European Commission
DG-17

Evaluation of Codes for Analysing the Drop Test Performance of Radioactive Material Transport Containers

Phase 1 Final Report

ISSUE 1

Ove Arup & Partners International
and
Gesellschaft für Nuklear-Behälter mbH

European Commission DG-17

Evaluation of Codes for Analysing the Drop Test Performance of Radioactive Material Transport Containers

Phase 1 Final Report

March 1998

Report Ref 53276/02

Issue 1

Ove Arup & Partners International Ltd
Consulting Engineers
13 Fitzroy Street
London W1P 6BQ
UNITED KINGDOM
Telephone +44 (0)171 636 1531
Facsimile +44 (0)171 465 2150

Gesellschaft für Nuklear-Behälter mbH
Hollestrasse 7A
D-45127 Essen
GERMANY
Telephone +49 (0) 201 109 1706
Facsimile +49 (0) 201 109 1124

OAPIL Job number 53276; EC Contract No. 4.1020/D/96-016

ARUP

GNB
Gesellschaft für Nuklear-Behälter mbH

Document Verification

Job title	Evaluation of Codes for Analysing the Drop Test Performance of Radioactive Material Transport Containers
Job number	53276
Document title	Phase 1 Final Report

Prepared by	DJ Bernasconi (OAPIL)	
Signed	DJ Bernasconi	
Date	31 March 1998	
Checked by	CF Tso (OAPIL)	R Hüggenberg (GNB)
Signed	Chi Fung Tso	pp Chi Fung Tso
Date	31/3/98	31/3/98
Approved by	MR Willford (OAPIL)	D Methling (GNB)
Signed	M.R. Willford	pp M.R. Willford
Date	31/3/98	31/3/98

Revision record

Revision number	Date	Description	Prepared	Checked	Approved
Issue 1	31/3/98	Phase 1 Final Report	DJB	CFT/RH	MRW/DM

Job title	Evaluation of Codes for Analysing the Drop Test Performance of Radioactive Material Transport Containers	Job number	53276
Document Title	Phase 1 Final Report		

Revision	Date of issue	Description
Issue 1	31/3/98	Phase 1 Final Report.

CONTENTS

1. INTRODUCTION

This is the final report for Phase 1 of the project *Evaluation of Codes for Analysing the Drop Test Performance of Radioactive Material Transport Containers* for the European Commission DG-17 (Contract Ref no. 4.1020/D/96-016). This report supersedes the previous interim report [1].

The work was jointly undertaken by Ove Arup and Partners International (OAPIL) and Gesellschalft für Nuklear-Behälter mbH (GNB). OAPIL acted as the project leader.

The aims of the project were threefold:

- To conduct a survey of existing finite element software, with a view to finding codes that may be capable of analysing drop test performance of radioactive material transport containers, and to produce an inventory of them
- To develop a set of benchmark problems to evaluate software used for analysing the drop test performance of radioactive material transport containers
- To evaluate a number of finite element codes by testing them against the benchmarks.

These three tasks comprise Phase 1 of the project.

It is hoped that the project will promote harmonisation of the methods of safety evaluation of packages by furthering the acceptance of computer analysis methods among the national Competent Authorities which grant approval to transport packages under the IAEA Transport Regulations [2], and by making a standard approach to qualification of such codes available throughout the European Union.

2. BACKGROUND

Currently the demonstration of the drop test performance of a package is invariably carried out by means of physical drop testing, although the IAEA Transport Regulations [2] allow calculation methods which have been shown to be sufficiently reliable or conservative to be used. In the future it is likely that computer calculations will increasingly be used as evidence of drop test performance when seeking Competent Authority approval. This will have benefits for both Competent Authorities and the Applicants. The greater level of detailed information available from a computer analysis will allow a more thorough assessment of the package by the Competent Authorities. Furthermore it will be easier to identify the worst impact attitude (as required by the Regulations) using computer techniques, so that it will be possible to confirm that the required level of safety has been achieved. For the Applicant, computer analysis will have significant time and cost saving benefits.

For a computer code to be acceptable it must ideally be shown to be sufficiently reliable in all relevant cases; or if this is not possible, it must instead be shown to be reliably pessimistic in all relevant cases. The term "sufficiently reliable" in this context means that the code should be capable of producing results which are within the band of experimental scatter which might be obtained from physical drop testing. Such a code could be said to be as reliable as physical drop testing. Even if a code is not capable of producing results within the experimental scatter band, it would be acceptable if the predictions of damage are reliably pessimistic. By careful choice of material parameters it is possible to ensure pessimistic results.

What is lacking at the present, however, is a standardised method of assessing a computer code in order to determine whether it is sufficiently reliable or pessimistic. This project fulfils this need by producing standard benchmark problems that may be used for this purpose. By making the standard problems available to Competent Authorities throughout the European Union, greater harmony in the method of assessing the safety of packages will be achieved.

Benchmarking exercises of this type have been previously carried out for Thermal and Criticality codes. However, there has been only limited benchmarking for impact analysis using finite element (FE) codes, many of which are no longer in commercial use. This project rectifies this imbalance.

3. TASK 1: FE CODE SURVEY

The aim of this task was to conduct a survey of existing finite element software, with a view to identifying codes that may be capable of analysing the drop test performance of radioactive material transport containers. This means that they should capable of analysing dynamic events with non-linear geometry and non-linear material models. The following codes which fulfilled these basic requirements were identified:

- ABAQUS/Explicit *(Hibbitt, Karlsson & Sorensen Inc.)*
- ANSYS *(ANSYS Inc.)*
- DIANA *(Analysis BV)*
- DYNA3D *(LLNL, available in the Public Domain)*
- H3DMAP *(Ontario Hydro Technologies)*
- LS-DYNA3D *(Livermore Software Technology Corporation)*
- LUSAS *(FEA Ltd.)*
- NIKE3D *(Available in the Public Domain)*
- PRONTO2D/3D *(Sandia National Laboratories)*
- SOLVIA *(Solvia Engineering)*

Information on these codes was provided by the respective developers or distributors, and full results of the survey including a summary of the software's capabilities and details of maintenance and quality assurance procedures is given in Appendix A.

4. TASK 2: DEVELOPMENT OF SIMPLE BENCHMARK PROBLEMS

The aim of this task was to develop a set of benchmark problems that test the software's ability to model pertinent physical phenomena without requiring the extensive use of computer resources. Accordingly, three benchmark problems were formulated to represent three distinct categories of impact phenomena that occur during cask impact. They are geometrically simple, do not require complex or intricate modelling, and require only short analysis times.

To ensure that the benchmark results from different software developers or distributors would be comparable, all the essential modelling parameters including material details, geometric and physical details, boundary and interface conditions, initial conditions, and output requirements were defined. However, the benchmarks do not include specific instructions for discretisation of the given geometry into a finite element mesh. This was left to the judgement of the participants, because it may involve both engineering considerations and the requirements of the individual code. The effects of this are discussed in Section 5.5. Full specifications of the benchmark problems are given in Appendix B.

The following sections provide a description of the benchmarks, and a discussion of the aspects of the software that each benchmark is designed to test.

4.1 Benchmark 1: Flat side impact of concentric cylinders

Benchmark 1 represents the flat-side impact of a cylindrical cask comprised of an outer steel cylinder and an inner lead cylinder. This model represents a typical cylindrical transport or storage cask, such as the *Excellox* cask. In the benchmark, the geometry has been idealised as a 2-D plane strain problem in order to reduce complexity.

The aim of this benchmark is to test the software capability in representing:

- 2-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.

During the development of the benchmark, various configurations of geometry, impact velocity and friction coefficient were considered, with the objective of designing a benchmark that could demonstrate the desired effects most clearly. Appendix B gives details of the final benchmark, which involves an assumed drop height of 30m.

The problem could be analysed either in 3D using plane strain boundary conditions, or in 2D using plane strain elements. It would be possible to reduce computation time by analysing a half-model only, using symmetric boundary conditions on the vertical cut plane.

4.2 Benchmark 2: Corner impact of a cube

The aim of Benchmark 2 is to test the capability of the software in modelling large plastic deformation or 'solid metal flow', which is a common mode of deformation in, for example, integral shock absorbers of cuboidal casks.

The benchmark is intended to simulate the impact of a 50 tonne cuboidal cask on to one corner. For geometric simplicity, the corner is represented by a uniform cube of deformable steel. The full weight of the cask is applied by three rigid surfaces covering the upper faces of the cube, which is sufficiently large that the boundary condition does not significantly affect the results. The impact velocity corresponds to a drop height of approximately 9m.

This benchmark tests the software capability in representing:

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

This problem can only be analysed using a full 3D finite element code.

4.3 Benchmark 3: Impact of a wooden cylinder with steel cladding (GNB)

Benchmark 3 represents a regulatory 9m drop test [2] for a transport cask which is equipped with a wooden shock absorber. The deformable wooden shock absorber is intended to ensure that the loadings (stresses and strains) at the cask will be reduced so that the cask will maintain its safety function.

The model for Benchmark 3 consists of a solid wood cylinder, which is surrounded at the cylindrical surface by a thin steel plate. For the purposes of the benchmark, this structure is placed on the unyielding target surface and is struck by a falling rigid body with a velocity corresponds to a drop height of approximately 9m. The geometry of the wood sample (100mm diameter, 50mm height, 1mm liner thickness) was selected to correspond to an actual experimental test. The mass of the impact body was chosen to be 100kg, which leads to an equivalent area load which typical for such a shock absorber loading. In order to simplify the material characterisation for wood, it was assumed that the wood has ideal plastic behaviour.

The aim of this benchmark is to test the software capability in representing:

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

5. TASK 3: FE CODE EVALUATION

The aim of Task 3 was to evaluate the finite element codes identified in Task 1 against the benchmarks derived in Task 2.

5.1 Codes evaluated

The analyses of the benchmarks with LS-DYNA3D and the version of DYNA3D available in the public domain were carried out in-house by OAPIL and GNB respectively. The analyses of the benchmarks with other codes was carried out by developers or distributers who agreed to take part in the benchmarking exercise. These organisations are listed in the following table, in which the ticks show which benchmarks were analysed by which organisations. Where an organisation did not analyse all three benchmarks, it was because of a lack of resources rather than an inability of the respective code to analyse the particular benchmark.

Code	Analysis Organisation	Abbreviation	Benchmarks Analysed		
			1	2	3
ABAQUS/Explicit	Hibbitt, Karlsson & Sorensen Inc. (User / Distributor)	ABAQUS	✓	✓	✓
LS-DYNA3D	OAPIL (User / Distributor)	LS-DYNA	✓	✓	✓
LUSAS	FEA Ltd. (User / Developer)	LUSAS	✓		
H3DMAP	Ontario Hydro Technologies (User / Developer)	H3DMAP	✓	✓	✓
DYNA3D (Public Domain)	GNB (User)	PD-DYNA	✓		✓
PRONTO3D	Sandia National Laboratories (User / Developer)	PRONTO (SNL)	✓	✓	✓
PRONTO3D	University of Texas (User)	PRONTO (UTX)	✓	✓	

Although NIKE3D is available in-house at OAPIL, it was not evaluated because it is solely an implicit code, impractical for the analysis of large-deformation, highly-dynamic impact events such as these. The organisations mentioned in the code survey (Task 1) but not listed above either declined to take part in the exercise, or did not have sufficient resources available in the required timescale.

The following three sections give details of the benchmark results obtained by the various finite element codes. Following these results, a discussion of the differences between the results is presented.

5.2 Results of Benchmark 1

Figures 5.1, 5.2 and 5.3 show the displacement histories of points A to F on the concentric cylinder model, as calculated by the seven organisations that took part in the exercise. The results of y-displacement at points A and B are similar: maximum displacements differ by about 9% (70mm) in the worst case. These curves correspond to points at the top of the two concentric cylinders, where overall displacements are relatively high.

The scatter of results for points C and D at the bottom of the cylinders appears to be much greater. However, at these points the actual displacements are smaller than at A and B, and the spread of results is only about 3mm in the worst case. Until the time of rebound, displacements at these points are caused by concave curvature of the cylinders at the initial point of impact, as shown in Figure D.5. ABAQUS and PRONTO (SNL) predict the greatest and least curvatures respectively, with corresponding displacements of 50mm and 18mm at point C on the outer steel cylinder at 0.04s after initial contact. Results from the other finite element codes fall within this band. Despite these differences, all the codes predict very similar initial rebound velocities, indicated by the gradients of the curves at approximately 0.06s after initial contact.

The curves showing the x-displacements of points E and F are the most consistent of the set. Here, maximum displacements differ by only 2% (12mm). At these points, the two cylinders remain in contact with each others for most of the analysis time, so the displacements being measured are for a combined, thick-walled cylinder rather than for either of two thinner walled ones. This may account for greater similarity between the results.

5.3 Results of Benchmark 2

Figures 5.4 and 5.5 show the y-displacement histories of points A and B on the cube model, as calculated by the five organisations that took part in the exercise.

Point A is on the top of the cube, most distant from the impact zone, and would also represent a general point on the main body of the cuboidal flask. For this point, the displacement curves as calculated by four of the five analyses are very similar: the maximum displacements differ by only 3% (6mm). The PRONTO (SNL) analysis predicts a maximum displacement of 15mm which is 8% less than the average of the other four.

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. The displacement curves corresponding to this point differ by about the same amounts as they do for point A in magnitude, but the actual displacements being measured are much smaller. For example, ABAQUS predicts that the displacement of point B at 0.025s after impact is 10mm, whereas H3DMAP predicts it to be approximately 5mm.

The ABAQUS and the H3DMAP curves for point B both show a fluctuation at approximately 0.02s after impact. Following this fluctuation, the ABAQUS curve becomes very jagged, indicating high frequency vibration of point B. This may be caused by an hourglass vibration mode of the elements in the impact zone (discussed further in Section 5.5.2).

The time of rebound, or loss of contact between the cube and the target, varies quite significantly between the codes. The shortest impact duration of 0.013s is predicted by ABAQUS, the longest of 0.021s by H3DMAP. However, despite this variation in contact times, the predicted rebound velocities are generally quite similar. This can be seen in Figure

5.5 which shows the velocity histories obtained by differentiating[1] the displacement histories at point A. Once again, the PRONTO (SNL) curve differs from the rest and predicts a slightly higher rebound velocity, indicating that less energy (approximately 98.3% as opposed to 98.8% in the other analyses) has been absorbed in the impact.

Figure 5.5 also shows the subtraction B(y) - A(y) histories for the five analyses, equivalent to the compression in the steel cube. The peak compression, which occurs approximately 0.018s after initial impact in all cases, ranges between 173mm and 176mm for four of the five analyses, and is 158mm for the PRONTO (SNL) analysis. All of the codes then predict some elastic rebound, although this varies between 1.2mm for LS-DYNA and 3.7mm for H3DMAP.

In general, the overall compressed dimension of the cube remains constant after rebound. However, the curve for ABAQUS predicts a gradual reduction in compression for several milliseconds after this. This could not represent a real effect (the fundamental vibration mode of the cube has a time period of the order 1ms), and could be a side-effect of the possible hourglassing described above and discussed in Section 5.5.2.

5.4 Results of Benchmark 3

Figure 5.6 shows the displacement histories of the rigid mass, as calculated by the four organisations that took part in the exercise. The wooden cylinder, originally 50mm tall, is compressed by about 27mm in the H3DMAP analysis, and by about 25mm in the ABAQUS analysis. The other codes predict amounts of compression within this range.

Figure 5.6 also shows the velocity histories of the rigid mass, obtained by differentiating the displacement histories. The time of rebound, when the rigid mass loses contact with the wooden cylinder and steel annulus, is indicated by the beginning of the flat constant velocity part of the curves. Rebound time ranges between 0.033s and 0.037s, as predicted by ABAQUS and LS-DYNA respectively. The rebound velocity varies significantly between the codes: the highest value of 1.6m/s is predicted by H3DMAP; the lowest value of 0.3m/s is predicted by LS-DYNA. This means that in the H3DMAP analysis, 98.5% of the initial kinetic energy of the mass is absorbed in the impact by plastic deformation, whereas LS-DYNA predicts the energy absorbed to be 99.9%.

The top half of Figure 5.7 shows force histories for the contact between the rigid mass and the test specimen. The ABAQUS curve is noticeably quite different from the rest: whereas the general trend is a smooth curve rising to a peak approximately 3.2ms after impact, the ABAQUS curve is very 'spiky' and exhibits high-frequency components.

Figure 5.7 also shows a second plot of the force histories for the contact surface, this time with the ABAQUS curve filtered at 1000Hz. The overall curve shapes are similar, with the peak contact force ranging between 660kN for PRONTO (SNL), and 750kN for ABAQUS.

5.5 Discussion

5.5.1 Software trends

Although the various finite element codes produced results which are measurably different from one another, the three benchmarks do not clearly separate any of them from the rest. For example, in Benchmark 1 three of the codes give very similar displacement histories for point A, with H3DMAP predicting slightly higher displacements. However, when comparing displacement histories for point C, the H3DMAP curve follows the average of the other curves,

[1] The velocity curve obtained from the ABAQUS analysis has been filtered to remove the high-frequency vibration.

with ABAQUS and LS-DYNA producing the extremes of the range. In Benchmark 2, PRONTO (SNL) gives results significantly different from the other analyses (including a second analysis also carried out with PRONTO, by the University of Texas). However, the PRONTO (SNL) analyses have produced results close to the average for the other two benchmarks.

If one or two of codes had produced results that were consistently different from the rest, it might have been clear which ones were accurate, and which were not. However, that is not the case, and in the absence of definitive analytical results it is much more difficult to determine whether any of the codes are performing better than the others.

The following sections discuss possible sources of errors within the various analyses, the significance of the differences between the results, and the accuracy of the codes in the context of predicting experimental results.

5.5.2 Analysis assumptions

The benchmark problems were specified such that the impact scenarios they described would be unambiguous in terms of geometry, boundary conditions and material properties. Reports from the participating organisations indicated that there were no problems in interpreting the exercises, which confirms that the results presented above all correspond to analyses representing the same physical situations. The results, in general, show extremely good agreement. However, there are some differences between the results obtained, and it is important to understand where these differences arise from.

In all of the benchmark cases, for all of the results, there is in theory a single 'correct' answer. The benchmarks were specified in terms of idealised material data, rather than by providing, for example, experimental tensile test data. Thus, if all of the analyses were perfect, they would all produce identical results. Any deviations from the theoretically correct answers can only be due to differences in the code-specific modelling assumptions or engineering judgement of the analyst, or the analysis theory and software methodology. It is interesting that in Benchmark 2, two analyses which were carried out using the same software but by different organisations produced noticeably different results. This discrepancy can only be due to differences in the analysts' modelling assumptions or engineering judgement.

The design of a finite element mesh is crucial to the accuracy of an analysis. The mesh density study carried out for Benchmark 1 using LS-DYNA (see Appendix C) found that in this case a mesh with only 2 elements through the steel thickness, and 4 through the lead thickness, was not adequate; however, double this density was sufficient for a convergent solution. In certain cases a coarse mesh can be compensated for by the use of fully integrated elements, but at the expense of computation time. The table below shows the number and type of elements used for the various analyses:

Code	Integrat. Scheme	Through Steel	Around Steel*	Through Lead	Around Lead*	Total*
Req'd for LS-DYNA	Full	4	128	8	128	768
ABAQUS	Reduced	4	80	4	160	480
H3DMAP	Unknown	3	180	6	180	810
LUSAS	Unknown	Unknown	Unknown	Unknown	Unknown	Unknown
PD-DYNA	Unknown	Unknown	Unknown	Unknown	Unknown	Unknown

PRONTO (SNL)	Reduced	5	100	5	150	625
PRONTO (UTX)	Unknown	Unknown	Unknown	Unknown	Unknown	Unknown

*For a half-model

Because the number, type and distribution of elements in the models varied significantly, a study was carried out to determine whether the design of the finite element meshes could be solely responsible for the differences in results. An additional LS-DYNA analysis was carried out by OAPIL using a mesh identical to the one used for the ABAQUS analysis, as shown in the Figure 5.8, and employing first order reduced integration elements to match the ABAQUS analysis. Although the 2D plane strain conditions used in ABAQUS are not possible in LS-DYNA, they were simulated by restraining all nodes from moving out of plane.

Figure 5.8 also shows the displacement histories at point C for the two analyses, and the corresponding displacement history from the LS-DYNA analysis with 6144 elements. This measurement point was chosen because it was one at which variation between codes was greatest. It can be seen that the LS-DYNA analysis with the ABAQUS mesh, rather than giving results similar to the ABAQUS analysis, has produced a displacement history more similar to the LS-DYNA analysis with the fine mesh. This shows that in this case at least, there are more significant causes of differences than mesh density and element formulation.

Without a more detailed investigation into the various codes it is not possible to determine the other factors that could have caused the differences between the results. One likely contributor is the ‘hourglass control’ mechanism. When using reduced integration elements, for which stresses are calculated only at the elements’ centres, ‘hourglass’ deformation modes involving zero internal strain energy can occur. Most finite element codes have systems for reducing the effect of such hourglassing modes, but these systems are likely to differ between the different codes. Such differences could account for variations between the results.

There are similar differences between the results in Benchmarks 2 and 3, and again it is not possible to determine from where these differences arise. For Benchmark 3, ABAQUS made use of axisymmetric elements, whereas the other codes used 3D models, but this should not have significantly affected the results. Variations between the finite element meshes could account for some of the differences, but it is unlikely that this was the only factor.

5.5.3 Application to prediction of experimental results

The benchmark exercises have shown that there is some variation in results produced by different finite element codes, even though they appear to be analysing identical physical situations. In this section, the significance of these differences is discussed in the context of using computer analysis to predict results subject to experimental scatter.

Although the concept of an ‘ideal solution’ was introduced in the previous section, it is unlikely that this theoretical result would be obtained in experiment: real materials are not uniform, they cannot be described by idealised bi-linear models, and manufactured test pieces will be flawed with imperfections. Also, in the case of radioactive material transport container drop tests, experimental conditions would not be completely repeatable: the impact velocity and orientation of the container would be subject to slight variations.

If experiments representing the benchmark problems were carried out, the results would exhibit scatter. The only way to determine the actual degree of this scatter would be to perform several real drop tests, with a different container each time, but in the absence of such experimental results it is still possible to give an indication of the order of magnitude of scatter that would be observed.

Some of the variables causing experimental scatter would be the material properties. In order to estimate the effect of variation in the material properties on the benchmark results, two additional LS-DYNA analyses of Benchmark 1 were carried out by OAPIL. In the first, the yield stress and hardening modulus of both the steel and the lead were increased by 10%; in the second these parameters were reduced by 10%.

Figure 5.9 shows the y-displacement histories produced by these extra analyses for points B and D. Also shown are the benchmark results from the various codes. At point B, where displacements are large and representative of overall container behaviour, it can be seen that all the benchmark results fall within the 'simulated experimental scatter band' produced by the LS-DYNA analyses. However, at point D, where displacements are much smaller and controlled more by local container behaviour, some of the results fall outside of the scatter band.

A full study of experimental scatter would involve investigating the effects of many parameters, for example geometric imperfections, friction and material non-uniformities. However, this simple exercise has demonstrated a probable trend:

- Global effects are predicted with reasonable consistency; results from all the codes are likely to lie within the experimental scatter
- Local effects are predicted with less consistency; different codes produce greater variations in the results; accurate prediction of experimental data is less certain

To establish the correlation between experimental and computer results more quantitatively, for both global and local parameters, further investigation would be required. In particular, comparison with reliable drop test data would be very valuable.

6. CONCLUSIONS

A survey of existing finite element software was carried out, and ten codes were identified as having the potential to be usable for drop test analysis of radioactive material transport containers. Summary tables comparing the codes were produced, and are included in Appendix A.

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.

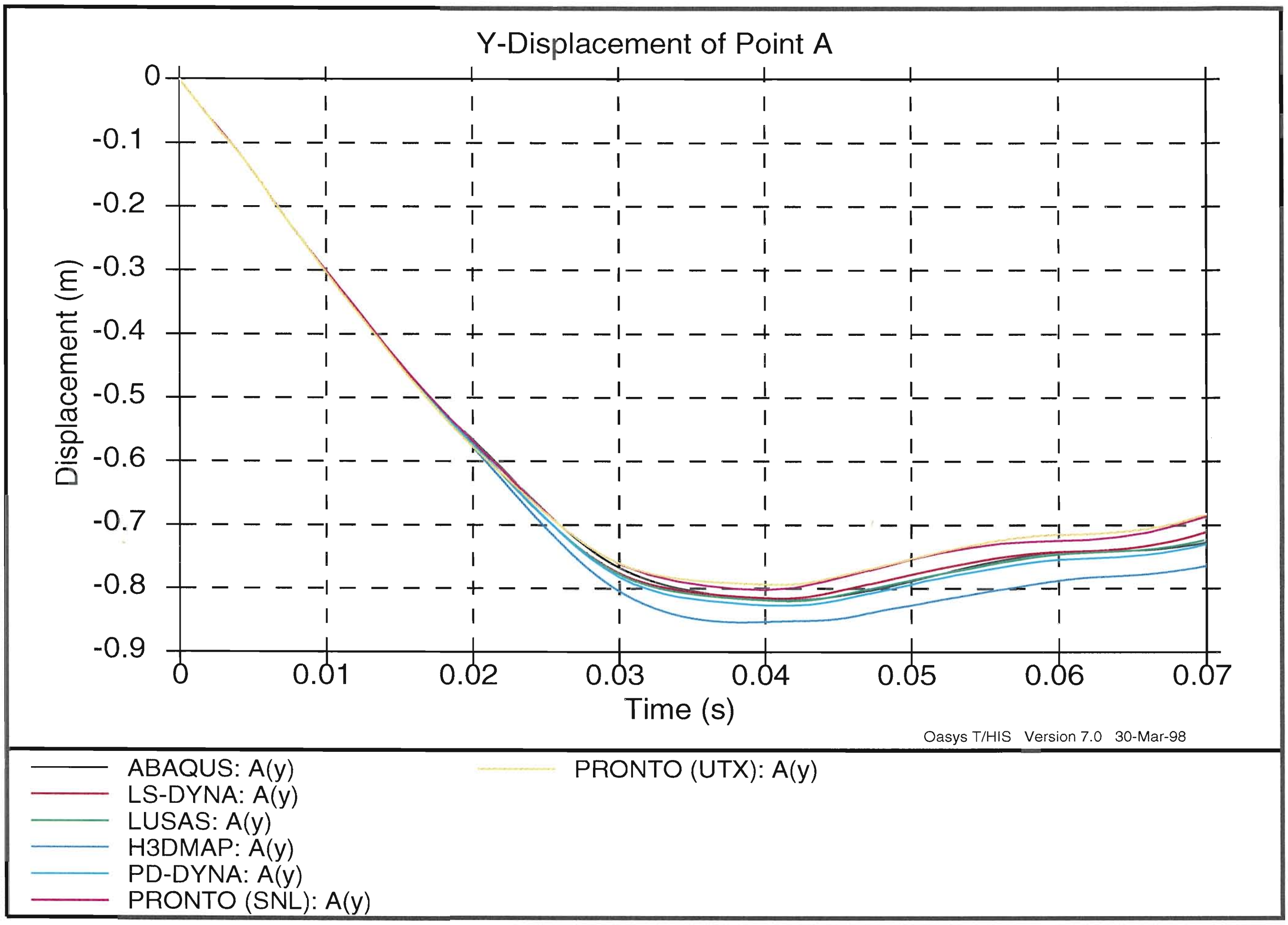
Seven organisations took part in an exercise to analyse the three benchmarks, using the finite element software which they develop, distribute or use in-house. The results from the various analyses were compared, and it was found that no one code consistently produced results which were significantly different from the rest. An investigation into the differences that were present concluded that in terms of global behaviour, the various codes produced results that were in agreement with each other, within reasonable experimental error. However, detailed local behaviour was predicted with less consistency.

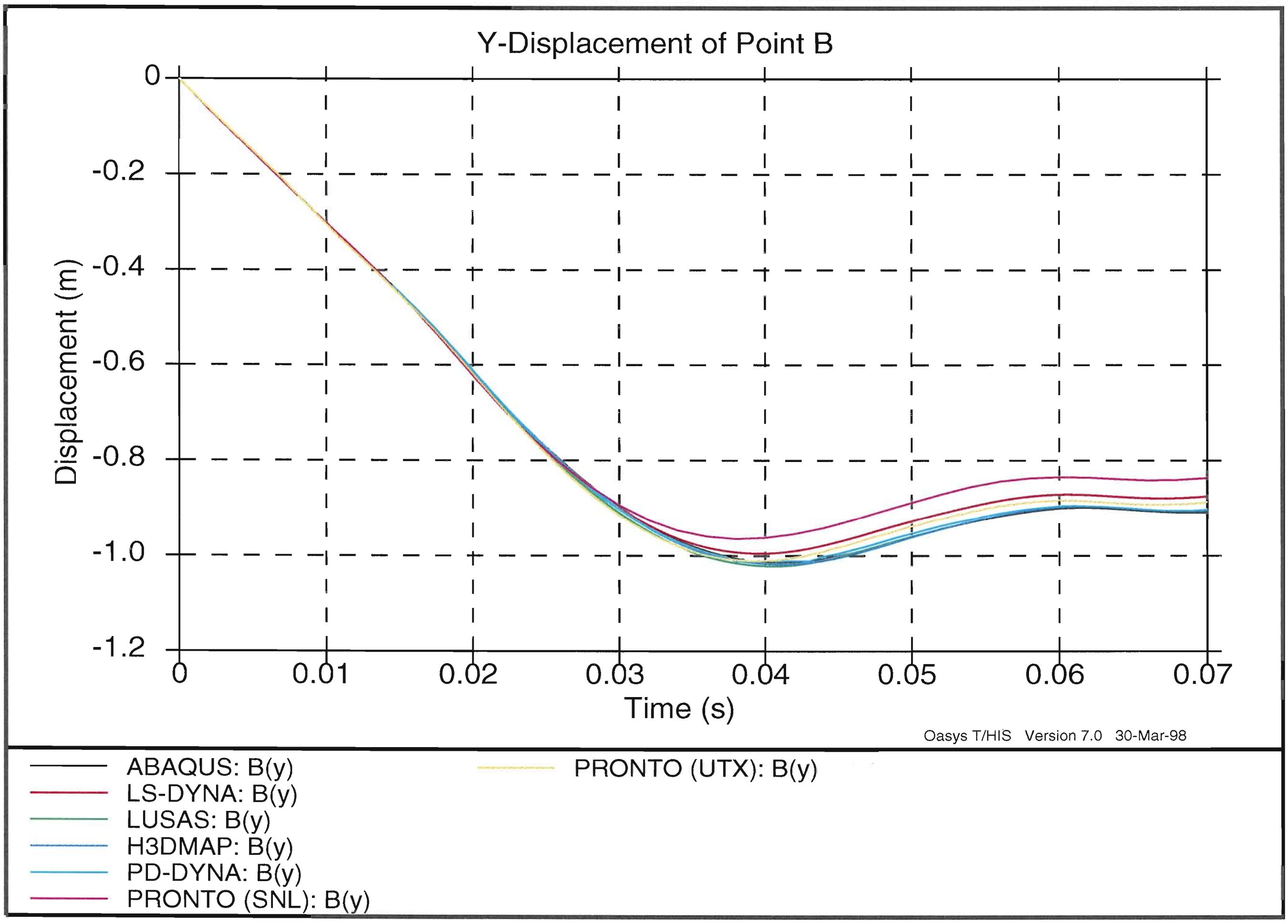
Further work, including the comparison of finite element analysis to real drop tests, would be needed in order to evaluate the software more quantitatively.

7. REFERENCES

[1] Ove Arup & Partners International, *Evaluation of Codes for Analysing the Drop Test Performance of Radioactive Material Transport Containers*, Ref No. C3/TMR/96,110, June 1996.

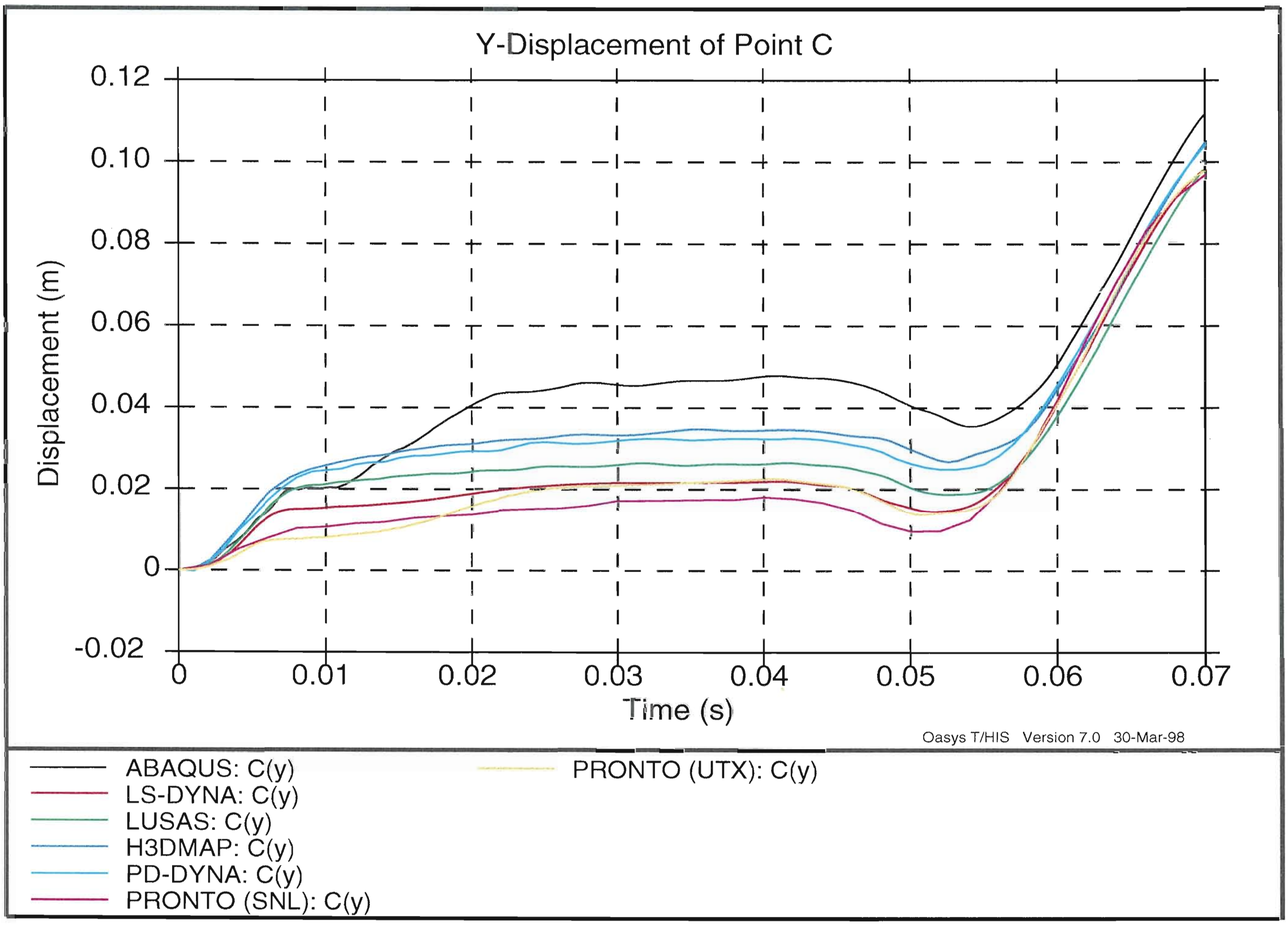
[2] International Atomic Energy Agency, *Regulations for the Safe Transport of Radioactive Material, 1996 Edition*. IAEA Safety Standards ST-1, Vienna 1996.

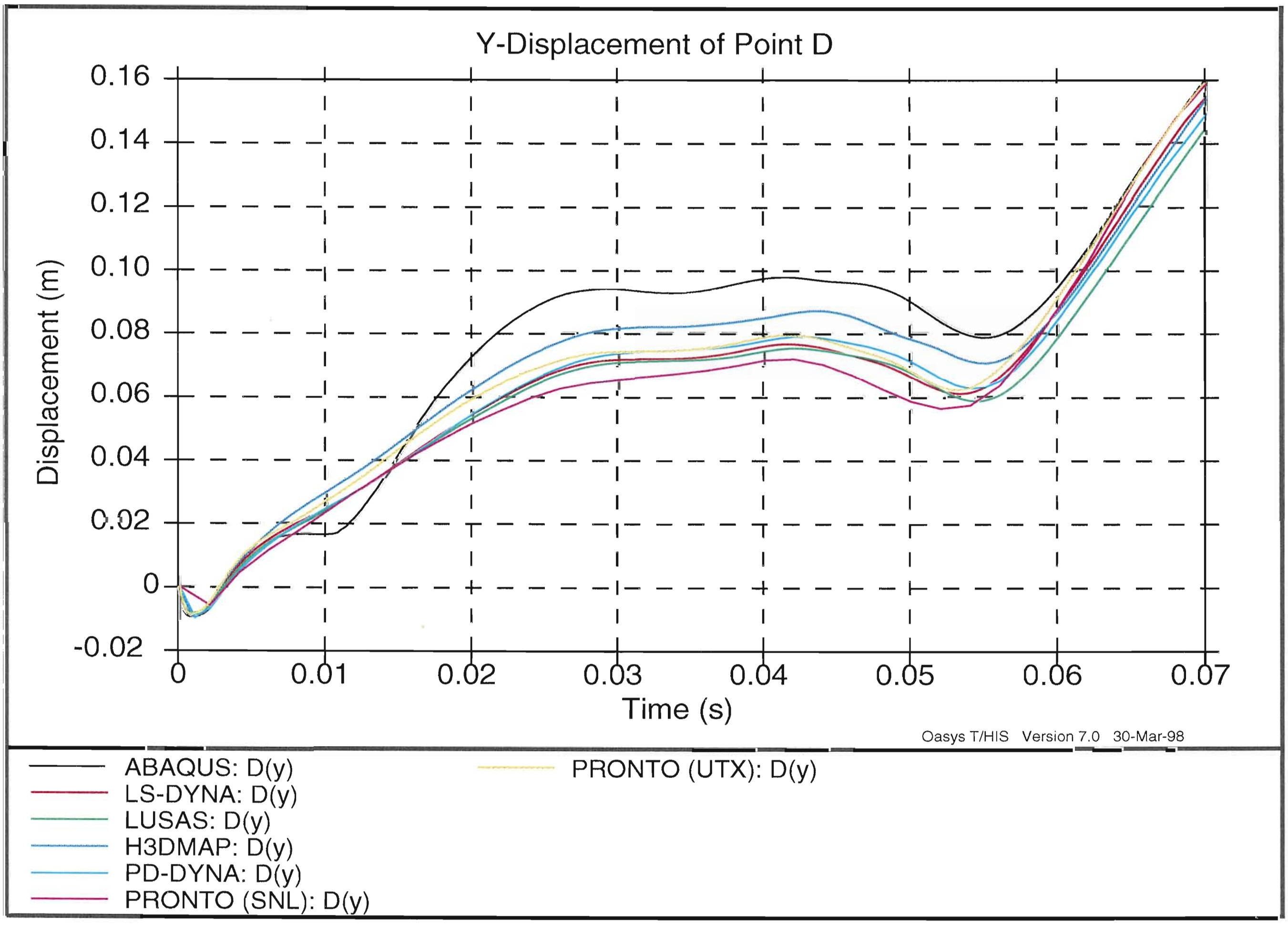




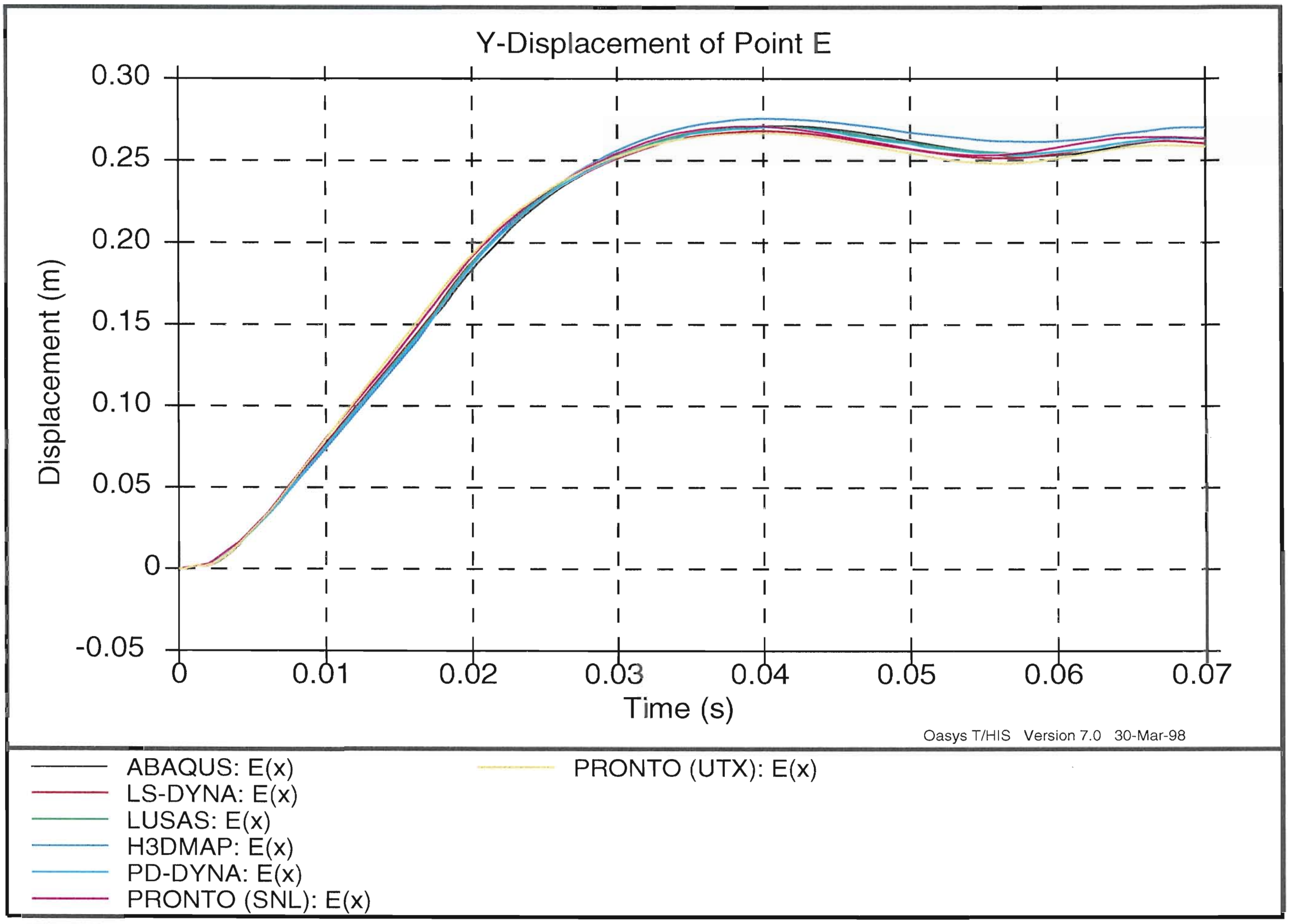
Benchmark 1: Results for Points A & B

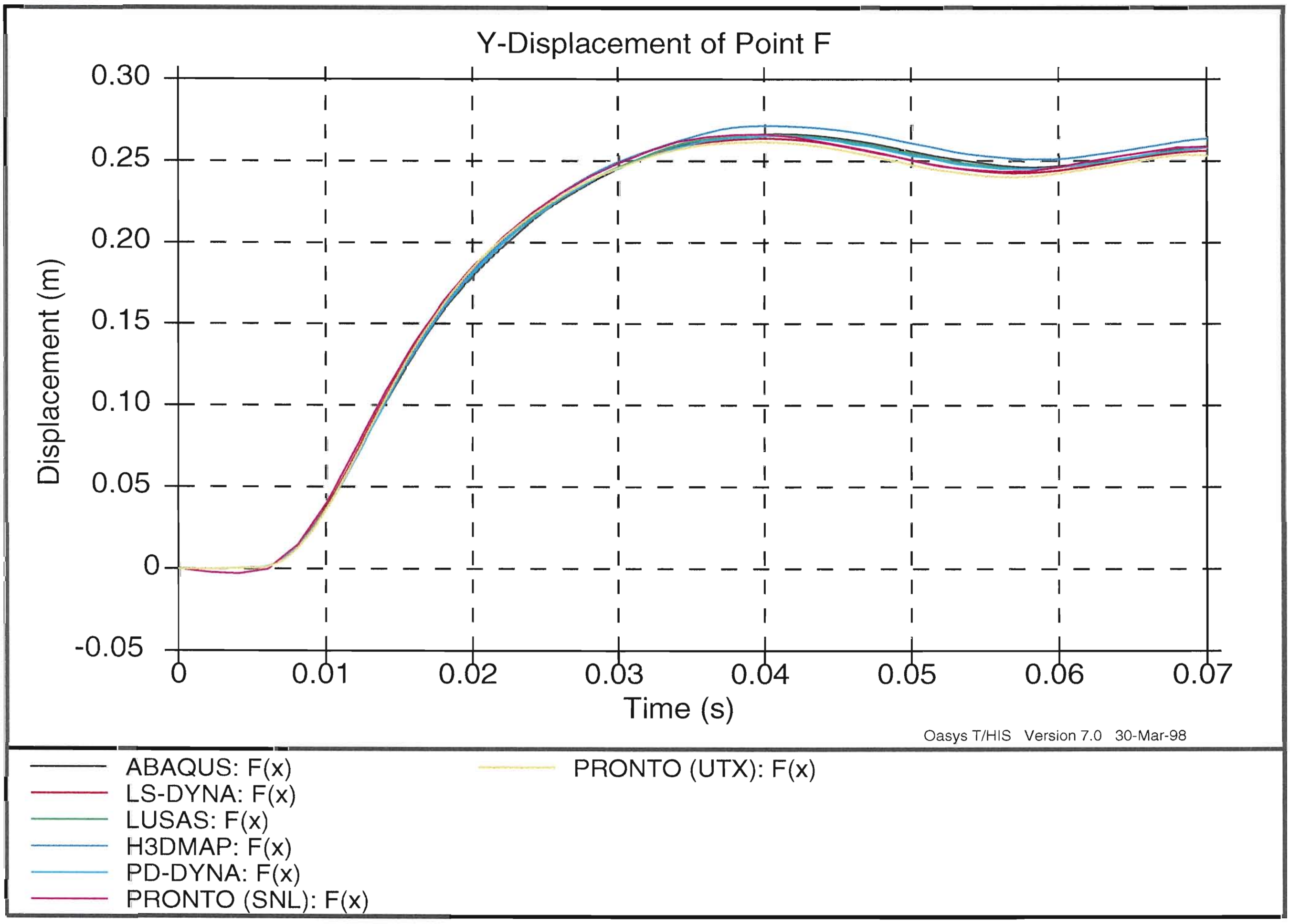
figure **5.1**



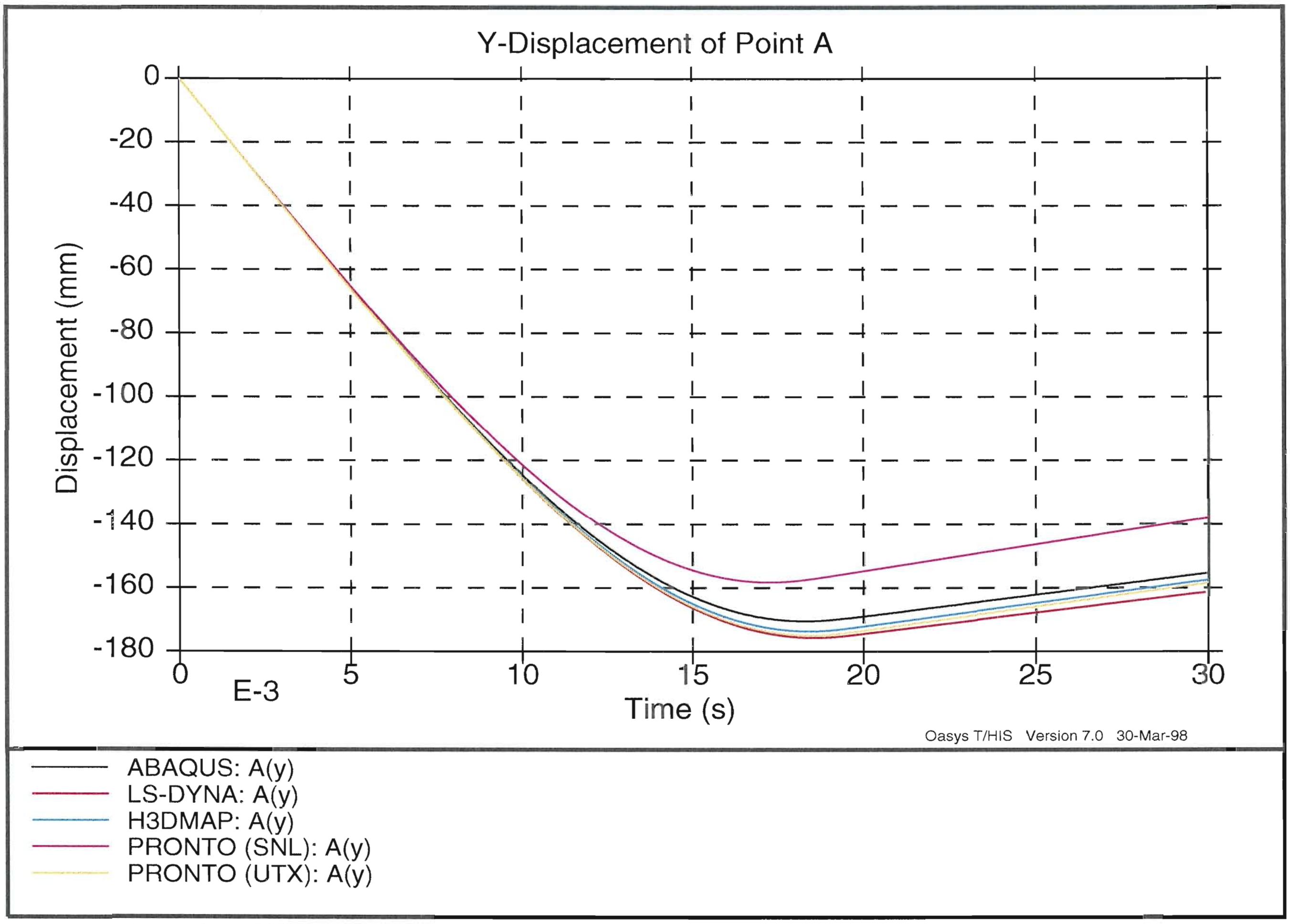


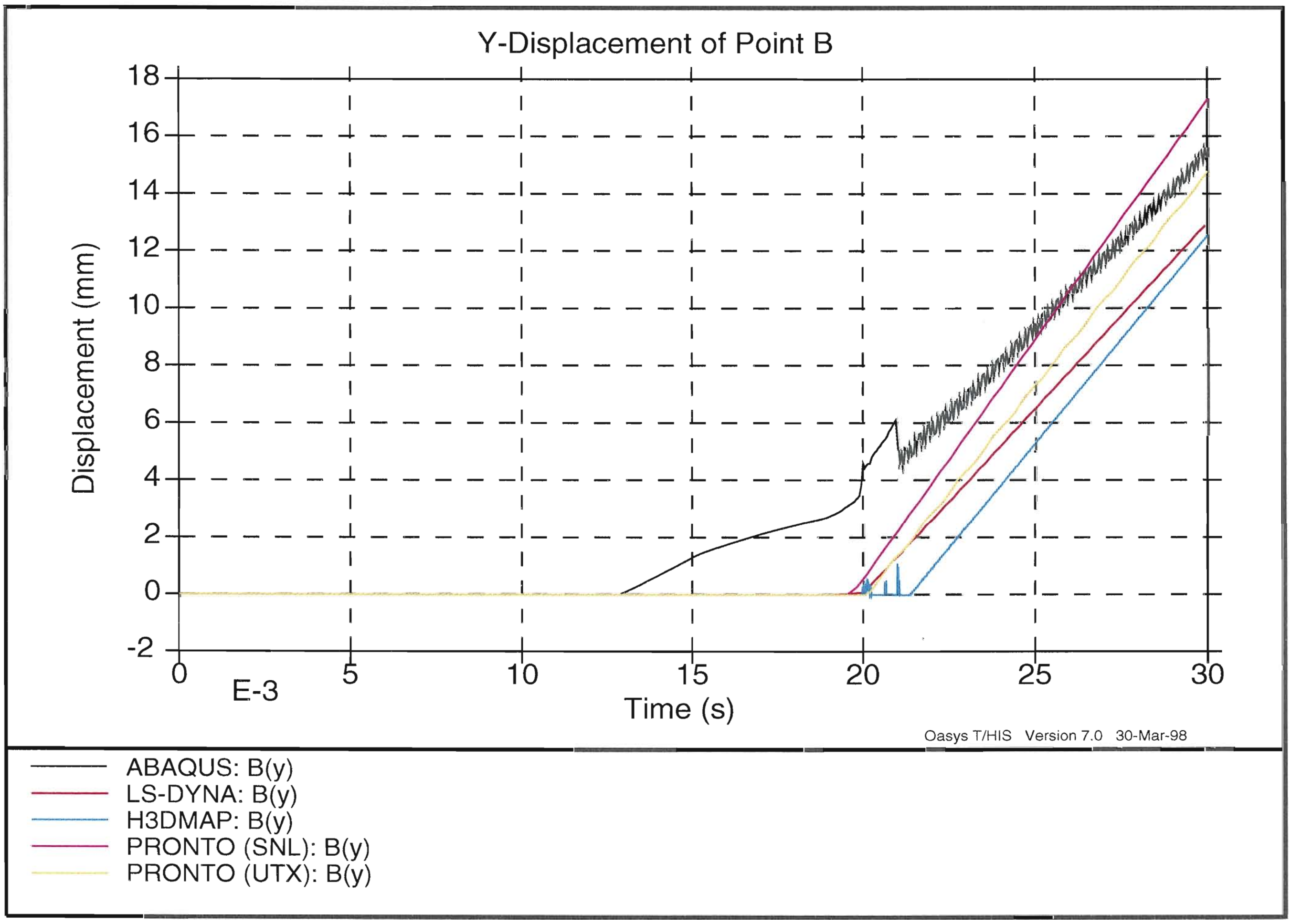
Benchmark 1: Results for Points C & D figure **5.2**





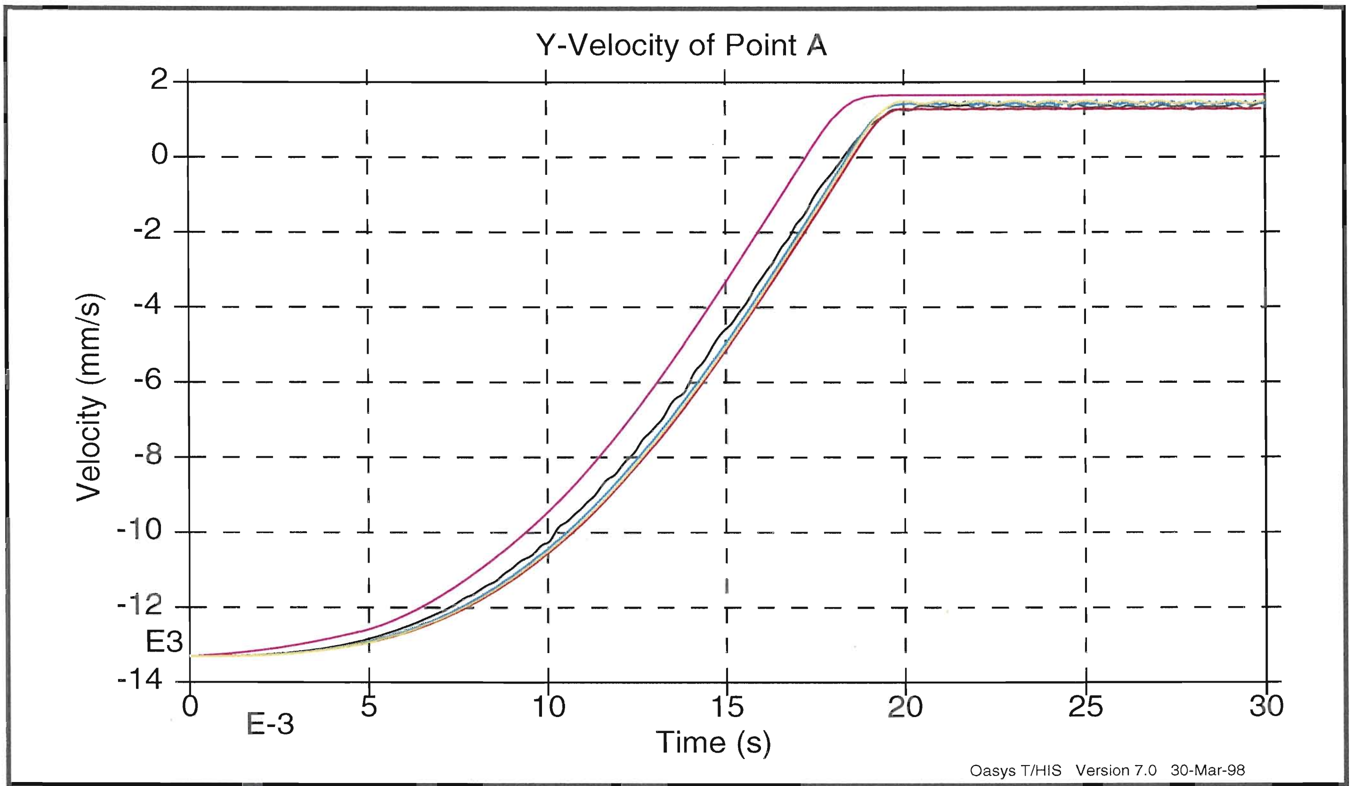
Benchmark 1: Results for Points E & F figure **5.3**



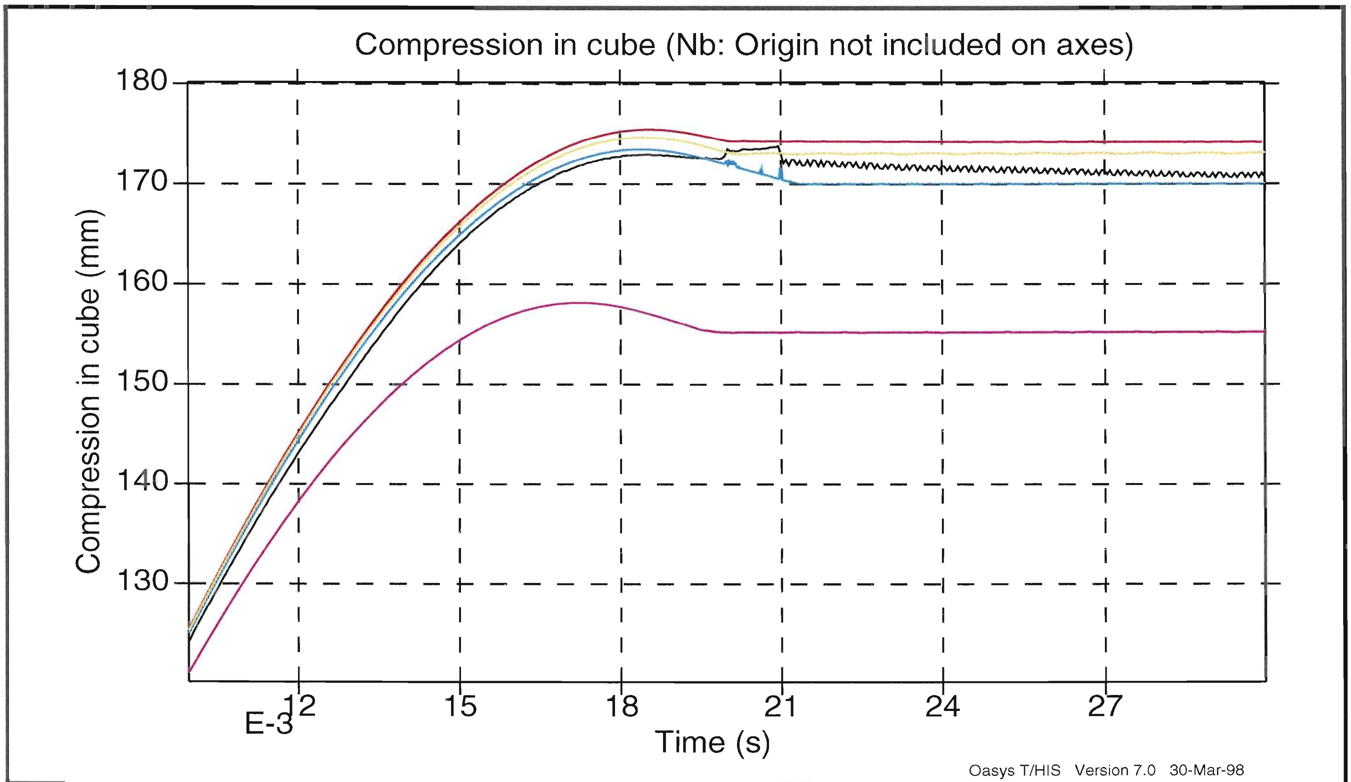


Benchmark 2: Results for Points A & B

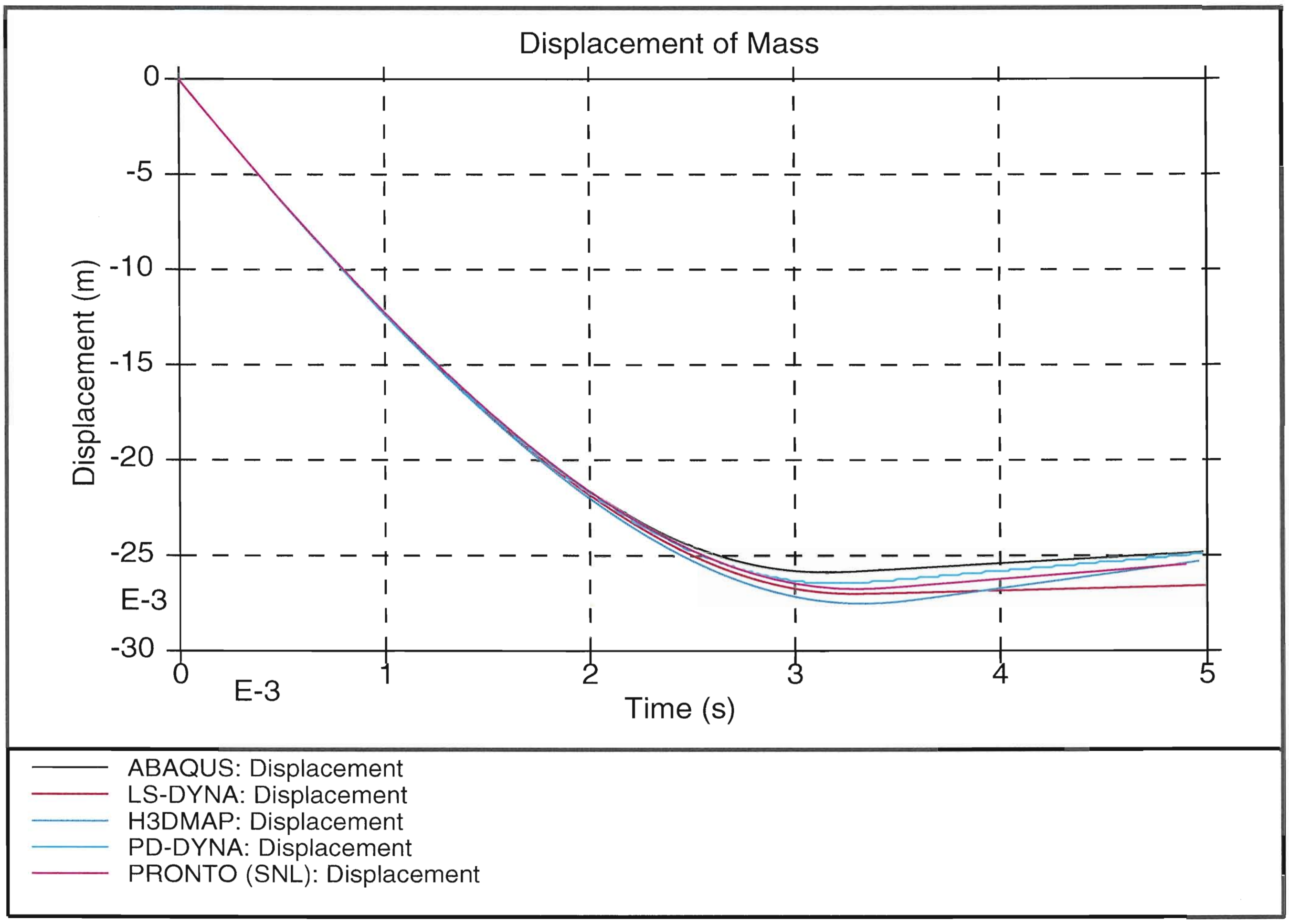
figure **5.4**

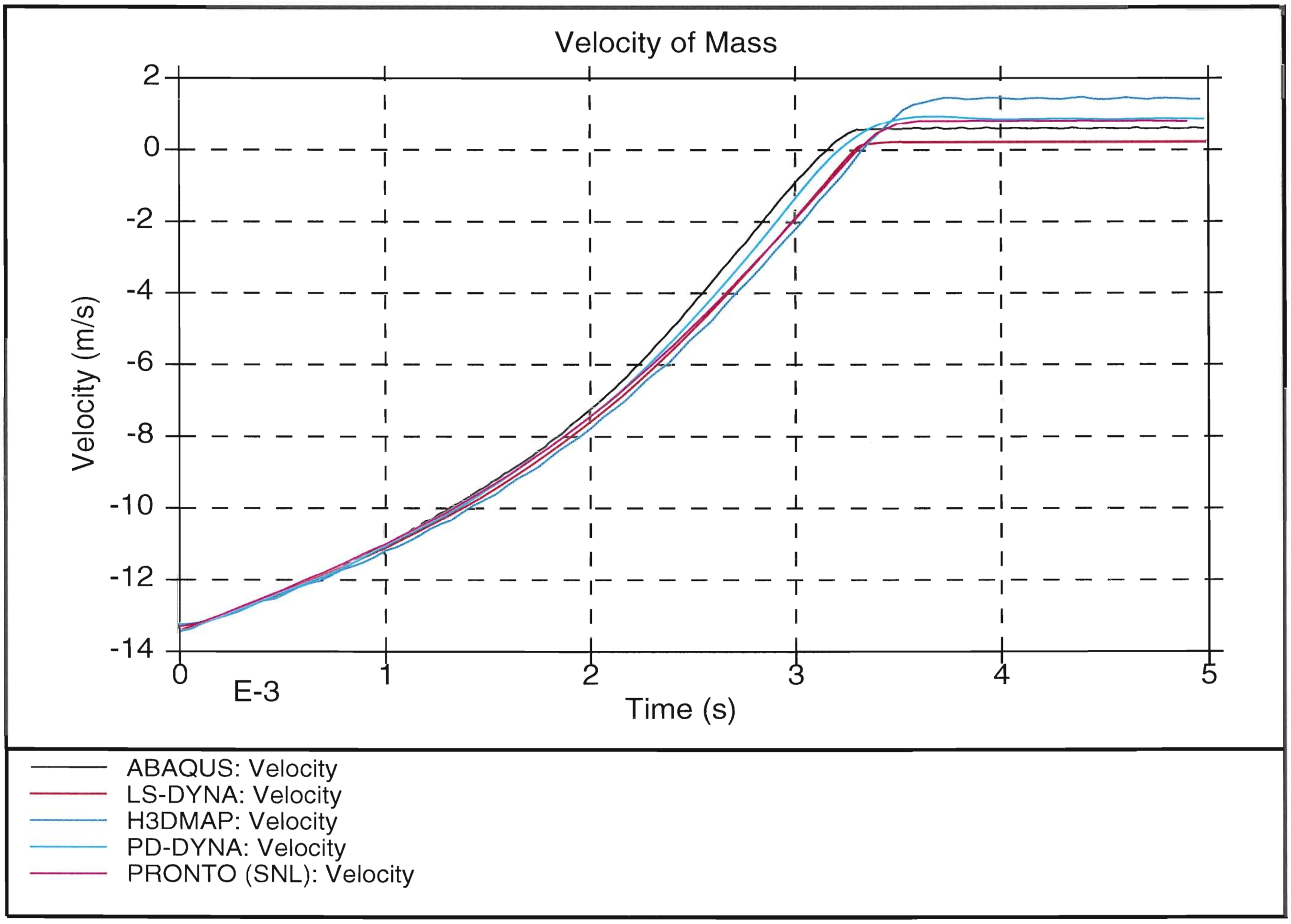


ABAQUS: VA(y) (Filtered at 600Hz)
LS-DYNA: VA(y)
H3DMAP: VA(y)
PRONTO (SNL): VA(y)
PRONTO (UTX): VA(y)

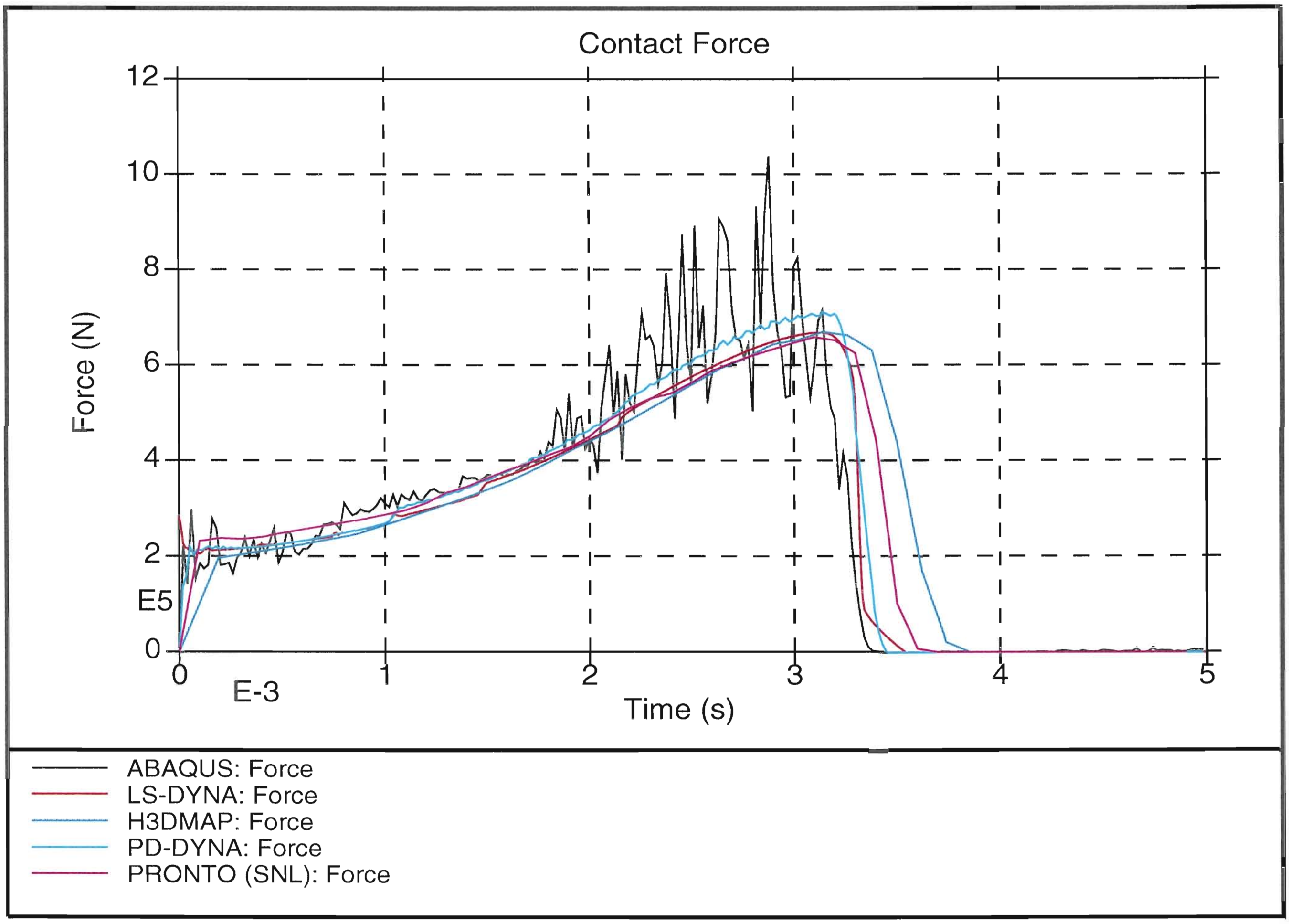


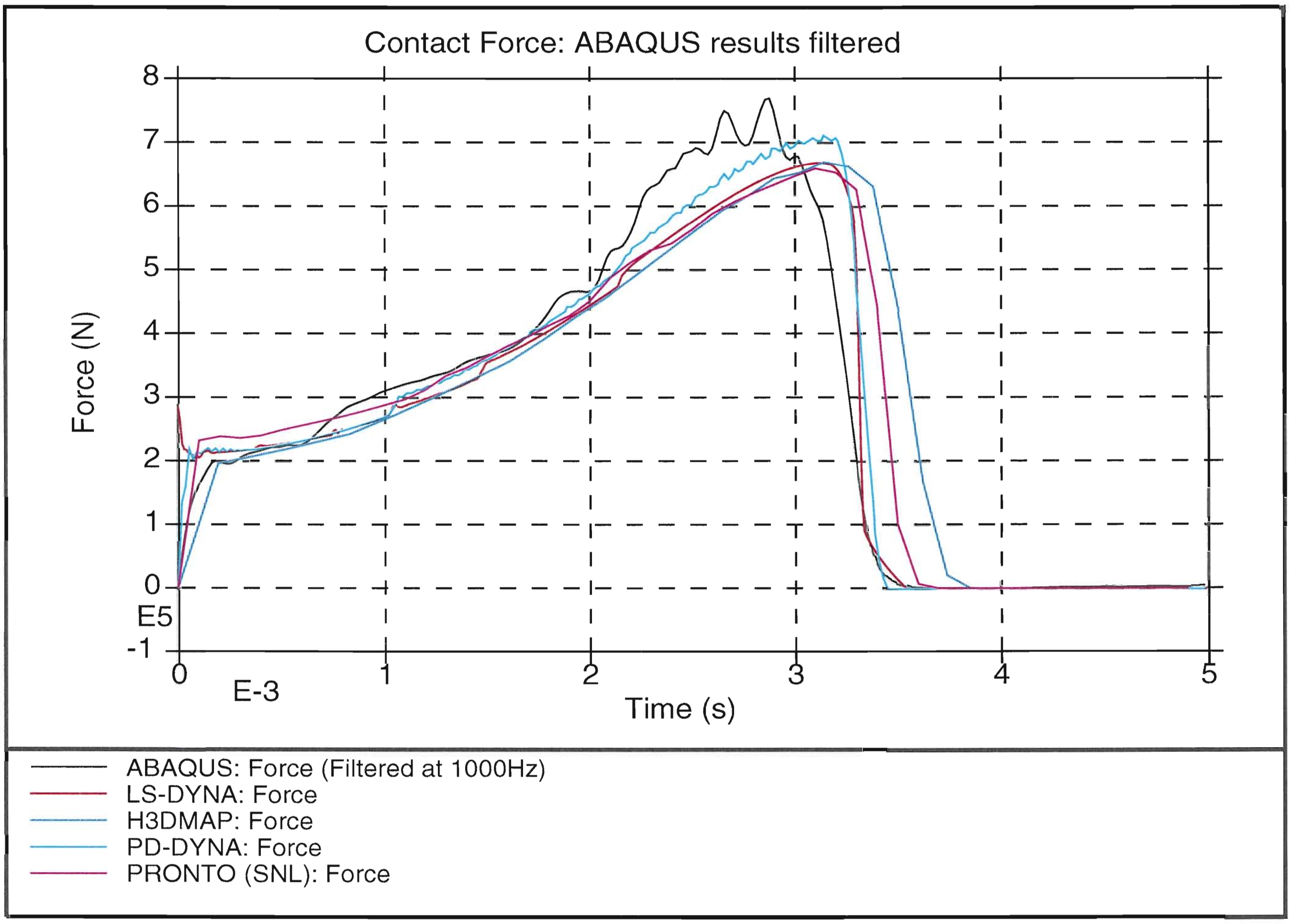
ABAQUS: B(y) - A(y)
LS-DYNA: B(y) - A(y)
H3DMAP: B(y) - A(y)
PRONTO (SNL): B(y) - A(y)
PRONTO (UTX): B(y) - A(y)





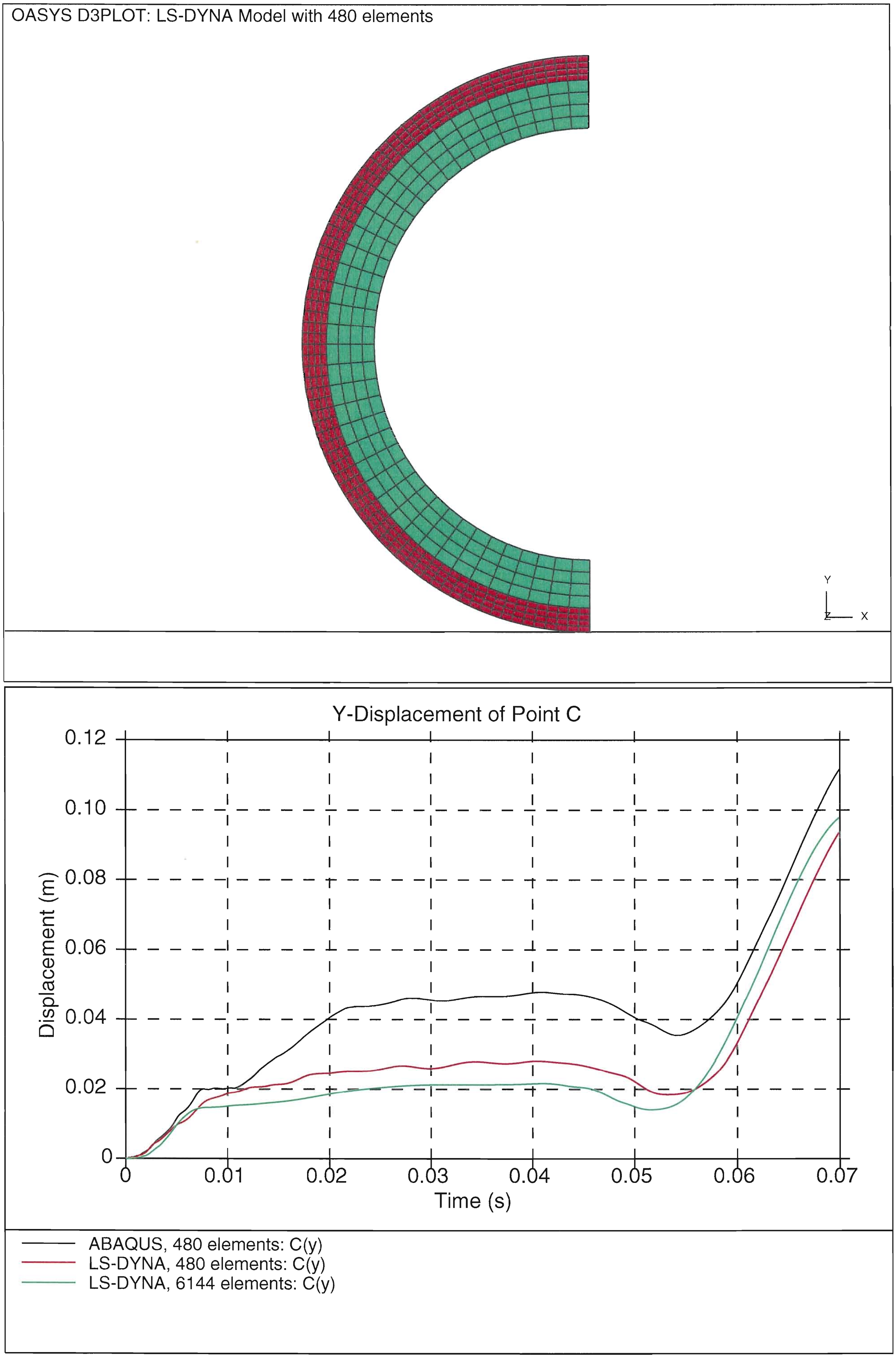
Benchmark 3: Displacement and Velocity Results **figure 5.6**



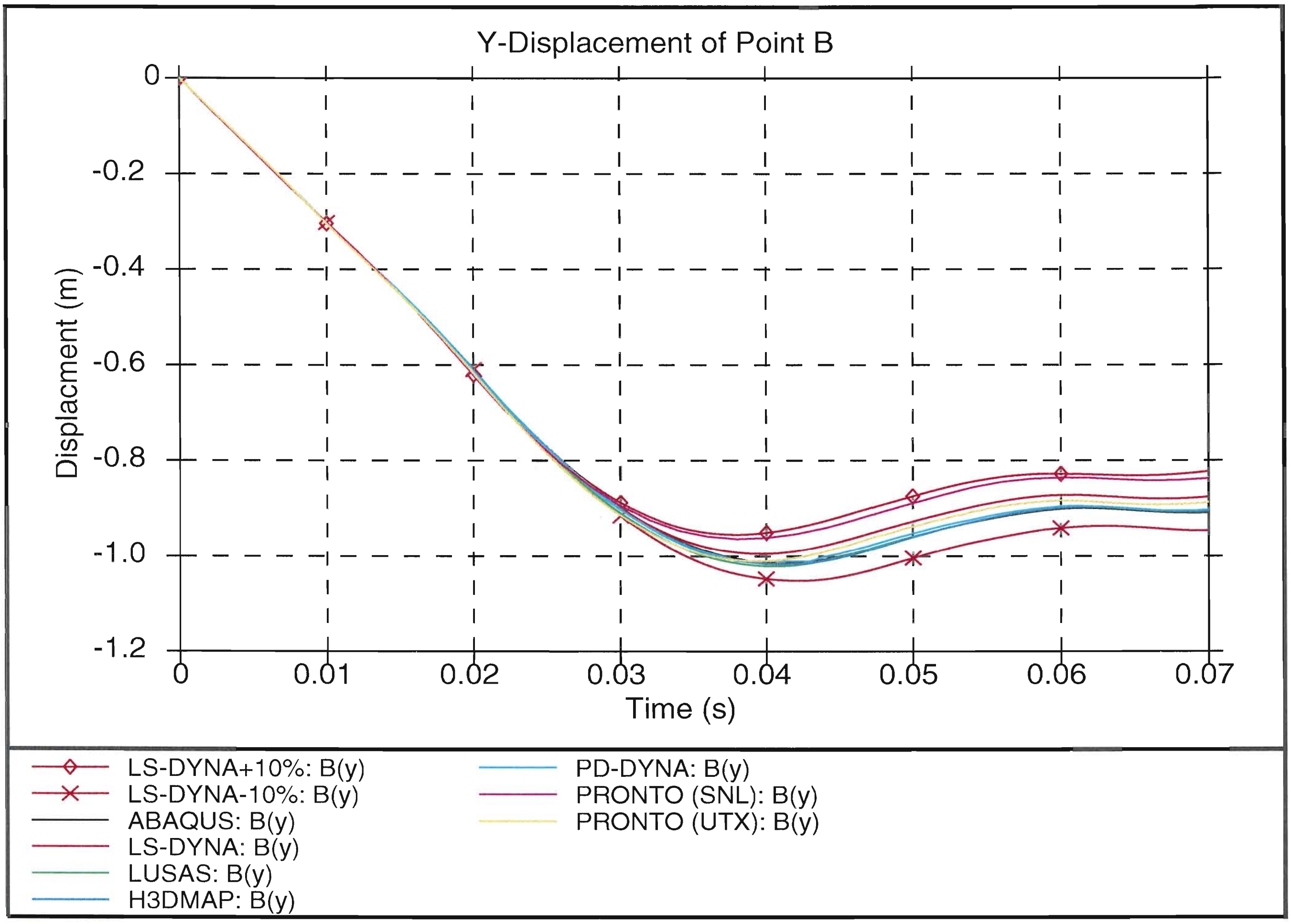


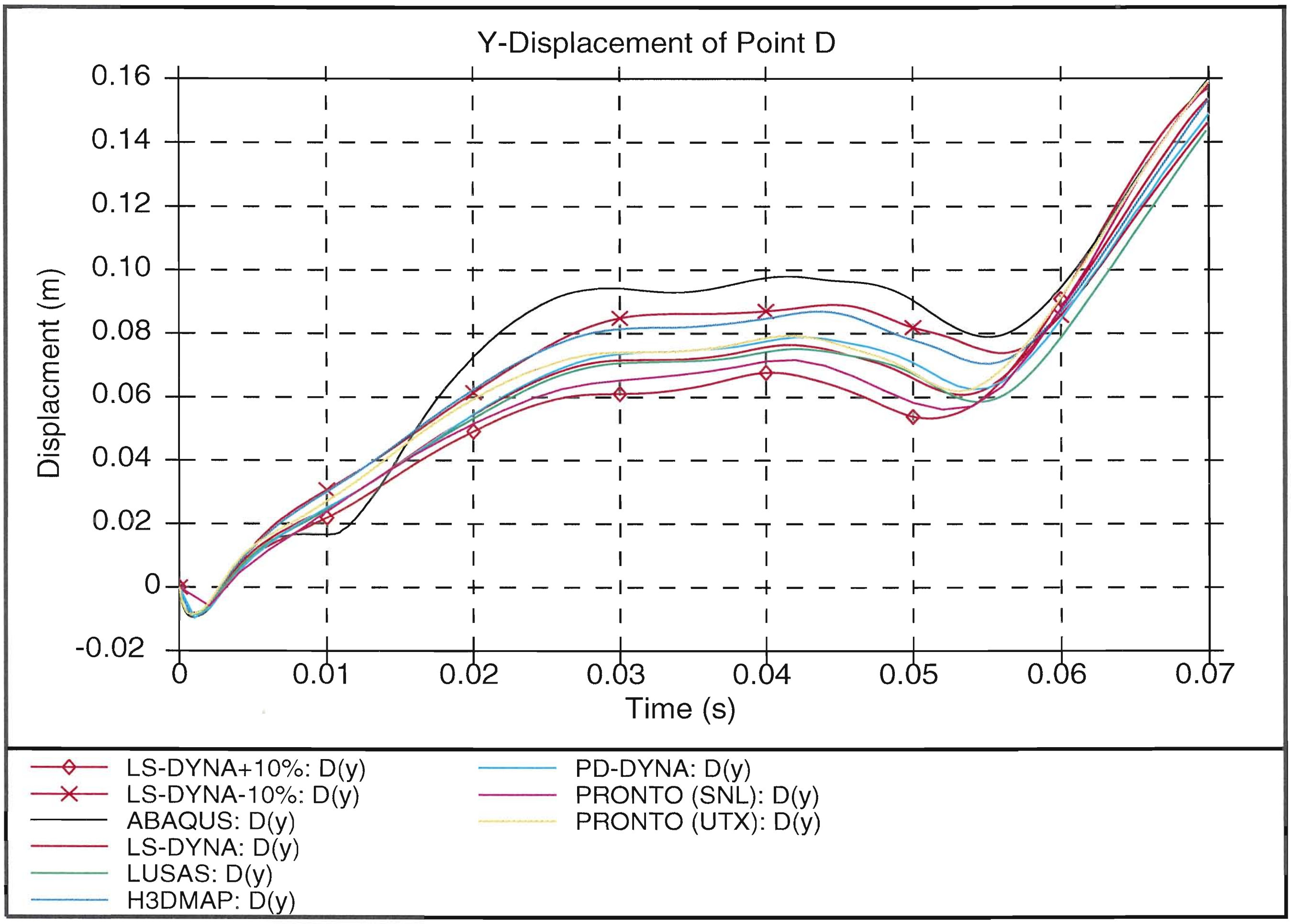
Benchmark 3: Force Results

figure 5.7



Benchmark 1: Extra LS-DYNA Analysis with 480 elements figure 5.8





Benchmark 1: FE Code results and simulated experimental scatter

figure **5.9**

Appendix A

Finite element code survey: summary tables

FE Code Survey - Summary Table by OAPIL

March 1998

Product name	ABAQUS/Explicit	H3DMAP	PRONTO2D, PRONTO3D	LS-DYNA3D	LUSAS
Version	Version 5.7	Version 6	Version 9.3.30	Version 93602.02	Version V12.3
Developer	Hibbitt, Karlsson & Sorensen, Inc. 1080 Main Street Pawtucket RI 02860-4847 USA	Ontario Hydro Technologies 800 Kipling Avenue Toronto Ontario M8Z 5S4 Canada	Sandia National Laboratories Albuquerque New Mexico 87185 USA	Livermore Software Technology Corporation 2876 Waverley Way Livermore CA94550 USA	FEA Ltd. Forge House 66 High Street Kingston upon Thames Surrey KT1 1HN UK
Distributer	Distributors Worldwide	In-house only	In-house only	Distributors Worldwide	Distributors Worldwide
Maintenance and Support	Release every 12 months Continuing support Enhancements to order	Periodic release Continuing support Enhancements to order	Release every 12 months Continuing support nhancements to order	Release every 12 months Continuing support Enhancements to order	Release every 12 months Continuing support Enhancements to order
QA	ANSI/ASME NQA-1 ISO9001	ISO9001	Internal QA	ISO9001 (by Oasys)	Internal QA
Benchmarking	Over 1000 benchmarks and example problems Comparisons to theory and experiment	Over 500 benchmarks Comparisons to theory and experiment	100 benchmarks Comparisons to theory and experiment	Extensive Comparisons to theory and experiment	Extensive Comparisons to theory and experiment

Product name	**ABAQUS/Explicit**	**H3DMAP**	**PRONTO2D, PRONTO3D**	**LS-DYNA3D**	**LUSAS**
Current applications	Engine design Aircraft Turbine blades Metal & plastic forming Metal cutting etc.	Transport container impact Fulid-structure interaction Metal forming Vehicle crashworthiness Creep Flow induced vibration Residual stress Coupled stress/diffusion	Transport container impact	Transport container impact Metal forming Vehicle crashworthiness Seismic Product drop tests Biomechanics	Ground engineering Bridge design Seismic Automotive Product design
Operating platforms	UNIX Windows NT	UNIX MS/DOS	UNIX	UNIX Windows 95 Windows NT	UNIX Windows 95 Windows 3.1
Solver type	Implicit Explicit Dynamic relaxation	Implicit Explicit Dynamic relaxation	Explicit Dynamic relaxation (with ancillary code, 'JAS')	Explicit Dynamic relaxation	Implicit Explicit
Pre-processing	ABAQUS/Pre External proprietory	External proprietory	External proprietory	External proprietory	Internal External proprietor
Post-processing	ABAQUS/Post External proprietory	Internal External proprietary	External proprietory	Oasys T/HIS & D3PLOT External proprietory	Internal External proprietory

Product name	**ABAQUS/Explicit**	**H3DMAP**	**PRONTO2D, PRONTO3D**	**LS-DYNA3D**	**LUSAS**
Material options	Elastic Elastic-plastic (isotropic/kinematic) Viscoelastic Reinforced concrete Foam Soil User defined ...	Elastic Elastic-plastic (isotropic/kinematic) Hydrodynamic (fluid) Hydrodynamic elastic-plastic Foam Soil Anisotropic creep Hydride stress interaction ...	Elastic Elastic-plastic (isotropic/kinematic) Viscoplastic Hydrodynamic Hydrodynamic elastic-plastic Foam Soil	Elastic Elastic-plastic (isotropic/kinematic) Viscoelastic General hydrodynamic Foam Soil Concrete ...	Elastic Elastic-plastic (isotropic/kinematic) Soil Concrete ...
Strain rate dependance	On all metal materials	On some materials	Yes	On some materials	On some materials
Failure criteria	Plastic strain	Plastic strain Hydrostatic stress	Strain energy von Mises stress Hydrostatic stress Maximum principal stress	Plastic strain on some materials	None
Elements types	1D, 2D, 3D Continuum (Full & red. integration) Shells Membranes Beams Springs etc.	1D, 2D, 3D Continuum Shells Beams Springs	3D Cont. (PRONTO3D) (8-noded, red. integration) Shells (PRONTO2D) (4-noded)	3D Continuum (Full & red. integration) Shells Membranes Beams Springs	2D, 3D Continuum Shells Membranes Beams Springs
Contacts	Surface-to-surface Singe-surface Surface-rigid Coulomb friction with optional shear stress limit	Surface-to-surface Surface-to-rigid wall Friction Stick/slip model	Surface-to-surface Coulomb or general friction	Surface-to-surface Single-surface Surface-to-rigid wall General friction model	Surface-to-surface Coulomb or general friction

Product name	ABAQUS/Explicit	H3DMAP	PRONTO2D, PRONTO3D	LS-DYNA3D	LUSAS
Prestressing of elements	For 'Geostatic' and 'Rebar' elements	Yes	Yes	Yes	Yes
Thermal expansion	Yes	Yes	Yes	Yes	Yes

REFERENCES

ABAQUS/Explicit
- *ABAQUS/Explicit: Product description*, Hibbitt, Karlsson & Sorensen, Inc., 1997.
- Letter from Rosemary Fowler of Hibbitt, Karlsson & Sorensen (UK) Ltd. to Chi-Fung Tso of Ove Arup and Partners, received on 13 January 1998 (OAP file ref. 53276).

H3DMAP
- *H3DMAP Version 6: A general three-dimensional finite element computer code for linear and nonlinear analyses of structures: User documentation*, pp1-5. Ontario hydro technologies.
- Sauvé R.G., Nadeau E., *H3DMAP Version 6: A general three-dimensional finite element computer code for linear and nonlinear analyses of structures: Validation case documentation*. Ontario hydro technologies.
- Letter from Greg Morandin of Ontario Hydro Technologies to Chi-Fung Tso of Ove Arup and Partners received on 17 January 1998 (OAP file ref. 53276).

PRONTO2D/PRONTO3D
- Taylor L.M., Flanagan D.P., *PRONTO2D, A two-dimensional transient solid dynamics program.* Sandia National Laboratories, 1987.
- Taylor L.M., Flanagan D.P., *PRONTO3D, A three-dimensional transient solid dynamics program.* Sandia National Laboratories, 1989.
- Attaway S.W., *Update of PRONTO2D and PRONTO3D transient solid dynamics program.* Sandia National Laboratories, 1990.
- E-mail from Douglas Ammerman of Sandia National Laboratories to Chi-Fung Tso of Ove Arup and Partners received on 20 February 1998 (OAP file ref. 53276).

LS-DYNA3D
- Livermore Software Technology Corporation, *LS-DYNA3D User's Manual, Version 936,* Report #1082, Aug 1, 1995.

LUSAS
- *LUSAS User Guide, Version 12.* FEA Ltd.
- *LUSAS Element Library, Version 12.* FEA Ltd.
- Letter from Gary Wyatt of FEA Ltd. to Chi-Fung Tso of Ove Arup and Partners, 8 October 1997, (FEA ref. GAW/071097, OAP file ref. 53276).

SELECTED PUBLICATIONS

ABAQUS/Explicit
- Awaiting details

H3DMAP

- Morandin G.D., Nadeau E., *Accident impact analysis of a spent-fuel storage transportation package,* PATRAM 95, Las Vegas, 1995.
- Sauvé R.G., Morandin G.D., Nadeau E., *Impact simulation of liquid-filled containers including fluid-structure interation - Part 1 and 2*, Journal of Pressure Vessel Technology ASME, Vol. 115, No. 1, 1992.
- Morandin G.D., *Irradiated spent fuel bundle impact analysis*, ASME PVP, Vol. 351, 1997.

PRONTO2D/PRONTO3D

- Hoffman E.L., Ammerman D.J., *Benchmarking of 2D and 3D finite element calculations with dynamic pulse-buckling tests of cylindrical shells under axial impact*, PATRAM 95, Las Vegas 1995.
- Glass R.E., *Impact testing and analysis for structural code benchmarking,* PATRAM 95, Las Vegas, 1995.
- Ludwigsen J.S., Ammerman D.J., *Analytical determination of package response to severe impacts*, PATRAM 95, Las Vegas 1995.

LS-DYNA3D

- Ajayi F., Dutton T., Donelan P., Sievwright R.W.T., Barlow S.V., *Recent advances in computer impact analysis for radioactive materials transport containers*, International Journal of Radioactive Materials Transport, Vol. 4, No. 2, pp.125-131, 1993.
- McGuinn P., Donelan P., Willford M., Everett D., *The use of computer impact analysis in licensing a container for the transport of fresh fuel,* PATRAM 95, Las Vegas, 1995.
- Akamatsu H., Taniuchi H., Fujimoto T., Ouchi M., *The Analysis of Drop Accident on to Real Target*, Institution of Nuclear Engineers, 4th International Conference on Transportation for the Nuclear Industry, Bournemouth, 1997.

LUSAS

- Crook A.J.L., Irving D.J., Lyons L.P.R., *Explicit and Implicit Impact Analysis with LUSAS*, 3rd Conference of Numerical Methods in Engineering, Swansea, 1990.
- Lyons L.P.R., Bell A.P., Crook A.J.L., *Contact Analysis with Finite Strains using LUSAS*, Conference Proceedings, Paris, November 1991.

Prepared by: D Bernasconi
Date 31 March 1998

Checked by: CF Tso
Date 31/3/98

Evaluation of Codes for Analysing the Drop Test Performance of Radioactive Material Transport Containers

EC Contract: 4.1020/D/96-016

FE Code Survey – Summary Table by GNB

Product name	DIANA	SOLVIA	ANSYS	DYNA3D	NIKE3D
Version			5.2	4.01	2.3.0a
Developer	DIANA ANALYSIS BV PO Box 113 2600 AC Delft Netherland	SOLVIA ENGINEERING AB Östra Rinvägen 4, S-722 14 Västeras, Sweden Tel 021-144050 Fax 021-188890	ANSYS, Inc. 201 Johnson Road Houston PA 15342-1300	Methods Development Group Lawrence Livermore National Laboratory (LLNL) P. O. Box 808 Livermore, California 94550	Methods Development Group Lawrence Livermore National Laboratory (LLNL) P. O. Box 808 Livermore, California 94550
Distributer	DIANA ANALYSIS BV PO Box 113 2600 AC Delft Netherland	SOLVIA ENGINEERING AB Östra Rinvägen 4, S-722 14 Västeras, Sweden Tel 021-144050 Fax 021-188890	Network of worldwide distributors	- NEA - In-house	- NEA - In-house
Maintenance and Support	DIANA ANALYSIS BV PO Box 113 2600 AC Delft Netherland	SOLVIA ENGINEERING AB Östra Rinvägen 4, S-722 14 Västeras, Sweden Tel 021-144050 Fax 021-188890	Continuing support	LLNL	LLNL
QA	Unknown	unknown	- ISO9001 - USNRC	?	?
Benchmarking	Unknown	- Comparisons to theory and experiments	- Extensive: 4500 benchmarks - comparisons to Theory and experiments	Some	Some
Current applications	Non-linear static, dynamic, thermal and flow stress analysis, specialized concrete models	- linear/nonlinear, - static/dynamic, - response spectrum analysis, - harmonic response	- Transport container development - Metal forming - Earthquake design - Product drop tests - Fluidmechanics - Multiphysics etc.	- Transport container impact - Metal forming - Vehicle crashworthiness - Seismic - Product drop tests etc.	- Transport container development - Earthquake design - Stress initialization for DYNA3D - Natural frequency and mode shape analysis etc.

Operating platforms	WINDOWS 95 WINDOWS NT UNIX running on 486 PC, workstations and mainframes	WINDOWS 95 WINDOWS NT UNIX running on 486 PC, workstations, mainframes and supercomputers	- Windows NT - UNIX - Parallel OS	- UNIX - Parallel OS	UNIX
Solver type	Explicit	implicit and explicit	Implicit	Explicit	Implicit
Pre-processing	flexible, interfaces to FEMGV, PATRAN, I-DEAS, DISPLAY III	external pre processor SOLVIA PRE	Internal/External ANSYS/PRE	External (LLNL) INGRID	External (LLNL) INGRID
Post-processing	flexible, interfaces to FEMGV, PATRAN, I-DEAS, DISPLAY III	external post processor SOLVIA-POST	Internal/External ANSYS/Post	External (LLNL) TAURUS	External (LLNL) TAURUS
Material options	Extensive concrete material models, easy modelling of reinforced concrete, elastic, nonlinear-elastic, plastic, plastic-creep	Elastic, orthotropic, nonlinear-elastic, thermo-elastic,thermo-orthotropic, laminated, concrete, curve-description material, drucker-prager, plastic, plastic-multilinear, plastic-creep and rubber	Elastic,Elastic-plastic (isotropic/kinematic),Viscoelastic, Reinforced concrete, Soil,Multifield etc.	Elastic, Elastic-plastic, (isotropic/kinematic), Viscoelastic, General hydrodynamic, Soil, foam, Reinforced concrete etc. Approx. 40 material models	Elastic-, Elastic-plastic (isotropic/kinematic)Viscoelastic, Thermo-elastic creep, Soil, foam etc. Approx. 15 material models
Strain rate dependance	unknown	Available on some materials	no	yes	no
Failure criteria	- dircerete and smeared cracking, - crushing, - creep, - shrinkage and bond slip	- Element birth and death options, - concrete model with failure surfaces in tension and compression	Plastic strain (pressure, tension)	Plastic strain (some materials) (pressure, tension)	no
Elements types	several types in 2D and 3D	- 2, 3 or 4-noded truss , - 1-noded ring , - 3 to 9-noded plane for plane strain, - 4 to 27-noded solid , - 2-noded beam, 2 3, or 4-noded isoparametric beam , - 4-noded general shell , - 16-noded general shell, - 3-noded plate, - 2 to 4-noded pipe	- 3D Continuum (Full & red. integration) - Shells - Membrans - Beams - Springs etc.	- 3D Continuum (Full & red. integration) - Shells - Membrans - Beams - Springs	- 3D Continuum - Shells - Membrans - Beams - Springs
Contacts	soil-structur interaction	- Contact conditions in 2D and 3D with variable	- Point-to-point - Point-to-surface	- Surface-to-surface - Single-surface	- Surface-to-surface

		contact area and large relative displacements	- Point-to-ground	- Surface-to-rigid wall - General friction model - Automatic contact	- Single-surface - Surface-to-rigid wall - General friction model
Prestressing of elements	unknown	unknown	yes	no	no
Thermal expansion	yes	yes	yes	yes	yes
Dynamic relaxation	unknown	unknown	no	yes	no

References

DYNA3D

- A Nonlinear, Explicit, Three-Dimensional Finite Element Code for Solid and Structural Mechanics
  User's Manual
  R. G. Whirley, B. E. Engelmann; Methods Development Group, Mechanical Engineering
  Orginated by: J. O. Hallquist
  UCRL-MA-107254, Rev. 1
  Lawrence Livermore National Laboratory, November 1993

NIKE3D

- A Nonlinear, Implicit, Three-Dimensional Finite Elemente Code for Solid and Structural Mechanics
  User's Manual
  Bradley N. Maker; Methods Development Group, Mechanical Engineering
  Robert M. Ferencz; Consultant
  Orginated by: J. O. Hallquist
  UCRL-MA-105268
  Lawrence Livermore National Laboratory, January 1991

INGRID

- A 3-D Mesh Generator For Modeling Nonlinear Systems
  User's Manual
  M. A. Christon, D. Dovey; Methods Development Group, Mechanical Engineering
  Orginated by: D. W. Stillman, J. O. Hallquist, R. B. Rainsberger

UCRL-MA-109790 (Draft)
Lawrence Livermore National Laboratory, September 1992

TAURUS

- An Interactive Post-Processor for the Analysis Codes NIKE3D, DYNA3D and TOPAZ3D
  T. Spelce; Methods Development Group, Mechanical Engineering
  Orginated by: B. E. Brown, J. O. Hallquist
  UCRL-MA-105401
  Lawrence Livermore National Laboratory, May 1991

ANSYS

- Revision 5.2
  ANSYS Inc.

Appendix B

Benchmarks 1, 2 and 3

Benchmark 1: Flat Side Impact of Concentric Cylinders

Order of problem

2D plane strain problem

Geometrical and material details

Steel cylinder

Dimensions:
Outer diameter 2.4m, Inner diameter 2.2m

Material:
Model as bilinear elastic-plastic material

Young's modulus	E	=	210×10^9 N/m^2
Density	ρ	=	7850 kg/m^3
Yield Strength	σ_y	=	200×10^6 N/m^2
Hardening modulus		=	1000×10^6 N/m^2
Hardening model		=	isotropic hardening
Poissons ratio	ν	=	0.3
Strain rate effect			none

Lead cylinder

Dimensions
Outer diameter 2.2m, Inner diameter 1.8m

Material:
Model as bilinear elastic-plastic material

Youngs Modulus	E	=	17×10^9 N/m^2
Density	ρ	=	11340 kg/m^3
Yield Strength	σ_y	=	1.9×10^6 N/m^2
Hardening modulus		=	790×10^6 N/m^2
Hardening model		=	isotropic hardening
Poissons ratio	ν	=	0.45
Strain rate effect			none

Interface Conditions

Between the steel cylinder and the lead cylinder
- unconnected
- Coefficient of Friction 0.2

Between the steel cylinder and target
- Coefficient of Friction 0.2

Initial velocity

30.0m/s perpendicularly towards the rigid target.

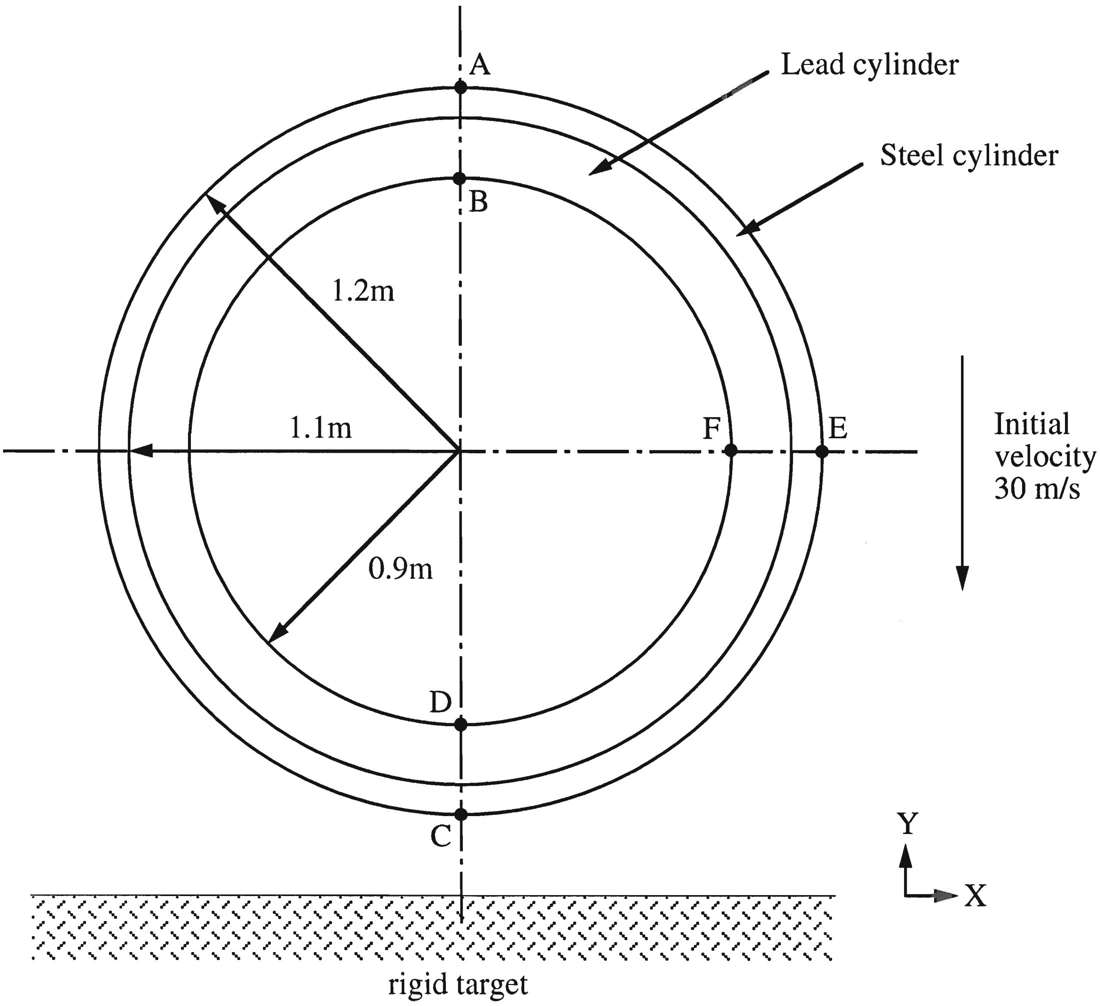
Analysis Time

70ms from initial contact of steel cylinder with the target

Required output

(1) Displacement vs Time in Y direction for points A, B, C and D
(2) Displacement vs Time in X direction for points E and F

Note:
- For both (1) and (2) above, take Displacement = 0 at Time = 0. Take Time = 0 when the steel cylinder make first contact with the target
- Please supply the required displacment vs time data in (i) graphical format and (ii) ASCII data format on 3.5" floppy disc


A
Lead cylinder
Steel cylinder
B
1.2m
1.1m
F
E
Initial
velocity
30 m/s
0.9m
D
C
Y
X
rigid target

Benchmark 2: Corner Impact of Steel Cube

Order of problem

3D problem

Geometrical and material details

Steel cube

Dimensions:
1.0m × 1.0m × 1.0m

Material:
Model as bilinear elastic-plastic material

Density	ρ	=	7850 kg/m^3
Young's modulus	E	=	210×10^9 N/m^2
Yield strength	σ_y	=	200×10^6 N/m^2
Hardening Modulus		=	1000×10^6 N/m^2
Hardening model			isotropic hardening
Poissons ratio	ν	=	0.3
Strain rate effect			none

Rigid shells

Dimensions:
3 no. 1.0m × 1.0m × 0.01m each. Each shell covers a face of the cube which faces away from the target. The three shells shall be rigidly connected to each other.

Material:
Model as Rigid material

Density	ρ	=	1666.667×10^3 kg/m^3

(to achieve a total mass of 50,000kg in the rigid shells)

Interface conditions

Between rigid shell and steel cube
- perfectly connected

Between steel cube and rigid target
- Coefficient of Friction 0.2

Initial Conditions

Orientation	- As illustrated on p.3 to 5 - Point A, Centre of Gravity and Point B (initial point of impact) are co-linear and perpendicular to the target.
Velocity	- 13.3m/s perpendicularly towards the rigid target.

Analysis time

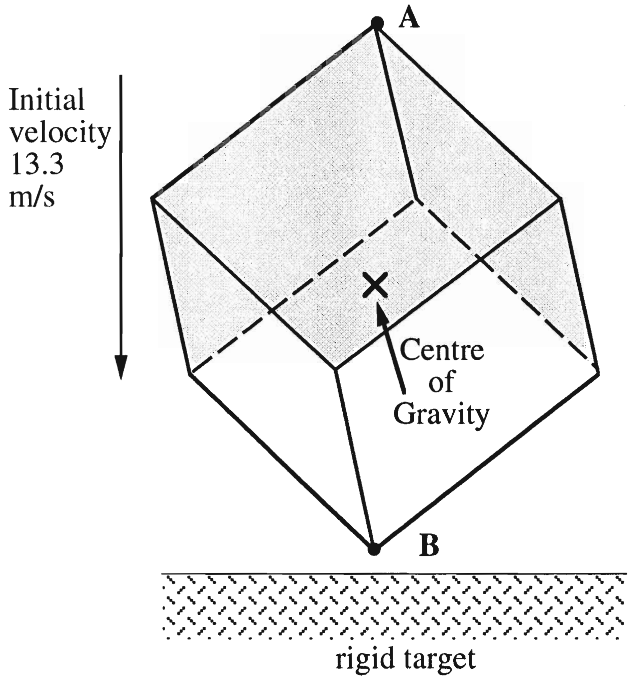
30ms from initial contact of steel cube with the target

Required output

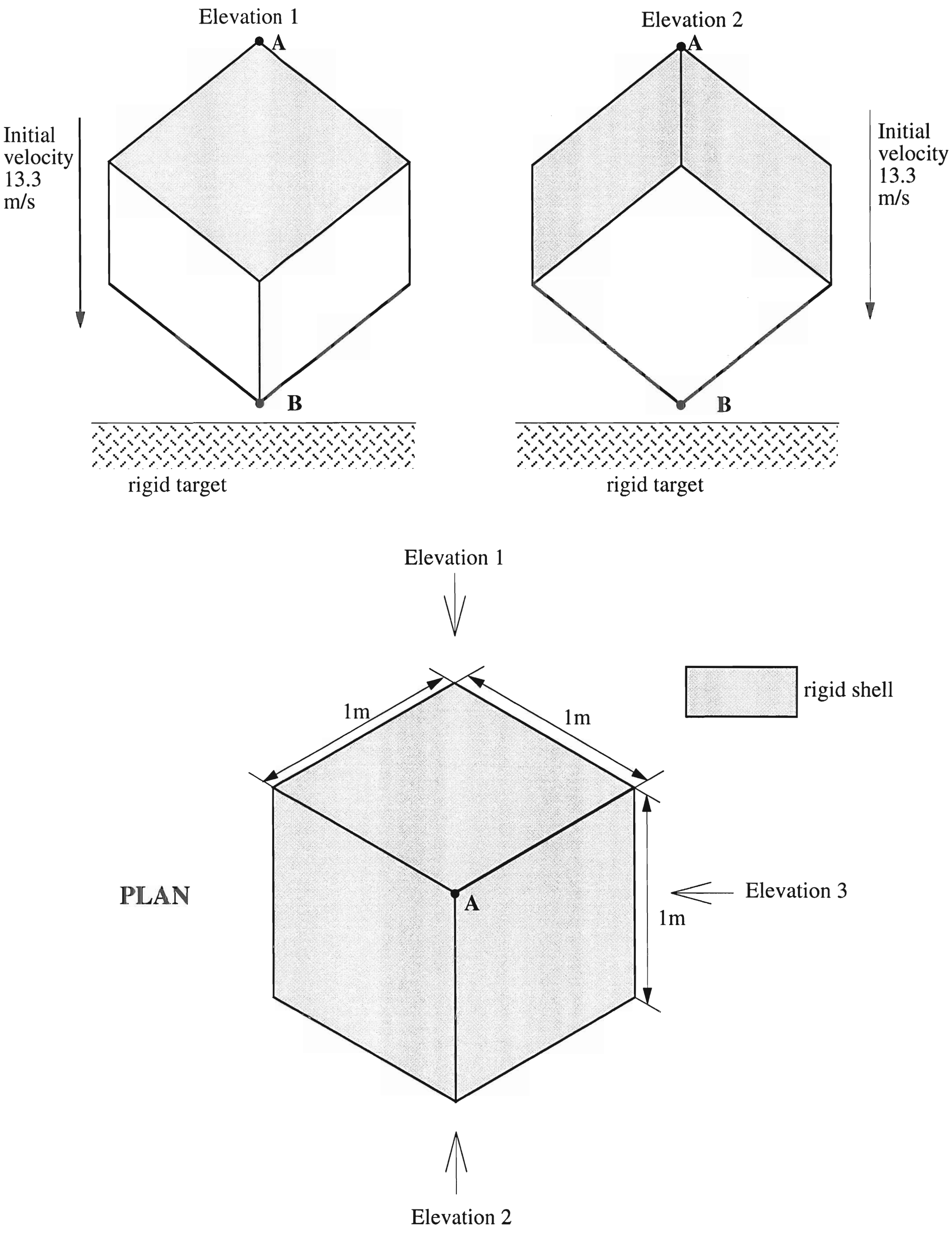
Displacement vs Time in Y direction for points A and B

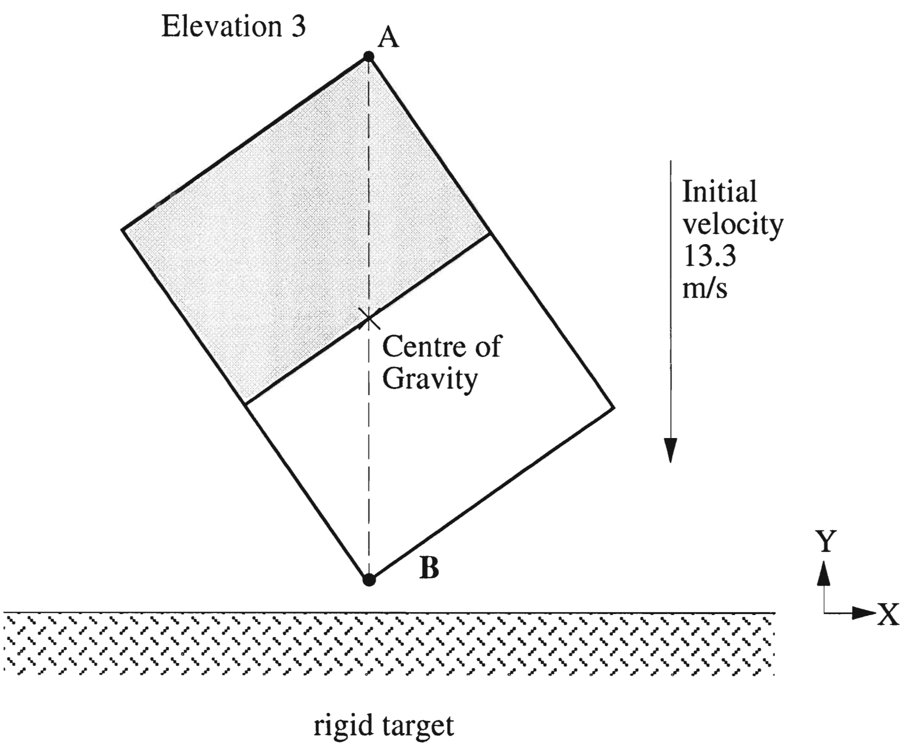
Note:

- Take Displacement = 0 at Time = 0. Take Time = 0 when the cube make first contact with the target
- Please supply the required displacment vs time data in (i) graphical format and (ii) ASCII data format on 3.5" floppy disc

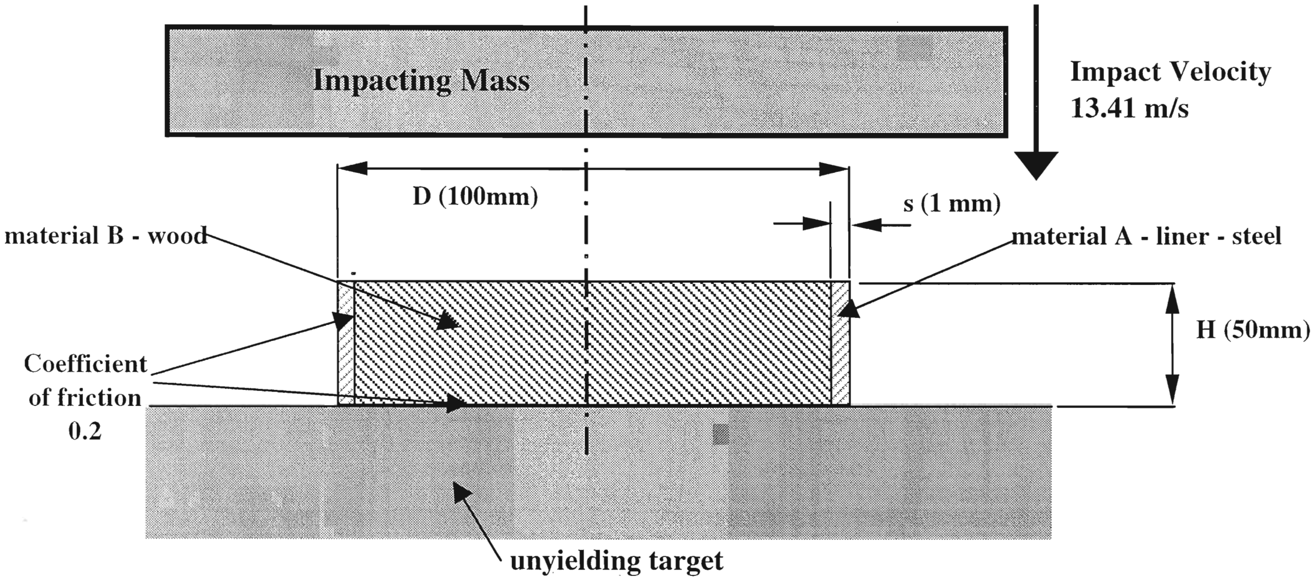


rigid shell


Elevation 1
A
Initial velocity 13.3 m/s
B
rigid target
Elevation 2
A
Initial velocity 13.3 m/s
B
rigid target
Elevation 1
1m
1m
rigid shell
PLAN
A
Elevation 3
1m
Elevation 2


Elevation 3
A
Initial
velocity
13.3
m/s
Centre of
Gravity
B
Y
X
rigid target

Benchmark 3 : Axis Vertical Impact of Wooden Impact Limiter with Liner



(DIAGRAM NOT TO SCALE)

Impacting Mass		
- Material Type	Rigid	
- Diameter	D	150 mm
- Mass	M	100 kg
- Impact velocity	v	13.41 m/s
Wooden Impact Limiter		
- Overall Height	H	50 mm
- Overall Diameter	D	100 mm
- Liner thickness	s	1 mm
Material A - Liner	Mild Steel	
- Density	rho	7850 kg/m³
- Young´s modulus	E	210000 N/mm²
- Poisson´s ratio	nu	0.3
- Yield strength	SigY	200 N/mm²
- Hardening modulus	Eh	1000 N/mm²
Material B	Wood	
- Material type	Perfectly Plastic	
- Density	rho	500 kg/m³
- Yield strength	SigY	17 N/mm²
- Hardening modulus	Eh	0 N/mm²

Interface Conditions

Between impact limiter and target
Between impact limiter and impactor
Between wood and liner

- Coefficient of Friction	mu	0.2

Required Output

- maximium compression of the wood
- displacement of impacting mass vs time
- maximum impact force and time of occurance
- impact force vs time

Note

- Take displacement = 0 at time = 0. Take time = 0 when the impacting mass make first contact with the wood
- Please supply the required time dependent data in a graphical format and ASCII data format on 3.5" floppy disk

Appendix C

LS-DYNA3D analysis of Benchmark 1 by OAPIL

C1. Introduction

This appendix describes the analysis of Benchmark 1 carried out by OAPIL using LS-DYNA3D. Studies were carried out to assess the effect of mesh density and element formulation on the results, and to determine the number and type of elements to be used in the analysis. The analysis results required by the benchmark test are presented and discussed.

C2. Model description

The details of the model were taken from the description of Benchmark 1 given in Appendix B. The geometry and loading are symmetric about a vertical plane, and so only a half-model needed to be analysed. Accordingly, a half-model was created and nodes which lay on the plane of symmetry were given symmetry boundary conditions, such that they were free to translate only in the vertical direction.

The benchmark is defined as a 2D plane strain problem, whereas LS-DYNA3D caters only for 3D solutions. However, a 2D plane strain problem can be simulated by using 3D elements and restraining all the nodes from moving in a direction normal to the 2D plane. In this case, a single layer of 8-noded solid hexahedral elements was used.

The material model 'MAT_PLASTIC_KINEMATIC' was used to represent both the steel and the lead, with the relevant material properties being assigned. A hardening parameter 'BETA' of 1.0 was used, corresponding to an isotropic hardening law as specified in the benchmark.

The rigid target was defined as a 'RIGIDWALL' in LS-DYNA3D, and the interface between the outer and inner cylinders was represented by a 'surface-to-surface contact' with a Coulomb friction coefficient of 0.2. The model was positioned so that it was just in contact with the target at the beginning of the analysis, and an initial velocity of 30m/s towards the target was applied to all of the nodes.

C3. Mesh density study

A mesh density study was carried out to assess the effect that the number of elements used in the model would have on results. This involved creating three models, identical except for the number of elements. The three models, comprising 192, 768 and 6144 fully integrated elements, are shown in Figures C.1 and C.2.

Each model was analysed in the impact scenario specified in the benchmark, and selected results were compared. Figure C.3 shows the displacement histories for points A in the y-direction and E in the x-direction. It can be seen that the results for the two more detailed models are in good agreement, whereas the displacements predicted by the model with the least number of elements differ by up to about 10%. This implies that for benchmark 1, 192 fully integrated elements is not sufficient for an accurate analysis, but a model with 768 fully integrated elements does produce a convergent solution.

C4. Element formulation study

An element formulation study was carried out to assess the effect that the element integration scheme used in the model would have on results. Models with constant stress (reduced integration) elements were compared with fully integrated elements, for both coarse (192 element) and fine (6144 element) meshes. Figure C.4 shows selected results from the four analyses.

In the case of the 6144 element models, it can be seen that the results are not significantly affected by the element formulation, which implies that for this number of elements both types would be suitable. However, there is a noticeable difference between the results of the two 192 element models, indicating that element formulation is significant at this level of detail. Note that although, in general, fully integrated elements would be expected to give more accurate results than constant stress elements, this does not appear to be the case here: the model with the coarse mesh and constant stress elements has produced a surprisingly good answer. This is purely coincidence; errors due to the under-stiff nature of the elements in elastic bending have cancelled out errors due the plasticity phase and the interaction between the two cylinders.

C5. Results

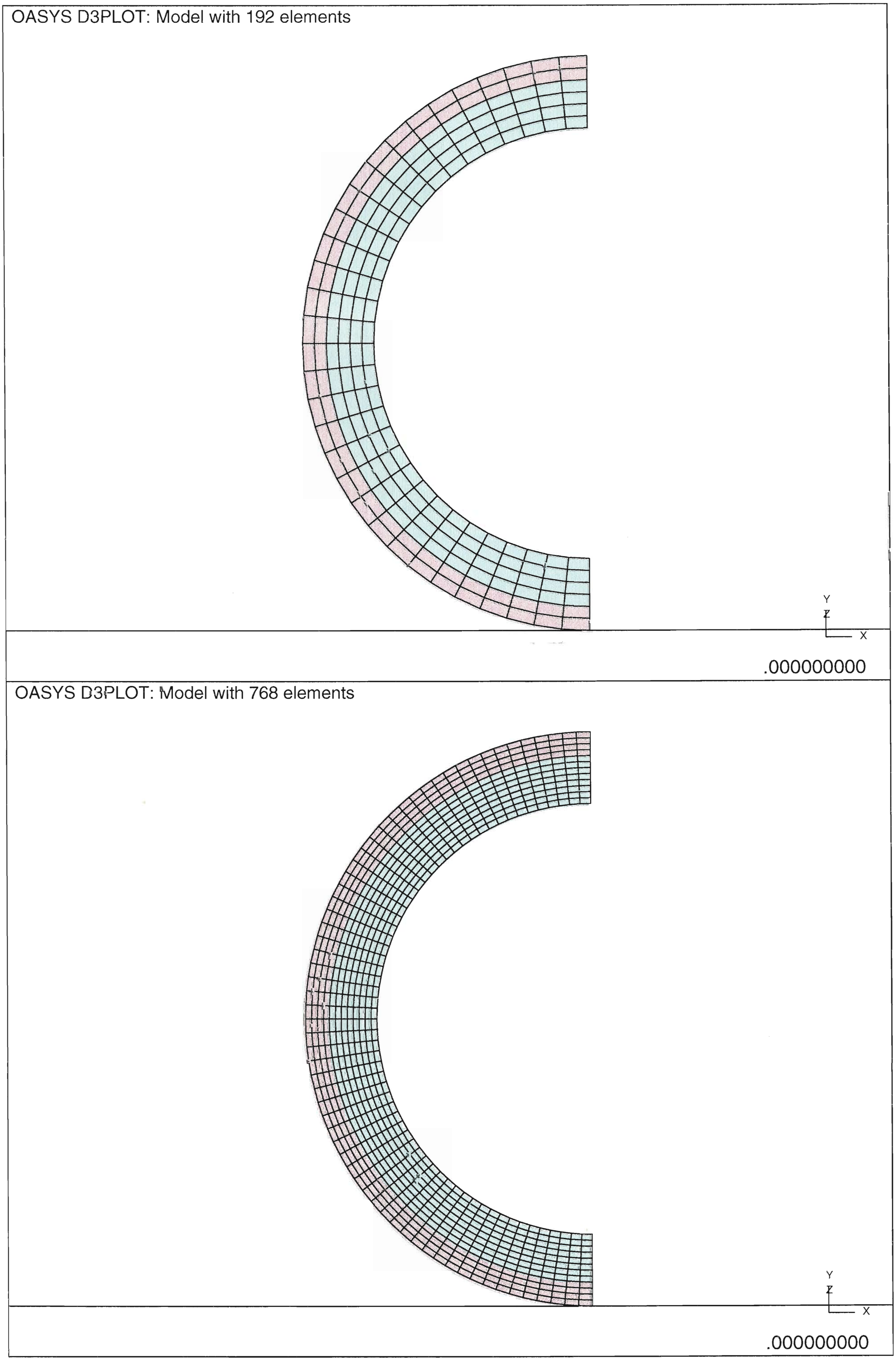
The results presented below are from the most complex model (6144 fully integrated elements). Although the 768 element model has been shown to be adequate, the more detailed model was used to provide marginally increased accuracy at the expense of computational efficiency.

Figure C.5 shows the sequence of events during the impact, from first contact of the outer cylinder with the target, to rebound. Soon after impact, a gap between the cylinders forms at the sides as the denser, softer lead 'slumps' more quickly towards the target. This gap moves around to the top of the cylinders as the lead deforms to fill the shape of the lower portion of the steel cylinder. A small gap also forms between the lead and the steel at the base of the cylinders

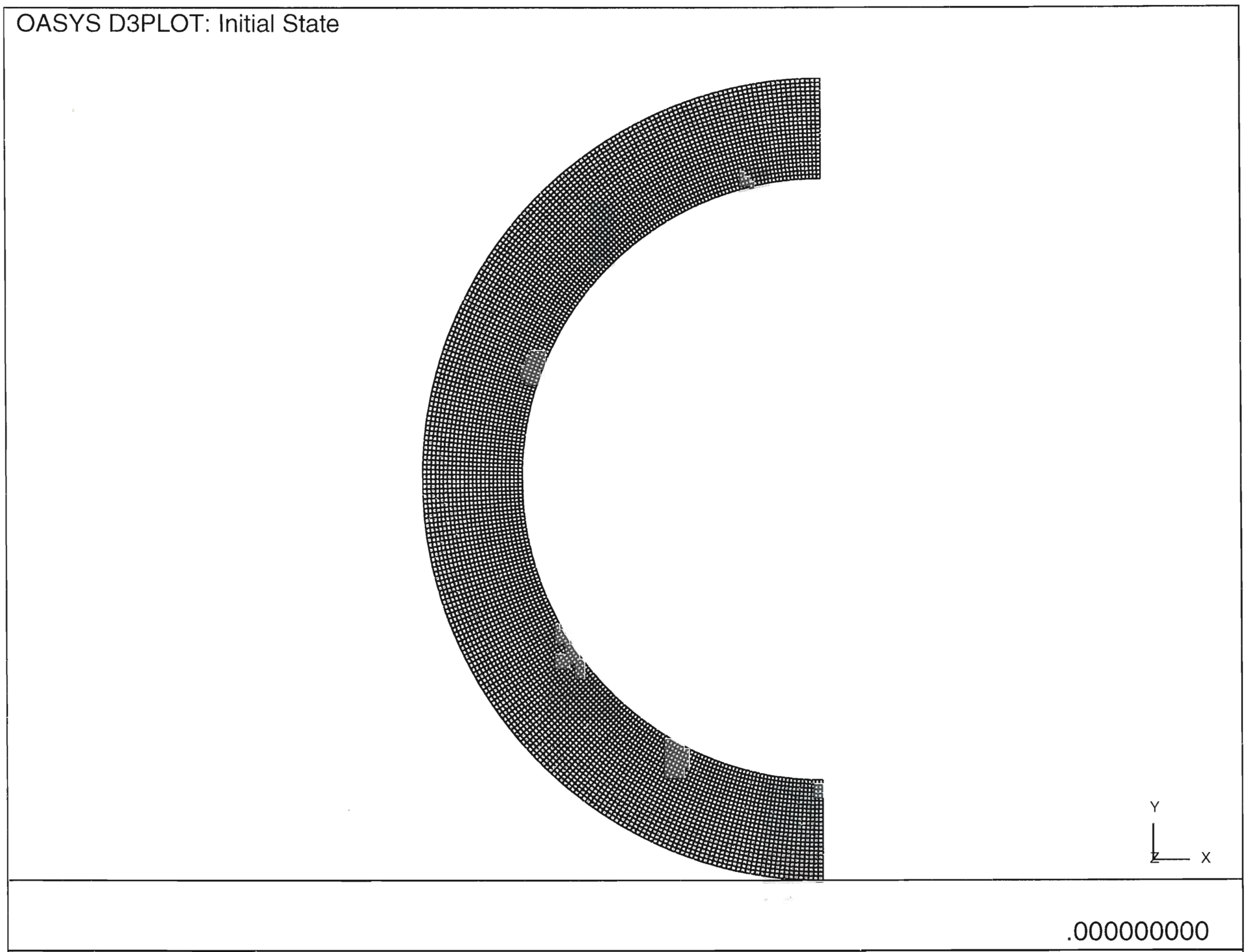
Figure C.6 shows the y-displacement histories for points A, B, C and D, and the x-displacement histories for points E and F, as required output for the benchmark. It can be seen that the maximum overall compression of both cylinders occurs at approximately 40ms after impact. Following this, there is a period of elastic recovery before the cylinders rebound, losing contact with the target at approximately 55ms after impact.

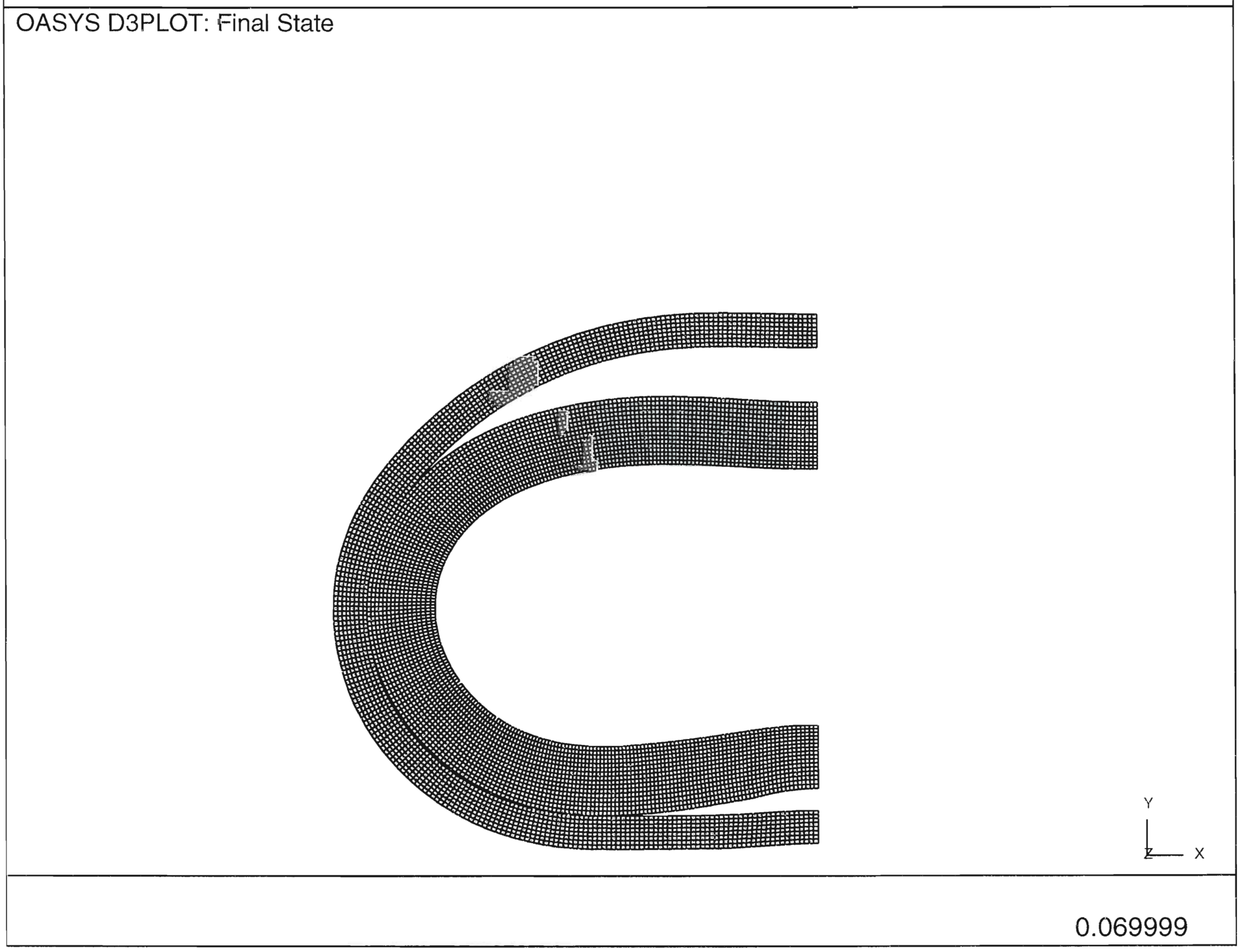
The final vertical compressions of the steel and lead cylinders are approximately 800mm and 1000mm respectively; the final horizontal expansions are both about 250mm.

Plastic strains are indicated in the contour plots (Figure C.5). By the time of rebound, maximum plastic strains of approximately 21% are present in the inner lead cylinder. The outer steel cylinder has suffered much less permanent deformation, with maximum plastic strains of only 11%.

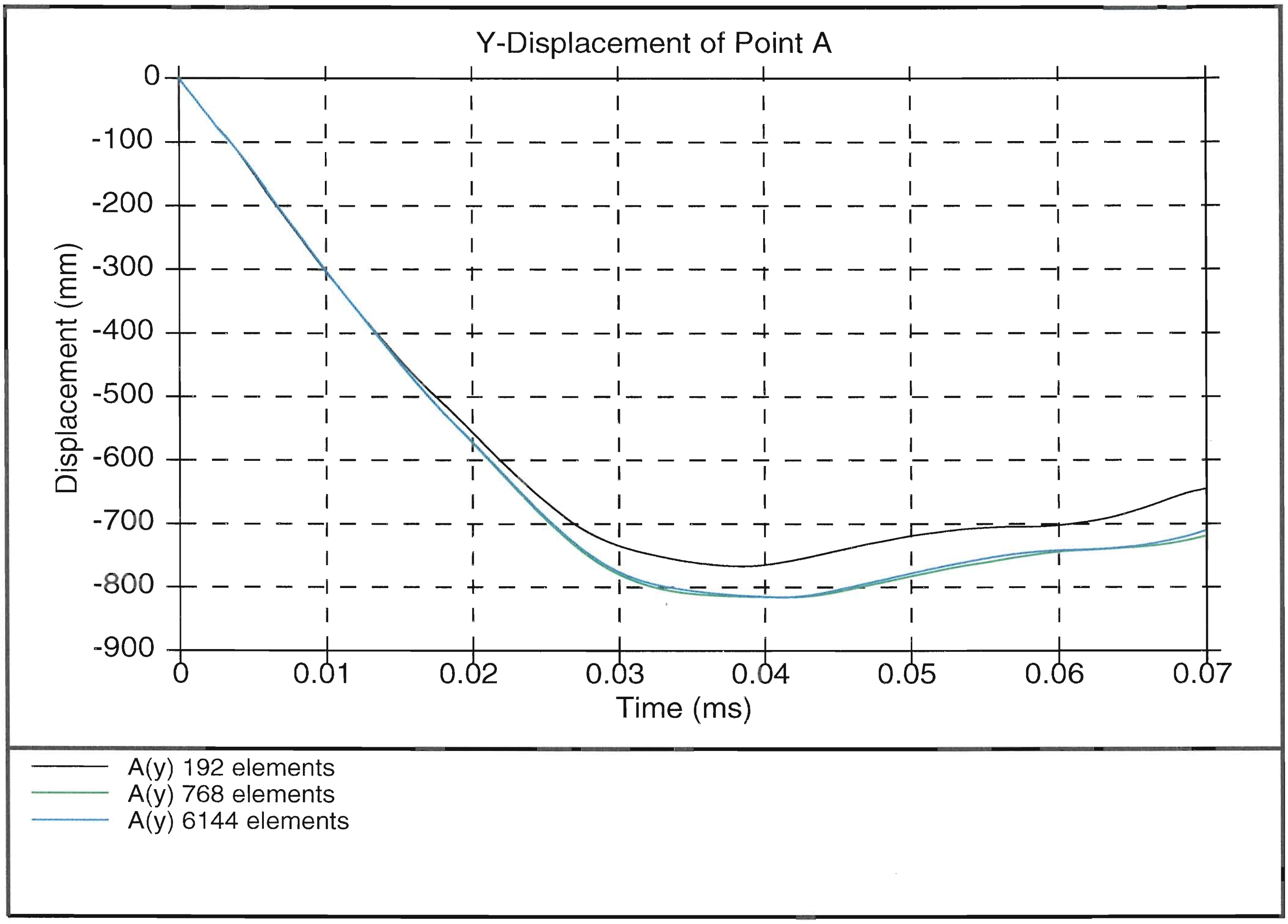


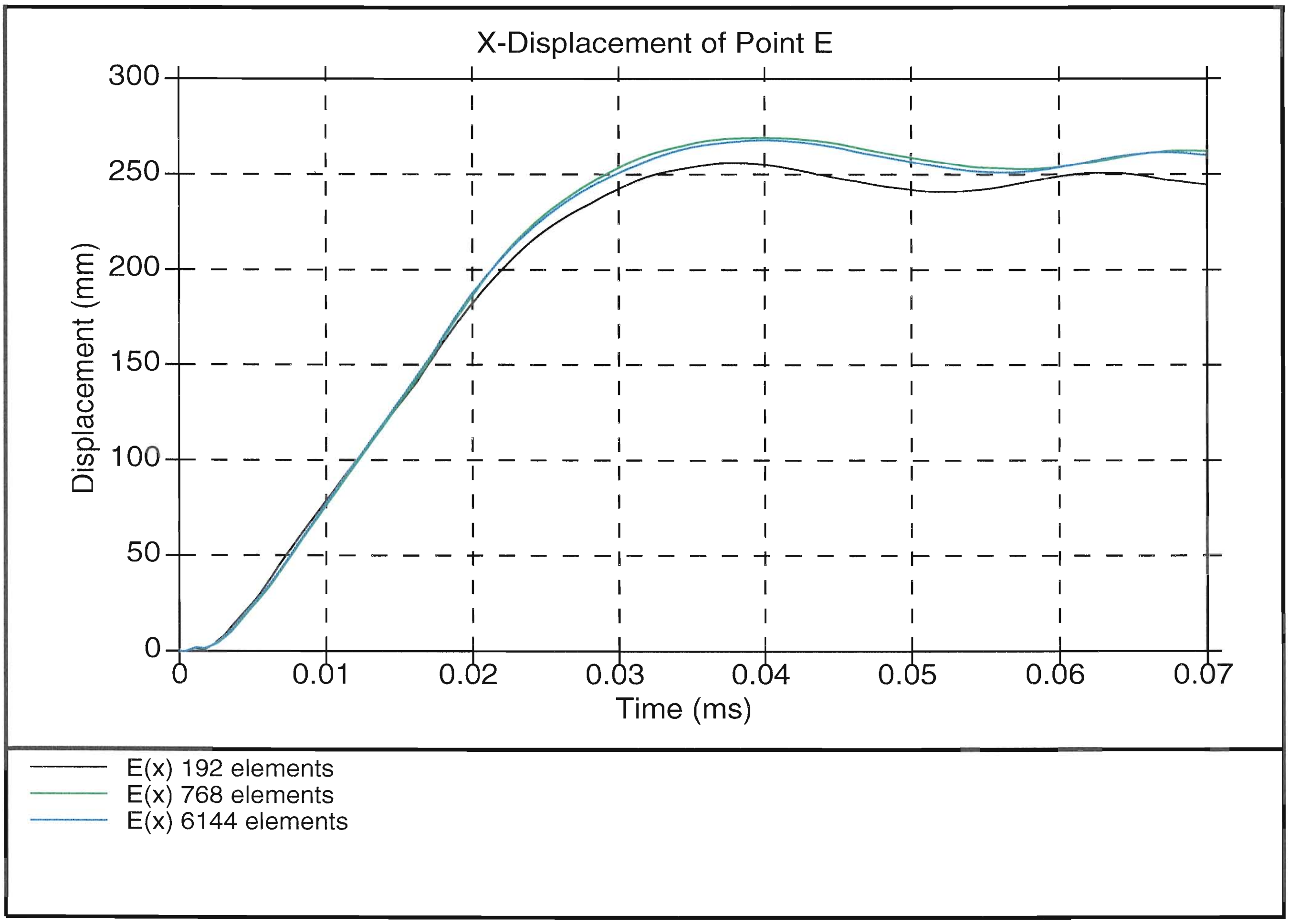
Benchmark 1 : Models used for mesh density study **figure C.1**





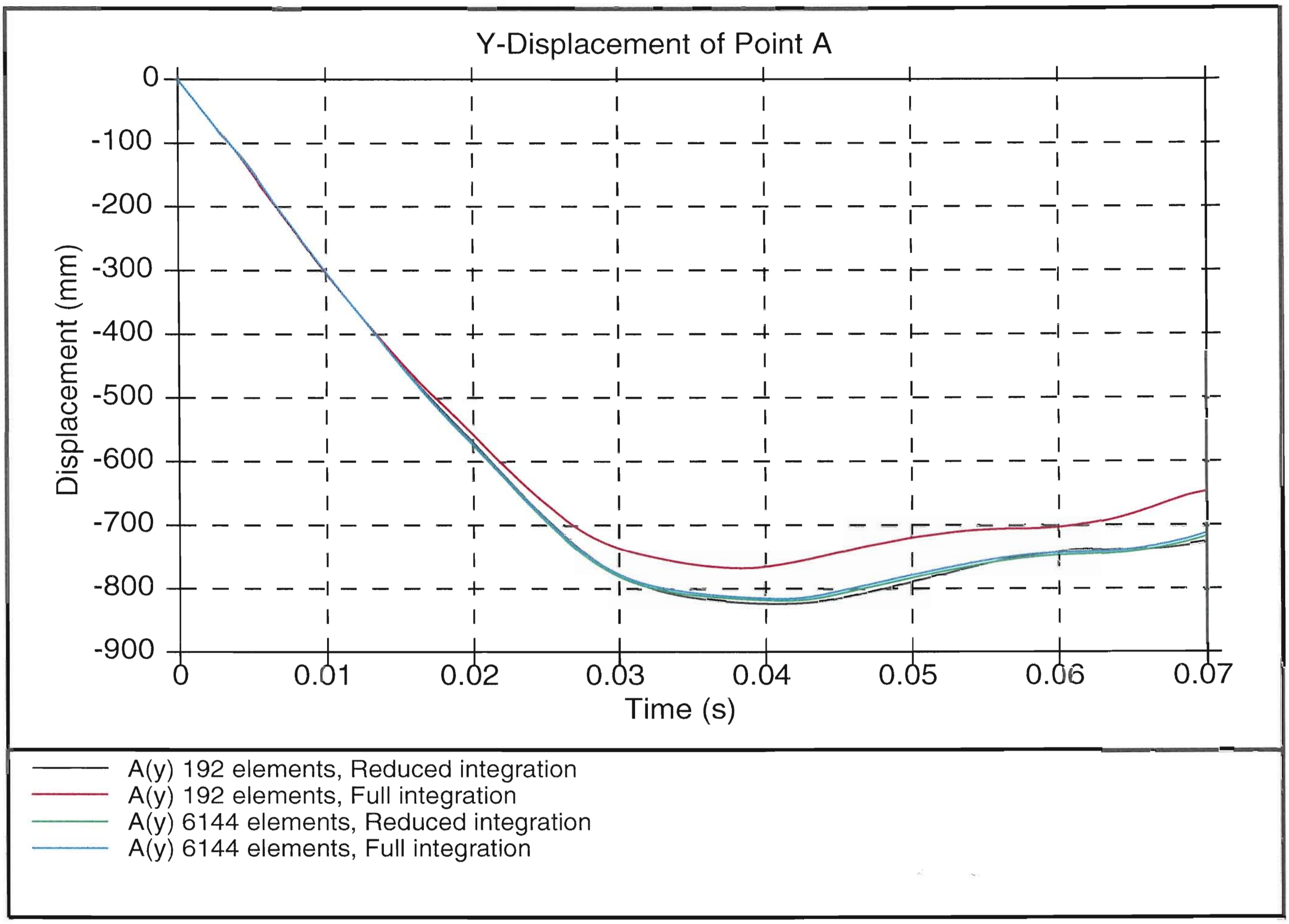
Benchmark 1 : Model with 6144 elements figure **C.2**

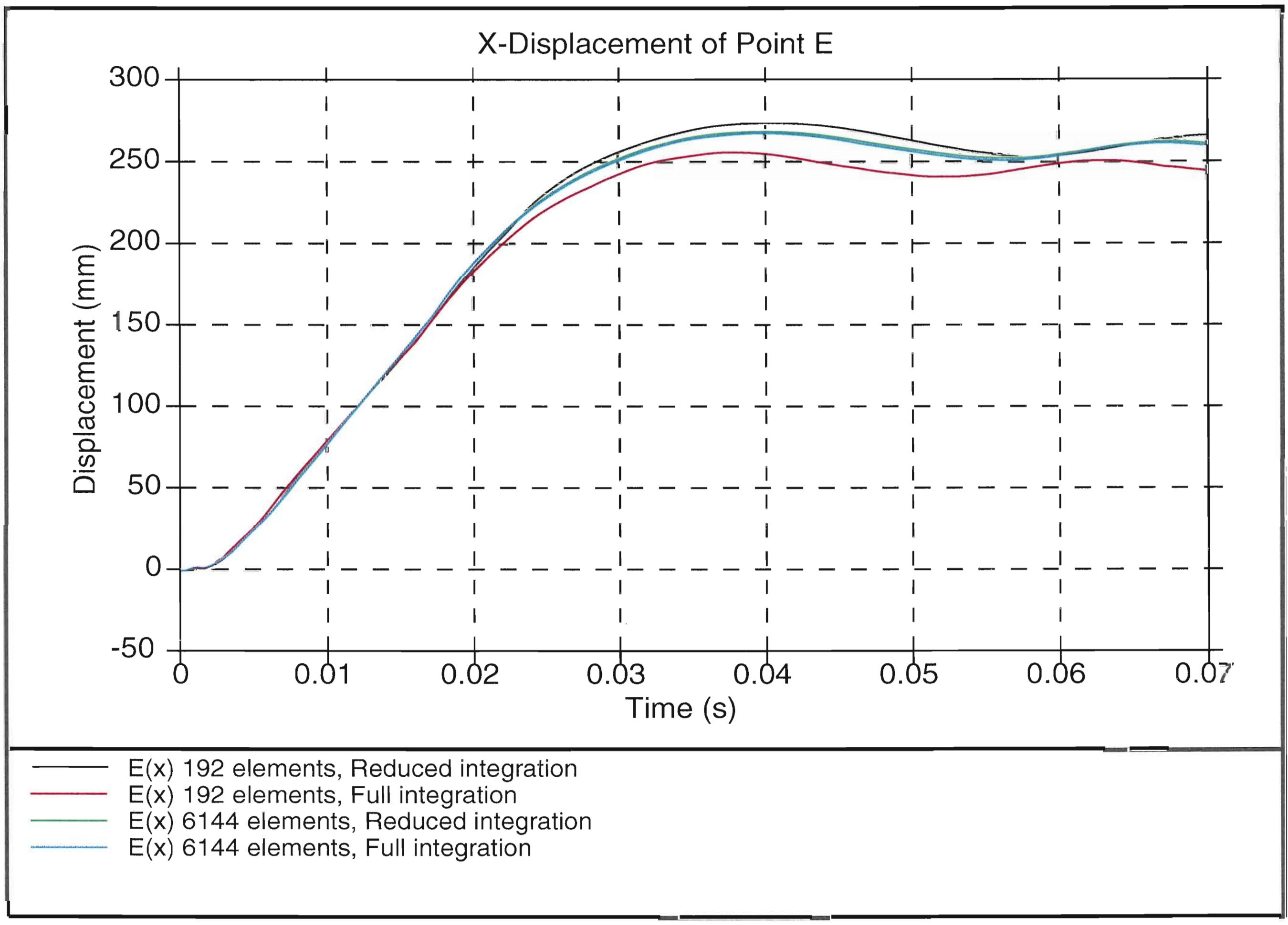


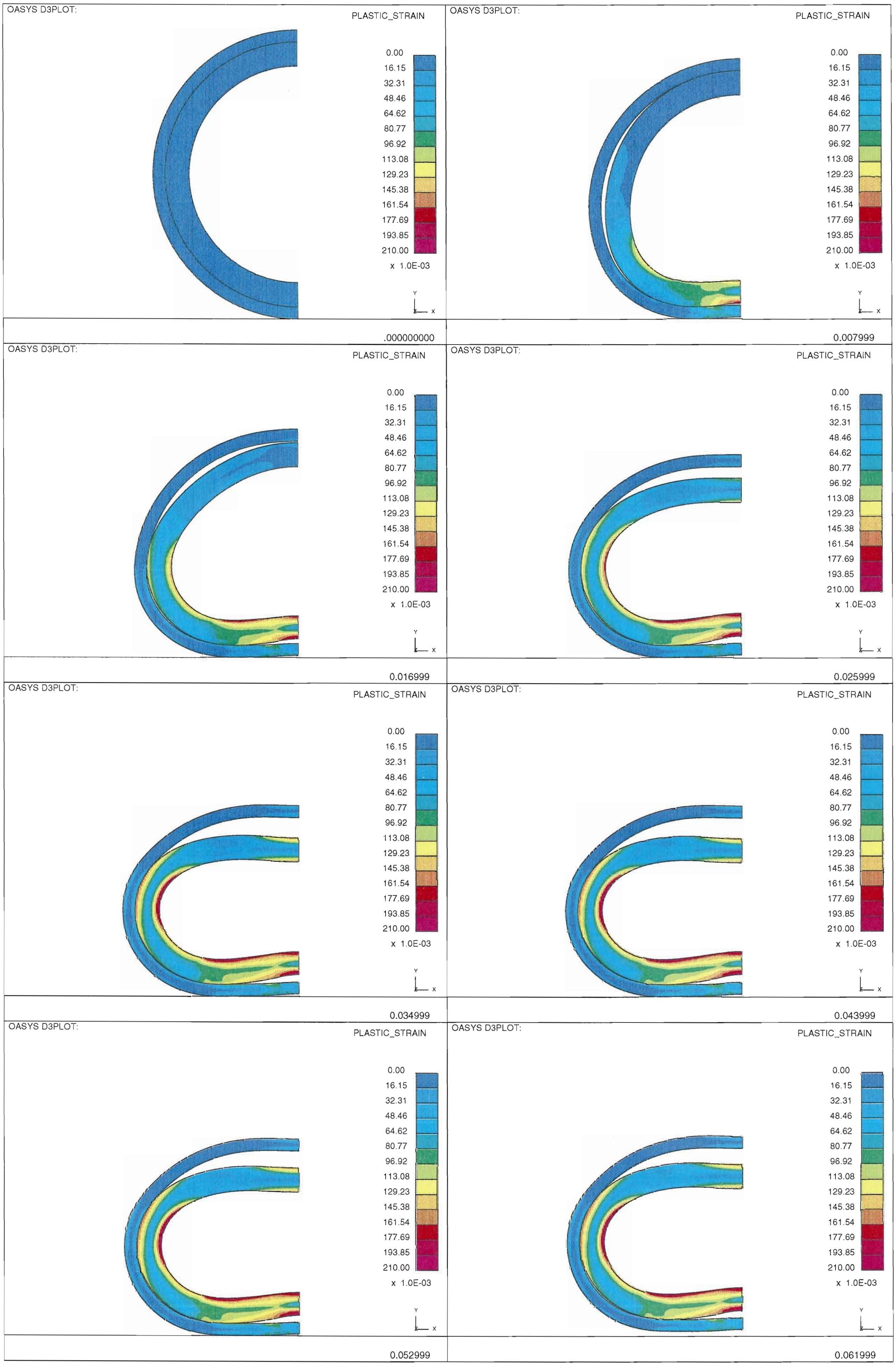


Benchmark 1: Mesh Density Study

figure C.3

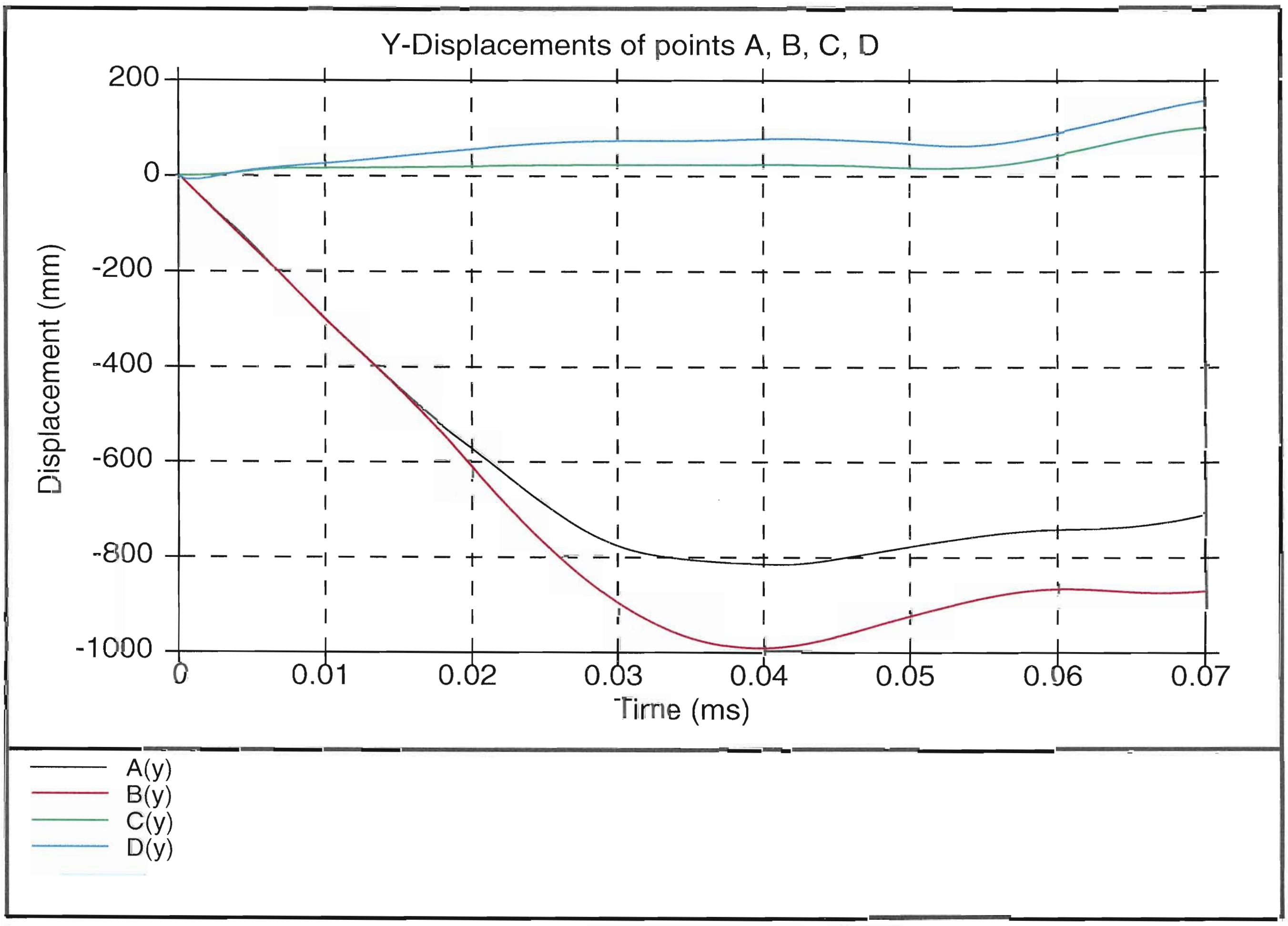

Y-Displacement of Point A
Displacement (mm)
0
-100
-200
-300
-400
-500
-600
-700
-800
-900
0
0.01
0.02
0.03
0.04
0.05
0.06
0.07
Time (s)
A(y) 192 elements, Reduced integration
A(y) 192 elements, Full integration
A(y) 6144 elements, Reduced integration
A(y) 6144 elements, Full integration

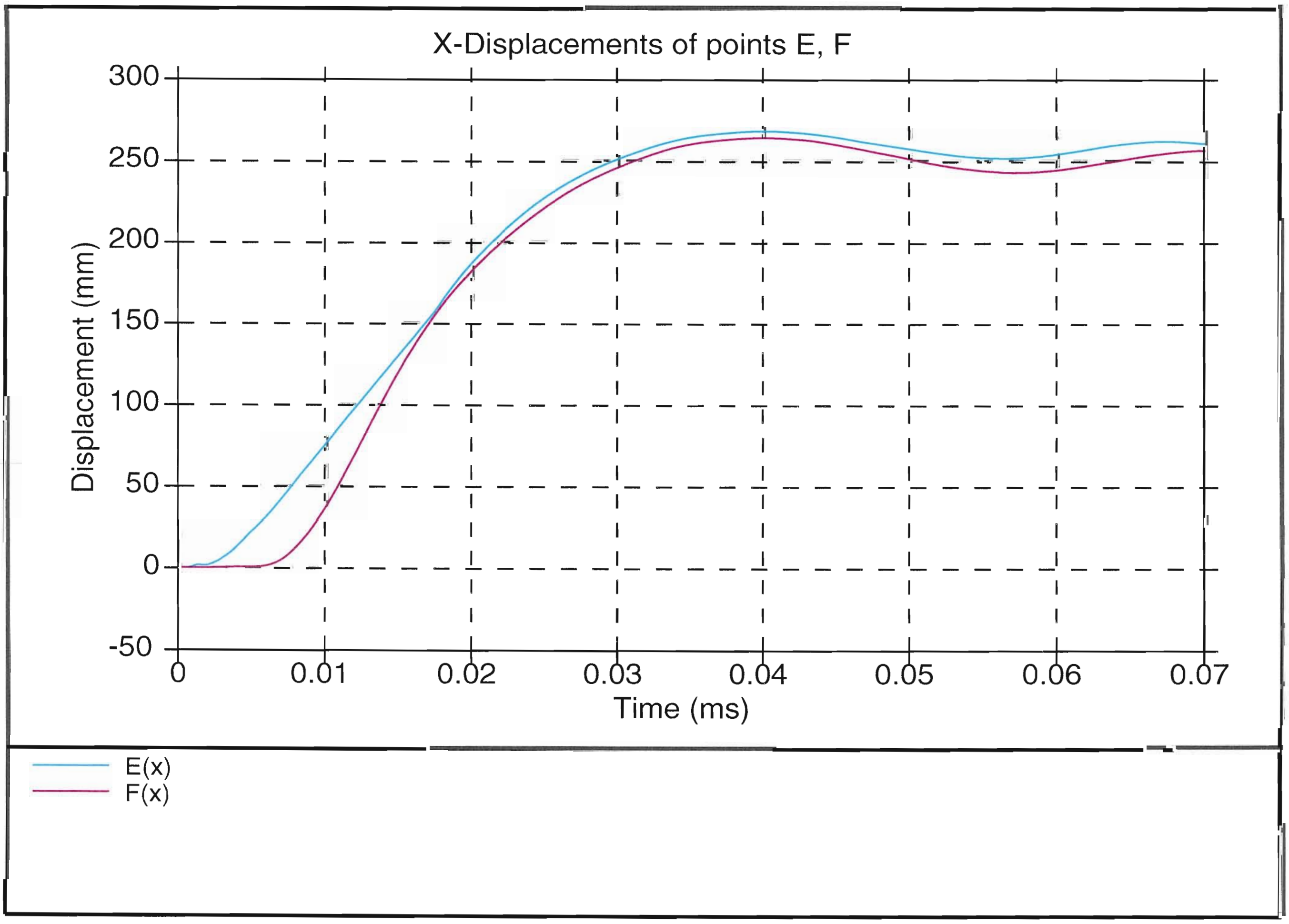

X-Displacement of Point E
Displacement (mm)
300
250
200
150
100
50
0
-50
0
0.01
0.02
0.03
0.04
0.05
0.06
0.07
Time (s)
E(x) 192 elements, Reduced integration
E(x) 192 elements, Full integration
E(x) 6144 elements, Reduced integration
E(x) 6144 elements, Full integration



Benchmark 1: Impact Sequence

figure **C.5**





Benchmark 1: Results from DYNA3D

figure **C.6**

Appendix D

LS-DYNA3D analysis of Benchmark 2 by OAPIL

D1. Introduction

This Appendix describes the analysis of Benchmark 2 carried out by OAPIL using LS-DYNA3D. A study was carried out to assess the effect of mesh density on the results, and to determine the number of elements to be used in the analysis. The analysis results required by the benchmark test are presented and discussed.

D2. Model description

The details of the model were taken from the description of Benchmark 2 given in Appendix B. The deformable cube was created from 8-noded hexahedral elements, with a high mesh density at the point of impact and a steady transition to a coarser mesh away from this point. Rigid 4-noded shell elements were fully connected to the top three faces of the deformable cube.

The material model 'MAT_PLASTIC_KINEMATIC' was used to represent the steel, with the relevant material properties being assigned. A hardening parameter 'BETA' of 1.0 was used, corresponding to an isotropic hardening law as specified in the benchmark.

The rigid target was defined as a 'RIGIDWALL' in LS-DYNA3D, and the interface between the target and the cube was assigned a Coulomb friction coefficient of 0.2. The model was positioned so that it was just in contact with the target at the beginning of the analysis, and an initial velocity of 13.3m/s towards the target was applied to all of the nodes.

D3. Mesh density study

A mesh density study was carried out to assess the effect that the number of elements used in the model would have on results. This involved creating three models, identical except for the number of elements. The three models, comprising 1000, 3375 and 8000 fully integrated solid elements are shown in Figures D.1 and D.2.

Each model was analysed in the impact scenario specified in the benchmark, and selected key results were compared. Figure D.3 shows the displacement histories for two points on each of the cubes. For Point A on the top of the cube, it can be seen that the results for the two more detailed models are in good agreement, whereas the displacements predicted by the model with the least number of elements differ by up to approximately 5mm. At point B, all analyses produce displacement histories within 1mm. This implies that 1000 elements is not quite sufficient for an accurate analysis, but a convergent solution can be obtained from a model with 3375 elements.

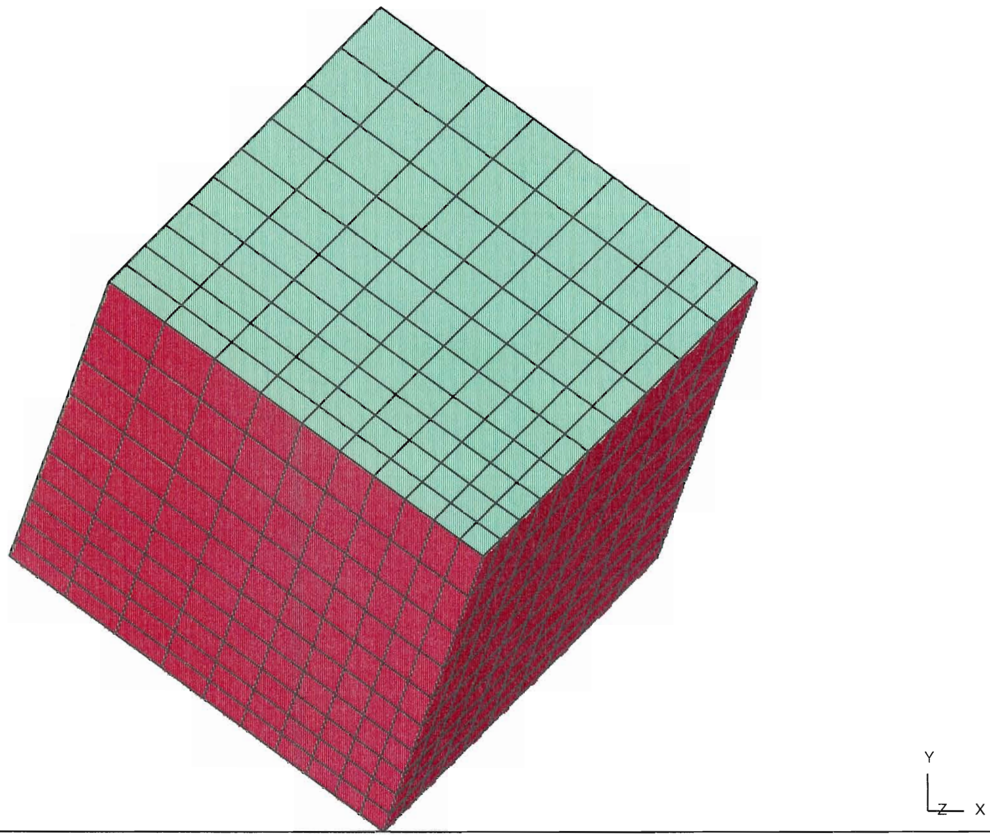
D4. Results

The results presented below are from the most complex model (8000 elements). Although the 3375 element model has been shown to be adequate, the more detailed model was used to provide marginally increased accuracy at the expense of computational efficiency.

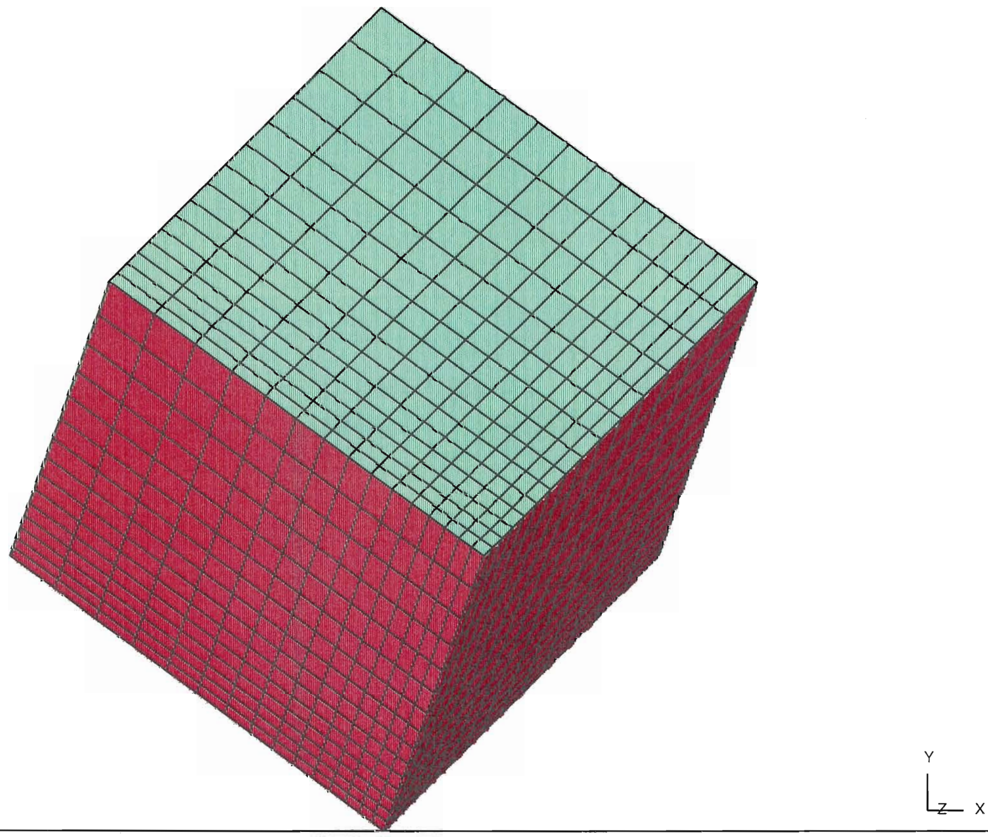
Figure D.2 shows the final state deformed shape of the model. The cube has begun to rebound from the target, and it can be seen from the flat profile of the deformed corner that very little elastic recovery has taken place. The plastic flow of material away from the impact zone has resulted in 'bulging' on the three downward faces of the cube.

Figure D.4 shows the y-displacement histories for points A and B, as required for the benchmark. It can be seen that the maximum compression of the cube occurs at approximately 18ms after first contact with the target. Following this, there is a short period of very slight elastic recovery before the cube rebounds, losing contact with the target at approximately 20ms after first contact.

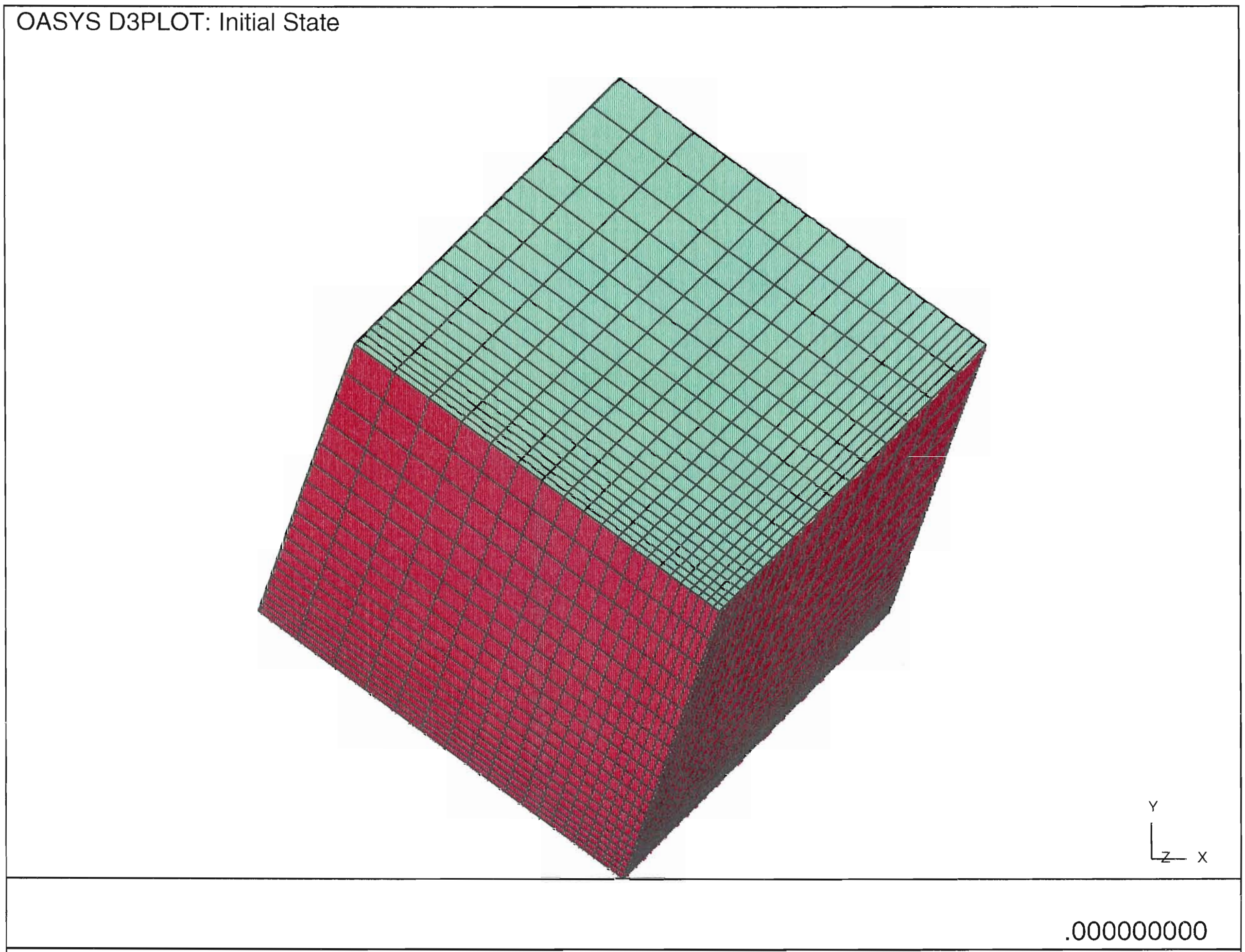
OASYS D3PLOT: Model with 1000 elements

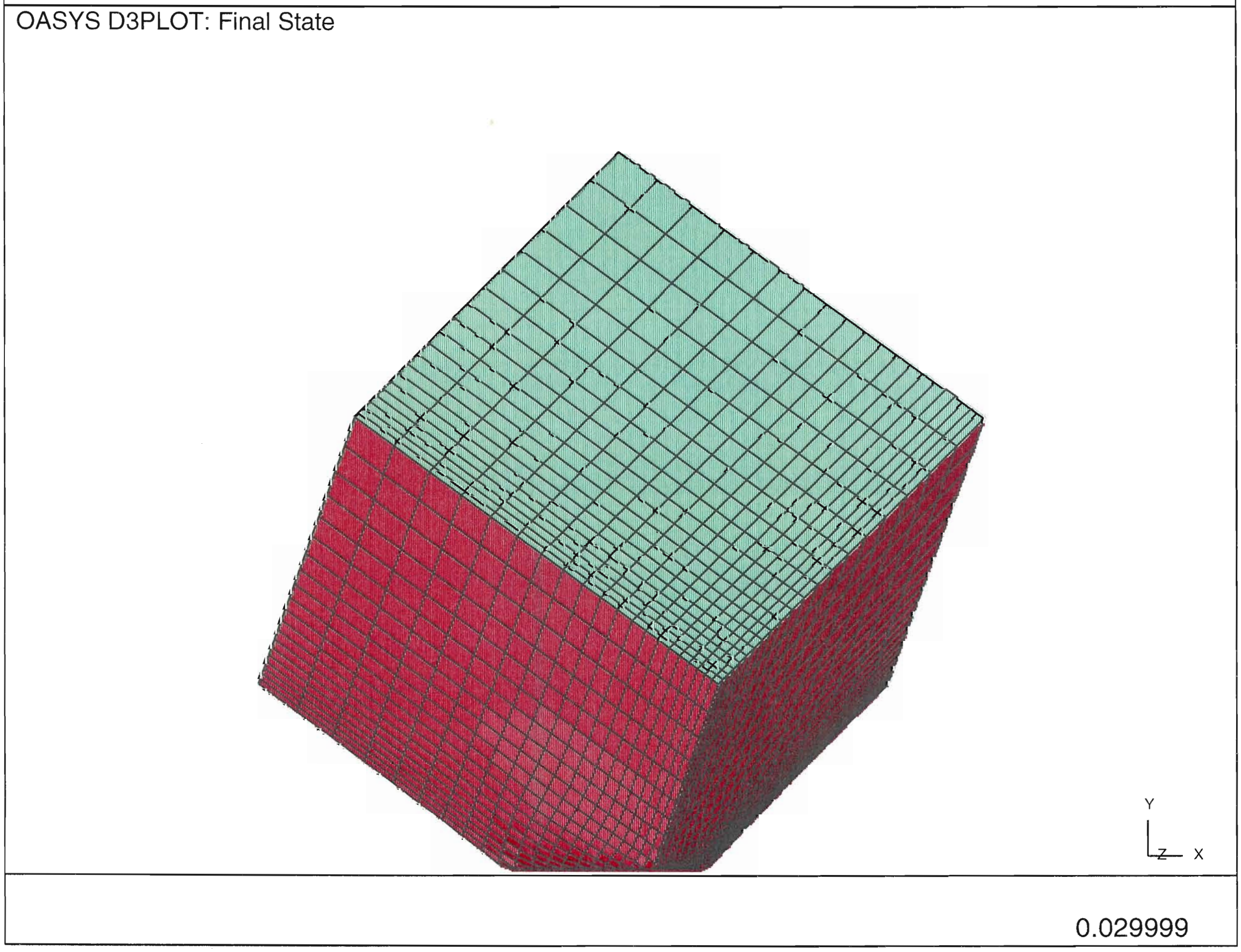


OASYS D3PLOT: Model with 3375 elements



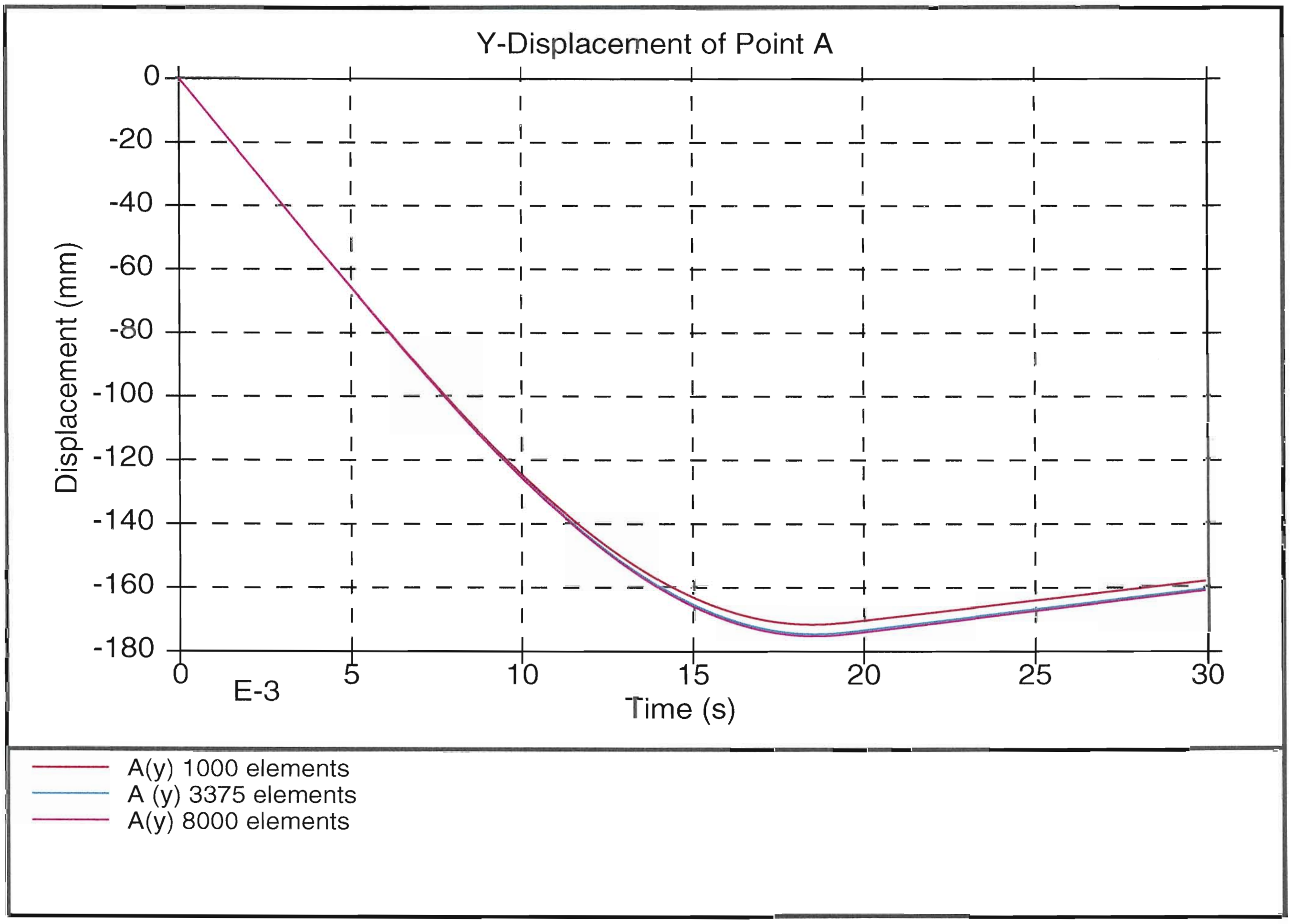
Benchmark 2: Models used for mesh density study figure **D.1**



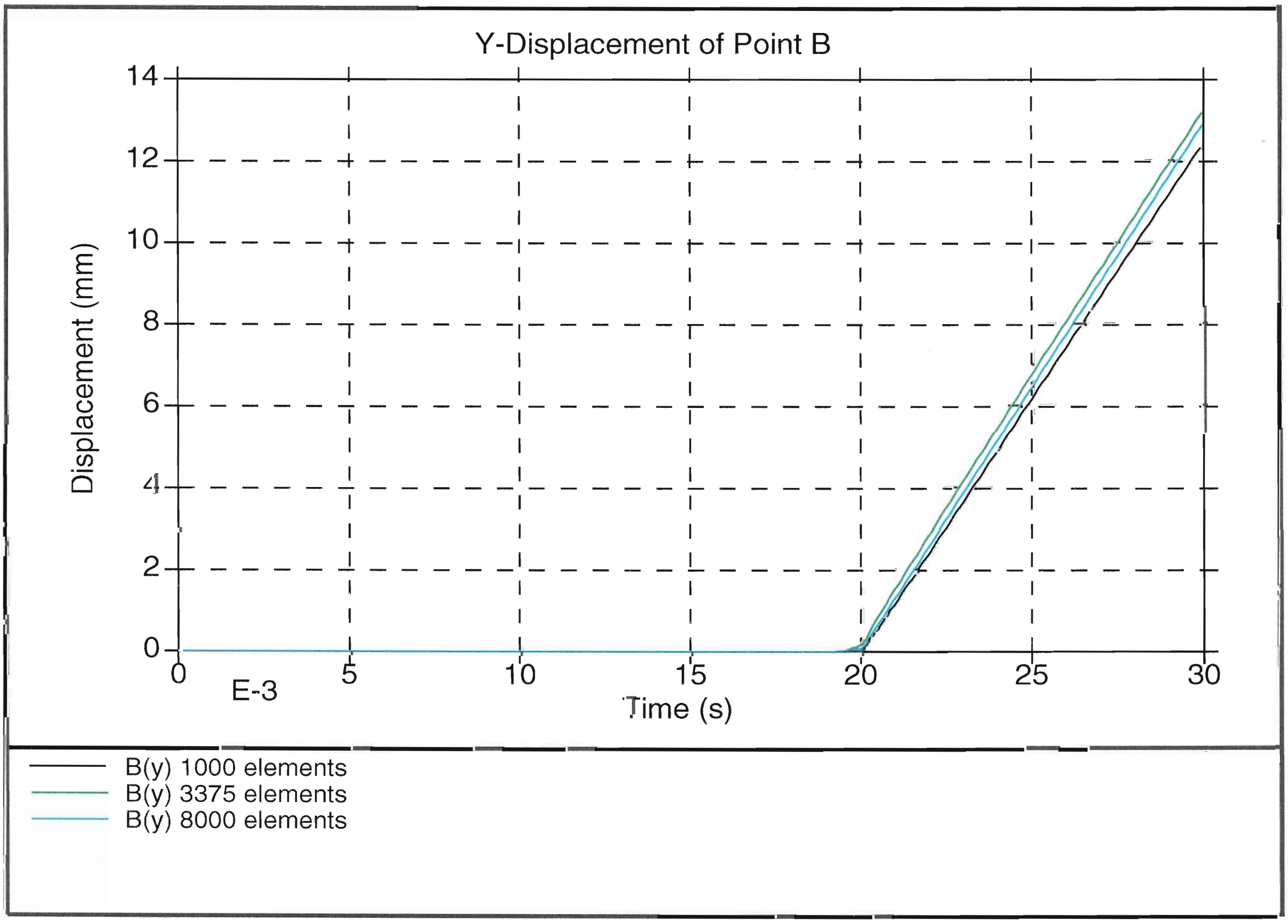


Benchmark 2: Model with 8000 elements

figure D.2



Y-Displacement of Point A
Displacement (mm)
0
-20
-40
-60
-80
-100
-120
-140
-160
-180
0
5
10
15
20
25
30
E-3
Time (s)
A(y) 1000 elements
A (y) 3375 elements
A(y) 8000 elements



Y-Displacement of Point B
Displacement (mm)
14
12
10
8
6
4
2
0
0
5
10
15
20
25
30
E-3
Time (s)
B(y) 1000 elements
B(y) 3375 elements
B(y) 8000 elements

Appendix E

Public Domain DYNA3D analysis of Benchmark 3 by GNB

E1. Introduction

This Appendix describes the public domain DYNA3D analysis of Benchmark 3. A mesh density study was carried out to determine an appropriate number of elements for the analysis. Additionally the influences of the modelled diameter of the impacting mass and the diameter of the target were studied parametrically. A full analysis of Benchmark 3 was carried out and the results required for the benchmark test were obtained.

E2. Model Description

The details of the model were taken from the description of Benchmark 3 given in Appendix B.

For analysing this problem the public domain version of the finite element code DYNA3D was chosen. This code is able to analyse 3-dimensional geometries under dynamic loadings with large deformations of the structure by using an explicit method. For mesh generation and preparation of the input-data file a pre-processor called INGRID was used. The post-processor TAURUS was used for output generation and obtaining the results.

Owing to the axisymmetric geometry of the benchmark problem a 2-dimensional model would be possible, but the code DYNA3D requires a 3-dimensional description of the geometry. By applying symmetry boundary conditions only a quarter section of the cylinder was modelled.

Eight-noded volumetric elements were used for modelling the wood, the impact mass and the target. The outer steel liner at the cylindrical surface of the cylinder was modelled by the use of 4-noded shell-elements.

An elastic-plastic material model was used to represent both the steel and the wood, with the material properties specified in Appendix B. To represent the specified ideal plastic behaviour of the wood, a hardening parameter of 1.0 was used.

The target and the impacting mass were each defined as a rigid, and the interfaces at the contact surfaces of different material were represented by a contact algorithm which was able to take Coulomb friction into account.

E3. Input description

In order to determine an appropriate finite element model for Benchmark 3 a mesh density study was performed, and also a study of the impact and target diameter.

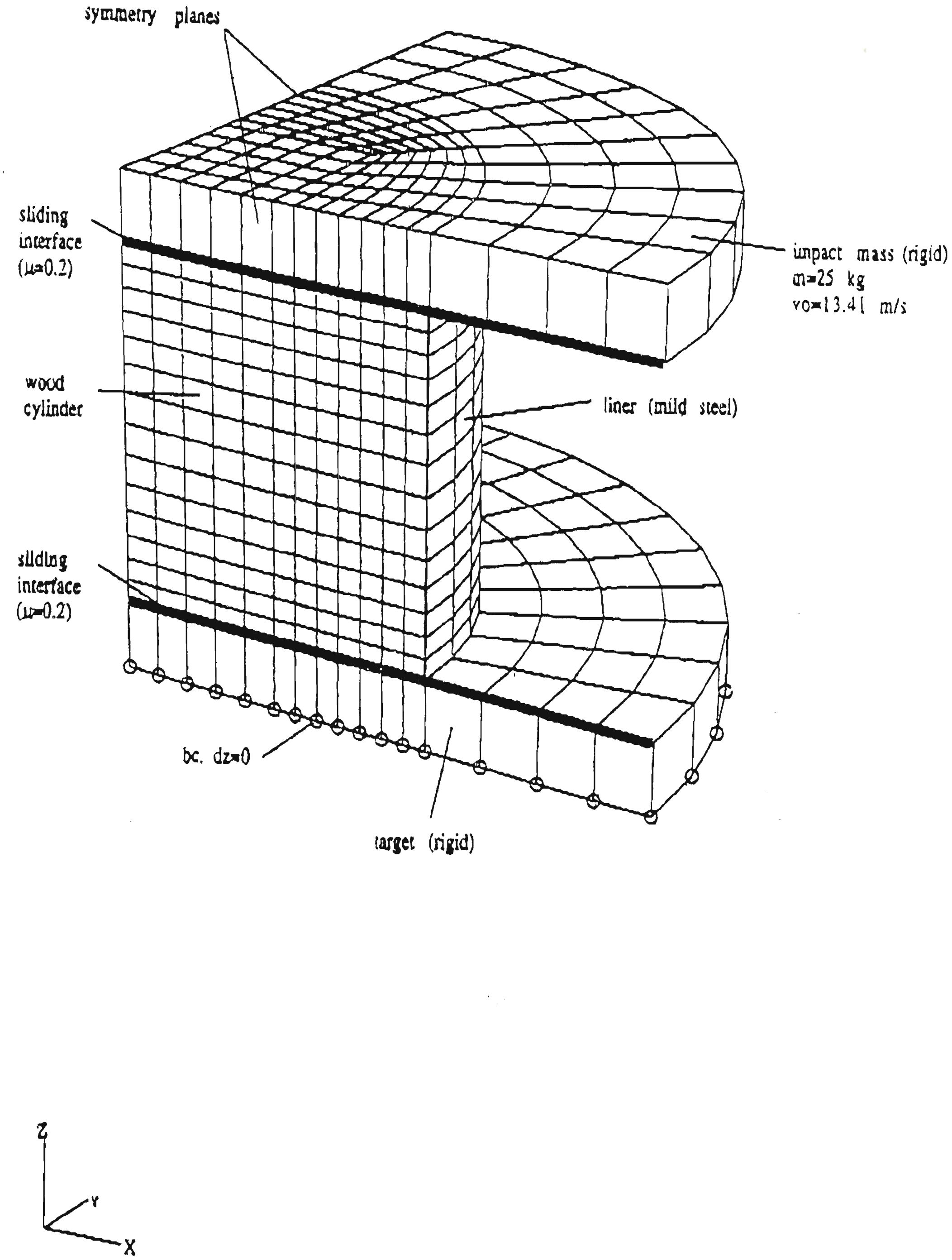
The geometry and mesh size shown in Figure E.1 were chosen for the final calculation. The model consists of :

- for wood, 1680 volumetric 8-noded elements
- for the outer steel liner, 140 shell elements
- for the impacting mass and target, 160 volumetric elements each.

E4. Results

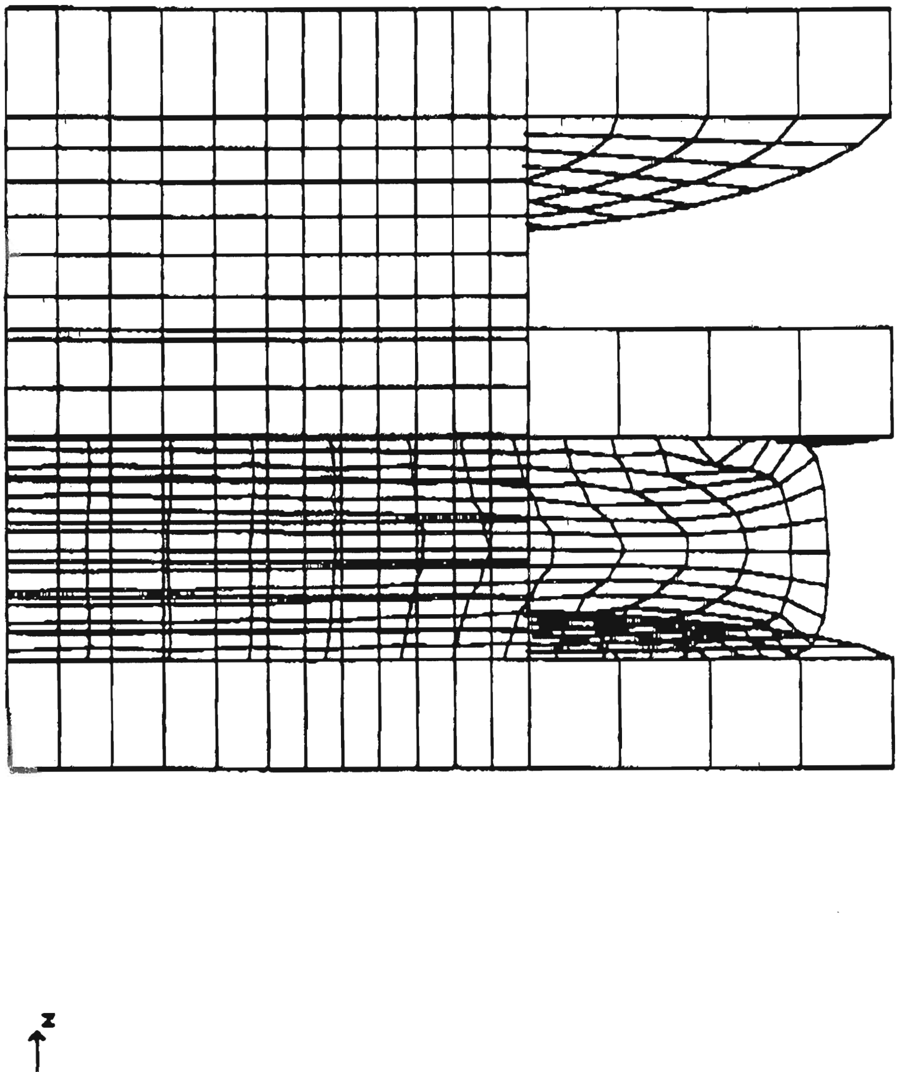
The shapes of the undeformed and deformed geometries are shown in fig. E.2. The maximum deformation of the wood reaches a value of 29.7mm.

Fig. C.3 shows the deformation history during the impact. The highest deformation of the cylinder is reached after an impact duration of 0.037s. Fig. C.4 shows the impact force history. The maximum force of 0.54MN is obtained after approximately 0.0345s.



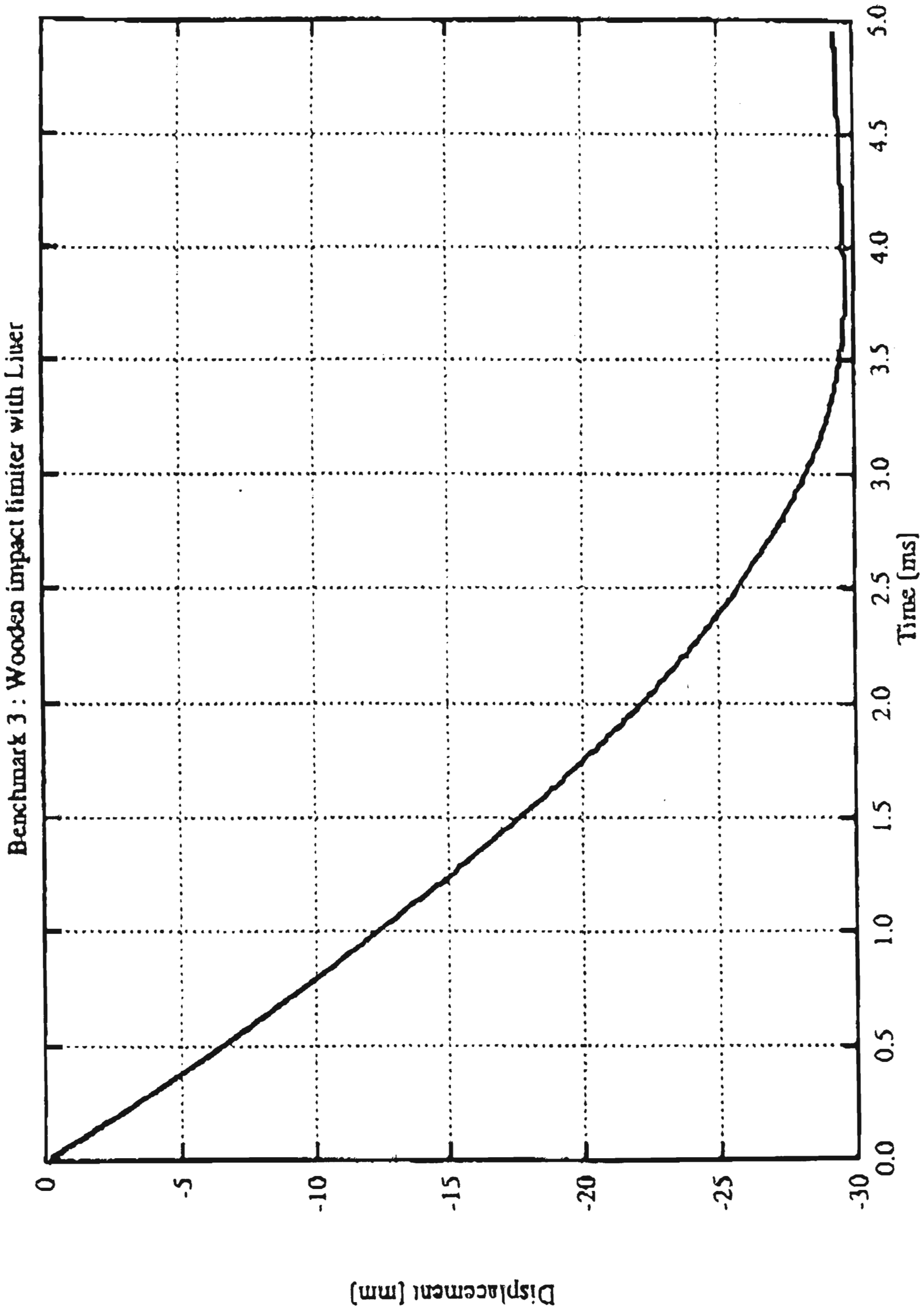
GNB	Benchmark 3 Meshed Model of the Geometry	Fig. E.1

Benchmark 3 : Wooden impact limiter with Liner
time = .37000E-02

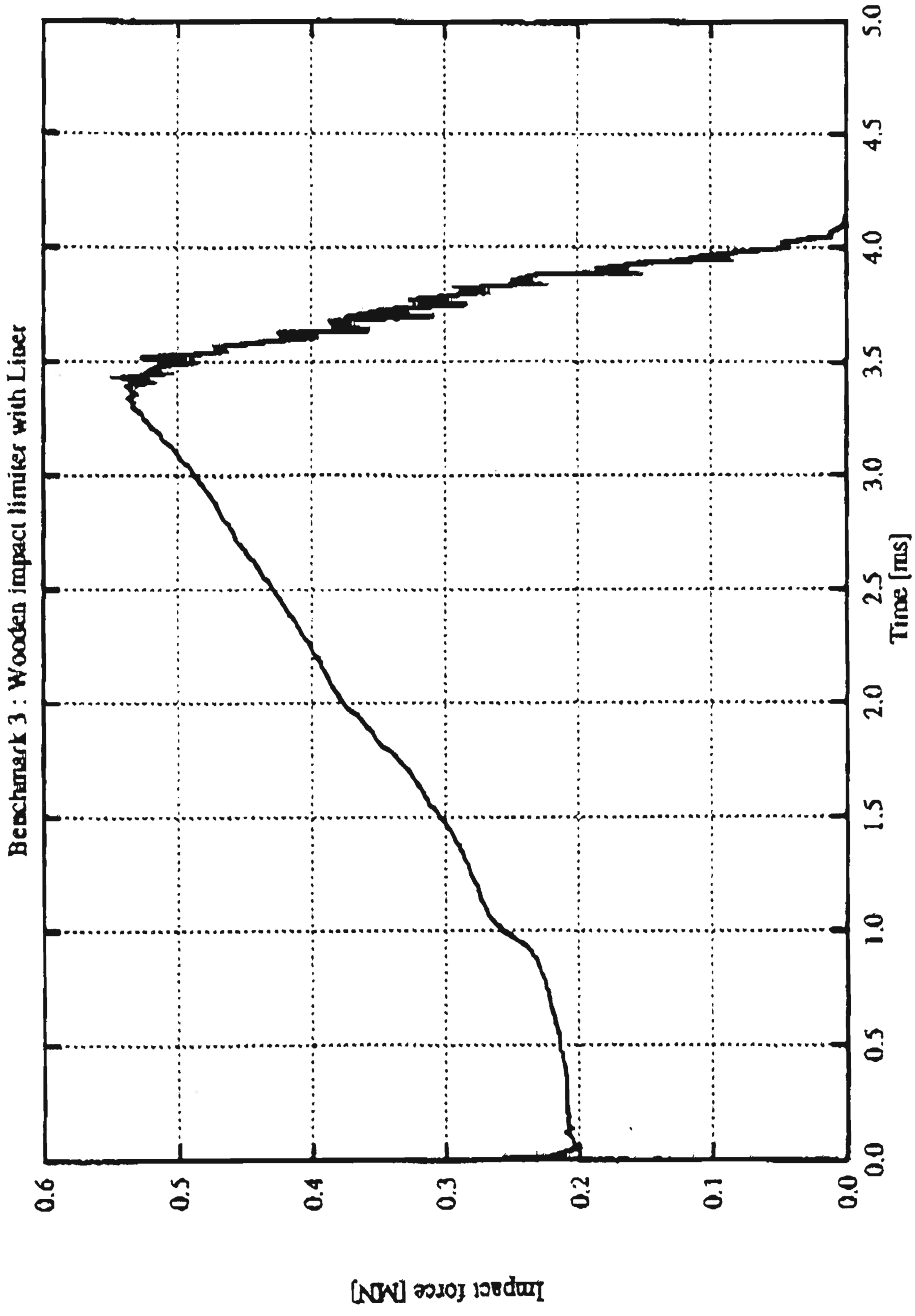


disp. scale factor = .100E+01 (default)

GNB	Benchmark 3 Comparison of Deformed and Undeformed Geometry	Fig. E.2



GNB	**Benchmark 3** Deformation versus Impact Time	Fig. E.3



GNB	Benchmark 3 Impact Force versus Impact Time	Fig. E.4