INTRODUCTION

The development of non-linear analysis methods for a large-scale finite element system involves much more than transcribing analytic equations into computer codes. A large-scale system such as MSC/NASTRAN must be required to efficiently solve a diverse range of non-linear applications, ranging from soil mechanics to automotive crash dynamics, with problem sizes ranging up to thousands of degrees of freedom. Many types of material and geometric non-linearities must be considered along with the many types of finite elements currently required for the different applications. This paper describes the new nonlinear system which will provide a unique framework for these applications.

The capability for both geometric and material nonlinear static analysis is provided as a self-contained solution sequence, SØL 66. The capability to perform static analyses of structures with only geometric nonlinear effects, provided in SØL 64, is a subset of SØL 66. Compatibility is provided with existing MSC/NASTRAN modeling options such as load, constraint and element definitions. These developments are not related to the original MSC/NASTRAN provisions for "Differential Stiffness" or "Piecewise Linear" analysis. These are now considered obsolete and are superseded in MSC/NASTRAN by the capabilities of SØL 64 and SØL 66.

Applications for this first delivery of the general nonlinear capability include many typical structural problems. Frames and trusses with plastic yielding and local collapse may be simulated with the BEAM and RØD elements. Shell or plate structures, ranging from nuclear containment vessels to oil tankers, may be represented with QUAD4 and TRIA3 plate elements. In addition, a GAP element is included for modeling structural separation and sliding effects, such as a building resting on a flexible foundation.

This paper provides an overview of the system design and implementation of the general nonlinear analysis provided by SØLution 66. Although the forthcoming documentation provided by new User and Application manual updates will help the analyst use the system, the descriptions provided below will provide a better "picture" of the overall system. The goals of the system are summarized first, followed by details of the actual implementation. The final section contains a summary of the results and conclusions obtained from implementing and testing the system.

SYSTEM DESIGN GOALS

The basic design intention for the V61 nonlinear analysis system was to serve as the first step in developing a fully general nonlinear structural analysis system. The goals of the system design were subdivided into three categories: 1) capabilities in terms of modeling options for the user, 2) performance of the code in terms of efficiency and computer requirements, and 3) reliability in terms of solution accuracy and numerical stability. Since these goals tend to interfere with each other within present-day computer hardware and operating systems, no single optimum system will exist. Thus, the overall approach of MSC/NASTRAN is to provide many alternate paths, much the same as the existing linear solutions. The user then has the means to adjust his input to provide the best combinations for his particular requirements.

User Capabilities

The emphasis of the V61 system was to provide basic nonlinear capabilities and serve as the framework for the long term system. The design was directed towards efficiency for large solutions, compatibility with existing linear models, and simplicity of the user input data.

Some of the capability requirements were:

- o Non-linear materials defined by general purpose material and tabular input data. For plastic materials, several different yield functions and hardening rules are implemented. Nonlinear elasticity is also an option.
- o The BEAM element for both plastic hinge and coupled bending-extensional effects.

- The QUAD4 and TRIA3 elements for both membrane and out-of-plane bending nonlinear effects.
- The RØD element family for uniform inelastic extension strain
- A new GAP element for the analysis of inter-structure contact problems.
- Nonlinear large displacement capabilities similar to the DMAP4 solution in V60 are provided for the nonlinear elements. Other element types in the model are automatically treated as linear elements.

For V62 and later versions, MSC is planning a full set of element types (3D solids, shear panel, scalars, etc.) as well as material nonlinear effects (creep, fatigue, viscoelasticity, along with anisotropic and brittle materials).

Efficiency Requirements

The V61 nonlinear system was designed to take full advantage of the superelement data base system for static analysis. Some of the solution module requirements were:

- The solution procedure contains options which will simulate most popular methods (Incremental, Newton Raphson, Initial Stress, etc.). Automatic methods are also provided to perform internal selection of the optimum path.
- Linear portions of the structure may be eliminated from the solution set with the ØMIT (or ASET) method.
- Economical memory-held vector operations may be performed for moderate size problems having second-order nonlinear effects.
- Restarts may be performed starting from previous plastic solutions without repeating the calculations.

For Version 62, MSC is planning to implement transient dynamic analysis with most of the same capabilities as the static analysis. Efficiency improvements will include:

- Quasi-Newton and line search solution methods

- o More selective storage of intermediate files on the data base to reduce file size requirements
- o Condensed data files for nonlinear element property, load, and stress data which are repeatedly cycled through the system.

Compatibility with Other Solutions

The V61 nonlinear statics system design was required to be compatible with the superelement statics analysis (DMAP1 in SØLution 60). This approach allows the use of the superelement data base controls as well as the ability to reduce the costs of large order linear structural components. The nonlinear elements occur only in the "residual" superelement which may produce a small order matrix problem for an economical solution process.

The system compatibility requirements were:

- o Superelement restarts controlled by the existing case control segments for matrix and load generation, assembly, etc. (SEMG, SELG, SEMA, etc.).
- o All MSC/NASTRAN load options to be included, including thermal loads and enforced grid point displacements.
- o All types of constraints, including multi-points, rigid elements, and the Guyan reduction permitted with restrictions.

The V61 release of the general static nonlinear SØLution 66 will not be directly compatible with the geometric nonlinear and buckling solutions. However, the design of V62 and later systems will be compatible between statics, creep, dynamics, and buckling solutions. The user will be able to restart from a yielded condition in one solution to a new load in a different solution.

ADDITIONS AND CHANGES TO MSC/NASTRAN

Nonlinear structural solutions are typically obtained from a trial-and-error search procedure for a particular loading or displacement increment. The search procedure starts from a particular stress and position state and terminates when the basic equations are satisfied within a known tolerance.

In the MSC/NASTRAN approach, the algorithm was selected to maintain compatibility with the existing program organization and usage. The modular organization of MSC/NASTRAN allows much of the nonlinear code to be isolated in efficient special-purpose "modules". Small, simple problems will be processed primarily with memory-held matrices. Large order problems are accommodated by use of MSC/NASTRAN's data base storage. Much of the matrix generation, load processing, and data recovery steps are performed with existing MSC/NASTRAN algorithms.

The fundamental method used to obtain a static solution is to minimize the error vector $\{\delta\}$, given by the following equation:

$$\{\delta\} = \{P\} + \{Q\} - \{F\} \qquad (1)$$

where $\{\delta\}$ = the error vector of unbalanced forces acting at all grid point components

$\{P\}$ = the known vector of applied external loads,

$\{Q\}$ = the unknown vector of constraint forces due to single and multipoint constraints,

$\{F\}$ = the vector of grid point forces due to forces generated by element motion and stress. The terms are functions of displacement, temperature, and stress history.

The method of solving the equation is to iterate on a trial and error basis using a new displacement vector, u^i, which is obtained by solving the Newton Raphson equation:

$$\lfloor K^r \rfloor \{u^i - u^{i-1}\} = \{\delta(u^{i-1})\} \qquad (2)$$

where $[K^r]$ is an approximation to the actual tangent matrix. $[K^r]$ may be calculated at any step in the process, but results generally are obtained easier when $[K^r]$ is evaluated at a state near the solution.

The primary difference in the solution procedures of the many nonlinear structure analysis programs lies in the method and frequency of updating the matrix $[K^r]$, the selection and load increments, and the internal methods used to calculate unbalanced forces $\{\delta\}$. Details of the actual algorithm will be given in the MSC documentation. In MSC/NASTRAN, controls have been provided for:

1. Corrective load iteration loops when the matrix $[K^r]$ remains constant as in initial-strain methods. These calculations are generally very efficient and are usually performed in real memory.
2. Load increment loops whereby the applied loads $\{P\}$ grow by steps. Only a minor amount of additional calculation is involved with these steps.
3. Matrix update loops whereby the matrix $[K^r]$ is recalculated using the current best estimate of $\{u\}$. Matrix updates may be scheduled for selected load increments or load iterations. The advantage is that the convergence rate will be much faster near a solution.
4. Load and boundary subcases may be scheduled for sequential loading or unloading as well as for changes to the boundary constraints. Each new subcase uses the previous solution as a starting point and may use an independent set of iteration controls.

Matrix and Vector Operations

The solution procedure, or DMAP program, in SØLution 66 is illustrated in Figure 1. The DMAP sequence is subdivided into six logical blocks with each block performing a sequence of matrix operations. Some of these blocks represent standard MSC/NASTRAN operations such as matrix assembly, load generation, and displacement recovery steps. Of particular significance for nonlinear problems is the NLITER module block in which the actual solution and convergence operations are performed.

Figure 2 illustrates the logical flow of the NLITER module. The NLITER module itself is subdivided into smaller subroutines which remain in the computer memory during the solution calculations. The major subroutines are streamlined versions of the familiar linear static modules, namely (a) reduction of the load vectors, (b) solution of the analysis set displacements, and (c) recovery of the dependent displacements. The major difference from linear static analysis is that this process operates only on a single solution vector.

Element Processes

Nonlinear element processing occurs in three modules in the MSC/NASTRAN system. These are:

1. The EMG module generates the element structural stiffness matrices. For nonlinear elements, a special file (ESTNL), which contains the current stress and plastic strain data as well as tangent material properties and large displacement transformations, is input instead of the element summary table.
2. The NLITER module, described above, calculates the actual element nonlinear forces and maintains the ESTNL file. Each element subroutine (NXXX where XXX is the element name) may perform one or more of the following tasks:
   - Determine internal equilibrium loads due to nonlinear properties. Iterations may be performed.
   - Calculate nonlinear corrective loads due to current displacements.
   - Update ESTNL file with current nonlinear properties and stress data for the individual element.
3. The SDR2 module has been modified to process the output stress data for the nonlinear elements. The linear calculations are bypassed and the stress data is obtained directly from the ESTNL file.

The elements and their type of nonlinear behavior are given in Table 1. The characteristics of each element are described below.

The RØD (CØNRØD and TUBE) element has material nonlinear extensional properties, with linear torsion. The user may supply plastic or nonlinear elastic material properties. Since the stress-strain curve for compression need not be the same as for tension, this element can, for example, be used to model cables which cannot carry tension.

The GAP element is intended to model surfaces which may come into contact. When positive pressure exists, the gap can carry any transverse shear load which is less than the coefficient of friction times the normal load. The GAP element connects two grid points which may be initially coincident. There is no geometric nonlinear behavior, which implies that the orientation of the contact plane does not change during deflection. (The physical shape of the two contact surfaces would have to be specified and it would require a solution of a difficult analytic problem to determine the location of the actual contact point or points.) Due to the requirements of the solution algorithm, there must be a finite compression stiffness for the GAP. The value must be carefully chosen, since a very large value may lead to numerical problems. A finite extensional stiffness may be used to prevent structures supported only by GAPs from drifting. The user may also supply an initial opening of the gap. An orientation vector is required (see CGAP data card documentation in the Users Manual for details) to define the transverse axes, even if no shear is to be carried.

The BEAM element has been modified to provide plastic hinges at the ends of an otherwise elastic element. This element is intended for use in frameworks with loads at the joints, and materials with small work hardening. The user need not specify the cross-section axis about which the yielding occurs, since the implementation allows for combinations of bending moments in two directions plus an axial load. The flexibility of the plastic hinge is based upon eight idealized rods at each end, chosen to match the total area, c.g. and moments of inertia of the cross-section. The material specified on a MATS1 bulk data card may be plastic.

The QUAD4 and TRIA3 plate elements account for combined bending and membrane strains found in shell structures. The state of stress is assumed to be constant in each of twelve equal layers. Only the plane stress formulation is available. The user input is the one-dimensional stress-strain curve for tension.

Nonlinear Material Processing

The actual nonlinear material calculation for all elements is performed in a single subroutine, NMAT, which resides in the NLITER module. The interface with the element routines are simple stress, strain, and tangent modulus data. For each iteration, the element routine provides the starting state, based upon the previous converged solution, along with an incremental strain vector. The NMAT routine returns the new stress, plastic strain, and tangent modulus data.

Four commonly used yield criteria (Von Mises, Tresca, Mohr-Coulomb, and Drucker-Prager) are presently provided. Of these, the Von Mises and Tresca theories are well verified in the plasticity analysis of metals; the Von Mises theory, being the simpler of the two and the easier to apply is the one largely used in plasticity analysis of metals. Two broad categories of material nonlinearity analysis, viz, nonlinear elasticity and plasticity, have been implemented. In both cases, the stress-strain relationship is nonlinear; however the unloading follows the stress-strain curve in the case of nonlinear elasticity whereas elastic unloading takes place in plasticity analysis.

For a material that strain hardens (or work hardens), the yield surface must change for continued straining. Such change of the yield surface is governed by the hardening rules. Three frequently used options, viz, the isotropic hardening rule, the kinemetic hardening rule, and the combined isotropic-kinematic hardening rule, are presently available in SØL 66. These options can be used to model most materials irrespective of whether they show Bauschinger effect or not.

The flow-rule used in the derivation of plastic stress-strain relations is quite general. This flow rule reduces to the well-known Prandtl-Reuss relations for the case of Von Mises yield surface under isotropic hardening rule.

Virtually any type of material nonlinearity can be accommodated with this system.

CONCLUSIONS

This first phase of the nonlinear development at MSC has been a major effort on a substantial scale, involving half of the MSC staff at least part of the time during development.

The first simple execution occurred on the VAX 11/780 computer in June, 1980 for a problem with RØD elements, and as development proceeded, the test problems became larger and more general. By the time of delivery of the new code to the version 61 implementation and Q/A team (on October 3), over 1,000 executions had been run, and 32 separate test problems were delivered. Since then, the effort has been concentrated on the conversion of the code to the other computers and providing a full set of documentation for the user community.

In summary, the original design goals have been fully achieved or exceeded in nearly all areas of the new system. The design of the next phase of the system is underway at this date, with a minimum effort expected for recoding the existing subroutines or changing the user interface. We expect the code to expand horizontally, with new capabilities to be added within the present framework in a systematic process.

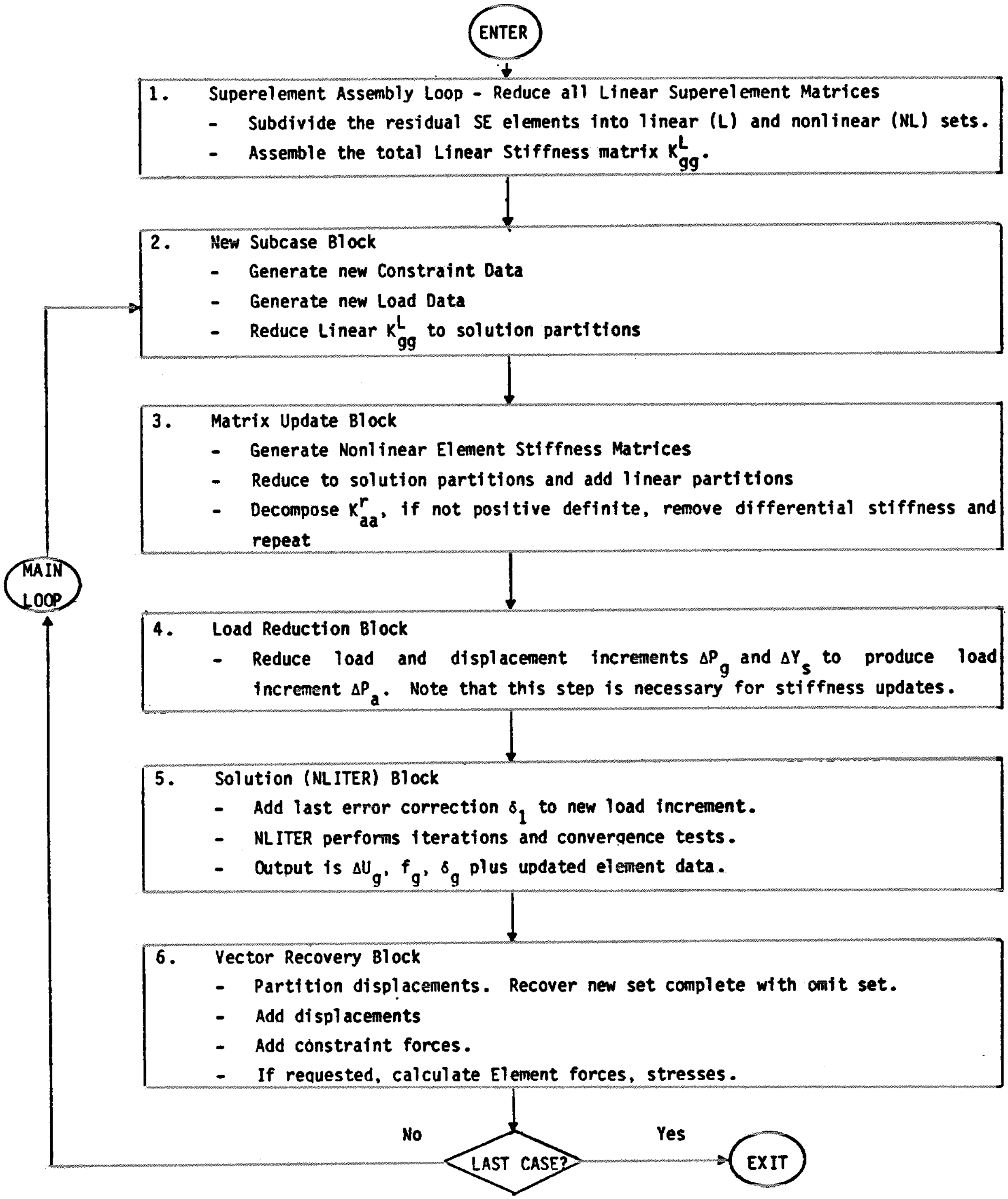


Figure 1. SØLution 66 Block Flow Chart.

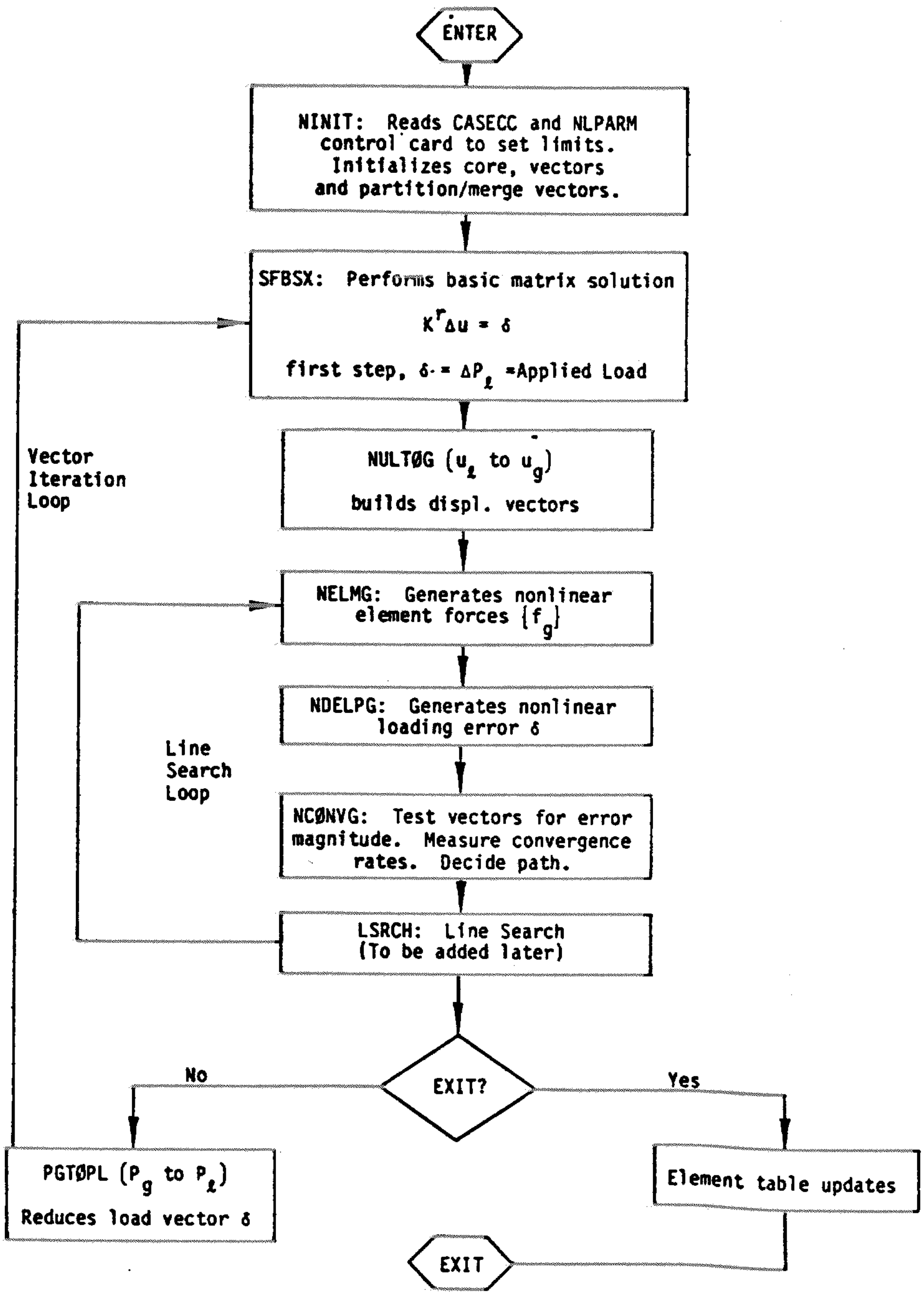
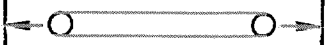
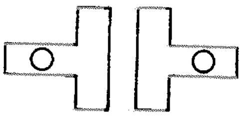
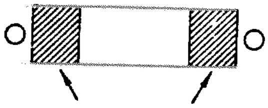
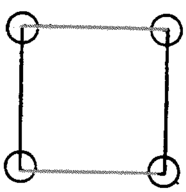
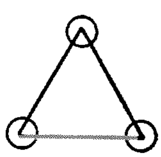


Figure 2 NLITER Major Subroutines

Table 1. Elements for SØL 66.

Name	Material Nonlinearity	Geometric Nonlinearity
RØD CØNRØD TUBE 	Extensional may be plastic or nonlinear elastic. Torsion is elastic.	Yes
GAP 	Extension has different spring for compression and tension. Shear forces less than friction coefficient times compression	No, the contact surface does not rotate.
BEAM  Plastic Hinges	Plastic hinge at each end, which couples axial motion and rotations Linear material for center section, transverse shear and torsion	Yes
QUAD 4 	Two-dimensional plasticity with twelve layers, for membrane and bending. Transverse shear and "HOURGLASS" modes are linear.	Yes
TRIA3 	Two-dimensional plasticity with twelve layers, for membrane and bending. Transverse shear is linear.	Yes