Using MSC.Nastran for Explicit FEM Simulations

Patrick Doelfs, Dr. Ingo Neubauer

MSC.Software GmbH, D-81829 München, Patrick.Doelfs@mscsoftware.com

Abstract:

Within MSC.Nastran 2005 an Explicit Solution Sequence, the so-called SOL700 is introduced. Based on LS-Dyna's proven explicit FEM-Solver, it makes accessible explicit solution techniques to the users of MSC.Nastran. This article briefly summarizes the basic functionality of the implementation.

Keywords:

MSC.Nastran, SOL 700, LS-Dyna, Explicit Solution Methods

1 MSC.Nastran Explicit Nonlinear (SOL 700)

MSC.Nastran Explicit Nonlinear (SOL 700) is an application module similar to SOL 600 (the implicit Nonlinear solution based on MSC.Marc) except that it is based on the Dytran LS-DYNA solver and offers a powerful explicit solution to analyze a variety of problems involving short duration, highly dynamic events with severe geometric and material non-linearities.
MSC.Nastran SOL 700 will allow users to work within one common modeling environment using the same Bulk Data interface. The NVH, linear and nonlinear models can be used for explicit applications such as crash, crush and drop test simulations. This will dramatically reduce the time spent to build different models for MSC.Nastran and LS-DYNA and it will prevent users from making mistakes because of unfamiliarity between the two programs. MSC.Nastran SOL 700 is being developed in phases. The first phase, available in Nastran 2004R3 supports the explicit structural applications. Later phases will include advanced fluid-structure interaction capabilities, seat belts, air bags and dummy passengers and will be available in various Nastran 2005 point releases.

2 Defining the Model

A finite element model consists of a geometric description, which is given by the elements and their nodes and a set of properties associated with the elements, describing their attributes. These properties include material definitions, cross-section definitions in the case of structural elements like beams and shells, and other parameters for contact bodies, springs, dashpots, etc. There may also be constraints that must be included in the model - RBE elements, or “multi-point constraints" or “equations" (linear or nonlinear equations involving several of the fundamental solution variables in the model), or simple “boundary conditions" that are to be imposed throughout the analysis. Nonzero initial conditions, such as initial temperatures, displacements, velocities, and even initial stresses and/or plastic strains may also be specified. The model is described and communicated to MSC.Nastran in the form of a text file, called an MSC.Nastran Input file. You can generate this file using a variety of preprocessor programs such as MSC.Patran or any text editor. It must adhere to MSC.Nastran conventions for the ordering and format of the model information.

3 User Interface

The user interface is the familiar Nastran card image interface comprised of executive control, case control and bulk data cards. In most cases, they will be identical to input formats defined for SOL 600 (and other solution sequences). The main differences will involve the use of the most important LS-DYNA materials and rigid joints (cylindrical, spherical, motor, screw, etc.) for various Nastran solution sequences. These materials and rigid joints may only be accessed using the SOL 700,ID executive control card. For example, for SOL 700,101 the linear portion of the LS-DYNA materials will be available for linear static analysis. Additionally, in some specialized cases, some familiar Nastran cards may need to small modifications and others added to specify items that are slightly different between Dytran and Nastran as well as new concepts such as moving coordinates.

4 Using MSC.Patran with SOL 700

The amount of information that needs to be conveyed in the MSC.Nastran Input file is extensive for even a modest size model. The amount of information and the complexity of most models makes it virtually impossible to generate the MSC.Nastran Input file with a text editor alone. Typically you benefit from using a preprocessor such as, MSC.Patran. MSC.Patran is another MSC Software simulation code that provides a graphical user interface and an extensive line of model building tools that you can use to construct and view your model, and generate a MSC.Nastran Input file. If you are using MSC.Patran as a preprocessor, you are required to specify an analysis code. Selecting MSC.Nastran Explicit Nonlinear (SOL 700) as the analysis code under the Analysis Preference menu, customizes MSC.Patran in five main areas:

- Material Library.
- Element Library.
- Loads and Boundary Conditions.
- MPCs.
- Analysis forms.

The analysis preference also specifies that the model information be output in the MSC.Nastran Input File format.

5 When to use Explicit Analysis

The time step for implicit solutions can be much larger than is possible for explicit solutions. This makes implicit methods more attractive for transient events that occur over a long time period and are dominated by low frequency structural dynamics.
Explicit solutions are better for short, transient events where the effects of stress waves are important. There is, of course, an area where either method is equally advantageous and may be used.
Explicit solutions have a greater advantage over implicit solutions if the time step of the implicit solution has to be small for some reason. This may be necessary for problems that include:

- Material nonlinearity. A high degree of material nonlinearity may require a small time step for accuracy.
- Large geometric nonlinearity. Contact and friction algorithms can introduce potential instabilities, and a small time step may be needed for accuracy and stability.
- Those analyses where the physics of the problem demands a small time step (e.g. stress wave effects).
- Material and geometric nonlinearity in combination with large displacements. Convergence in implicit methods becomes more difficult to achieve as the amount of nonlinearity for all types increases.

Explicit methods have increasing advantages over implicit methods as the model gets bigger. For models containing several thousand elements and including significant nonlinearity, SOL 700 may provide the cheapest solution even for problems dominated by low-frequency structural dynamics (see Figure 1).

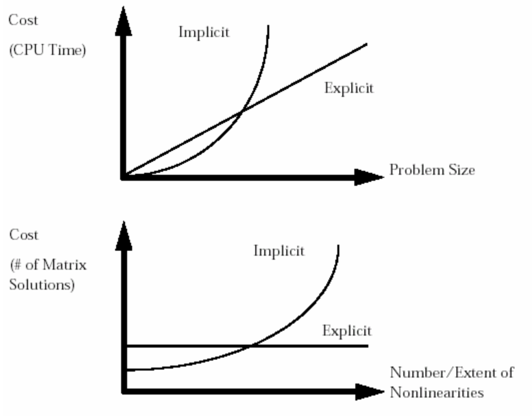

Fig.1: Comparison of Implicit and Explicit Solution Methods

The benefit of the SOL 700 Explicit solver is that the displacements, deformation, and stresses in structures can be monitored with a high degree of precision. However, extreme deformations may lead to drastically reduced time steps and extended run times.
Another case where explicit solutions may have advantages are in extremely large linear or nonlinear problems where implicit solvers may be limited to the number or parallel domains. There is no limit to the number of SOL 700 explicit parallel domains. Therefore, if a user has a large number of processors available, the explicit analysis may be faster even for linear static analyses.
The SOL 700 should normally be used for structural components that may undergo large deformation and for which the dimensions, deformed geometry, and residual stress state are of major importance. Table 1 summarizes the areas of overlap as well as the differences between the implicit and explicit analyses.

Material Nonlinearity	Implicit	Explicit
Linear isotropic elastic (metals)		
Nonlinear isotropic elastic (rubber materials)		
Linear orthotropic elastic (composites)		
Elastic-perfectly plastic (limit analysis)		
Elastoplastic, strain hardening (metals)		
Viscoelastic (polymers)		
Restricted orthotropic (metal-forming)		
Damage accumulation and failure		
Tearing and failure		
Explosive detonation		

Deformation Nonlinearity	Implicit	Explicit
Infinitesimal strains and rotations		
Infinitesimal strains and finite rotations		
Finite strains and rotations		
Large strains (100% plus) and large rotations		
(Multi)Material flow		

Contact Nonlinearity	Implicit	Explicit
Small displacement gaps		
Gaps with friction		
Large displacement gaps		
Contact surfaces		
Single surface contact		
Fluid-structure interaction		

Motion	Implicit	Explicit
Static (infinite)		
Quasi-static (noninertial)		
Vibration, fundamental modes		
Shock and vibration		
Stress wave propagation		
Shock wave propagation		
High Frequency Dynamics		
Detonation waves		

Table 1: Differences and areas of overlap in Implicit/Explicit Solution Methods

6 Typical Applications

Some of the typical structural applications which are well suited for the SOL 700 explicit analysis are:

- Automotive crash
- Crash/Crush simulations
- Transport container design and drop testing
- Ship Collision
- Projectile penetration
- Metal forming, stamping, and deep drawing
- Bird strike scenarios with “structural bird”
- Jet engine blade containment
- Golf Club simulation
- Rollover events

There are many other explicit applications that involve fluid-structure interaction (FSI). These applications are intentionally omitted here to reflect the current capabilities of the MSC.Nastran SOL 700.

7 Feature List

The complete features of MSC.Nastran Explicit Nonlinear (SOL 700) Phase I are:

1. MSC.Nastran Explicit Nonlinear (SOL 700) solves nonlinear (material, contact and/or geometric) static and transient dynamic structural finite element problems.
2. MSC.Nastran Explicit Nonlinear (SOL 700) supports the following elements/bodies:
   - 3 noded triangular shell/membrane/plane stress/(generalized) plain strain elements.
   - 4 noded quadrilateral shell/membrane/plane stress/(generalized) plain strain elements.
   - 4 noded solid tetrahedral elements.
   - 6 noded solid wedge elements.
   - 8 noded solid hexahedral elements.
   - 2 noded beam element.
   - 2 noded bar element.
   - spring elements.
   - damper elements.
   - Rigid and deformable contact bodies.
   - Point Mass element.
   - RBE elements and multi-point constraint equations are supported to tie specific nodes or degrees-of-freedom to each other. Special MPC entities are supported, (e.g. rigid links) which can be used to tie two nodes together or equate the motion of two DOFs.
3. MSC.Nastran Explicit Nonlinear (SOL 700) supports the following loads and boundary conditions:
   - Constrained nodal displacements (zero displacements at specified degrees of freedom).
   - Enforced nodal displacements (non-zero displacements at specified degrees of freedom)
   - Forces applied to nodes
   - Pressures applied to element edges or faces
   - Temperature applied to nodes.
   - Inertial body forces. Linear acceleration and rotational velocity can be applied
   - Beams can be loaded by nodal forces, body forces, prescribed displacements and loads due to thermal expansion and contraction.
   - Contact between two bodies can be defined by selecting the contacting bodies and defining the contact interaction properties.
4. MSC.Nastran Explicit Nonlinear (SOL 700) supports isotropic, orthotropic and anisotropic material properties. Nonlinear elastic-plastic materials can be defined by specifying piecewise linear stress-strain curves.
5. Physical properties can be associated with MSC.Nastran Explicit Nonlinear (SOL 700) elements such as the cross-sectional properties of the beam element, the area of the beam and rod elements, the thickness of shell, plane stress, and membrane elements, spring parameters and masses.
6. Laminated composite solid and shell elements are supported in MSC.Nastran Explicit Nonlinear (SOL 700) through the PCOMP card of the materials capability. Each layer has its own material, thickness, and orientation and may represent linear or nonlinear material behavior. Failure index calculations are also supported.

7. Analysis jobs consisting of complex loading time histories are available. All loading must be applied in a single subcase. The applicable subcase for a particular analysis may be chosen from many subcases if so desired.
8. MSC.Nastran Explicit Nonlinear (SOL 700) jobs are submitted using textbased input decks that may be generated manually with a text editor, or by a variety of pre/post processing programs such as MSC.Patran. The input file is read in and a number of text files, such as the .f06, .log, .f04, .dytr.dat, .dytr.dehsp, .dytr.d3plot files are generated.
9. Results can be requested in several output formats such as .dytr.deplot, .dytr.d3thdt, .f06, .xdb, .op2, or punch files. These files are typically read back into the pre/post processing programs for the purpose of evaluating the results with plots such as deformed shape plots, contour stress/strain plots, or X-Y history plots.
10. Nodal displacements, velocities and accelerations, element stresses, element strains, element plastic strains, nodal reaction forces and contact interface force values, element strain energy, are outputs when applicable. These may be visualized with results visualization tools such as MSC.Patran. Composite element results are returned for each layer of the composite.
11. A restart capability is available in MSC.Nastran Explicit Nonlinear (SOL 700). Any analysis can be saved from any point for a possible restart.
12. Crash, impact are supported by the Phase I release. Dynamic loading described by time histories as well as static loading is also supported.

Items not supported in the Phase I release are fluids, fluid-structure interaction, air bags, seat belts, dummy passengers and explosions (this is not a complete list so there may be others). All of these items will be supported in the Phase II release.

8 Summary

The solution sequence SOL 700 of MSC.Nastran will allow users to work within one common modeling environment using the same Bulk Data interface. The NVH, linear and nonlinear models can be used for explicit applications such as crash, crush and drop test simulations. It utilizes the LS-Dyna explicit finite element solver. In Phase I of the implementation, Crash and Impact analysis is supported. In Phase II of the implementation work, the functionality will be extended to cover items like fluid, fluid-structure interaction, air bags, seat belts and more.