  3. Nodal contact forces can now be output to plot file by specifying a 5 for the acceleration dump flag on C.C #5. Nodal force residuals can be output to the plot file by specifying a 4 for the acceleration dump flag. Exact nodal displacements can now be output to the plot file by specifying a 3 for the acceleration dump flag.
  4. A new brick to shell connection feature is available
-

The connection can work by entering the appropriate brick and shell nodes or can merge the bricks to shells automatically by material number specification. The connection basically forces the plane formed by the brick facets interface to remain plane like the shells and enforces the same rotation. The bricks and shells can strain in the thickness direction naturally at the interface without any ill effects. To input, enter the # of brick shell definitions to be used on card 9,

COL.	CARD	QUANTITY	FORMAT
61-65	1	# of brick shell interface definitions	I5

The interface definition can be done by entering each interface element individually by a node list or automatic generation of interface elements by material specification. The definitions are entered immediately after slide surface definitions. Two cards are needed for each definition.

COL.	CARD	QUANTITY	FORMAT
1- 5	1	# of nodes for interface element	I5
eq. 0		nodes are automatically generated by material specification given on card 2.	
eq. N		nodes are defined explicitly on card 2.	
6-10	1	X-boundary condition	I5
11-15	1	Y-boundary condition	I5
16-20	1	Z-boundary condition	I5
21-25	1	X-rotation condition	I5
26-30	1	Y-rotation condition	I5
31-35	1	Z-rotation condition	I5

Boundary condition flags:

eq. blank	boundary conditions taken from shell element
eq. 0	free boundary condition
eq.-1	forced

36-40	1	thickness boundary condition	I5
eq. 0		All nodes along interface are free to slide along interface element	
eq.-1		All nodes along interface are fixed to interface element (no sliding)	
eq.-2		Nodes maintain a constant strain in direction parallel to interface element (This b.c. is most ideal for connecting bricks to shells and is recommended for most problems)	
eq. blank		use -2 boundary condition	

1-N*5	2	If explicit node list w/ N entered on card 1 node list	N(I5)
		Starting with shell node and continuing all brick nodes. This list can wrap lines if necessary.	

If automatic node list w/ 0 entered on card 1

1- 5	2	brick material #	I5
6-10	2	shell material #	I5
11-15	2	imerge flag	I5

eq. 1        A shell node is merge to brick node on facet  
(faster search)

eq. 2        Closest point algorithm used to find where  
shell node is to meet brick element edge.

The automatic generation procedure traverses a path of brick edges starting at the shell node to generate the brick shell interface element. It first chooses the brick edges that are closes to normal to the shell facet. It then traverses the brick faces. If two edges are within 10 degrees of parallel the algorithm will continue on the path, otherwise it will stop at that point.

5.    Single pass slide surfaces can now be used for type 3, 8 and 9 slide surfaces. The single pass algorithm is more computationally efficient since it only calculates the slave nodes penetrating the master surface facets. The double pass does one pass where the gaps from the slave nodes penetrating the master surface are calculated and one pass where the gaps from the master nodes penetrating the slave surface are calculated. The single pass option is effective for many problems and is more efficient. To use the single pass option, use a negative type number i.e. -3, -8 or -9 when specifying the slide surface.
6.    A new slide surface option 9 has been added. This option allows the current shell thickness for Hughes-Liu shells with thinning to be considered in the contact gap calculation. This option is similar to type 8 contact (described below) but uses the most updated thickness. Type 8 only considers the undeformed thickness.

--- 8/23/97    VERSION 3.3.1    ---

1.    Shell thickness can now be considered in contact problems. Type 3 contact, sliding with gaps, currently doesn't consider shell thickness. The new contact type 8 is the same as 3 but consider shell thickness. To use, specify 8 in the third field of the slide surface deck.

COL.	QUANTITY	FORMAT
1- 8	Number of slave facets	I8
9-16	Number of master facets	I8
17-20	Interface type	I4

2. The number of eigenmodes output to the hsp file when doing eigenanalysis can be set by specifying the desired amount in the output printing interval field on control card 5. The default 0 value will give no eigenmodes in the hsp file.
3. A new fast eigensolver has been added to NIKE3D. The new solver uses the implicitly restarted Lanczos method. The new solver can calculate the lowest or highest eigenvalues. In addition, other new eigenvalue analysis options have been added on c.c. #7.

The eigensolver type, desired tolerance and the maximum number of iterations for determining convergence of eigenvalue analysis can now be specified.

COL.	QUANTITY	FORMAT
1- 5	Analysis type	I5
6-10	Initial condition parameter	I5
11-15	Thermal effects option	I5
16-20	Temperature profile input flag	I5
21-25	The number of eigenvalues	I5
26-35	Frequency shift	E10.0
36-45	First Newmark integration param, gamma	E10.0
	EQ.0.0 default set to .50	
46-55	Second Newmark integration param, beta	E10.0
	EQ.0.0 default set to .25	
56-65	HHT alpha parameter	E10.0
	EQ.0.0 default set to 1.0	
66-75	Eigenvalue convergence tolerance	E10.0
76-80	Eigensolver maximum iteration number	I5
81-85	Eigensolver method	I5
	EQ. 0 Implicitly restarted Lanczos (low frequencies)	
	EQ. 1 Implicitly restarted Lanczos (low frequencies)	
	EQ.-1 Implicitly restarted Lanczos (high frequencies)	
	EQ. 2 Lanczos method	
	EQ. 3 Subspace iteration	

4. A bug which caused slide surface type 1, frictionless sliding with no gaps, to not converge was fixed.
6. Users can now specify zero number of time steps on card #2 to get an initial plot state. This can be useful for model verification.
7. Gauss Lobatto Integration rules are now available for shell elements. These integration rules are one order lower than Gauss Integration and therefore require an extra integration point for the same accuracy. The advantage of these integration rules is that they include the outer fibers in shell. This can capture initial yield better and display the outer most stresses in post-processing. Gauss Lobatto integration is activated by specifying a 2

on the Quadrature rule field on Card 9 of the material cards:

COL.	CARD	QUANTITY	FORMAT
31-40	9	Quadrature rule (thru thickness)	E10.0

8. A new timing summary has been added at the bottom of the n3dhsp file.

--- 4/02/97 VERSION 3.2.9 ---

1. The Boeing Harwell type sparse direct solvers are now the default solvers in NIKE3D versions which have them available. In the EN (old) deck the entries 0,1,2 in the third field of control card #8 will now activate the sparse direct solver. In the FL deck 0,1,2,3 will do the same. The FISSLE solver is only necessary for out of core solutions when the stiffness matrix will not fit in core. The out of core FISSLE solver will be activated using a 6.

COL.	CARD	QUANTITY	FORMAT
16-20	8	direct linear solver	I5

2. The Hilbert Hughes Taylor (HHT) method is now available for dynamics with NIKE3D. This method provides high frequency damping while retaining second order convergence. It requires an input parameter alpha in addition to the usual Newmark parameters. The alpha parameter modifies the discrete equilibrium equations as such:

$$M a(n+1) = (\alpha) * (Fe(n+1) - Fi(n+1)) + (1-\alpha) * (Fe(n) - Fi(n))$$

where  $Fe(n)$  and  $Fi(n)$  are the external and internal nodal forces at time  $n$  and  $a(n)$  is the acceleration at time  $n$ . With Newmark's method alone,  $\alpha = 1$ . This is the default. Other values of  $\alpha$  can provide numerical damping when chosen with the appropriate Newmark parameters (see "The Finite Element Method" TJR Hughes 1987). The alpha parameter is entered on C.C. #7 as such:

COL.	QUANTITY	FORMAT
1- 5	Analysis type	I5
6-10	Initial condition parameter	I5
11-15	Thermal effects option	I5
16-20	Temperature profile input flag	I5
21-25	The number of eigenvalues	I5
26-35	Frequency shift	E10.0
36-45	First Newmark integration param, gamma	E10.0
	EQ.0.0 default set to .50	
46-55	Second Newmark integration param, beta	E10.0

---



EQ.0.0 default set to .25  
56-65 HHT alpha parameter E10.0  
EQ.0.0 default set to 1.0

3. A modified version of the rigid body dynamics algorithm which conserves energy and momentum was added to NIKE3D. Energy conservation requires the Newmark parameters to satisfy the constraint  $\gamma/\beta = 2$ .
4. Rigid body spherical contact is now available with NIKE3D. This feature allows rigid body spheres to be smooth such that they can roll without bumping due to the discrete faceted surfaces.

--- 6/12/96 VERSION 3.2.7 ---

1. Rigid body inertia has been added and included in dynamic analysis. NIKE3D calculates moment of inertia at the center of mass by default. User defined moments of inertia, center of rotation and initial velocities can be prescribed on the material card as follows:

COL.	CARD	QUANTITY	FORMAT
1-10	6	Center of Rotation input flag	E10.0
11-20	6	X-coordinate center of rotation	E10.0
21-30	6	Y-coordinate center of rotation	E10.0
31-40	6	Z-coordinate center of rotation	E10.0
1-10	7	Moment of inertia input flag	E10.0
11-20	7	I-xx moment of inertia	E10.0
21-30	7	I-yy moment of inertia	E10.0
31-40	7	I-zz moment of inertia	E10.0
41-50	7	I-xy moment of inertia	E10.0
51-60	7	I-xz moment of inertia	E10.0
61-70	7	I-yz moment of inertia	E10.0
1-10	8	Initial velocity input flag	E10.0
11-20	8	Vx translation velocity	E10.0
21-30	8	Vy translation velocity	E10.0
31-40	8	Vz translation velocity	E10.0
41-50	8	Wx rotational velocity	E10.0
51-60	8	Wy rotational velocity	E10.0
61-70	8	Wz rotational velocity	E10.0

--- 6/12/96 VERSION 3.2.5 ---

2. Dump files are written at the end of analysis by default when an autostepper is not used (since auto steppers write running restart files). By specifying a negative value for the number of time steps between restart file generation in the seventh field on control card 5, no

dump files will be written at any time during the analysis.

3. NIKE3D now uses dynamic memory allocation on the UNIX workstations. Users now do not need to use specific versions of NIKE3D to manage memory, since the memory is allocated according to problem size.
4. New direct sparse solvers have been added to NIKE3D. These solver only store the non-zeros in the factorized stiffness so many applications use only 25% the memory used by fistle. Furthermore the solvers are two to five as fast. The "Boeing Harwell" type solvers are activated by specifying a 2 for the EN (old) format and a 3 for the FL (new) format on control card #8 i.e.

COL. CARD	QUANTITY	FORMAT
16-20	8 direct linear solver	I5

The old ACTOL solver has been disable

--- 4/12/96 VERSION 3.2.3 ---

1. A bug which caused bus errors when strains are output using the YASE shell element was fixed.
2. A bug which caused the void growth input parameters in material model 16 to be wrong was fixed.
- 3.. The pressure controller now has a bisection routine that traps the pressure when strain rates occur above and and below the target strain rate on a give time step.
4. An additional entry to the pressure controller card has been added to start the pressure controller at a given time in the analysis. The new format is as follows  
An additional card #11 is added for SPF specific control parameters.

C.C. #11	COL.	QUANTITY	FORMAT
	1-10	Target strain rate	E10.0
	11-20	Minimum strain rate threshold	E10.0
	21-30	Maximum strain rate threshold	E10.0
	31-40	Pressure controller initiation time	E10.0
	41-50	Ramp up time	E10.0
	51-60	Maximum allowable pressure (magnitude)	E10.0
	61-65	Load curve controlling scheduled load	I5
	66-70	Controlling material number	I5
	71-80	Top percentage of elements. used for	E10.0

average strain rate calculation

Prior to the initialization time the strain rate is not controlled. Furthermore, multiple materials numbers (using material 19) can be used with the controller. The controlling material number in columns 31-40 specifies the number at which all materials with numbers at or below will be controlled.

5. Shell elements can now be prestrained (see below for details).

6. A membrane element with thinning has now been added. The membrane element is a special form of the Hughes-Liu shell such that bending moments are ignored by forcing element normals to remain perpendicular to the shell midsurface. This element is typically used for very thin membranes. Its main advantage is that it has only 12 d.o.f. as opposed to 24 d.o.f. for the shell. To specify a shell as a membrane, the H-L shell with thinning is specified by by entering a 4 or 14 in columns 36-40 on c.c. 8 as usual. By specifying only one through thickness gauss point in columns 11-20 on card 9 of the cross-section definitions for shell materials (after material properties) the element normals will be forced to be perpendicular to the midsurface. Furthermore, nodal rotations must be fixed for all membrane elements (as with bricks) by specifying a 7 for the rotational boundary condition code in the node point deck. Although a single gauss point shell element could be used previously, the membrane mode would not work correctly since element normals were not computed appropriately. It is typically necessary to prestrain membrane elements since they otherwise have no out of plane stiffness. Furthermore the shell geometric stiffness flag on c.c. 8 should be set to one. Membranes or shells can be prestrained in the in plane 11-22 direction by entering a load curve number in columns 41-50 and a magnitude in columns 51-60 on card 9 of the cross section definitions for shell materials:

COL.	QUANTITY	FORMAT
1-10	Shear factor (not applicable to membranes)	E10
11-20	Number of thickness integration points	E10
21-30	Printout option	E10
31-40	Quadrature rule	E10
41-50	Load curve # for prestrain	E10
51-60	Magnitude for prestrain	E10

Typically a prestrain of .0001 is appropriate

7. File size for plot files and out of core scratch files can now be modified on the command line. By entering l=N on the command line, family file sizes of N Mb will be generated. GRIZ and and certain versions of TAURUS will accept plot files other than 1 Mb.

--- 2/23/96 VERSION 3.2.1 ---

1. The double precision version of NIKE3D requires twice as much memory as single precision for the same problems. Consequently a bottleneck may occur during the linear solve due to memory throughput. The new VBS solver (parallel-vector solver for the CRAY Y and J-90) runs at least twice as fast on the workstations for many large problems since it is more efficient at data movement. The VBS solver is activated by using a 4 for the EN deck (old format) and a 5 for the FL deck (new format) on c.c. #8 i.e.

COL.	CARD	QUANTITY	FORMAT
------	------	----------	--------

16-20	8	direct linear solver	I5
-------	---	----------------------	----

On the other hand, the VBS solver has a different storage scheme for the stiffness matrix and tends to be much less space efficient when the maximum bandwidth is much larger than the average bandwidth.

2. By default incompatible modes elements print average stress at the element center to the print file. The gauss point stress in addition to the avg. stress may printed by entering a one in columns 71-75 on c.c. #5 for the EN deck i.e.

COL.	CARD	QUANTITY	FORMAT
------	------	----------	--------

71-75      5      incompatible modes print out flag      I5

and entering a one in columns 66-70 on c.c. #5 for the FL deck i.e.

COL.	CARD	QUANTITY	FORMAT
------	------	----------	--------

```
65-70      5  incompatible modes print out flag      I5
```

WARNING: the g.p. stresses are not as accurate as the avg.  
center stress.

3. Beam element gauss point strains can be printed out to an auxiliary file "n3data" for material model #22 when a one is entered in the second field of material card #3
- |       |   |                                    |       |
|-------|---|------------------------------------|-------|
| 1-10  | 3 | Compressive stiffness (neg. value) | E10.0 |
| 11-20 | 3 | Strain printout option             | E10.0 |
- A filename other than n3data may be selected on the command line by entering j=filename

--- 1/31/96 VERSION 3.2.0 ---

1. NIKE3D is now available in double precision. A double precision version is recognized by appending the name with dp e.g. nike3ddp.sun. This version will use twice as much memory and run slower for some types of problems but is also a necessity for some particularly difficult problems. My recommendation is to first try single precision on a problem and resort to double if necessary.
2. Discrete elements and their forces can now be included in the plot file database. Discrete element numbers will start after the last beam element.
3. A bug which gave wrong results using auto stepper and the dynamic analysis option was fixed.

4. A new concrete beam damage model has been added. The material model number is 22 and the material parameters are specified on material cards as follows:

COL.	CARD	QUANTITY	FORMAT
1-10	3	Compressive stiffness (neg. value)	E10.0
1-10	4	Strain to softening (pos. value)	E10.0
1-10	5	Damage modulus (pos. value)	E10.0
1-10	6	Strain to zero stiffness (pos. value)	E10.0
1-10	7	Plasticity ratio (0-1)	E10.0

5. A new steel beam cyclic plasticity model has been added. The material model number is 24 and the material parameters are specified on material cards as follows:

COL.	CARD	QUANTITY	FORMAT
1-10	3	Elastic modulus	E10.0
1-10	4	Asymptotic hardening modulus	E10.0
1-10	5	Initial yield stress	E10.0
1-10	6	Hardening transition parameter R	E10.0
1-10	7	Hardening transition rate parameter a0	E10.0
1-10	8	Hardening transition rate parameter a1	E10.0

6. Multiple material models can now be used for composite beam elements. Using user defined integration, an alternate material type number is specified in the fourth column as the material for that integration point

COL.	CARD	QUANTITY	FORMAT
1-10	2-NIP	s coordinate of integration	E10.0
11-20	2-NIP	t coordinate of integration	E10.0
21-30	2-NIP	Weighting Factor	E10.0
31-40	2-NIP	Material type number.	E10.0

If no material type number is given in the fourth column, the default material type number as specified on the element card is used.

--- 10/10/95 VERSION 3.1.1 ---

1. Specifying a negative residual norm tolerance will cause NIKE3D to do stiffness reforms only when iteration limits are reached or upon a new time step. This is only valid with the BFGS solution method (1 on Card #6 col. 1-5). Otherwise by default stiffness reforms occur when residual norm increases on subsequent steps. By default the residual force tolerance is 1.e10. This effectively eliminates the residual norm from consideration of convergence. Using -1.e10 would do the same, but would eliminate the reform due to increasing residual norms.
2. NIKE3D can now allow more than 24 user defined integration points for beam elements.
3. Fixed bugs in material model 16.
4. A new direct linear equation solver has been added to NIKE3D. The VBS solver is a parallel vector solver which greatly improves speed on CRAYS and other vector/parallel based platforms. It is activated for the EN deck by setting the "stiffness matrix storage option" on Control Card #8, col. 16-20 to a value of 3. It is activated using the FL deck by setting the "Direct linear equation solver option" on Control Card #8, col. 16-20 to a value of 4. This solver appears to work better than the PVS.

--- 08/15/95 VERSION 3.1.0 ---

1. The super plastic forming controller was modified to allow use with auto, ratl and cost time steppers. The usual input for the time step controller is used with the addition of the string SPF in the last column 66-70 for pressure control.  
C.C. # 3  

COL.	QUANTITY	FORMAT
66-70	Pressure Control Keyword	1X,a3
EQ."	": no pressure controller	
EQ."SPF"	use pressure controller and input from c.c. #11	

An additional card #11 is added for SPF specific control parameters.

C.C. #11	COL.	QUANTITY	FORMAT
	1-10	Target strain rate	E10.0
	11-20	Minimum strain rate threshold	E10.0
	21-30	Maximum strain rate threshold	E10.0
	31-40	Ramp up time	E10.0

41-50	Maximum allowable pressure (magnitude)	E10.0
51-55	Load curve controlling scheduled load	I5
56-60	Controlling material number	I5
61-70	Top percentage of elements. used for average strain rate calculation	E10.0

The process is explicit until the strain rate exceeds the strain rate thresholds whence an implicit loop is used to bring the strain rate within the thresholds. The ramp up time is the time where the minimum strain rate threshold is enforced thereafter.

2. A bug that was occasionally causing the following error: "fatal error in subroutine slavel" was fixed.

--- 07/28/95 VERSION 3.0.7 ---

1. Added necessary stiffness contributions to the general nonlinear discrete element routines.
2. Fixed the bugs in the single surface contact routines that occasionally caused segmentation faults.
3. The minimum and maximum allowable time steps for the auto solvers can now be modified upon restart. Control card 2 is modified as such:

COL.	QUANTITY	FORMAT
1-5	Number of Additional Steps	I5
6-15	Initial time Step	E10.0
16-20	Output printing interval	I5
21-25	Output plotting interval	I5
26-35	First Newmark integration parameter	E10.0
35-45	Second Newmark integration parameter	E10.0
45-55	Minimum allowable time step size	E10.0
55-65	Maximum allowable time step size	E10.0

Zero entries will retain original values.

4. The swplot sense switch now outputs current iteration values from i+1 equilibrium step as opposed to the i th equilibrium step as previous.

5. Fixed restart bugs in rsrund, rstrt and dump routines. iadd did not reflect actual common block length.
6. Fixed Mat. #8 Power Law Creep such that  $b=1$  can be used without division by zero.
7. NIKE3D now allows application of applied nodal displacements at locations of hex, beam and shell elements when the number of steps between stiffness reformations (Control Card 6, field 3) exceeds one.

--- 05/09/95 VERSION 3.0 ---

All previously undocumented features are now included in the NEW 1995 MANUAL!