CONF-960706--22

VERIFICATION OF MAXIMUM IMPACT FORCE FOR INTERIM STORAGE CASK FOR THE FAST FLUX TESTING

S. J. Chang

Research Reactors Division
Oak Ridge National Laboratory
Oak Ridge, Tennessee

Presented at

American Society for Mechanical Engineers (ASME)
Pressure Vessels, Piping, and ICPVT-8 Conference

Montreal, Canada
July 22-26, 1996

Prepared by the
Research Reactors Division
OAK RIDGE NATIONAL LABORATORY
Oak Ridge, Tennessee 37831
managed by
LOCKHEED MARTIN ENERGY RESEARCH CORP.
for the
U.S. DEPARTMENT OF ENERGY
under contract DE-AC05-96OR22464

VERIFICATION OF MAXIMUM IMPACT FORCE FOR INTERIM STORAGE CASK FOR THE FAST FLUX TESTING FACILITY

William W. Chen
Westinghouse Hanford company
Richland, WA 99352

Shih-Jung Chang
Oak Ridge National Laboratory
Oak Ridge, TN 37831

ABSTRACT

The objective of this paper is to perform an impact analysis of the Interim Storage Cask (ISC) of the Fast Flux Test Facility (FFTF) for a 4-ft end drop. The ISC is a concrete cask used to store spent nuclear fuels. The analysis is to justify the impact force calculated by General Atomics (General Atomics, 1994) using the ILMOD computer code. ILMOD determines the maximum force developed by the concrete crushing which occurs when the drop energy has been absorbed. The maximum force, multiplied by the dynamic load factor (DLF), was used to determine the maximum g-level on the cask during a 4-ft end drop accident onto the heavily reinforced FFTF Reactor Service Building's concrete surface. For the analysis, this surface was assumed to be unyielding and the cask absorbed all the drop energy. This conservative assumption simplified the modeling used to qualify the cask's structural integrity for this accident condition.

Because the impact analyses were based on the maximum g-levels calculated in a simplified approach, it is necessary to verify whether the results are acceptable and conservative. Therefore, an independent impact analysis was performed to determine the maximum force using the general-purpose finite-element computer code (ABAQUS, 1994; ANSYS 1993).

A four-element reduced-scale model using both ANSYS and ABAQUS was developed to verify the modeling approach. Both results agreed very well. The axisymmetric full-size finite-element model was developed and analyzed using the ABAQUS code without modeling the detailed structural components, fuel section, loose parts and gaps. However, different values of Modulus of Elasticity were used to simulate a more compact structure. This approach is conservative since it results in a larger impact force.

The results indicated that the impact force calculated by two full size models was higher than the maximum force used by General Atomics (GA). However, the impact force calculated by another model using the concrete properties was

less than that used by GA. Because the models used for this analysis are conservative, the g-levels used by GA for the design analysis are acceptable.

INTRODUCTION

Spent nuclear fuel shipping containers are designed to handle the transportation between different facilities. However, there were no unique design of a container that may handle different level of spent fuel. Thus, it requires a safety analysis almost for each individual designed container. In order that the structure suffers no catastrophic damage following the accident without radioactive release, it is essential that the container should be qualified for impact analysis.

The local effects resulting from structural impact depend on the material properties and stiffness of both the impacter and the target, and on the impacter weight, shape, size, and impact velocity. The complexities were further enhanced by large inertia effects, buckling, warping, and inelastic constitutive relationships to predict reasonable responses of the composite materials, all of which lead to gross deformation. Lack of understanding of the probable failures modes and collapse mechanisms associated with the impacted structural system represents a major impediment for successful modeling representation using either finite difference or finite element techniques.

Destructive testing of scale model container offers one means of identifying the regions and modes of collapse. However, difficulties always arise to reconcile the test results from those predicted by finite element technique to say nothing of obtaining a comparable results based on the approximated theoretical formulas.

The paper describes an impact analysis of the Interim Storage Cask (ISC) of the Fast Flux Test Facility (FFTF) for a 4-ft end drop using the general- purpose finite-element computer code (ABAQUS, 1994; ANSYS 1993). The result is to justify the impact force calculated by General Atomics (General Atomics, 1995) using the ILMOD computer code. ILMOD determines the maximum force developed by the concrete crushing which occurs when the drop energy has been absorbed. The maximum force, multiplied by the dynamic load factor (DLF), was used to determine the maximum g-level on the cask during a 4-ft end drop onto an unyielding surface so as to qualify the cask's structural integrity.

A four-element reduced-scale model using both ANSYS and ABAQUS finite element computer code was developed to verify the modeling approach. Both results agreed very well. The axisymmetric full-size finite-element model was developed and analyzed using the ABAQUS code without modeling the detailed structural components, fuel section, loose parts and gaps. However, different values of Modulus of Elasticity were used to simulate a more compact structure. This approach is conservative since it results in a larger impact force.

The results indicated that the impact force calculated by two full size models was higher than the maximum force used by General Atomics (GA). However, the impact force calculated by another model using the concrete properties was less than that used by GA. Because the models used for this analysis are conservative, the g-levels used by GA for the design analysis are adequate.

PROBLEM DISCUSSION

The analyses performed by GA applied a simplified quasi-static method based on D'Alembert's principle to substitute equivalent static forces for inertial forces created by the impact. This method treated the cask as a rigid body ignoring the flexibility of the falling body. The difference between calculating the peak dynamic oscillatory response for impact analysis and the impact force predicted by quasi-static method is accounted for by applying the DLF. The method applied by GA using static properties of the material may be justified on the basis that any dynamic effects which would increase the decelerating force would also increase the strength of the material resisting the decelerating force. This approach has a limitation when the results were applied to oblique impact cases.

The problem of impact analysis between two bodies is to determine the duration of impact and the contact force. The duration of impact consists of the combination of the three phases, namely,

- the phase of penetrations at t_1,
- the phase of reaction at t_2, and
- the phase of release at t_3,

as described in Tschedi (1921) and Andrews (1930). The duration of impact and the contact force are related by the Principle of Conservation of Momentum as described in Goldsmith (1960). Once the duration of impact and impact force have been determined, the deformation responses may be used to compute the stress and strain distributions.

The ratio of these three intervals are nearly constant for a given material and independent of the initial velocity. For a given impact velocity the magnitude of dynamic force will decrease as the target flexibility increases (or target thickness decreases). Increase of target flexibility will also increase contact duration and decrease the area of contact. For infinitely thick and rigid target, impact force will increase.

Analytical calculations for impact analysis for even a simple geometry have insurmountable mathematical problems. Simplified assumptions have to be made to avoid the mathematical difficulties. Finite-element analysis has rendered the approximate solution of impact problem tractable. However, mounting computer time and core storage are required to obtain a converged solution. Therefore, a conservative and simplified approach is required to solve the impact problem.

The impact process is really much more complicated than that conceived by general notion of compression. The possible simultaneous existence of zones of sticking and sliding at the contact surface makes the stress and strain distribution nonuniform and induces nonuniform strain rate distributions in the axial direction.

The concrete bottom surface under impact may be subjected to pulverization or spalling depending on its crushing strength and the impact velocity. For small velocity impact, such as that occurs from a 4-ft drop, spalling or crushing of the concrete will occur on the concrete at the outside of the reinforcing bar region near the surface. The reinforced steel in the concrete still retains the ability to confine the concrete mass and its shielding capability.

ANALYSIS METHOD

This analysis used nonlinear finite-element computer codes (ABAQUS, 1994, ANSYS 1993) to perform the impact analysis. The results were compared with those from GA for the 4-ft end drop. The end drop generally represents the stiffest orientation with the highest deceleration loads, while the center-of-gravity-over-bottom-corner drop produces the greatest deformations and localized bending stresses.

The nonlinear analysis may give a series of solutions depending on the modeling approach, therefore, a simple reduced-scale problem was first analyzed to gain some insight and confidence in the computer code capability without burning too much computer time.

Each computer code has unique input requirements and applies different solution procedures. ANSYS requires longer time input to stabilize the structure before achieving the full contact between the structure and the target. No unique criteria exist to apply the time step size and the total time to predict the actual contact. ABAQUS inputs the impact velocity directly, which makes the model inputs more accurate and simple. The nonlinear analysis may give a series of solutions depending on the modeling approach; therefore, a simple reduced-scale problem was analyzed first to gain some insight and confidence in the computer code capability without using too much computer time.

Two different finite-element models were developed. A simple reduced-scale model, with increased density to simulate the total weight of the cask, was developed using the ABAQUS and ANSYS codes. The results of the simple analyses may be compared. The full-size axisymmetric models were developed for both codes to perform the impact analysis. These models applied the different material properties to simulate the concrete outer shell, inner fuel section and part of the impact limiter. The materials used are tabulated in Table 1.

ABAQUS Model

The four-element reduced-scale model and the axisymmetric full-size finite-element model, as shown in Figures 1 and 2, were analyzed using the ABAQUS code. Both models were run using reasonable computer times compared with the ANSYS time. The four-element model uses eight-node solid elements with gap elements as the contact elements with gap closed initially.

Three axisymmetric full-size ABAQUS models, using the 4-node axisymmetric solid elements, were developed for analyses so that the results could be compared.

- Model A uses gap elements as the contact elements with gap closed initially.

- Model B uses interface element for contact between a single surface node of a rigid body and nodes of the axisymmetric mesh. Models A and B were developed to be structurally more stiffer to obtain the higher impact force. Because the results were much higher than that predicted by GA (1995).

- Model C was developed to reconcile the results predicted by GA (1995). Model C is also an axisymmetric model with gap elements but used concrete Young's Modulus, concrete compression yield stress, and increased density to simulate the total weight of the cask.

ANSYS Model

A four-element reduced-scale model using 8-node solid elements, as shown in Figure 3, was analyzed using the ANSYS code with scaled-up density to simulate the total weight of the cask. The axisymmetric full-size solid element model simulating the cask using the ANSYS code required more than 70 hours of computer time on CEA2 work station and could not reach numerical convergence. Therefore, the impact analysis using the ANSYS code was abandoned.

DISCUSSION OF RESULTS

Reduced-Scale Model

The reduced-scale finite-element ABAQUS model with node and element numberings is shown in Figures 1. The contact and separation sequences of the ANSYS analysis using the reduced-scale model are shown in Figure 3. Table 2 compares the reaction forces for the four-element reduced-scale ANSYS and ABAQUS model. The node reaction forces appear to have distributed unevenly on the bottom nodes in the ANSYS model while ABAQUS results have an uniform distribution. The user-defined stiffness input for gap elements in the ANSYS model is the key parameter and may contribute to the differences. However, the total impact force is in close agreement for both models even though different boundary conditions were applied.

Full-Size Model

The ABAQUS full-scale axisymmetric finite-element model is shown in Figure 2. The reaction forces on all the bottom nodes for Models A, B, and C are tabulated at Tables 3, 4 and 5, respectively. The force-time histories of all the bottom nodes in ABAQUS analyses using gap and interface elements as the contact elements for Models A, B, and C are plotted in Figures 4, 5, and 6, respectively.

Because the force-time histories represent the variation of forces over a finite period of contact, a time-average impact force F was computed as follows:

$$F = \frac{1}{T} \int_0^T f(t)\ dt, \qquad (1)$$

where:
T is the contact duration,
and f(t) is the calculated contact force at time t.

Tables 3 and 4 show that the time-averaged impact force was calculated at 242.6g and 262.0g for Models A and B, respectively. It is seen that Model B calculates the impact force at 8 percent higher than that of Model A. In comparison, the Model A acceleration (242.6g) is 38 percent higher than that calculated by GA (175.8g) (1995). When considering the modeling conservatism, i.e., no concrete properties modeled, the actual impact forces for the cask are expected to be lower than the model predicted.

By comparing further with the results from GA (1995), an analysis was performed using Model C which applied the same concrete properties and yield strength as those used by GA. The force-time histories of all the bottom nodes in ABAQUS analyses using Model C are plotted in Figure 6. This approach makes Model C more structurally flexible than that of Models A and B. As a result, the time-averaged impact force was calculated at 10,925 kips (96g), shown in Table 5, which is less than those calculated by Models A and B. In terms of acceleration, Model C is 45 percent lower than the maximum acceleration (175.8) calculated by GA, including the dynamic load factor. Because the contact surface following the accident condition occurs mostly on the concrete surface, lower impact force may be expected with large deformation. Thus, the maximum impact force used by GA to perform the static equivalent analysis is judged acceptable.

Increasing the impact force may result in concrete crushing or spalling outside the reinforced region near the surface. The reinforced steel retains the ability to confine the concrete mass and its shielding capability.

CONCLUSION

Analyses of the reduced-scale model using ANSYS and ABAQUS code verified the modeling approach for impact analysis. The results of impact analyses using the ABAQUS models A and B demonstrated that the impact force acting on the bottom nodes is higher than that used by General Atomic to structurally qualify the Interim Storage Cask under 4 ft end drop accident condition. However, modeling conservatism and very high stiffness built into the computer models A and B in the analysis overpredicted the impact force. Thus, models A and B predicted higher impact forces than that used by GA for design analysis. However, Model C calculates the impact force lower than that used by GA. Because of the conservatism in the analyses, the impact force used by GA in performing the design analyses is conservative.

REFERENCES

ABAQUS, 1994, "*User's Manual*," Hibbitt, Karlsson, & Sorenson,Inc., Providence, Rhode Island, Version 5.4.

Andrews, J. P., 1930, "Theory of Collision of Spheres of Soft Metals," *Phil. Mag.*, S.7, Vol.9, No.58, April, pp.593-610.

ANSYS, 1993, "*Engineering Analysis Computer Program*," Swanson Analysis Systems, Inc., Houston, Pennsylvania, Version 5.0 A.

General Atomics, 1995, "*FFTF Spent Fuel Interim Storage Cask*," Design Analysis Report, 910683/A, General Atomics, San Diego, California, Sept., 1995.

Goldsmith, W., 1960, "*Impact*," *Edward Arnold (Publishers) Limited*, London, Great Britain.

Tschedi, E. W., 1921, "Duration of Impact of Bars," *Phys. review*, Vol. 18, pp.423-430.

Wilkins, M. L. and M. W. Guinan, 1973, "Impact of Cylinders on a Rigid Boundary," J. Appl. Phys., Vol. 44, No. 3, pp.1200-1206.

Zukas, J. A., T. Nicholas, H. F. Swift, L. B. Greszczuk, L. B., and D. R. Curran, 1982, "*Impact Dynamics*," John Wiley & Sons, Inc., New York.

Table 1. Material Properties

Material	Modulus of Elasticity (psi)	Poison's Ratio	Yield Stress (psi)
Concrete	3.12×10^{-6}	0.3	3.000
Steel	29.5×10^{-6}	0.3	30.000
Aluminum	10.0×10^{-6}	0.3	20.000

Table 2. Reaction Forces (kips)
(Reduced-Scale Model)

Node No.	ANSYS	Node No.	ABAQUS
1	2.818	101	3.650
2	4.182	102	3.650
3	6.228	111	3.650
4	4.182	112	3.650
Total	17.410	Total	14.600

Note: The weight of the loaded cask is approximately 114.2 kips.

Table 3. Reaction Forces of the
Axisymmetric ABAQUS Model A

	Reaction Forces (Kips)								
Time	3.386E-5	.7358E-4	1.163E-4	1.589E-4	2.033E-4	2.453E-4	2.950E-4	3.123E-4	Remarks
ΔT	3.386E-5	.3972E-4	.4272E-4	.4260E-4	.444E-4	.420E-4	.497E-4	.173E-4	
Node 1	34	94	174	206	229	247	254	257	
2	213	529	976	1,184	1,347	1,480	1,534	1,522	
3	465.	1,080	1,888	2,280	2,689	3,038	3,229	3,110	
4	861	1,623	2,607	3.411	3,922	4,491	5,021	4,905	
5	1,820	2,698	3.679	4,569	5,336	6,226	6,845	6,759	
6	2,384	2,729	2,680	2,965	3,571	3,655	3,660	3,610	
7	2,407	2,425	2,434	2,489	2,780	2,403	1,981	2,168	
8	3,030	2,477	2,652	2,520	3,055	2,871	2,250	2,290	
9	3,177	2,718	2,530	2,920	2,885	3,126	2,618	2,338	
10	3.234	2,522	2,830	3,211	3,406	3,476	3,152	3,038	
11	2,473	1,716	1,986	1,907	1,967	1.976	1,843	1,868	
Total	20,098	20,611	24.436	27,662	31,187	32,989	32,387	31,865	

Time-averaged total force = 27,706 kips or 242.6g.

Table 4. Reaction Forces of the Axisymmetric ABAQUS Model B

	Reaction Forces (Kips)								
Time	.4133E-4	.8362E-4	1.278E-4	1.702E-4	2.151E-4	2.567E-4	2.900E-4	3.097E-4	Remarks
ΔT	.4133E-4	.4229E-4	.4418E-4	0.424E-4	0.449E-4	0.411E-4	0.333E-4	0.197E-4	
Node 1	31	88	149	180	219	277	280	272	
2	273	536	907	1,117	1,309	1,511	1,641	1,579	
3	584	1,241	1,979	2,329	2,677	3,059	3,339	3,138	
4	1,062	2,064	3,037	3,728	3,950	4,382	5,037	4,880	
5	1,820	3,217	4,758	5,191	5,908	6,790	7,288	7,189	
6	2,355	3,195	4,310	4,611	5,237	5,507	5,612	5,624	
7	2,215	2,526	2,612	2,408	2,769	2,313	1,985	2,131	
8	3,173	2,558	2,754	2,558	3,097	2,684	2,263	2,278	
9	3,614	3,142	2,685	3,041	2,816	3,197	2,727	2,365	
10	3,919	3,311	3,238	3,285	3,459	3,423	3,182	3,058	
11	2,210	1,829	1,941	1,937	1,968	1,943	1,840	1,865	
Total	21,256	23,707	28,370	30,385	33,409	35,086	35,194	34,379	

Time-averaged total force = 29,926 kips or 262.0g

Table 5. Reaction Forces of the Axisymmetric ABAQUS Concrete Model C

	Reaction Forces (Kips)								
Time	1.711E-4	3.458E-4	5.122E-4	6.755E-4	8.590E-4	10.21E-4	12.00E-4	13.92E-4	Remarks
ΔT	1.711E-4	1.747E-4	1.664E-4	1.633E-4	1.835E-4	1.620E-4	1.790E-4	1.920E-4	
Node 1	96	88	25	16	17	32	62	60	
2	576	515	166	105	105	208	367	356	
3	1,152	972	375	233	222	452	715	688	
4	1,724	1,339	593	339	389	717	1,020	1,004	
5	2,301	1,595	799	468	555	1,023	1,269	1,312	
6	2,841	1,818	1,022	640	751	1,287	1,541	1,532	
7	3,228	1,993	1,160	812	995	1,506	1,768	1,731	
8	2,967	1,634	1,072	734	945	1,446	1,662	1,636	
9	2,150	1,456	847	632	823	1,294	1,475	1,523	
10	1,634	1,152	689	343	810	987	1,319	1,251	
11	1,414	883	926	550	1,015	876	1,196	1,007	
Total	20,083	13,445	7,674	4,872	6,627	9,828	12,394	12,100	

Time-averaged total force = 10,925 kips or 96g.

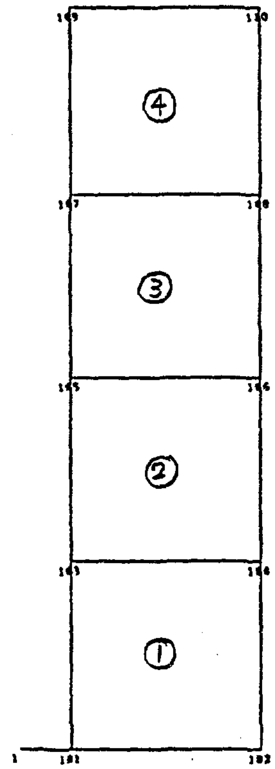

Figure 1. ABAQUS Reduced Scale Model (Plane View).

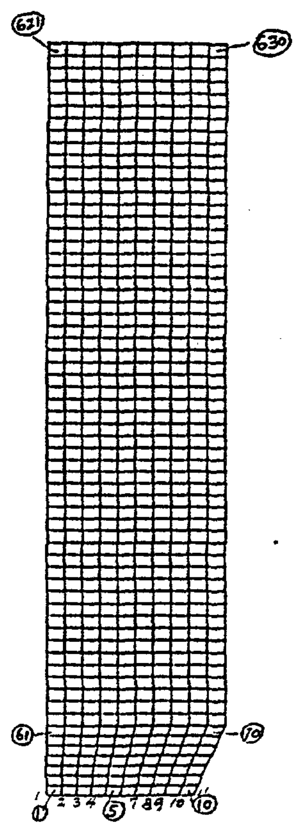

Figure 2. ABAQUS Full Size Axisymmetric Finite-Element Model.

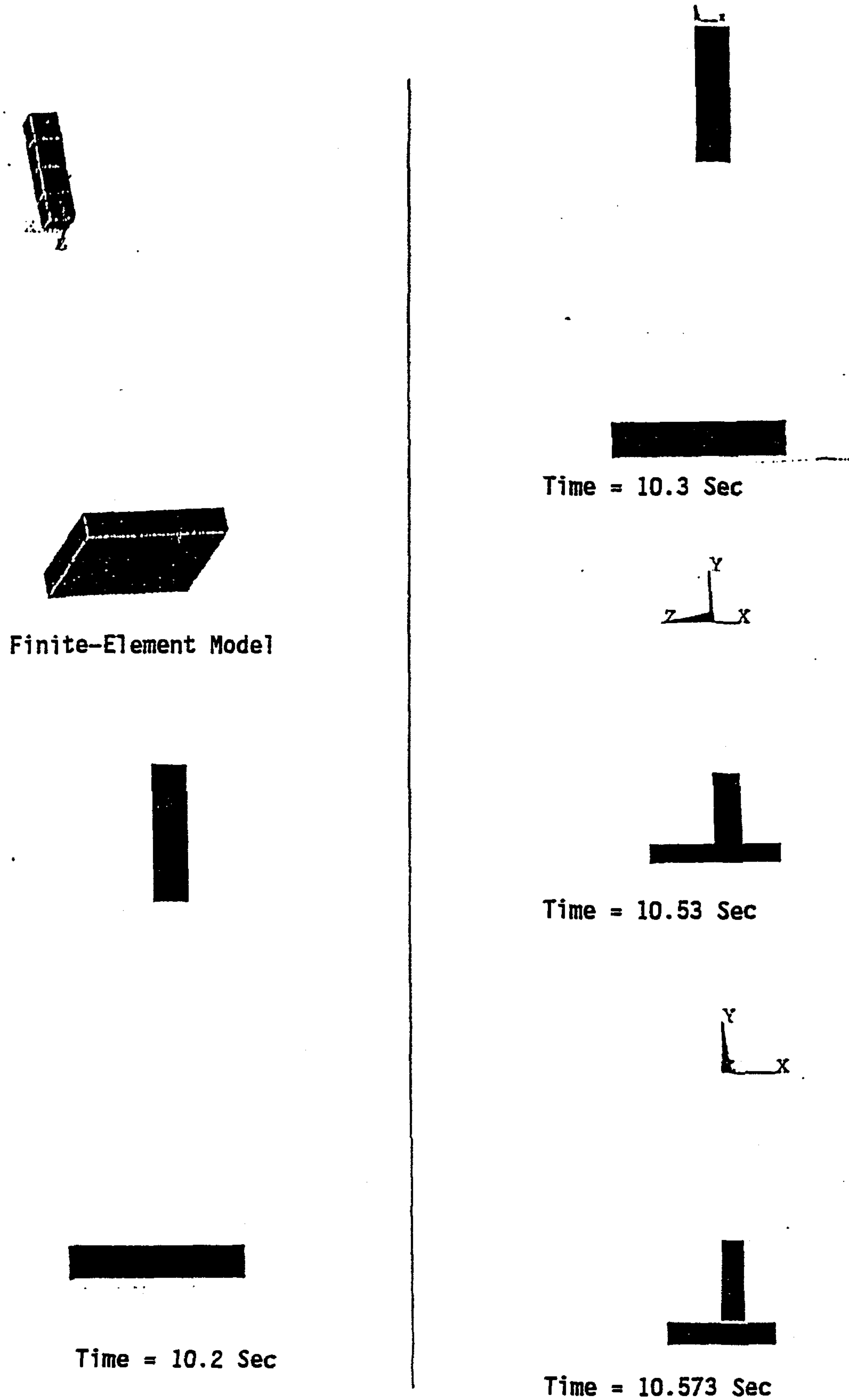

Figure 3. ANSYS Reduced-Scale Model Drop and Contact Sequence for Impact Analysis

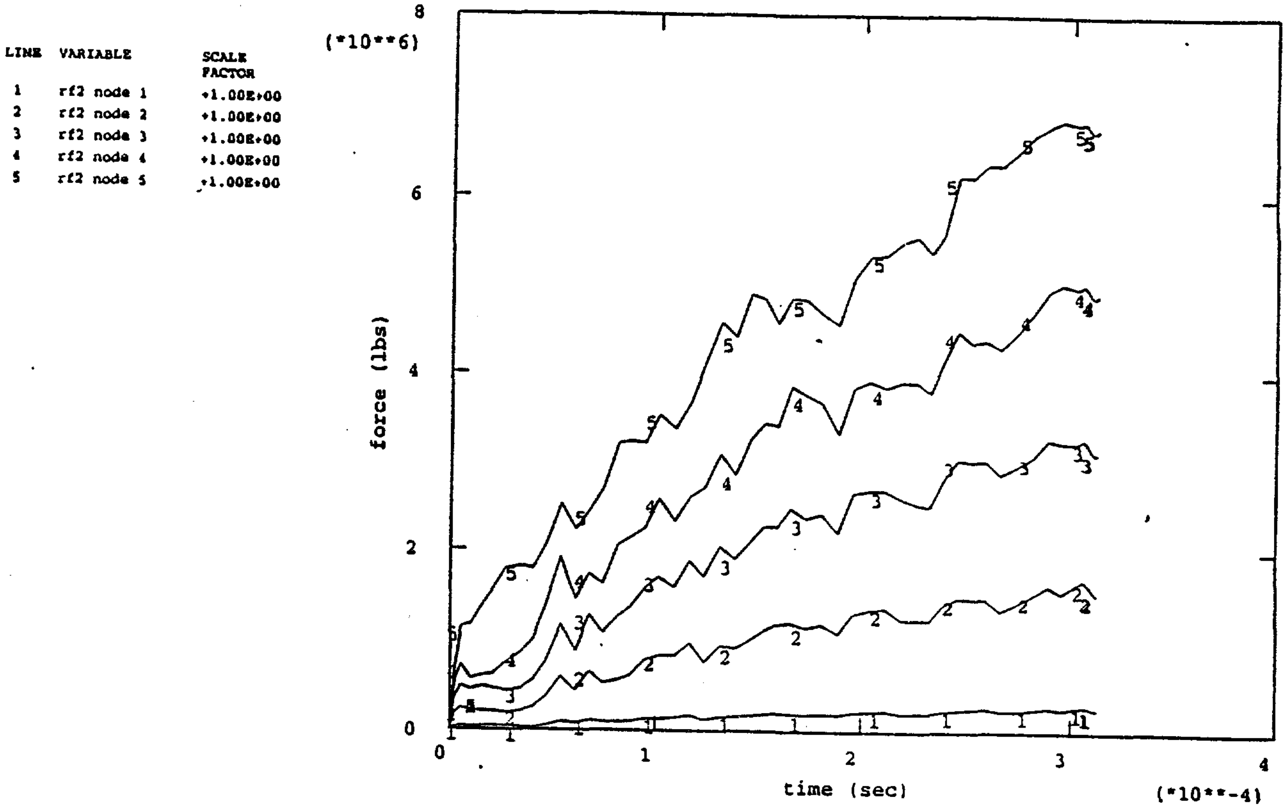

Figure 4a. Reaction Force Histories for Bottom Nodes 1, 2, 3, 4 and 5 of ABAQUS Full-Size Model (Model A).

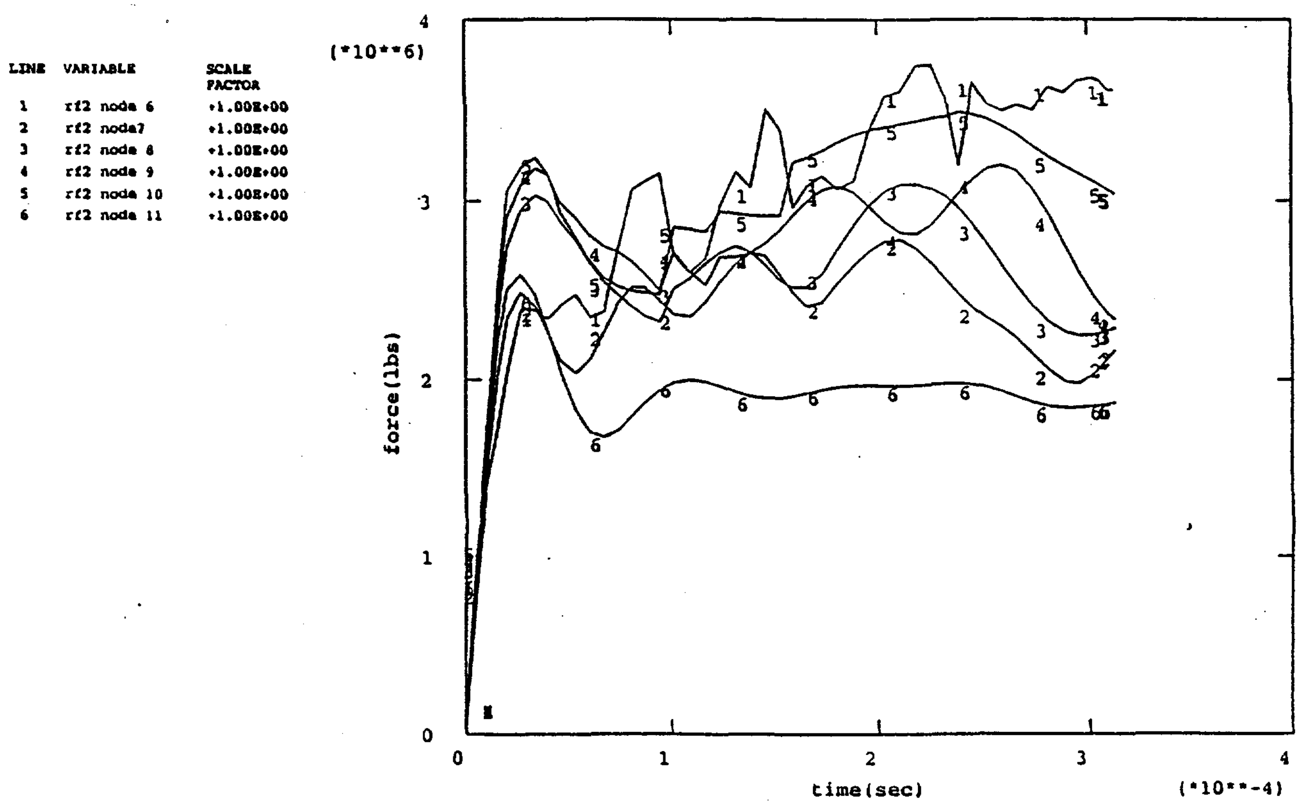

Figure 4b. Reaction Force Histories for Bottom Nodes 6, 7, 8, 9, 10 and 11 of ABAQUS Full-Size Model (Model A).

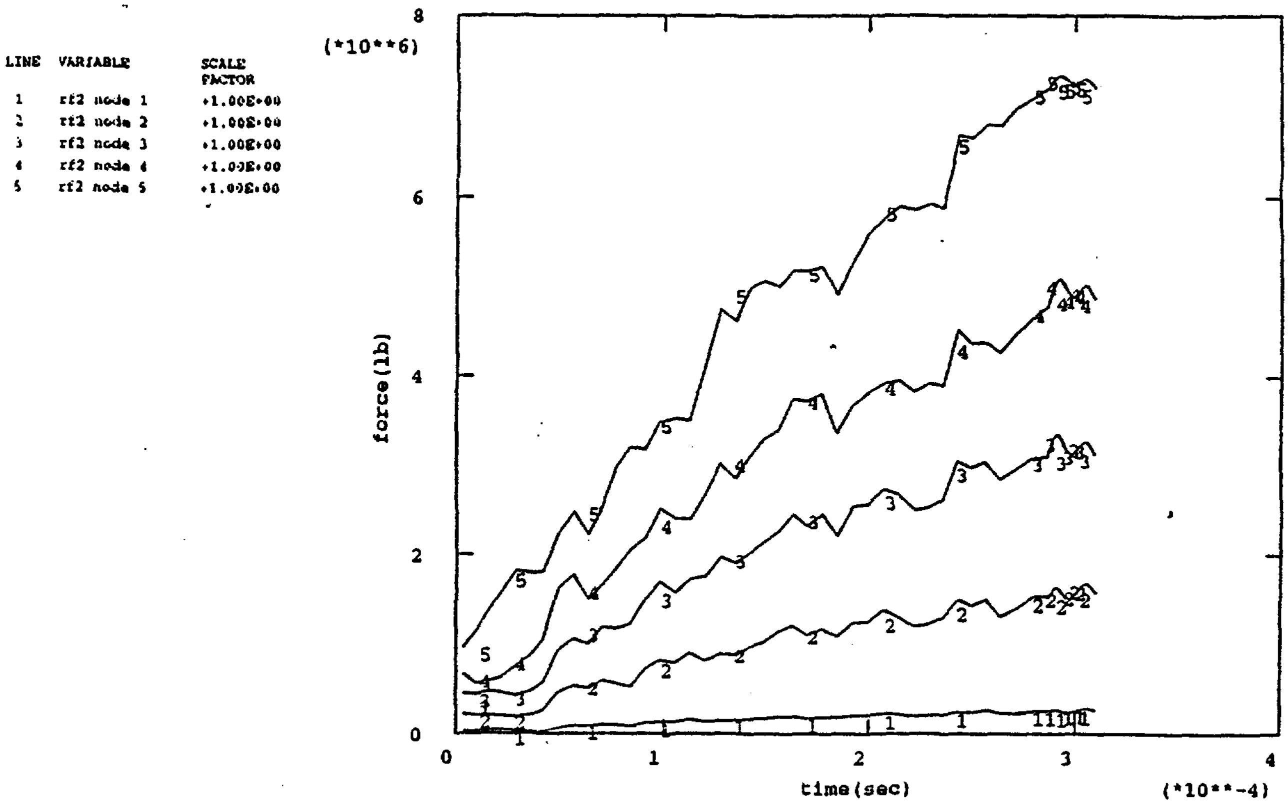

Figure 5a. Reaction Force Histories for Bottom Nodes 1, 2, 3, 4 and 5 of ABAOUS Full-Size Model (Model B).

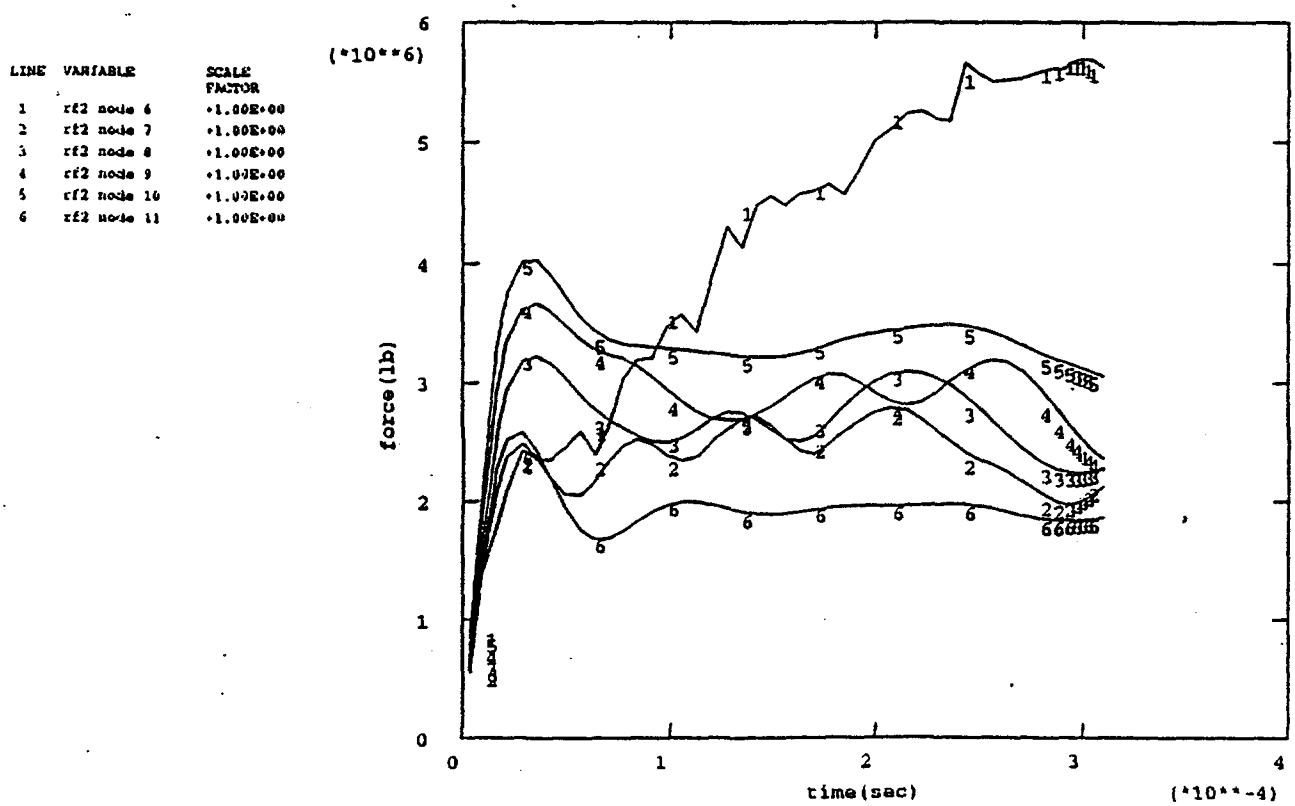

Figure 5b. Reaction Force Histories for Bottom Nodes 6, 7, 8, 9, 10, and 11 of ABAQUS Full-Size Model (Model B).

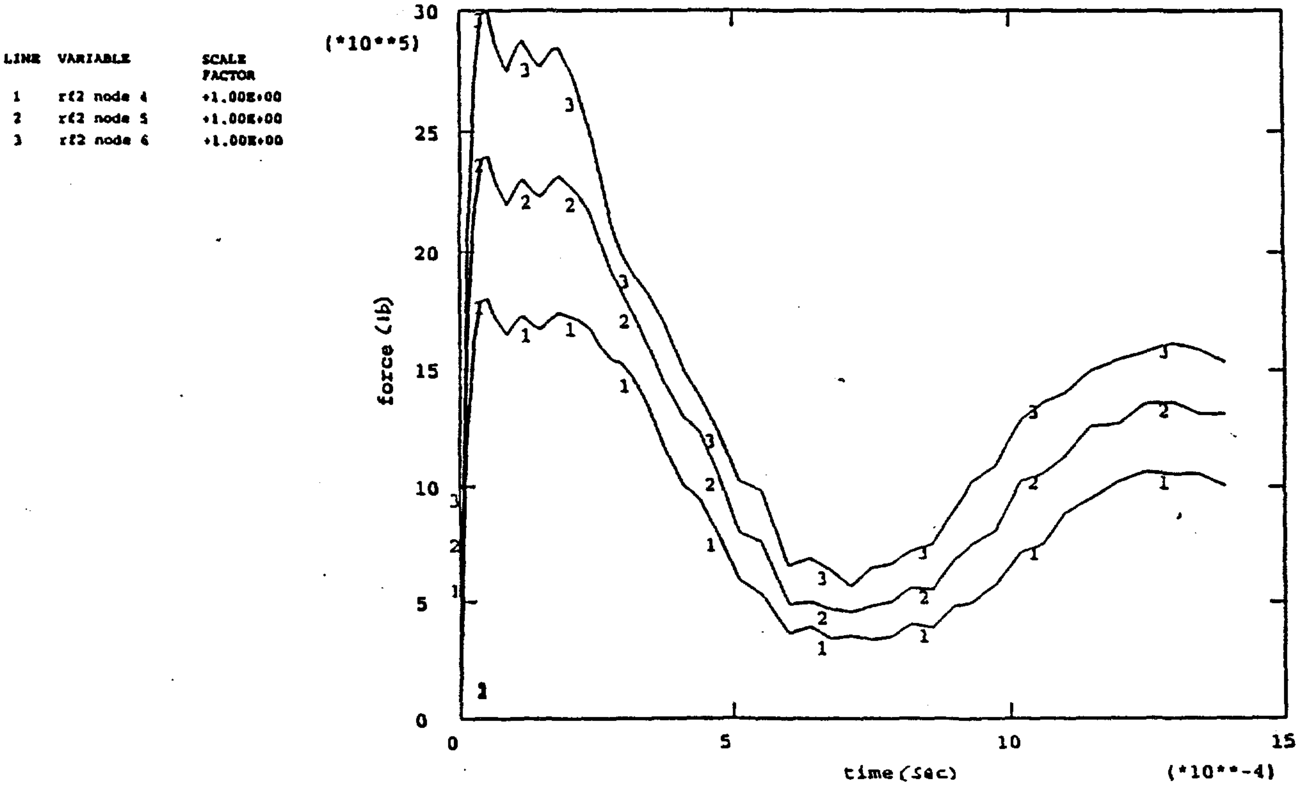

Figure 6a. Reaction Force Histories for Bottom Nodes 1, 2, and 3 of ABAQUS Full-Sized Model (Model C).

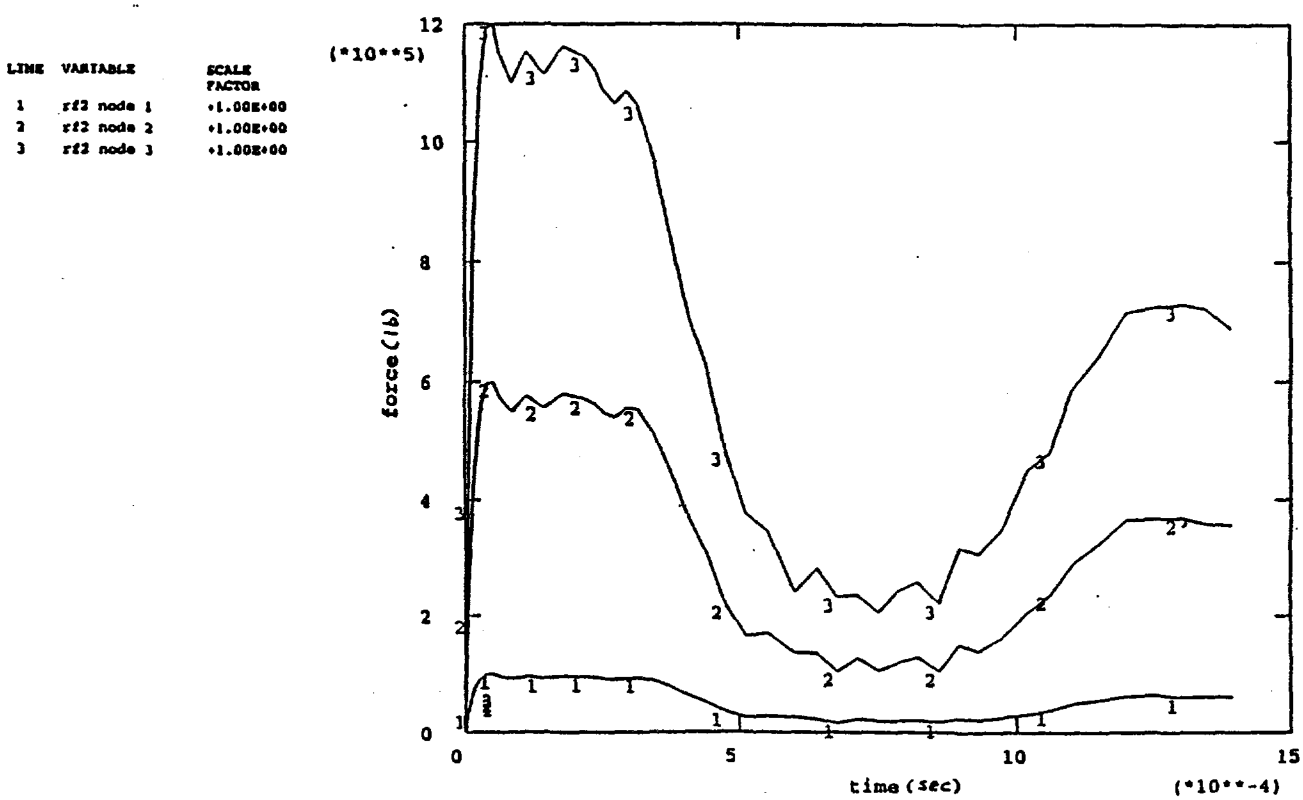

Figure 6b. Reaction Force Histories for Bottom Nodes 4, 5, and 6 of ABAQUS Full-Sized Model (Model C).

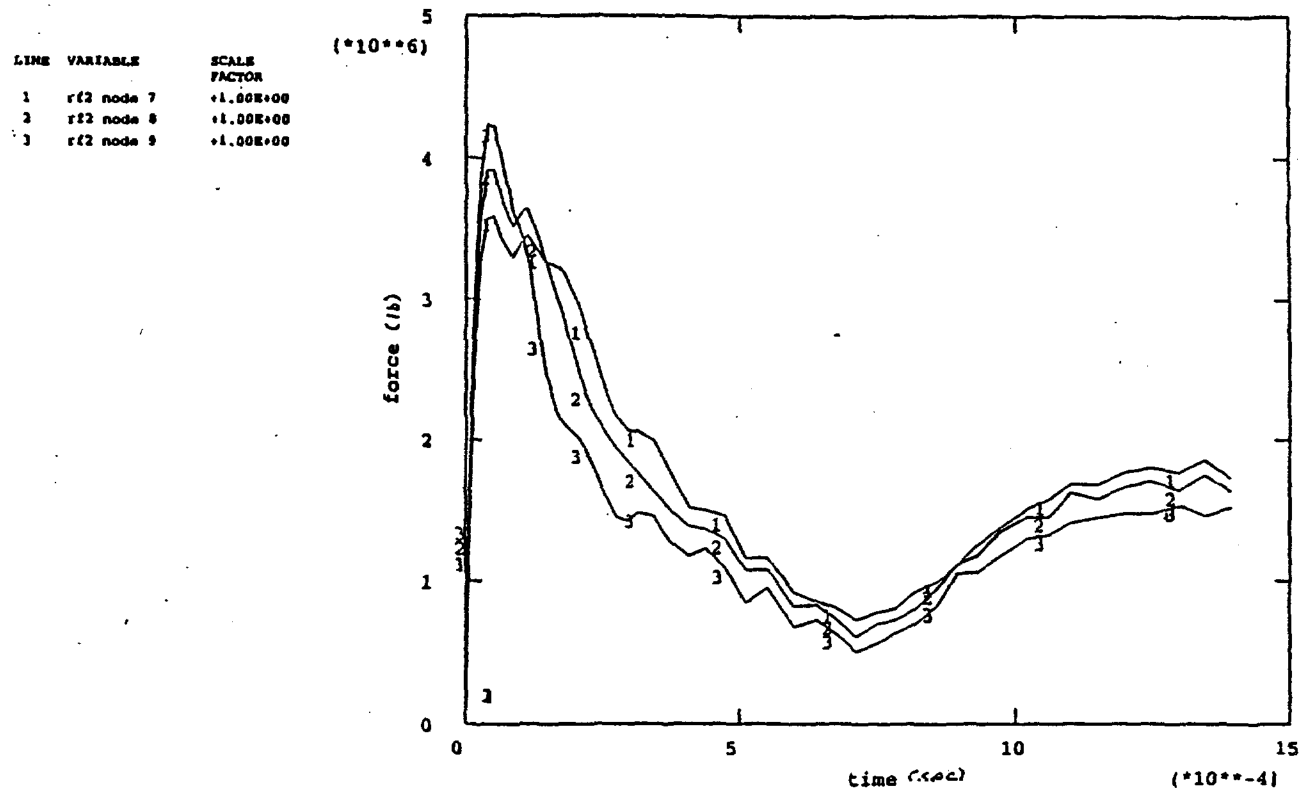

Figure 6c. Reaction Force Histories for Bottom Nodes 7, 8 and 9 of ABAQUS Full-Sized Model (Model C).

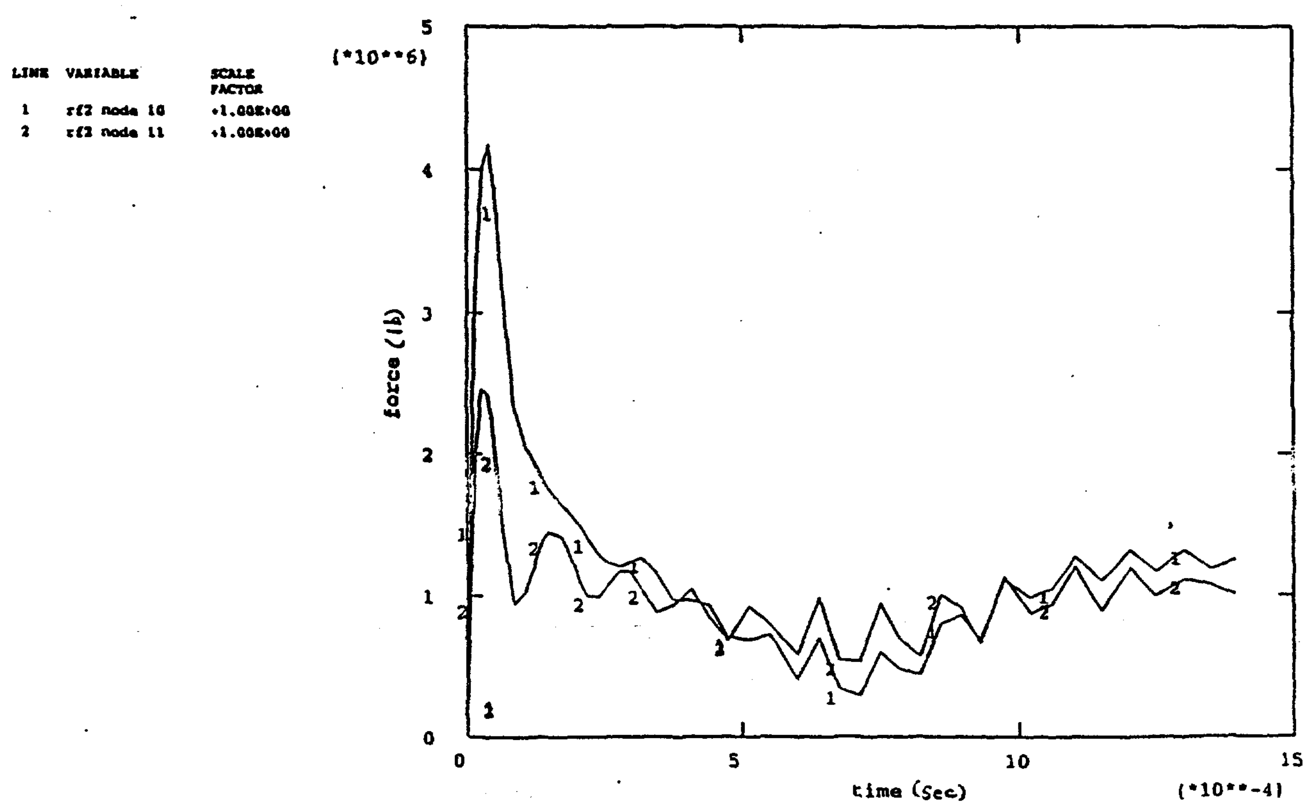

Figure 6d. Reaction Force Histories for Bottom Nodes 10 and 11 of ABAQUS Full-Sized Model (Model C).