# GACAP: A New Computer Code for Analysis of Symmetric and Nonsymmetric Cask Cross Sections

H. Shatoff and M. Koploy

*General Atomics, San Diego, California, United States of America*

## INTRODUCTION

There are a number of different two-dimensional dynamic analysis methods used for analyzing impacts of nuclear shipping casks. The simplest method assumes the cask is a rigid bar and uses conservation of momentum and energy. An approximation of acceleration values and energy absorption in the impact limiters at each end of the bar results from this simple analysis.

A more accurate analysis performs a time integration of the equations of motion. The simplest time integration treats the cask as a rigid bar with three degrees of freedom: horizontal and vertical translation of the center of gravity (CG), and rotation about the CG. Drop energy can be absorbed only in the impact limiters in this model.

The next level of analysis treats the cask as a beam or set of beams. This is a finite element representation. The number of degrees of freedom is then three times the number of nodes and the beam elements can absorb some drop energy. Unless the beams are assumed to undergo plastic deformation, however, the energy pickup in the beam elements is small compared to the impact limiters.

Beyond the beam representation, there are more extensive finite element representations which provide more realistic elastic plastic analyses with finer geometric detail.

The General Atomics Cask Analysis Program (GACAP) Code uses the elastic beam treatment of the cask. It has a single axis with multiple beams and is limited to small deformations in the beam elements while going through large rigid body motions of the whole cask. There is no damping allowed in the beam elements. All inelastic absorption of drop energy takes place in the impact absorbers. The calculation is an explicit step-by-step time integration. Figure 1 shows the cask model.

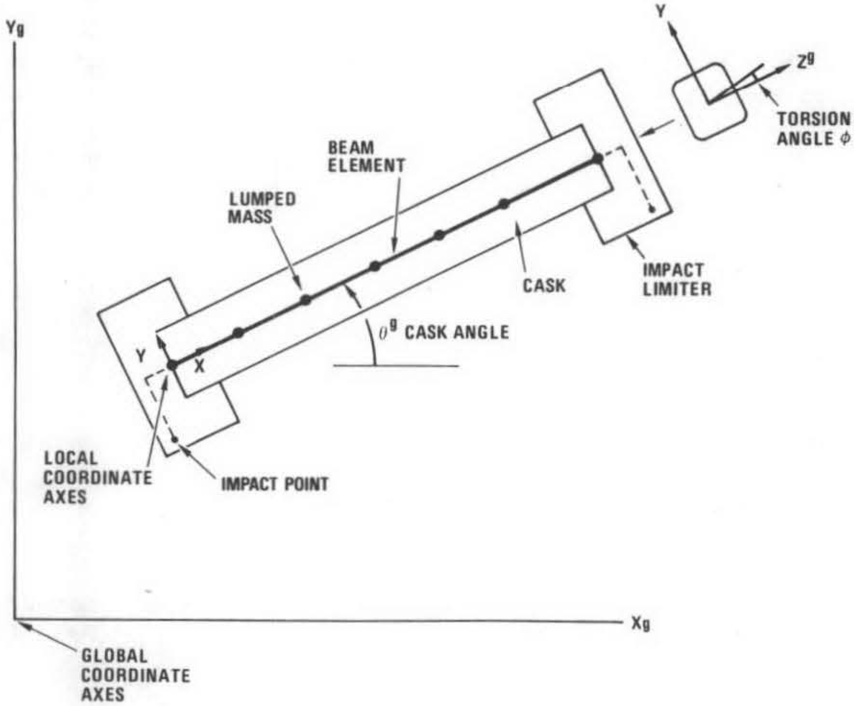


**Figure 1. Model of a Cask Using Lump Mass and Beam Representation**

The basic formulation of the code is standard. The code, however, has many features which make it useful as a design tool for a variety of cask designs. There are also aspects of the code which make it easier to analize a cask design developed at General Atomics (GA) in San Diego, California. The GA-4 and GA-9 casks have a rounded corner box cross section for which the code calculates the properties. The user can input the impact limiter force vs deflection tables without regard to order since the code reorders them. The GACAP code interpolates in between these tables as the cask angle varies during the impacts.

The code prints cask node motions and beam and limiter loads at user-specified time intervals. A summary of maximum values is provided at the end of the run. Also, code outputs the full energy state information. The energy state informs the user of the makeup of the remaining kinetic potential and elastic energy.

GACAP provides the user with the resultant beam loads from the flexible element analysis and the corresponding beam loads with assumed rigid body modes. The effect of beam flexibility on the loads is evident. The code and theory are well documented. It is verified by comparisons with the DYNA3D and SCANS codes.

Presently the GACAP code runs on the CRAY XMP-48 computer at the San Diego SuperComputer Center. It can easily be converted to any other computer.

## THEORY OF MODELING

### Formulation

The code models the cask with a series of aligned beam elements and mass nodes numbered from left to right. The basic mass and stiffness formulations of the code are standard. Equation 1 characterizes the equations of motion in the local coordinate system for a straight line of massless beam elements and n lumped node masses (each with three degrees of freedom, X, Y, $\theta$).

$$[M] \quad \{\ddot{X}\} = \{F_{lim}\} + \{F_g\} + [K] \quad \{X\} \qquad (1)$$

row, col 3*n,3*n 3*n,1 3*n,1 3*n,1 3*n,3*n 3*n,1

where [M] = the mass/mass moment of inertia matrix, which is diagonalized,
$\{\ddot{X}\}$ = the displacement and rotation acceleration vector,
$\{X\}$ = the displacement and rotation vector,
$\{F_{lim}\}$ = the force vector imposed by the limiters,
$\{F_g\}$ = the body force vector from the acceleration of gravity,
[K] = the stiffness matrix of the beam structure.

This equation is in the local beam coordinate system which has the coordinates and directions of node 1 of the cask model. There are 3*n degrees of freedom in Eq. 1. The sequence is $X_1$, $Y_1$, $\theta_1$, $X_2$, $Y_2$, $\theta_2$ ... $X_n$, $Y_n$, $\theta_n$.

The stiffness matrix [K] is assembled from the 6 x 6 stiffness matrices of the individual beam elements. This 6 x 6 symmetric element matrix can be characterized for element i as

$A_i \; B_i$ where A, B, and C are 3 x 3 submatrices and $B^T$

$B_i^T \; C_i$ is the transpose of B .

The element stiffness matrices are assembled into the total stiffness matrix [K]

$$[F] = [K]\ [X] \qquad (2)$$

Here [X] vector $X_i$, $Y_i$, $\theta_i$ are the coordinates of the nodes in the local coordinate system and the [F] vector $F_{X_i}$, $F_{Y_i}$, $M_{\theta_i}$ are the resultant loads on the nodal points also in the local system.

Equation 1 is used to compute the accelerations in the directions of the local coordinate system. The local accelerations are then rotated into the directions of the global coordinate system and integrated to obtain incremental displacements:

$$\{\ddot{X}^g_j\} = [R_{ji}] \{\ddot{X}_i\} \qquad (3)$$

$$3{*}n,1 \quad 3{*}n,3{*}n \quad 3{*}n,1$$

where $n$ = the number of nodes,
$\{\ddot{X}^g_j\}$ = the acceleration vector in the global coordinate system,
$\{\ddot{X}_i\}$ = the acceleration vector in the local coordinate system,
$[R_{ji}]$ = the rotation matrix.

$\theta$ is the angular position of node 1 with respect to the global system.

GACAP solves these equations explicitly using central difference integration of the accelerations. The local deformed state of the beam is updated using the resulting deflection and rotation.

The local displacement/rotation vector [X] is then multiplied by the stiffness matrix [K] to produce the forces and moments imposed by the beams on the nodes.

## Impact Limiter Forces

GACAP provides the user with flexibility in the treatment of the impact limiters. The impact limiter is "slaved" to a node by a rigid connection between the impact limiter contact point and the node. Their masses are lumped into the respective nodal mass. The limiters impose forces and moments on specified model nodes. The model for the impact limiter can be seen in Fig. 1. The left side limiter is shown connected to node 1, however, limiters can be connected to any node in the model. The model can also include several impact limiters connected to different nodes, each with their own load vs deflection tables. The code positions the cask vertically so that the initial impacting limiter just contacts the impact plane at the start of the run.

The impact limiter normal force will always act at the contact point, producing moments on the connected cask node. The code calculates the impact limiter force by interpolating from the user defined force vs deflection tables, using the vertical displacement overlap of the contact point with the impact plane.

The code accommodates impact limiter designs in which the behavior can change depending on the direction (torsion angle, see Fig. 1) relative to the cross-section of the cask. Each force deflection table for an impact limiter is associated with a cask angle $\theta^g$ and a torsion angle $\phi$ (see Fig. 1). Since the model is two-dimensional, the initial torsion angle is used throughout the calculation. A linear interpolation of the tables is made for both the cask angle and the torsion angle. Cask angle specification for the tables must be between 0 and 90 deg.

The user may include a horizontal friction force in the analysis. This force is dependent on the magnitude of the impact limiter normal force. The code treats the friction force at each limiter as a viscous damper which opposes the horizontal velocity of the limiter on the impact plane.

### Section Properties

GACAP provides the user with flexibility to input the desired section properties. The program computes stiffness properties for two of the four cross section inputs provided. These two cross sections are the circular cylinder and the rounded-corner square box. The moments of inertia for either of these cross sections are independent of the torsion angle. The user may also input the properties of the beam sections as shown in Figure 2.

| CROSS-SECTION | TYPE OF INPUT |
|---|---|
| CYLINDERS | E, G |
| ROUNDED-CORNER BOX SECTIONS | E, G |
| ARBITRARY SHAPE | $EA/_L$, $EI/_L3$, $EI/_L2$, $EI/_L$, f |
| | E, G, A, I, f |

WHERE

E = ELASTIC MODULUS
A = AREA OF THE CROSS-SECTION
I = MOMENT OF INERTIA
L = LENGTH OF BEAM
G = SHEAR MODULUS
f = SHEAR FORM FACTOR

**Figure 2. Different Types of Section Properties Input Accepted by GACAP**

The code allows for mixed multiple beam input, between adjacent nodes. The parallel beam stiffnesses are simply added together to create the model. The user may define the properties of the beams between each adjacent nodes independently. Therefore, the code can be used to analyze cask designs with variable cross-sections.

The codes allows the user to either input directly the mass moment of inertia of each node or to have the code calculate this parameter. The user may also provide a value for the shear form factor in order to calculate the appropriate shear deformation of the beam.

### Flexible and Rigid Body Beam Loads

Along with the beam loads from the flexible element analysis, the code also provides beam loads based on the cask acting as a single rigid body with only three degrees of freedom, see Fig. 3. In the flexible element model, each node has its own accelerations computed from the 3*n degree of freedom model. A comparison of the rigid body results with the flexible element results provides information on the so called "Dynamic Amplification Factor".

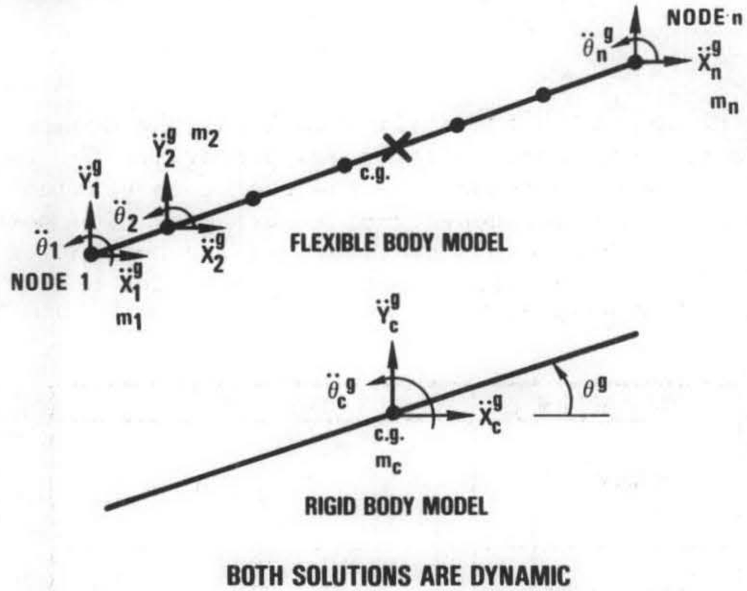


**Figure 3. GACAP Provide Both the Flexible Body and Rigid Body Dynamic Solutions**

To calculate the beam loads from the rigid body accelerations, each node is given accelerations in the local coordinate system dependent on its position from the CG. The rigid body nodal accelerations are used to compute body forces and moments on the nodes. The body forces along with the forces imposed by the impact limiters and gravity comprise the rigid body force-moment loading on the cask.

The printout for the rigid body beam loads conforms with that of the dynamic beam loads.

## VERIFICATION

### Verification with DYNA3D

GACAP was initially verified using DYNA3D. DYNA3D can perform some of the same cask calculations as GACAP. However, since DYNA3D is a general code developed for very complex geometries and varied problems, it is more difficult to use than GACAP and has significant limitations for cask analysis.

Figure 4 shows an example run in both GACAP and the DYNA3D code. The cask impacts from a 30-ft height at 30 deg from the horizontal plane. There are two impact limiters. GACAP and DYNA3D use identical geometry, initial conditions, nodal masses, and mass moments of inertia. The DYNA3D model simulates the impact limiters using the discrete spring input.

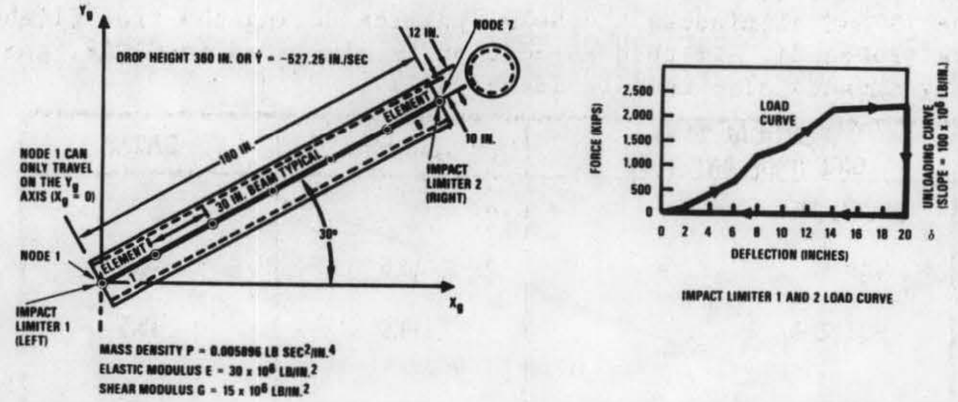


**Figure 4. GACAP Model Used For Verification With DYNA3D**

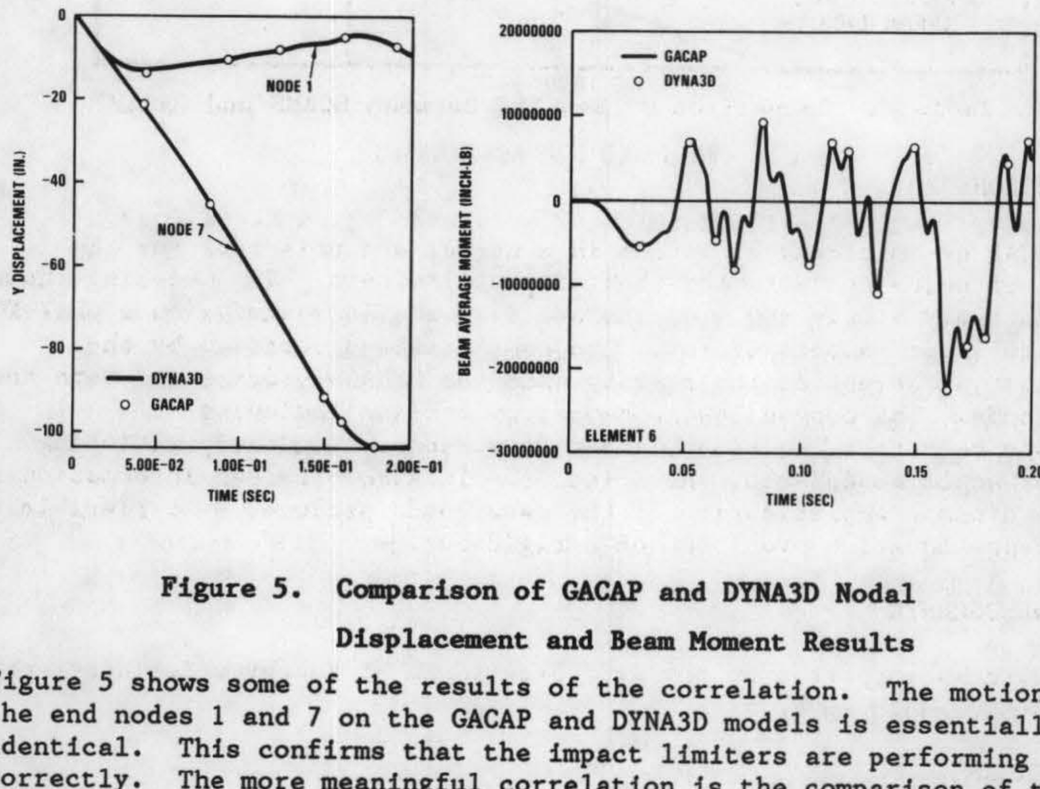


**Figure 5. Comparison of GACAP and DYNA3D Nodal Displacement and Beam Moment Results**

Figure 5 shows some of the results of the correlation. The motion of the end nodes 1 and 7 on the GACAP and DYNA3D models is essentially identical. This confirms that the impact limiters are performing correctly. The more meaningful correlation is the comparison of the time variation of beam loads where the high frequency beam structure modes appear. These have been plotted for beam element 6 in Fig. 5. The correlation is excellent and verifies the GACAP code.

**Comparison with the SCANS Code**

GACAP has also been compared with the Lawrence Livermore Laboratory SCANS code. SCANS is based on the same formulation as GACAP. Table 1 shows that both codes give equivalent results during primary impact or when the cask does not rebound.

The primary difference between GACAP and SCANS results arises from the fact that SCANS eliminates the beam dynamics during the free flight of the cask (rebound). If this assumption is simulated in GACAP, the secondary impacts also compare identically.

| IDENTICAL 30 FT SIDE DROP ANALYSES | SCANS | GACAP |
|---|---|---|
| CRUSH (IN.) | | |
| TOP | 10.5 | 10.5 |
| BOTTOM | 10.5 | 10.5 |
| MAX DYNAMIC MOMENT (IN.-KIP) (CENTER NODE) | 71868 | 71930 |
| MAX DYNAMIC SHEAR (KIPS) | 1134 | 1132 |

**Table 1. Comparison of Results Between SCANS and GACAP**

## CONCLUSIONS

The GACAP dynamic cask drop code is a useful analysis tool for the design of shipping casks and their impact limiters. The modeling input is relatively simple and the runs are fast and inexpensive on a CRAY-XMP to allow large numbers of runs. The code has been verified by the excellent agreement of the results with the DYNA3D program and with the SCANS code. The code allows a number of options including multiple beam cross sections, shear form factor, limiter positioning, limiter angle dependence, and friction. It also provides information as to the dynamic amplification of the cask loads produced by a flexible beam representation over that of a rigid bar.

## ACKNOWLEDGMENTS

This work is supported by the U.S. Department of Energy under Contract No. DE-AC07-88ID12698.

## REFERENCES

Hallquist, J. O., and D. J. Benson, "DYNA3D Users Manual," UCID-19592, Rev. 3, Lawrence Livermore Laboratory, July 1987.

"SCANS A Microcomputer Based Analysis System for Shipping Cask Design Review," Theory Manuals, Vols. 2 and 3, January 1989.