14th International Symposium on the Packaging and
Transportation of Radioactive Materials (PATRAM 2004),
Berlin, Germany, September 20-24, 2004

Paper # 075

Evaluation of Finite Element Codes for Demonstrating The Performance of Radioactive Material Packages in Hypothetical Accident Drop Scenarios

Chi-Fung Tso, Arup, UK
R. Hüggenberg, Gesellschaft für Nuklear-Behälter mbH, Germany

INTRODUCTION

Drop testing and analysis are the two methods for demonstrating the performance of packages in hypothetical drop accident scenarios. The exact purpose of the tests and the analyses, and the relative prominence of the two in the license application, may depend on the Competent Authority and will vary between countries.

The Finite Element Method (FEM) is a powerful analysis tool. A reliable finite element (FE) code when used correctly and appropriately, will allow a package's behaviour to be simulated reliably. With improvements in computing power, and in sophistication and reliability of FE codes, it is likely that FEM calculations will increasingly be used as evidence of drop test performance when seeking Competent Authority approval.

For a FE code to be acceptable it must ideally be shown to be sufficiently reliable in all relevant cases. The code should be able to produce results which are within the band of experimental scatter that might be obtained from physical drop testing. Even if a code were not capable of producing results within the band of experimental scatter, it would be acceptable if the predictions of damage were reliably pessimistic.

What is lacking at the moment, however, is a standardised method of assessing a FE code in order to determine whether it is sufficiently reliable or pessimistic.

To this end, the project Evaluation of Codes for Analysing the Drop Test Performance of Radioactive Material Transport Containers [1], funded by the European Commission Directorate-General XVII (now Directorate-General for Energy and Transport) and jointly performed by Arup and Gesellschaft für Nuklear-Behälter mbH, was carried out in 1998.

The work consisted of three components:

- Survey of existing finite element software, with a view to finding codes that may be capable of analysing drop test performance of radioactive material packages, and to produce an inventory of them.
- Develop a set of benchmark problems to evaluate software used for analysing the drop test performance of packages.
- Evaluate the finite element codes by testing them against the benchmarks

This paper presents a summary of this work.

There are different approaches to analysing the drop test behaviour of packages, ranging from "dynamic lumped parameter" method, to quasi-static methods in which the containment and the basket are analysed on their own using static implicit FE analysis, to detailed dynamic analysis with a model of a complete package using non-linear dynamic FE methods. Purpose and context of the analysis, requirement of the competent authority, behaviour of the package, budget and timescale, expertise of the analysis organisation, are just some of the factors that will influence the choice of the most appropriate methodology. This work concentrated on the dynamic non-linear method.

SURVEY OF FE CODES

A survey of existing FE codes was conducted to identifying codes that may be capable of analysing the drop test performance of radioactive material packages using the dynamic non-linear FE method – i.e. codes that are capable of analysing dynamic events with non-linear geometry and non-linear material models. The codes that fulfil these requirements are as follows, with their developer shown in brackets:

- ABAQUS/Explicit (*Hibbitt, Karlsson & Sorensen Inc, USA*.)
- ANSYS (*ANSYS Inc. USA*.)
- DIANA (*Analysis BV, Netherland*)
- DYNA3D (*Lawrence Livermore National Laboratory,USA)*
- H3DMAP (*Ontario Hydro Technologies, Canada*)
- LS-DYNA3D (*Livermore Software Technology Corporation, USA*)
- LUSAS (*FEA Ltd. UK*.)
- NIKE3D (*Lawrence Livermore National Laboratory,USA*)
- PRONTO2D/3D (*Sandia National Laboratories, USA*)
- SOLVIA (*Solvia Engineering, Sweden*)

For each code, the survey also identified the following: quality assurance, maintenance and support, benchmarking, current applications, operating platforms, solver type, pre-processing system, post-processing system, material options, strain rate dependence capability, failure criteria, element types, contact types, pre-stressing of elements, and thermal expansion modelling.

DEVELOPMENT OF BENCHMARK PROBLEMS

Three benchmark problems were developed. They were designed to test FE codes' ability to model pertinent physical phenomena without requiring extensive use of computer resources or intricate modelling. Each benchmark represents a distinct category of impact phenomena that could be found in package impact.

For each benchmark, the parameters that define its physical characteristics were exhaustively defined, including material properties, geometric details, boundary conditions, interface conditions, and initial conditions. Analysis time and details of required output were also defined. Details of the modelling, e.g. finite element mesh, element type, material model, etc, were not defined, as it will depend on the individual FE code.

Overview of Benchmark 1: Flat Side Impact of Concentric Cylinders

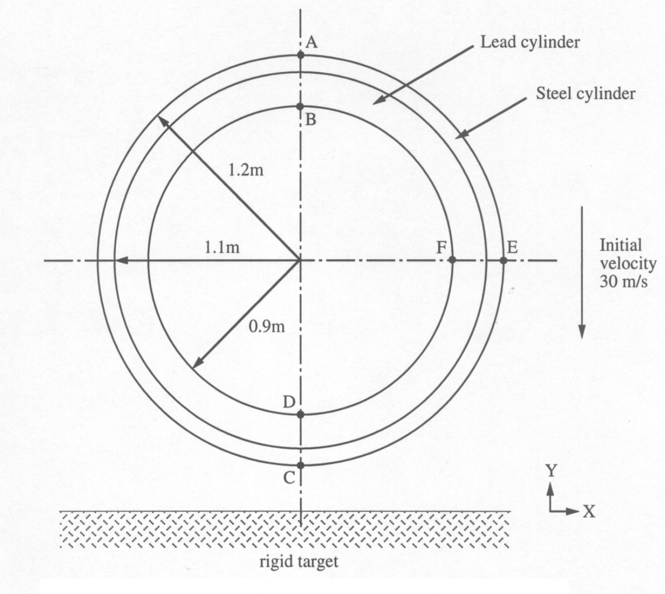

Figure 1: Benchmark 1

Benchmark 1 represents a severe side impact of a cylindrical cask body from 30m onto an unyielding target. The geometry is idealised as a two-dimension problem.

The aim of this benchmark is to test the software capability in representing:
- Two-dimensional plane strain behaviour
- Elastic and plastic deformation, followed by unloading
- Frictional interfaces between two deformable materials
- Frictional interfaces between a deformable material and a rigid target

The geometry, impact velocity, friction coefficient and material properties were selected so that the desired effect can be demonstrated most clearly.

Required outputs are:
- Displacement time histories in the Y direction (hereafter Y-displacement) from points A, B, C and D
- Displacement time histories in the X direction (hereafter X-displacement) in points E and F

Overview of Benchmark 2: Corner Impact of a Cube

Benchmark 2 was designed to test the ability of FE codes in modelling 'solid metal flow' type phenomenon - a common mode of deformation in integral shock absorbers of cuboidal flasks. The benchmark represents the impact of at a corner of a 50 tonne cuboidal flask onto a flat unyielding target from a drop height of 9m. For geometric simplicity, only the corner is defined for explicit modelling. The rest of the flask is assumed undeformable, with its inertia and boundary with the corner defined by rigid shells. The size of the cube is sufficiently large so that the location of this boundary does not affect the deformation behaviour of the corner.

This benchmark tests the software's ability to model:

- 3-Dimensional elastic and plastic deformation and unloading
- Solid metal flow material behaviour
- Severe deformation and distortion of finite elements.

Required output was displacement time histories in the direction perpendicular to the target from points A and B.

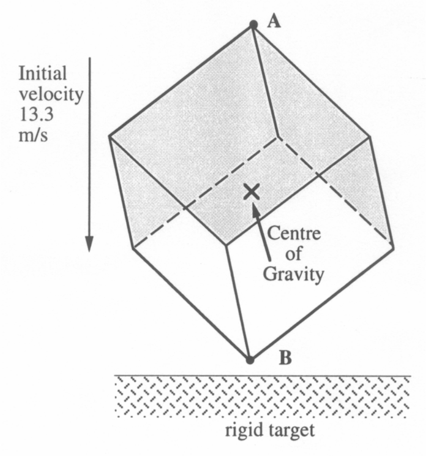

Figure 2: Benchmark 2

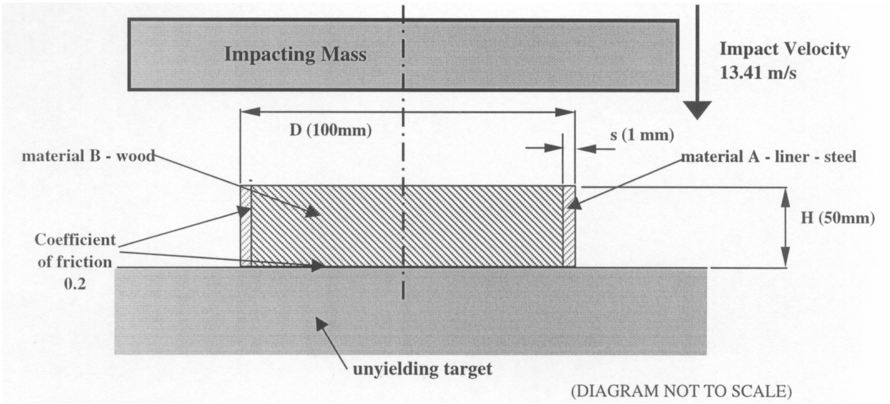

Figure 3: Benchmark 3

Overview of Benchmark 3: Impact of a Wooden Cylinder with Steel Cladding

Benchmark 3 represents a class of deformations typically found in impact limiters which consist of wood within a steel housing. The model consists of a solid wood cylinder, surrounded at the cylindrical surface by a thin steel plate. The assembled cylinder is placed on a flat unyielding surface and is impacted by a falling rigid body with a velocity corresponding to a 9m drop height. The geometry of the wood sample was chosen to correspond to an actual experimental test. The mass of the impact body was chosen to be 100kg, corresponding to an equivalent area load typical for such a shock absorber.

The aim of this benchmark was to test the software ability to model:

- Elastic and plastic deformation.
- Two material models with different deformation behaviour.
- Frictional interfaces between two deformable materials.
- Frictional interfaces between deformable materials and rigid bodies.

Required output were:

- Maximum compression of the wood.
- Displacement time history of the impacting mass.
- Maximum impact force and time of occurrence.
- Impact force vs. time.

EVALUATION OF FE CODES

Developer/suppliers of FE codes identified in the survey were invited to analyses the three benchmarks. The code, the analysis organisation, and the benchmark analysed are shown in the table below:

Where an organisation did not analyse all three benchmarks, it was because of lack of resources rather than inability of the respective code to analyse the particular benchmark. Organisations mentioned in the code survey but not listed above either declined to take part in the exercise or did not have sufficient resources available for the required timescale.

Code	Analysis Organisation	Benchmarks Analysed		
		1	2	3
ABAQUS/Explicit	Hibbitt, Karlsson & Sorensen Inc. (User/Distributor)	✓	✓	✓
LS-DYNA3D	Arup (User/Distributor)	✓	✓	✓
LUSAS	FEA Ltd. (User/Developer)	✓		
H3DMAP	Ontario Hydro Technologies (User/Developer)	✓	✓	✓
DYNA3D (Public Domain)	Gesellschaft für Nuklear-Behälter mbH (User)	✓		✓
PRONTO3D	Sandia National Laboratories (User/Developer)	✓	✓	✓
PRONTO3D	University of Texas (User)	✓	✓	

Results of Benchmark 1

Behaviour of the cylinders is illustrated in terms of plastic strains at three different times during the analysis in Figure 4.

Deformation of the cylinders Y-displacement time histories at points A and C are shown in Figures 5 and 6 respectively. X-displacement time histories at point E are shown in Figure 7.

Agreement between the analyses is extremely good. Scatter of Y-displacement at Point A from the different analyses is similar to the scatter of Y-displacement at Point B. In both, the difference between the largest and smallest prediction at maximum displacement was about 9%, or 70mm. Scatter in Y displacement at Points C and D between the different analyses appeared larger. However, the actual displacements were smaller than at Point A, and the maximum spread was only about 30mm. Scatter in X-displacements at points E and F were very similar. Predictions from the analyses were very consistent, with a difference of only 2% at maximum displacement. All the codes predicted very similar initial rebound velocities.

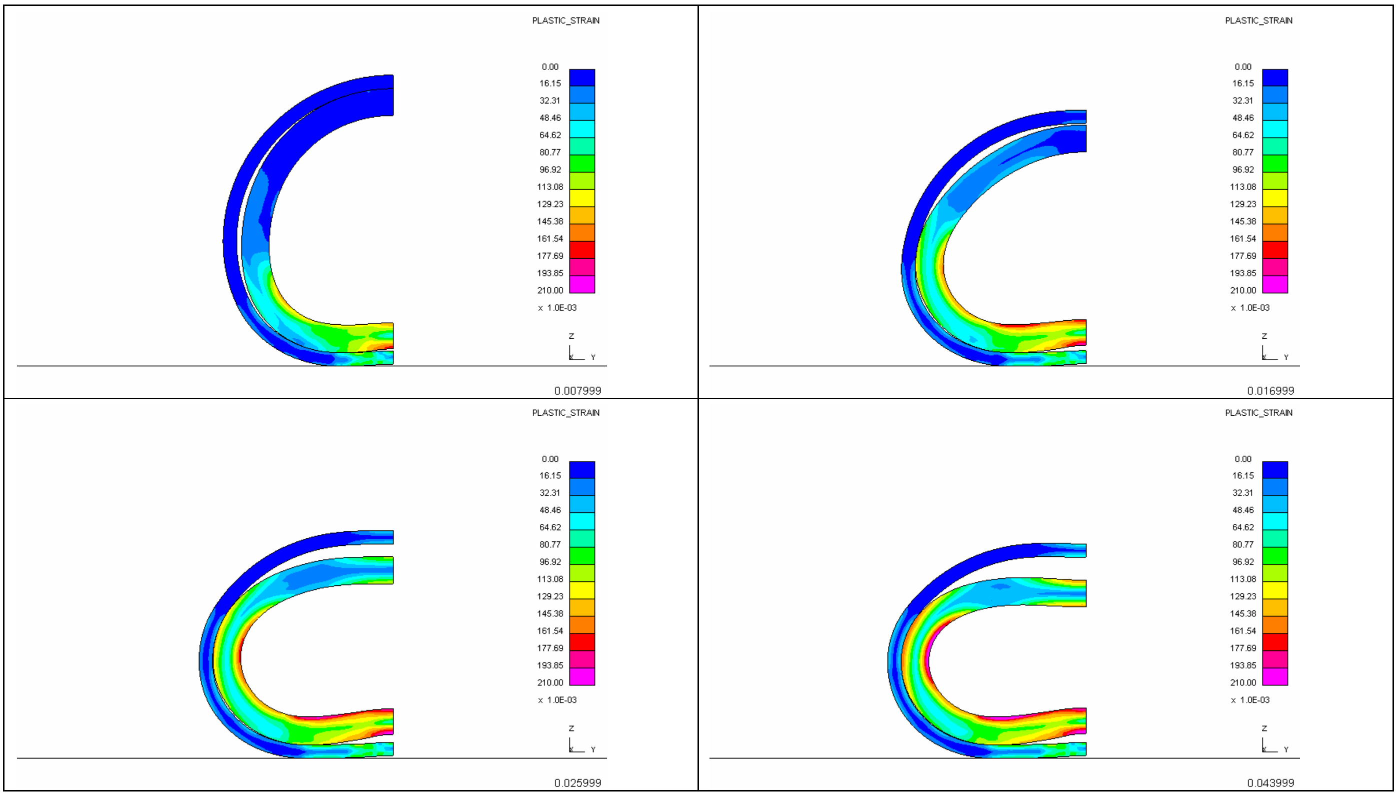

Figure 4: Deformation of the cylinders

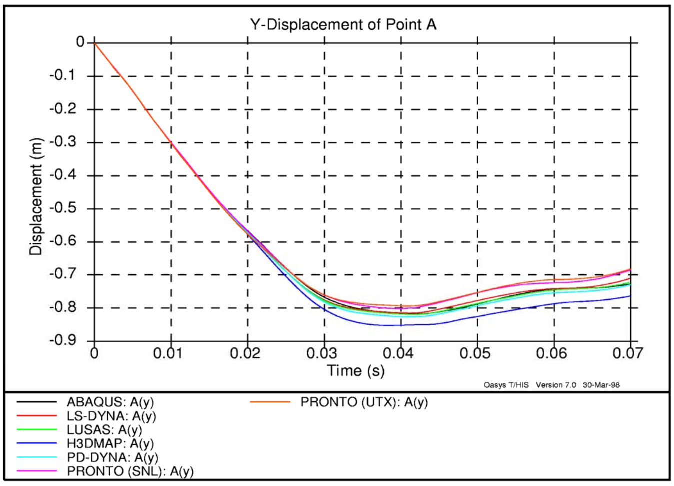

Figure 5: Y-displacement of Point A

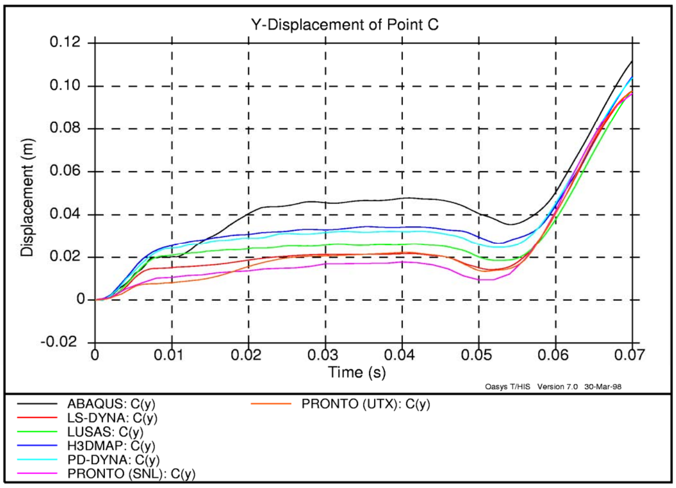

Figure 6: Y-displacement of Point C

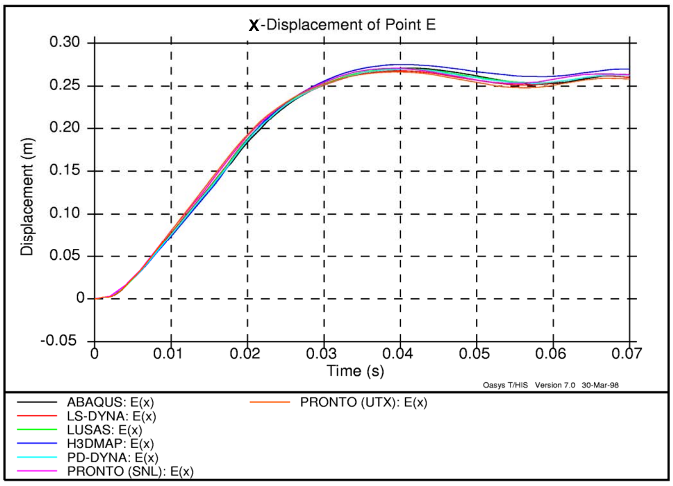

Figure 7: X-displacement of Point E

Results of Benchmark 2

Vertical displacements at Points A and B are shown in Figure 8 and 9.

Point A is on the top of the cube, and it would represent the displacement of the entire flask. With the exception of one analysis which predicted a displacement that is 8% lower than the average of the others, results from all the analyses agree well and lie within a maximum of 3% within each other.

Point B, on the base of the cube, is the first point to make contact with the rigid target, and the material around it is severely deformed during the impact. In terms of magnitude, the displacement time histories from the different analyses differed by about the same amounts as they did at point A. However, because the actual displacements were smaller, the percentage was larger. High frequency vibrations are seen in two of the analyses starting from about 20ms after impact, and in one of the analysis, the time history becomes very jagged afterwards. This may be caused by an hourglass vibration mode of the elements in the impact zone.

The time at which the cube losses contact with the target varied quite significantly between the analyses. The earliest was at 13ms, and the latest was at 21ms. Despite this variation, the rebound velocities agreed quite well.

Compression of the cube in the vertical direction is shown in Figure 10. Prediction from the analyses lie between 173mm and 176mm, i.e. a difference of less than 1% between them, except for the analysis which predicted a 8%

smaller displacement at point A, predicting a maximum compression of 158mm, i.e. about 9% less than the average of the rest.

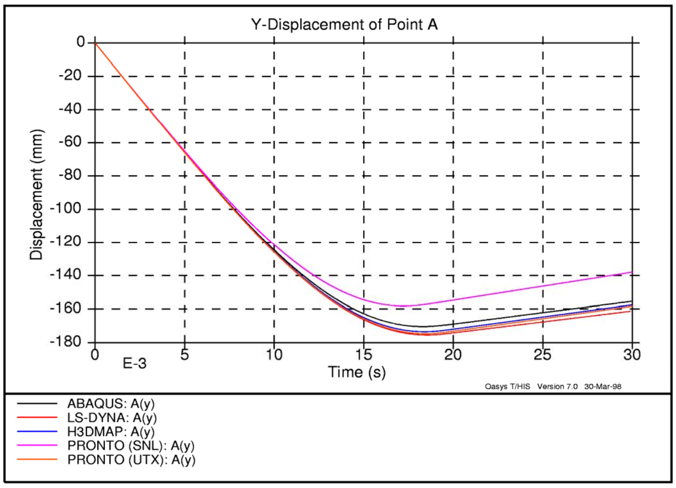

Figure 8: Vertical displacement of Point A

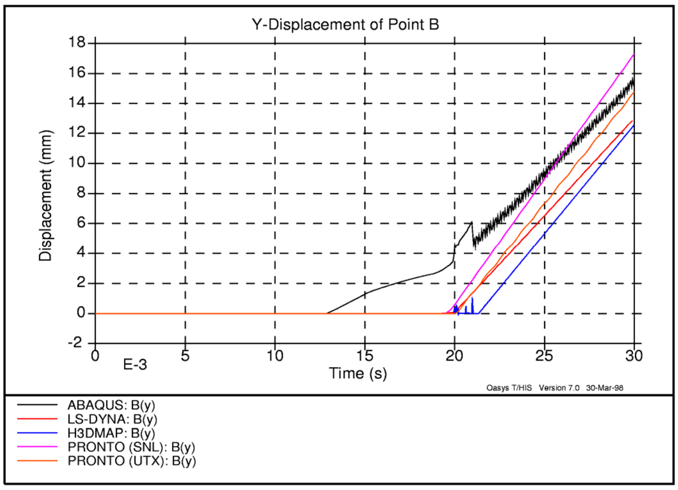

Figure 9: Vertical displacement of Point B

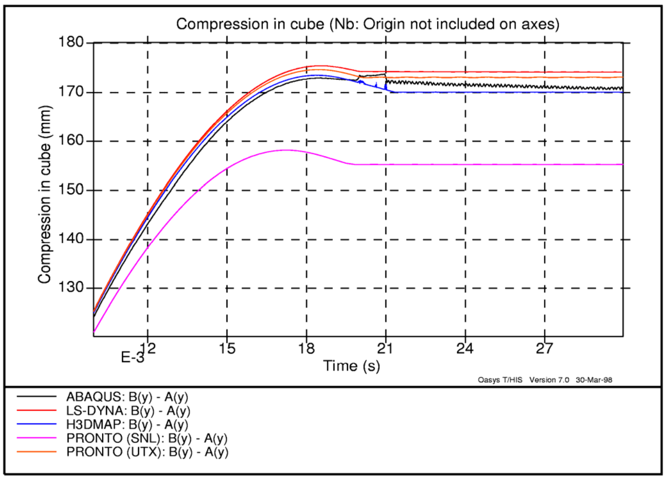

Figure 10: Compression of the cube

Results of Benchmark 3

Five organisations analysed this benchmark.

Displacement time histories of the rigid mass from the five analyses are shown in Figure 11. Until the rigid mass losses contact with the test specimen, the displacement time history of the rigid mass is equivalent to the compression time history of the test specimen. Maximum compression ranged between 25.5mm and 27.5m

Velocity time histories of the mass from the five analyses are shown in Figure 12. The beginning of the constant velocity part of the curves corresponds to the time when the rigid mass loses contact with the impact limiter. The time varied between the analyses, from 33ms, to 37ms. The rebound velocity varied quite significantly between the analyses – from 0.3m/s to 1.6m/s.

Contact force time history at the contact between the rigid mass and the test specimen is shown in Figure 13. While four of the analyses produced a smooth curve, one analysis produced a curve exhibiting high-frequency components. The curve from this analysis shown in Figure 13 had been filtered at 1000Hz. The overall curve shape from the different analyses is similar. Peak contact force lie between 660kN and 750kN.

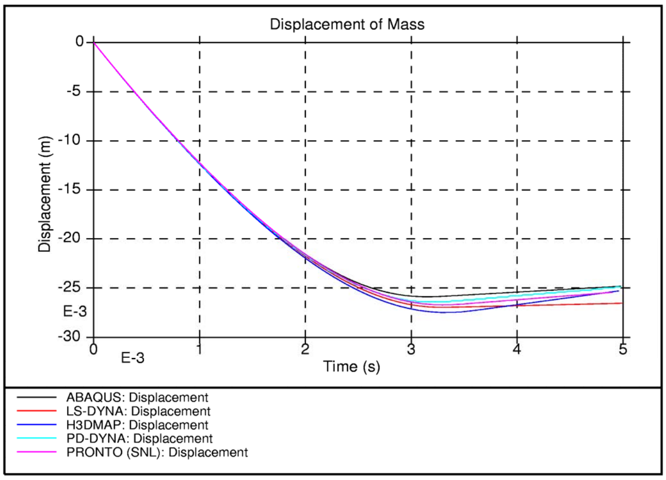

Figure 11: Displacement of the Mass

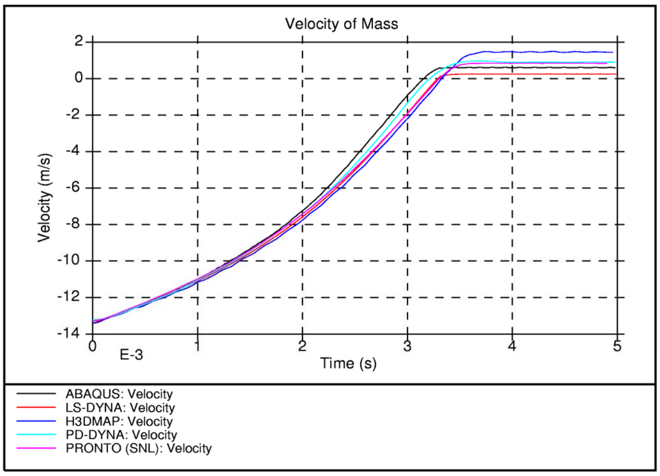

Figure 12: Velocity of the Mass

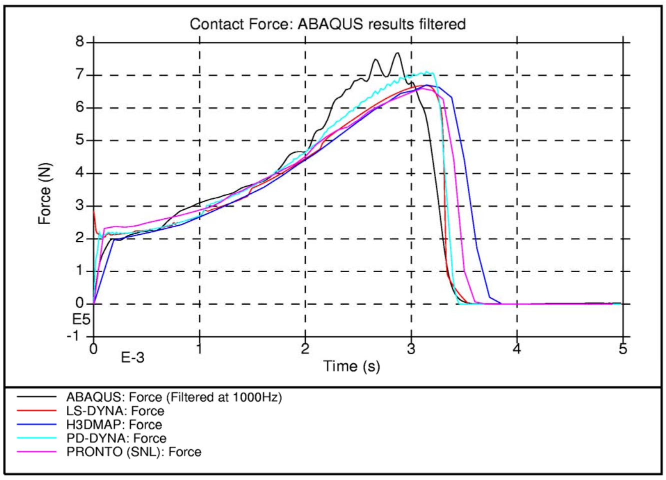

Figure 13: Contact Force (Note: ABAQUS results filtered)

DISCUSSION OF RESULTS

In general, results from the different codes-organisations agree extremely well. Although there are differences between the results, no single code-organisation stands out from the rest as being consistently different.

In interpreting the difference, it should be noted that the physical system represented by these benchmarks are sufficiently complex such that there is no practical method of working out a precise theoretical answer.

Reports from the participating organisations indicated that there were no problems in interpreting the benchmark problems. Hence the differences were not due to ambiguity in the physical characteristics or the definition of the benchmarks.

Mesh design, element type, code-specific modelling assumptions, analysis theory employed in the code and software methodology would all have contributed to the difference between the results. Analyst's technique, his judgement of adequacy of the results and his judgement of the required quality/accuracy of the analyses would also be important factors.

The differences in the results between the different codes-organisations should also be viewed in the context of scatter of experimental results – i.e. what scatter would there be in the results if the benchmark is tested physically instead of analysed. If physical testing of the benchmarks are carried out, and each benchmark is tested five time using five nominally identical samples, there will be scatter in the results. Slight variation in drop orientation, drop height, difference in geometry in the nominally identical samples, small non-uniformity in material properties, geometric imperfections, friction and material non-uniformities, are just some of the causes of scatter. The only way to determine the scatter would be to perform the drop tests. However, the scatter in the analyses results does not seem inconsistent with scatter would be seen in physical testing.

To illustrate the effect of mesh design on the results, Benchmark 1 was analysed in LS-DYNA with three different models which were identical except for the number of elements. The design of the mesh is summarised below.

	Elements through thickness of steel	Elements through thickness of lead	Element around circumference (in half model)	Total number of elements
Model 1	2	4	32	192
Model 2	4	8	64	768
Model 3	8	16	256	6144

All the elements through the thickness and around the circumference were evenly spaced in all three models. Full-integrated elements were used in all three models. The results from Model 2 and 3 agreed very well with a difference of less than 1%, while the results from Model 1 differed from the other two by up to 10%. This indicated, for Benchmark 1, the mesh in Model 2 was sufficient to produce a convergent solution. Generally, accuracy of an analysis improves with number of elements in the right positions. Purpose of an analysis is an important factor in determining the accuracy required.

CONCLUSIONS

A survey of existing finite element software was carried out, and ten codes were identified as having the potential to be used in drop test analysis of radioactive material transport containers.

A set of three benchmark problems were developed, each designed to test specific aspects of the software without requiring extensive use of computer resources. The benchmark problems represent three distinct categories of impact phenomena that occur during the drop testing of casks.

Seven organisations took part in an exercise to analyse the three benchmarks, using finite element software which they develop, distribute or use in-house. In general, there was good agreement in the results and no code consistently produced results which were significantly different from the rest.

Further work, including the comparison of FE analysis with different codes to real drop tests, would be beneficial needed in order to evaluate the software more quantitatively.

ACKNOWLEDGEMENT

The authors wish to acknowledge the effort and help of all the organisations who took part in the survey and the analyses of the benchmarks, especially Greg Morandin of Ontario Hydro Technologies, Canada and Doug Ammerman of Sandia National Laboratories, USA.

REFERENCES

[1] Ove Arup & Partners International and Gesellschaft für Nuklear-Behälter mbH, *Evaluation of Codes for Analysing the Drop Test Performance of Radioactive Material Containers – Phase 1 Final Report*, Report Ref 53276/02, Issue 1, March 1998