Containment Vessel Dynamic Analysis
Part 1: Base Configuration

MASTER

DISTRIBUTION OF THIS DOCUMENT IS UNLIMITED

Los Alamos
NATIONAL LABORATORY

*Los Alamos National Laboratory is operated by the University of California
for the United States Department of Energy under contract W-7405-ENG-36.*

Edited by Mable Amador, Group CIC-1

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither The Regents of the University of California, the United States Government nor any agency thereof, 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 Regents of the University of California, 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 Regents of the University of California, the United States Government, or any agency thereof. Los Alamos National Laboratory strongly supports academic freedom and a researcher's right to publish; as an institution, however, the Laboratory does not endorse the viewpoint of a publication or guarantee its technical correctness.

Containment Vessel Dynamic Analysis
Part 1: Base Configuration

R. Robert Stevens
John Benner

DISCLAIMER

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

Contents

I.	INTRODUCTION	1
II.	FINITE ELEMENT MODEL AND STRUCTURAL ANALYSIS CODE	5
III.	PRESSURE-TIME HISTORY DATA CALCULATED BY MESA2D	6
IV.	RESULTS	10
	A. Stresses Produced by the Blast Loading	10
	B. Dynamic Loading of the 6-ft-Diameter Closure	11
	C. Dynamic Loading of the 14-ft-Diameter Closure	12
	D. Support Stand Force	12
	E. Other Vessel Responses	16
	REFERENCES	16

Figures

1.	The finite element model of the main body of the DX-containment vessel	2
2.	The finite element models of the containment vessel's main heads	3
3.	A two-dimensional drawing of the containment vessel and extension modules	4
4.	The three-dimensional approximations of the vessel geometry used in MESA2D	7
5.	Locations of tracer points used in the MESA2D analysis.....	8
6.	A composite plot of all 46 pressure-time histories applied to the model.....	8
7.	The contact forces acting on the extension module closure at time $t = 7.0$ ms	11
8.	The longitudinal loading acting on the extension module closure	12
9.	The total force acting on the extension module closure	13
10.	The longitudinal loading acting on the main vessel closure	13
11.	The total force acting on the main vessel closure	14
12.	The force produced at a vessel support foot	14
13.	The vessel support foot transient load factor	15

Tables

1.	Material Properties for HSLA-100 Steel Plate	5
2.	Peak Stresses Caused by Blast Loading.....	10

Containment Vessel Dynamic Analysis

Part 1: Base Configuration

by
R. Robert Stevens and John Benner

Abstract

This report describes the approach and methods used in the analysis of a containment vessel that is being designed for use at the Dual-Axis Radiographic Hydrodynamic Test Facility (DARHT). The vessel will be subjected to transient pressure loads during explosive tests conducted at the facility and is required to contain all of the products of these tests. Computational simulations of the structural response of the vessel have been made using a combination of two numerical techniques: the transient pressure histories acting on the inner surfaces of the vessel were computed using an Eulerian hydrodynamics code, and the vessel's structural response to these pressures was analyzed using an explicit finite element code. The methods and approximations involved in combining these two numerical analysis tools will be discussed, and the stresses produced in the vessel and important measures of the dynamic response will be presented.

I. INTRODUCTION

The Dynamic Experimentation (DX) Division containment vessel that is being designed for use in the Dual-Axis Radiographic Hydrodynamic Test Facility (DARHT) will be subjected to large transient pressure loads caused by detonation of up to 50 kg of high explosive (HE). In order to estimate the structural response of the vessel to these loadings, a series of finite element models were created that represent the vessel in various configurations. The configuration studied in this report is the "base configuration": the vessel's four 6-ft-diameter ports are capped by extension heads, and no extension modules or re-entrant noses are used.

The finite element model of the four-extension-module configuration is shown in Figs. 1 and 2. The model was based on geometry and design drawings (see Fig. 3) that were current at the time the mesh was created (August 1996). Because this configuration possesses a 1/8 (45°) cyclic symmetry, the finite element model included only 1/8 of the vessel and extension modules.

The base configuration was analyzed in order to make estimates of the severity of the following responses during an HE detonation:

Fig. 1. The finite element model of the main body of the DX-containment vessel.

- The stresses and strains in the vessel walls, heads, and ports,
- The forces acting on the 6-ft-diameter closure mechanism,
- The forces acting on the 14-ft-diameter closure mechanism, and
- The forces produced at the support feet of the vessel.

The goal was to provide insight into the dynamic structural behavior of the vessel to the designers.

The mass of the blast armor (shrapnel-mitigation plating, assumed to be 3/4-in. steel plate that lines all inner surfaces of the vessel) was included in the finite element model. Although the blast armor does not carry structural loads, its added mass does have a significant effect on the response of the vessel.

The material from which the DARHT vessel is to be fabricated is HSLA-100 steel, manufactured as rolled plate. The vessel construction is primarily weldment. The material properties used for this material are described in Table I.

Fig. 2. The finite element model of the containment vessel's main head (14-ft diameter) and extension head (6-ft diameter).

The material response was assumed to be linear elastic. This assumption is not valid in certain regions of the vessel where plastic yielding is predicted to occur. The linear material model was used anyway, however, because the vessel designers intend to find a design that will give no yielding. All stresses greater than the yield stress of 100 ksi are therefore taken to be fictitious, but serve to show (1) where material yielding is calculated to initiate, and (2) by what margin the loading exceeds the maximum allowable loading so that no material yielding would occur.

Fig. 3. A two-dimensional drawing of the containment vessel and extension modules from which the finite element mesh was created. The geometry was based on an IGES file provided by Christopher Romero, DX-5, in August 1996.

Table I: Material Properties of HSLA-100 Steel Plate*

Density	0.282 lb/in. ³
Elastic modulus	28.6 10 ⁶ psi
Poisson's ratio	0.29
Yield stress	100 ksi
* These properties are taken from Ernest J. Czyryca, Richard E. Link, Richard J. Wong, Denise A. Aylor, Thomas W. Montemarano, and John P. Gudas, "Development and Certification of HSLA-100 Steel for Naval Ship Construction," <i>Naval Engineer's Journal</i> , May 1990, p. 63-82.	

II. FINITE ELEMENT MODEL AND STRUCTURAL ANALYSIS CODE

The finite element model was composed entirely of 8-node linear continuum ("brick") elements. The use of shell elements was considered but dismissed because of the need to represent accurately the stress concentrations occurring at the joint between the main cylindrical chamber and the extension heads. A minimum of five brick elements through the thickness of the vessel's walls were used in order to accurately represent bending behavior of the walls. The model contained 125,813 brick elements and a total of 154,883 nodes.

The major features of the vessel were included in the finite element model, including the 35-inch ports, the structural reinforcement around the 6-foot-diameter extension module ports, the support feet, and the "skirt" around the base of the vessel. The shrapnel-mitigation shielding that will protect the inner surfaces of the vessel from damage will probably consist of steel plates hung onto the inner walls of the vessel with a neoprene pad between the vessel and the plates. The plates do not carry any structural loads, but their added mass does have a significant effect on the response of the vessel. The mass of the shrapnel-mitigation shielding was included by simply increasing the density of the vessel's walls. The density increase used in the analysis corresponded to a shrapnel plate thickness of 3/4-inch.

The closures between both the 6-ft and the 14-ft heads were represented in an approximate way. The clevis-and-finger joint was modeled, but the tapered shear pins that connect the two halves were not explicitly modeled. Detailed analyses of these closures will be made by their designers.*

The finite element analysis code that was used in the analysis was DYNA3D.¹ DYNA3D is a non-linear, explicit, Lagrangian, three-dimensional, finite element code for solid and structural mechanics. This code was chosen because of its reputation and history in simulating similar

* The closures are being designed by Southwest Research Institute (San Antonio, Texas) and Hahn & Clay (Houston, Texas). The design work is being done at the time of writing of this report.

dynamic structural behavior, for its robust contact capabilities, and for its known computational efficiency.

III. PRESSURE-TIME HISTORY DATA CALCULATED BY MESA2D

The structural analysis of the containment vessel is based upon estimates of the time history of the pressure distribution and magnitude acting on the inner surfaces of the vessels. These estimates were made* using the explicit, Eulerian hydrodynamics code "MESA2D."² The primary uncertainties associated with the pressure loading were due to the restriction that MESA2D required an axisymmetric approximation to the three-dimensional geometry of the containment vessel. The use of the pressure data calculated by MESA2D was therefore considered as approximate and was used in the structural response calculations with caution. However, the use of MESA2D was partially validated in previous experimental tests involving a scaled model of the DARHT vessel. The scaled-model analyses and the comparison of the numerical and experimental data were described by Benner and Stevens.[†]

Many MESA2D analyses were conducted, using one of three overall approaches. The first used a model that completely ignored the 6-ft-diameter extensions. The second approach approximated the four 6-ft-diameter extensions as one continuous axisymmetric "bulge." Both of these approaches allowed the vessel's vertical axis to be used as the axis of symmetry for MESA2D. The third approach used the extension module axis as the axis of symmetry and approximated the main vessel as a cavity with volume equal to the volume of the actual vessel body. Pictures of the three-dimensional shapes assumed in the MESA2D analyses are shown in Fig. 4.

None of these axisymmetric models are faithful to the actual vessel geometry, but computational limitations required the use of a two-dimensional blast simulation code. The scheme used to associate pressure data calculated by MESA2D to particular regions of the DYNA3D finite element model was as follows: the inner surface of the main vessel was loaded by the pressures calculated in the first MESA2D approach, and the inner surface of the extension module was loaded by the pressures calculated in the extension module in the third MESA2D approach. The pressures calculated with the second MESA2D approach were not used.

The data from the MESA2D calculations consisted of 31 time-history data files from the first MESA2D model and 23 time-history data files from the second MESA2D model (15 of which were actually used). Each file corresponded to one "tracer particle," which is basically a point fixed in space at which the pressure was determined. The tracer particles were slightly offset from the inner surfaces of the vessel and also from the axis of symmetry of the vessel. These pressure

* All MESA2D analyses were conducted by Stan Marsh, DX-3, Hydrodynamic Experimentation.

† The results of a series of explosive tests conducted by DX-Division personnel on a scaled model of the containment vessel were presented at a DARHT design review held on June 20, 1996. Comparison of these results to a numerical simulation of the event was made by John Benner, ESA-EA.

Fig. 4. The three-dimensional approximations of the vessel geometry used in the MESA2D analyses. Upper left: the main vessel is modeled without any extension head. The axis of symmetry is the vertical axis (i.e., the axis of the vessel body). Upper right: the extension head is modeled as one continuous axisymmetric bulge. The axis of symmetry is still the axis of the main vessel body. Bottom: the extension head is represented faithfully, but the main vessel body is approximated by a structure that is symmetric about the extension module's axis and which has a volume equal to the actual main vessel body's volume.

histories were applied to the corresponding inner surfaces of the model using a "nearest neighbor" scheme whereby the pressure history used for each element on the inner surface of the model was the history belonging to the tracer particle closest to that element. In total, 46 tracer-particle time histories were used as pressure loading on the DYNA3D model. The location of the tracer points is shown in Fig. 5. A composite plot of all 46 pressure histories is shown in Fig. 6.

Fig. 5. Locations of the tracer points used in the MESA2D analysis. The 31 tracer points of the MESA2D model on the left side and the first 15 tracer points from the MESA2D model on the right side were used in a hybrid scheme.

Fig. 6. A composite plot of all 46 pressure-time histories applied to the model. The tracer points with the largest peak values are points #1 and #31 (shown in Fig. 5), which are on the bottom and top heads of the vessel on the centerline (poles).

The duration of the MESA2D analysis was 12 ms, which was sufficient to capture several reflected pressure waves. As a means of giving a "natural" time scale for the pressure-time histories, the time for the initial pressure wave to travel from the HE charge to the closest vessel wall was approximately 0.75 ms and approximately 2.0 ms to the furthest vessel wall.

The net impulse imparted to the vessel by the applied pressures was checked to verify that no net impulse was imparted to the vessel. Because the pressure loads represent internal forces, the net impulse acting on the vessel should be exactly equal to zero, resulting in no change in momentum to the system. The original MESA-2D data did not yield a zero net impulse on the vessel model, probably because the "nearest neighbor" assignment scheme with spatially piecewise-constant pressure assignments was not exact.

The total impulse acting on the vessel in the global y-direction is defined as

$$impulse = \int_T \int_A p(t) \bar{n}_y da dt = m \Delta v , \quad (1)$$

where $p(t)$ is the pressure-time history, and $(\bar{n}_y da)$ is the projection in the global y-direction of an element of area over which $p(t)$ is applied. The impulse is equated to the change in the momentum of the vessel in the y-direction, which is equal to the mass m times the change in y-direction velocity Δv . When this calculation was carried out using the MESA2D tracer-particle data (applied as described above), the net impulse acting on the vessel was not equal to zero at the end of the 12-ms time interval, as it must be if momentum is to be conserved, but rather had a value that corresponded to a y-direction velocity of -1.27 in./s. This net y-direction impulse prevented making any realistic estimate of the support foot reaction forces.

The following scheme was used to correct the MESA2D data to make the net impulse applied to the vessel by the internal pressures exactly zero. Because the net impulse was in the negative y-direction, causing the vessel to be "driven into" its support stand, the pressure histories of tracer particles on the top surface of the vessel were scaled by an amount that caused the net impulse on the whole vessel to equal zero. This scaling was applied to tracer particles #24 through #31 (see Fig. 5), and had a magnitude of 1.098751 over the entire 12 ms of the MESA2D pressure data. This adjustment caused the net y-direction impulse to be exactly zero at the end of the 12-ms interval. Scaling the pressures applied at the top of the vessel in this way was thought to be a conservative method of making the analysis more realistic.

The pressure acting on the vessel walls for all time after 12 ms was taken to be a constant 215 psi, uniform over all of the inner walls of the vessel. This value was determined from approximate blast calculations.

IV. RESULTS

A. Stresses Produced by the Blast Loading

The stresses produced by the vessel's response to the pressure history are a function of both the location on the vessel and of time. In general, the peak stresses produced at two different locations on the vessel do not occur at the same time. Whole-model "state dump" files (files required for generating color plots of the stress state) may only be written relatively sparsely (time spacing = 500 μ s), because of disk space limitations. Therefore, the stresses produced in a set of 163 specially chosen elements were written at a relatively fine time spacing (10 μ s). These elements were selected based on their location: elements in this set cover all highly stressed areas of the vessel and are considered to be representative of the most highly stressed regions of the vessel.

The peak stresses at various locations on the vessel are given in Table II. These are effective (von Mises) stress values and represent the maximum value at any time during the 20 ms of the analysis. Values of effective stress greater than the material yield stress of 100 ksi are fictitious (a con-

Table II: Peak Stresses Caused by Blast Loading

Maximum Effective Stress (ksi)	Location
113	Near the welded joint between the 35-in. port at the top of the vessel and the elliptical vessel head
70	12:00 position just above the upper 35-in. port on the 14-ft vessel body
70	Bottom of the vessel body (at the pole)
67	12:00 position just above the 35-in. port on the extension head
64	6:00 position just below the 35-in. port on the extension head
53	3:00 position near the lower 35-in. port on the vessel body
61	Knuckle area of the top 14-ft head
34	Knuckle area of the bottom 14-ft head
51	Knuckle area of the extension head
58	Just under the 14-ft head closure (on the vessel body)
44	Highest stresses in support foot gussets
32	12:00 position, outside, near the joint between the 6-ft extension and the 14-ft vessel body
52	9:00 position, outside, near the joint between the 6-ft extension and the 14-ft vessel body
37	6:00 position, outside, near the joint between the 6-ft extension and the 14-ft vessel body

sequence of using a linear material model), but they do indicate the location of initial yielding and the degree by which the applied loading exceeds the maximum allowable loading for no yield.

B. Dynamic Loading of the 6-ft-Diameter Closure

The finite element model of the 6-ft-diameter extension head was connected to the model of the main vessel by using 20 "tied-contact" patches. The force acting between the main vessel and the extension modules was thus broken down into 40 components around the circumference of the closure (because a plane of symmetry cut the extension port in half in the 1/2-symmetry model). The sum of the forces acting on all of the contact patches is thus the net force acting on the closure. This scheme provided the means to have the forces that the closure mechanism carries reported at a useful resolution (9.0°). The maximum tensile force that the closure mechanism must carry affects the design of the closure and its seal, as well as the preload required for the closure's fasteners.

The interface force at each contact patch was reported by the analysis code at short time intervals during the analysis. It was observed that the forces were not uniform around the perimeter of the closure. The tensile or compressive components of force (i.e., the component of force in the x-direction) tended to be greatest at the top and bottom of the closure, but the vertical shear forces (the component of force in the y-direction) was greatest near the middle of the closure's perimeter. Figure 7 shows the forces acting on the closure at one particular time.

Fig. 7. The contact forces acting on the extension module closure at time $t = 7.0$ ms. Each line represents a force vector. The negative-x direction represents tension on the interface.

The circumferential loading (force per unit length) acting on the contact patches during the time period of the analysis is shown in Fig. 8. The net (total) axial and vertical shear forces acting on

Fig. 8. The longitudinal loading (force per unit length) acting on the extension module's 20 contact patches during the first 20 ms of the vessel's response. Note that a positive value represents compression.

the closure during the time period of the analysis are shown in Fig. 9. This is the sum of the axial- or vertical-direction forces acting on the 20 individual tied-contact patches, multiplied by 2.0 to account for the symmetry of the model.

C. Dynamic Loading of the 14-ft-Diameter Closure

The finite element model of the vessel's main 14-foot-diameter head was connected to the main vessel using five tied-contact patches. Like the 6-foot-diameter closure, the use of the tied contact allowed the interface forces to be reported by DYNA3D.

The circumferential loading (force per unit length) acting on the five tied-contact patches is shown in Fig. 10. The net longitudinal (y-direction) force acting on the 14-foot-diameter closure during the time period of the analysis is shown in Fig. 11. This curve represents the total axial-direction force acting on the entire closure.

D. Support Stand Force

The forces produced at the vessel's support "feet" (i.e., the four gusseted platforms upon which the vessel is supported by the hydraulic jacks) is shown in Fig. 12. The force history shows that there is a large initial reaction followed by a period of zero force. The initial reaction occurs because the support feet are driven downwards by the (expanding) vessel when the HE detonates. The large upward reaction force acting on the vessel causes the vessel to separate slightly from the support feet (starting at about 6.0 ms).

Fig. 9. The total force acting on the entire 6-ft-diameter closure in the axial (x direction) and vertical (y direction). Note that a negative-x direction axial force is tensile.

Fig. 10. The longitudinal loading (force per unit length) acting on the five tied contacts holding the vessel's main (14-ft-diameter) head to the vessel itself.

Fig. 11. The total longitudinal (y-direction) force acting on the vessel's main 14-foot-diameter closure. Note that a negative value represents tension.

Fig. 12. The force produced at a vessel support foot. A negative value represents compression.

It is noted that the forces produced at the support feet and the lift-off behavior of the vessel are strongly dependent on the stiffness that was assumed for the vessel's support foot "foundation." The foundation consists of the vessel's hydraulic support stands and all the components and mate-

rials under the stands (including, for example, the concrete foundation block). The stiffness of the foundation is affected by the stiffness of all of these components. The value of support foundation stiffness used to produce the force history shown in Fig. 12 was $25.5 \cdot 10^6$ lbf/in. This value was determined from a finite element analysis of the support jack.*

The peak (compressive) force in the support stand was recalculated using several values of support stand stiffness. A "transient load factor" was then calculated, which simply represents the factor by which the peak force at a support foot exceeded the static load (i.e., one-fourth of the weight of the vessel). The results are shown in Fig. 13. For a very "soft" foundation stiffness, the

Fig. 13. The vessel support foot transient-load factor. The peak compressive force in the support stand is a function of the stiffness of the vessel's support "foundation." The "transient load factor" is the factor by which the peak support force exceeds static weight.

transient-load factor tends toward 1.0, meaning that the peak force at a support foot remains nearly constant at one-fourth of the vessel's weight. For a very stiff foundation, the transient-load factor tends toward a value of about 17.0 (the peak force at a support foot is 17 times larger than the static force).

It was observed that a larger foundation stiffness caused a larger peak support foot force and a larger total liftoff displacement. However, it was also observed that the support foundation stiffness did not significantly affect more "distant" responses, such as the forces at the extension module closure.

* This analysis was performed by Christopher Romero, DX-5.

E. Other Vessel Responses

The maximum deflection at the bottom and top of the vessel in the axial (vertical) direction was approximately 0.5 in. The maximum kinetic energy attained by the vessel was $3.9 \cdot 10^5$ in.-lbf, and the maximum kinetic plus potential energy was $6.5 \cdot 10^5$ in.-lbf.

REFERENCES

1. Robert Whirley and Bruce Engelmann, "DYNA3D User Manual," Lawrence Livermore National Laboratory report UCRL-MA-107254, Rev. 1 (November 1993).
2. "MESA2D Users Manual (Version 5)," Los Alamos National Laboratory report LA-CP-92-229 (June 1992).