CHALMERS

Evaluation of finite element tools for transient structural dynamic simulations of firing systems

Master's Thesis in the Solid and Fluid Mechanics Masters Program

MIKAEL BJÖRKMON

Evaluation of finite element tools for transient structural dynamic simulations of firing systems

Master's Thesis in theThesis in the Solid and Fluid Masters Program

MIKAEL BJÖRKMON

Evaluation of finite element tools of firing systems
Master's Thesis in the Solid and Fluid Masters Program

MIKAEL BJÖRKMON

Evaluation of finite element tools of firing systems
Master's Thesis in the Solid and Fluid Mechanics Masters Program
MIKAEL BJÖRKMON
Department of Applied Mechanics
Division of Dynamics
Chalmers University of Technology

BAE System Bofors observes a potential in using explicit finite element solvers for dynamic simulations in weapon platforms. Two of these solvers are AutoDyn and LS-Dyna. Several interfaces (Ansys Workbench, Ansys Classic, LS PrePost, AutoDyn Standalone) for using these solvers are available on the market, where Ansys is the most used one. A major part of the report covers an evaluation of the explicit method for the mentioned purpose, especially using the Ansys Workbench interface connected with the AutoDyn solver (AWBAD). Minor parts of the report cover other interfaces and solvers.

The AWBAD suite and several of other interfaces with both the AutoDyn and LS-Dyna solver are used for computing structural response on a 0.01x0.01x1m beam. The softwares give the same natural frequencies when the beam is free oscillating, but the displacement differs a bit between the softwares.

A real weapon platform (BAE System Bofors Lemur) has been modelled and simulated with AWBAD. Quick and easy transfer/workflow from CAD model to computational model is an advantage for the software. The software shows disadvantages in the pre-processing part for firing simulations. This depends on the lack of possibility to model point masses, springs and dampers which are important for firing simulations. There is also too few options for the boundary conditions for the software being considered as fully usable.

A snap back test has been performed on the Lemur where one of its consoles was pre-stressed and released. The test correlates well with the simulated results.

The recommendation is to use AWBAD as long it is possible due to its limitations. In other cases there are alternatives which can be found in the report.

Utvärdering av FE-verktyg för transientdynamiska förlopp för skjutande system
Examensarbete inom masterprogrammet Tillämpad mekanik
MIKAEL BJÖRKMON
Institutionen för tillämpad mekanik
Avdelningen för Dynamik
Chalmers tekniska högskola

SAMMANFATTNING

BAE Systems Bofors AB ser en potential i att använda explicita finita elementlösare för tidsberoende strukturdynamiska förlopp för vapenplatformar. Två av dessa lösare är AutoDyn och LS-Dyna. Flera gränssnitt för att använda dessa lösare (Ansys Workbench, Ansys Classic, LS PrePost, AutoDyn Standalone) finns tillgängliga på marknaden där Ansys är den mest använda. Största delen av denna rapport omfattar en utvärdering av explicita metoder för det nämnda syftet, där Ansys Workbench är användargränssnittet vilket är ihopkopplat med AutoDyn-lösaren (AWBAD). Mindre delar av rapporten täcker andra gränssnitt och den andra lösaren, LS-Dyna.

AWBAD och andra gränssnitt tillsammans med de båda lösarna AutoDyn och LS-Dyna används för att modellera och simulera dynamisk respons på en balk med dimensionerna 0.01x0.01x1m. Programvarorna ger ungefär samma frisvängningsfrekvenser inbördes. Denna frekvens överensstämmer med en analysisk modalanalys. Förskjutningarna för balken skiljer sig inbördes mellan programvarorna.

En verklig vapenplattform (BAE Systems Bofors Lemur) har modellerats och simulerats med AWBAD. En fördel med AWBAD är den enkla och snabba arbetsgången mellan CAD-modell och lösningsfärdig modell. Programvaran visar på brister i pre-processing delen när skjutande system ska modelleras. Dessa brister beror på avsaknaden i punktmassor, fjädrar och dämpare, vilka är viktiga att kunna modellera i skjutande vapensystem. Något för få randvillkor finns tillgängliga för att programvaran ska anses vara fullt användbar.

Ett snap back test har gjorts på vapenstationen Lemur där en av konsolerna spänns upp och släpps så att strukturen kan frisvänga. Resultatet av testet övensstämmer bra med den framsimulerade lösningen.

Rekommendationen är att använda AWBAD så länge det är möjligt med hänsyn till dess begränsningar. I rapporten ges det alternativ när andra programvaror kan komma att behöva användas.

Contents

Preface

The aim of this work was to investigate if explicit finite element methods could be used to simulate the firing of weapon platforms. This work has been carried out between February 2010 and July 2010 at BAE Systems Bofors in Karlskoga, Sweden with help from the department of Applied Mechanics, Chalmers University, Sweden. Supervisors at BAE Systems Bofors were Anders Ekberg and Sven Strömberg. Examiner at Chalmers was Professor Thomas Abrahamsson.

I want to say especially thanks to both my supervisors at BAE Systems. Further I want to thank all the other people who have helped me during this work.

Göteborg, September 100901

Mikael Björkmon

Notations

Ansys (Workbench and Classic), LS-Dyna and AutoDyn terms will be used for naming the pre-and post-processing parts for the simulations in this report. This will be done so the user can redo this work.

Notations for the softwares and modules:

AWB: Ansys Workbench

AWBIS: Ansys Workbench Implicit Static Structural

AWBIT: Ansys Workbench Implicit Transient Structural

AWBM: Ansys Workbench Modal Analysis

AWBAD: Ansys Workbench Explicit AutoDyn

AWBLSD: Ansys Workbench Explicit AutoDyn LS-Dyna

ACL: Ansys Classic

ACLLSD: Ansys Classic Explicit LS-Dyna

ACLI: Ansys Classic Implicit Transient Structural

SAD: Standalone AutoDyn

1 Introduction

1.1 Background

In the process of developing artillery and naval guns and other weapon systems the systems are generally designed against recoil forces. The recoil load has a time range from some few milliseconds up to several milliseconds. The recoil load will be smeared out in the platform.

Today explicit finite element solvers are included in the package that the company uses for structural simulations. The company observes a potential of simplifying the workflow by using the new explicit solvers.

1.2 Purpose

The purpose of this study is to examine if explicit methods can be used for firing simulations for evaluating transient response in the weapon platform. If so, then the wish is to examine how the workflow should run and how the usability is for these softwares in this type of simulations.

1.3 Task

The main task is to evaluate if an explicit solver can be used for simulating transient dynamic response for shooting systems. Advantages and disadvantages for an explicit solver should be evaluated.

Modelling should be done in Ansys Workbench and the available solvers are AutoDyn and LS-Dyna, at least one of these solvers should be used.

The response in the platform from the explicit transient simulations can be compared to old calculation methods, to measured data, or to implicit transient simulations.

1.4 Method

The method of evaluating the usability of the explicit method and the software has been carried out in mainly three steps:

1. Create some criterions that are important due to

- Numerical results

- Workflow/usability for the software

2. Evaluate the numerical results with AWBAD, AWBLSD, ACLI, ACLLSD softwares by modelling and simulating a beam

3. Model, simulate and test a Lemur so both numerical results and the workflow can be evaluated for this weapon platform

1.5 Limitations

One load case. For the simplicity one load case for the weapon platform is considered.

No other CAD software is to be used. External software as Solidworks can be used before the geometric modelling part in the finite element software.

Linear elastic material model. The firing of a weapon should not give any plasticity in the platform.

No structure optimization. The work performed in this report is to evaluate the explicit software, not use it for optimization.

2 Theory

An old handbook method has been used for a long time within the company. The old methods are shortly described in this chapter. Theory of explicit time stepping is described here. Theory for the solvers can be found in their respective manuals. General finite elements are not described in this report. Derivations for equations are not done in this report.

2.1 Old methods for transient structural dynamics

The paper *Dynamisk belastning, 1960* [1] describes methods used for calculating response in weapon platforms for a given shooting load.

In short terms the method uses the lowest active eigen frequency (time T) compared to the time (t) which is the duration of reaction force from the bullet in the barrel. For a given shape of the reaction force and quotient t/T, there can be found magnification factors, which can be multiplied with static stresses to get the dynamic stresses. This method has in some cases been found to give conservative results.

2.2 Explicit finite elements

AWBAD and several of other explicit softwares uses a Leap Frog scheme for the time stepping (equation (2)-(4) and Figure 2-1) when solving the conservation of momentum equation (1). It can be noticed that the Leap Frog scheme uses already computed values in the right hand side of the equation (explicit method). As for all explicit schemes the leap frog scheme must follow the Courant–Friedrichs–Lewy (CFL) (equation (5)) condition for the largest time step.

$$\rho \ddot{x} = b_x + \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \sigma_{xy}}{\partial y} + \frac{\partial \sigma_{xz}}{\partial z}$$

$$\rho \ddot{y} = b_y + \frac{\partial \sigma_{yx}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \sigma_{yz}}{\partial z} \qquad (1)$$

$$\rho \ddot{z} = b_y + \frac{\partial \sigma_{zx}}{\partial x} + \frac{\partial \sigma_{zy}}{\partial y} + \frac{\partial \sigma_{zz}}{\partial z}$$
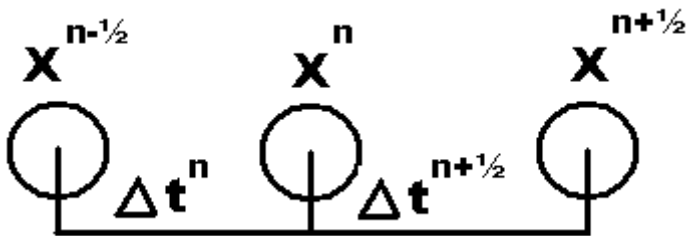
Figure 2-1: Leap Frog Scheme

$$\ddot{X}_i = \frac{F_i}{m_i} - b_i \qquad (2)$$

$$\dot{X}_i^{n+1/2} = \dot{X}_i^{n-1/2} + \ddot{X}_i^n \Delta t^n \qquad (3)$$

$$X_i^{n+1/2} = X_i^n + \dot{X}_i^{n+1/2} \Delta t^{n+1/2} \qquad (4)$$

Where

$\dot{X}_i$ components of nodal accelerations

F_i forces acting on the nodal points

b_i components of body acceleration

m_i mass attributed to the node i

n time step number

From these equations, combined with a characteristic size of the element e and the local material soundspeed the element e, the CFL condition can be derived to:

$$\Delta t \leq f \min\left[\frac{h_e}{c_e}\right] \qquad (5)$$

Where

Δt largest time increment or CFL time

f time step factor

c_e local material soundspeed in element e

h_e characteristic length of element e
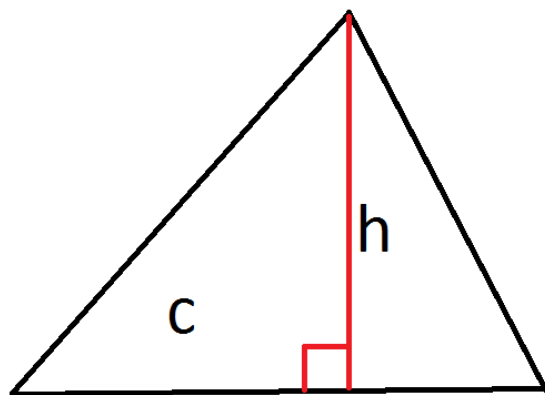
Figure 2-2: Element with characteristic length (red)

$$\dot{X}_i^{n+1/2} = \dot{X}_i^{n-1/2} + \ddot{X}_i^n \Delta t^n$$

2.3   Typical consequences for a explicit solver

From the Ansys Manual [1] due to calculation of accelerations and Leap from scheme these consequences are presented:

- The equations become uncoupled and can be solved directly (explicitly)

- No convergence checks are needed since the equations are uncoupled

- No inversion of the stiffness matrix is required. All nonlinearities (including contact) are included in the external force vector F_i

2.4   Extended theory

Two important things for explicit finite element methods are mass scaling and element formulations. The element formulations are improved all the time. Some of the theory of these two things is presented in:

Theory mass scaling/lumping can be studied in [4], [5]

Theory of Average Nodal Pressure can be studied in [6], [7], [8], [9]

3   Beam Simulation

Several softwares have been used for transient simulation of a beam made out of two shorter beams with a *bonded* contact between the two parts. The softwares are ABWAD, AWBLSD, ACLLSD, ACLI, SAD and AWBM. The aim of these simulations is to test how mutually equal the results will be and how these results will correlate with an analytical solution. For evaluating the amount of hourglass energy one simulation has been performed on the beam and on a similar beam but with three contacts made out of four smaller beams.

The following have been compared between the softwares:

- Total, Kinetic, Internal, Hourglass Energy, Energy error (SAD)

- Period time (Analytical, AWBM, ACLI, ACLLSD, AWBAD)

- Global deformation (ACLI, ACLLSD, AWBAD)

3.1   Simulation setup

The set up of the explicit simulations are described first generally for the beam and than separate for each software. For the ACL simulations the APDL scripts can be found in the appendix.

3.1.1   General setup for the explicit simulations

Dimensions: 0.01x0.01x1 m

Steel with density 7800 kg/m^3, Young's modulus 200 GPa

5 mm element size

Bonded contact in the middle of the beam between the two parts, see Figure 3-1

Fixed support at the end B, see Figure 3-1

Rectangular shaped impulse with a duration time of 1 ms and 100 N in amplitude at the end A, see Figure 3-1
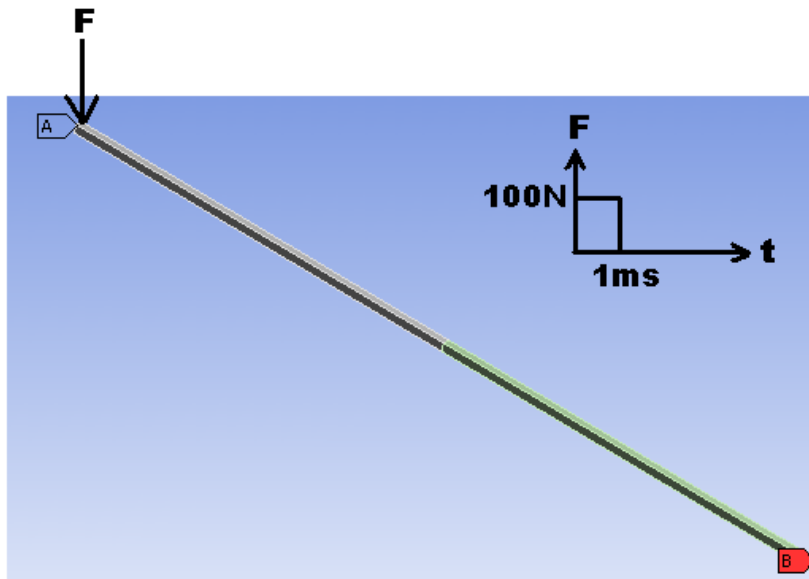
Figure 3-1: Beam with load (left end) and fixed support (right end)

3.1.2   Simulation setup for AWB AD/LSD and SAD

Tetrahedron elements

Double precision

Visualisation points: 250

The SAD simulation are directly imported from AWBAD and set to log hourglass energies. One and three contacts used equally spaced from each other.

3.1.3   Simulation setup for ACLLSD

Tetrahedrons

Script as can be seen in the appendix A2

3.1.4   Simulation setup for ACLI

Bricks

Script as can be seen in the appendix A3

3.2   Results

Results from the simulations are presented in Figures 3-2 to 3-5. The first natural frequency for the beam is 8.1 Hz. The results are discussed in section 3.3.

Figure 3-2: Results different softwares/solvers

Figure 3-3: Results hourglass energy from SAD
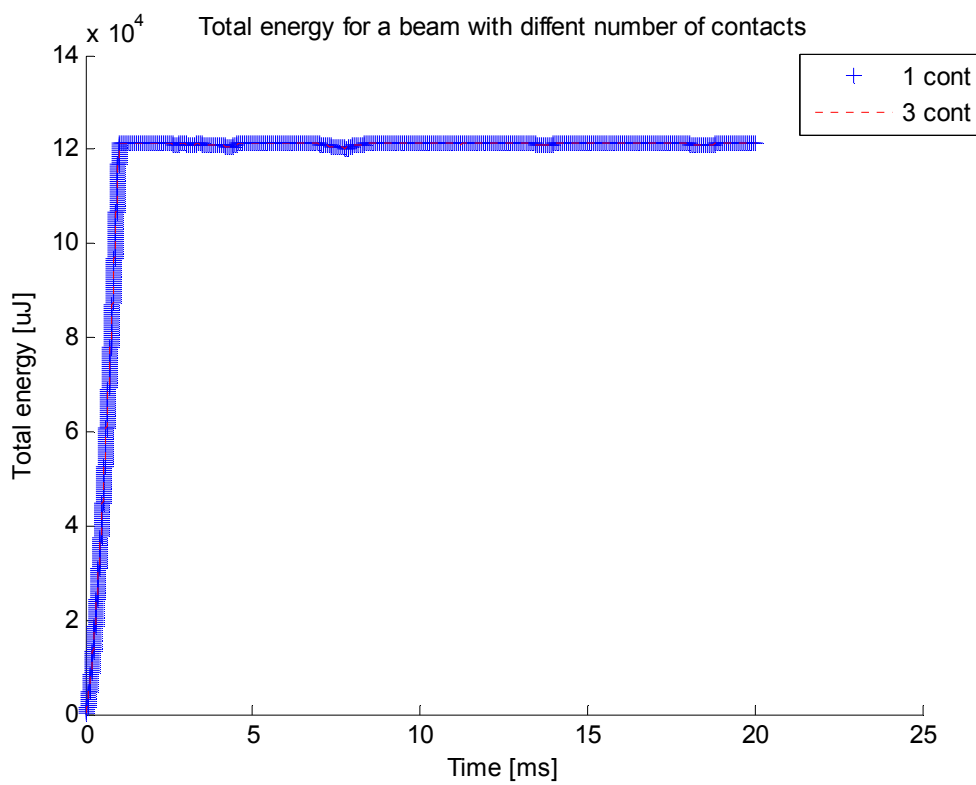
Figure 3-4: Total energy from SAD
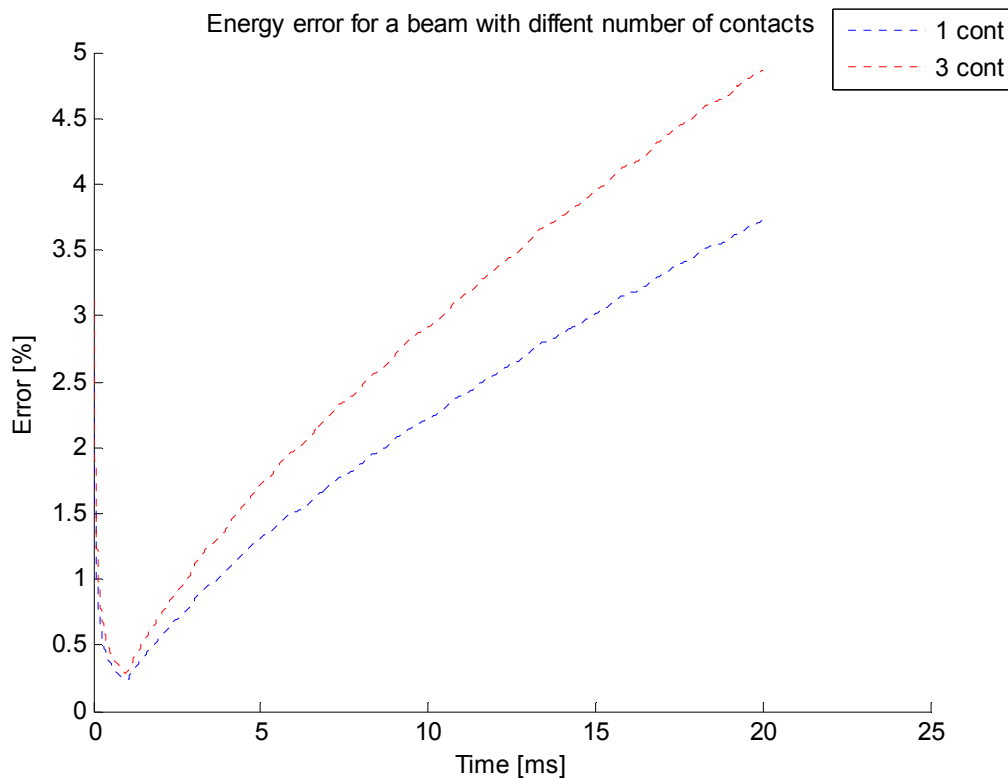
Figure 3-5: Energy error between total and hourglass energy for 1 and 3 contacts

3.3 Conclusions

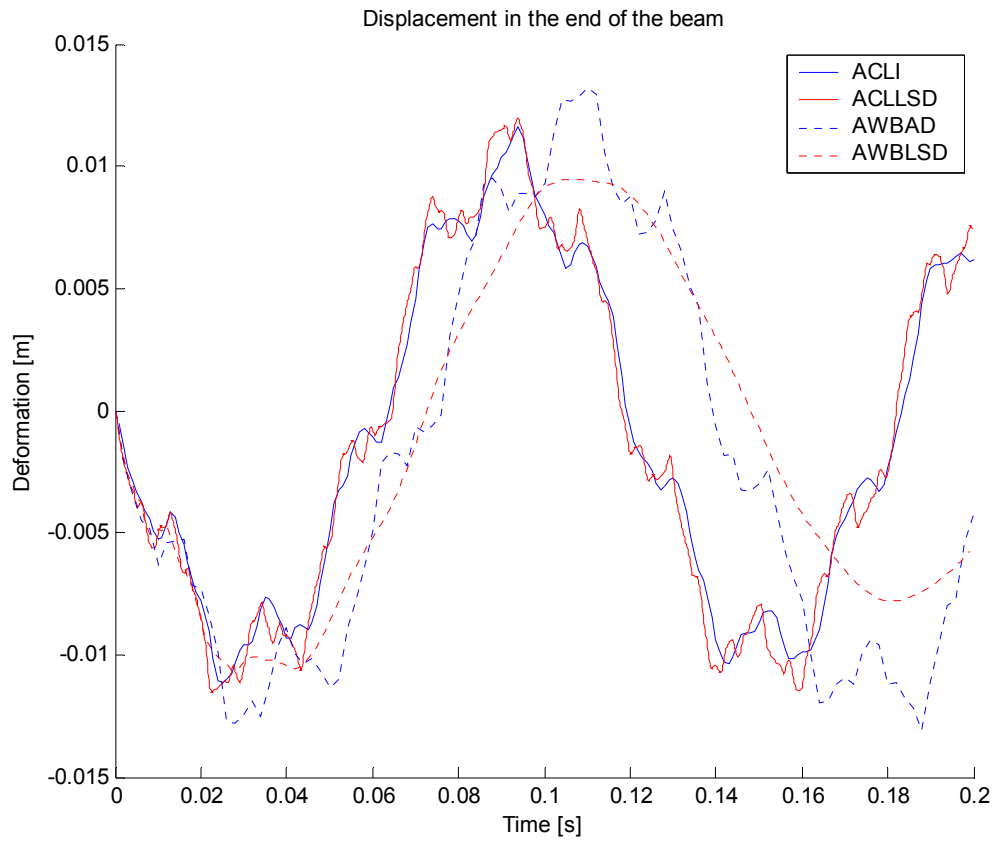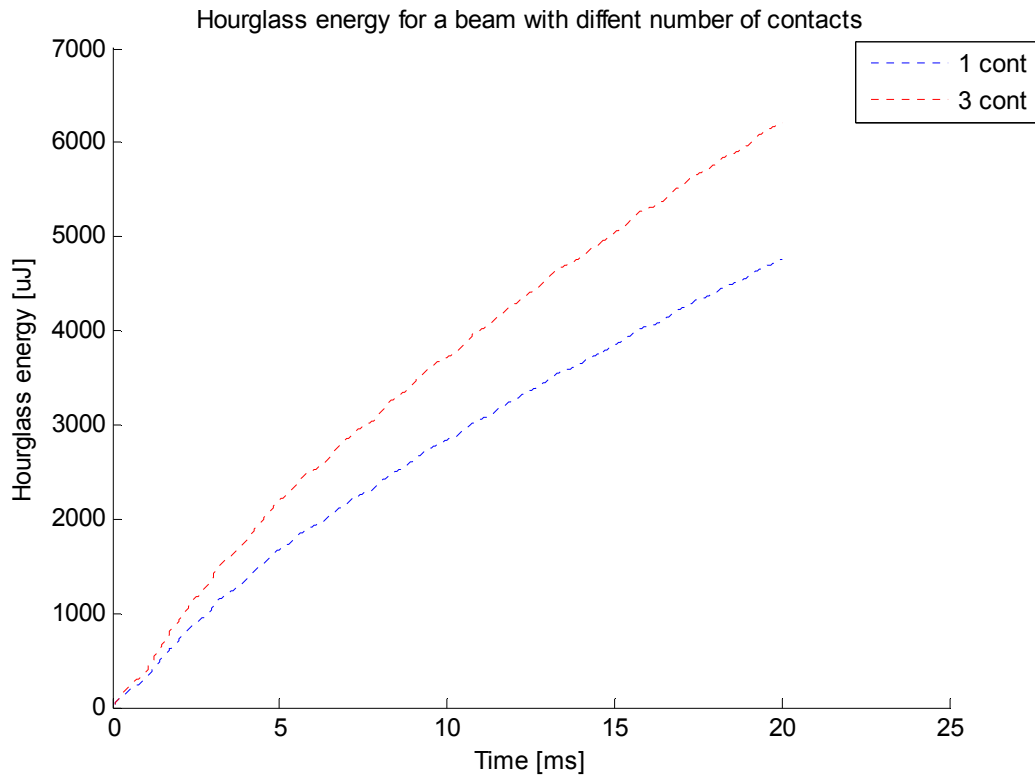The criterions for decide if the beam simulation are correct, total deformation, period time, limited hourglass energy.

The simulated transient period times do correspond well mutually. The simulated period times do correspond well to the analytical and the simulated period time from the modal analysis, seen from Figure 3-2.

The simulated transient responses do differ a bit mutually. It can be seen that the ACLLSD and ACLI are almost identical. The solution generated by AWBAD give some larger deflection. The simulation generated by AWBLSD seems to have faulty default damping settings due to the smooth behaviour, see Figure 3-2.

The energy plots shows that for AWBAD the hourglass energy grows a bit but never passes the 5 % limit during the time of simulation, see Figure 3-5. The total energy after the force drops to zero is constant, see Figure 3-4. It shall be noticed that the energy error does not grow during the time energy is added to the structure.

Due to period times and energies the beam simulation can be said to give sufficient similar results, especially under loading conditions. The difference in displacement after excitation may be evaluated more.

4   Ansys Workbench

One of the goals for this study is to evaluate how well suited explicit software is for computing transient structural dynamics on weapon systems. Especially AWB with different explicit solvers is evaluated. Some other interfaces are discussed but not evaluated in this chapter. The interfaces and solvers will differ a bit in the pre, post- and solver process. In this chapter the workflow with explicit solvers is described more practically. The Lemur will be the main example.

As for all finite element analysis there are several of steps that must be passed through but for an explicit solver the steps will be different. There are several conditions that have to been fulfilled during the workflow to consider the software fully usable. The steps are:

1. Geometrical modelling

2. Defining material properties

3. Meshing

4. Contact modelling

5. General post-processing/modelling

6. Stable and correct solution

7. Evaluating solving time

4.1   Geometric modelling

Ansys Workbench uses the Design Modeler for simplification and modelling of the CAD geometries. Design Modeler is a powerful tool for simplifying the geometries regarding to fillets and holes. Explicit methods require geometries where the smallest size of an edge in the geometry must be larger than the minimum specified length of the elements so the time step can be set manually and will not be controlled by the associated length of the smallest edge, this is illustrated in Figure 4-1.
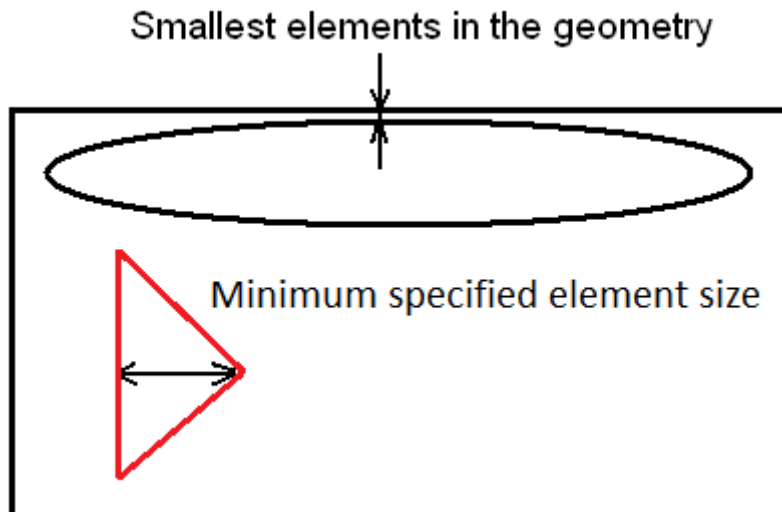
Figure 4-1: Geometry with small geometrical property

In pre-processing for an explicit analysis the model requires a lot of work on geometry for eliminating small edges so the time step can be controlled by the user. It can be said that this is a problem, so the user must do a lot of time consuming work in the geometrical parts of the pre-processing. One option is to remove the small features with abrupt geometry changes in the CAD software before editing it in the Geometric Modeller for AWBAD and AWBLSD. The problem of small elements also occurs if the geometry is simplified and/or meshed in the Design Modeller and transferred to SAD or ACLLSD. Note that this problem may occur if some other geometric modelling tool is used. Another option is to build the model from scratch and thereby control the smallest characteristic length.

A problem which is more general is that geometrical modelling changes the mass and the stiffness properties of the structure. The amount of change will of course depend on what the geometry look like from the beginning, but the problem will occur in some way. This may be a disadvantage of the explicit way of solving dynamic equations as both the stiffness matrix and mass matrix are changed.

In the Lemur console there are a lot of features like holes, fillets, chamfers and other milled features. This will cause a problem when trying to make the model simple because if too much of the geometry is changed the structural dynamic properties will be changed. On the other hand, if too small changes are done the geometry cannot be meshed in a good way.

To summarize this type of geometrical modelling for explicit simulation: If the time step should not be controlled by the geometry, the structural geomery has to be modified in some way, this will require resources. A geometric change will change the structural properties of the structure.

4.2   Mesh

In this section the type of discretization for the explicit way of simulation is discussed. Moreover the discretization in AWB is described.

First of all, in the discussed softwares, all elements for explicit simulations are constant stress elements. This will require a lot of elements for obtaining a not too stiff model. If the purpose of the analysis is to obtaining accurate stresses even more elements are required for the geometry. The focus in this examination is to obtain deformations so the discussion of creating a well suited mesh for stresses will not be presented deeply.

One wanted feature for the mesh generation is a control of the smallest element so it does not disturb the CFL condition. For elements around holes and corners are smaller than the CFL length. Holes are almost always deleted easily but there will still be corners in the structure. If deformations and not stresses are studied it is recommended to switch off the automatic refinement options around corners and edges. If this option is off and the global element size is set to a sufficient large value it should not spoil the CFL time step. This can be done in the mesh generation part of AWBAD and AWBLSD by the option *fixed*. However if these options is used and the geometry has geometrical features larger than the wanted mesh size some small elements may be created in some locations.

A wanted feature from the mesh generator is to avoid generating of small, thin elements. In the Lemur small thin elements may be generated in the consoles because of its complicated geometry. This may not be a problem if the geometry is fairly geometrical cleaned up and some mass-scaling is used. However there are several possibilities to check the geometry. One is to use the FE Modeler in Workbench and find out if there are some thin elements. It is not always easy to see where the thin elements are located but one can for example use a Matlab script to investigate. A third option is to transfer the elements into ACL and find which element sides are short. The method of transfer the mesh to ACL is more straightforward than using a Matlab script.

If shells or beams should be used, thin surfaces and lines must have been defined in the geometric modelling part. Solid shell elements cannot be used in AWBAD.

4.3   Contact modelling

One important thing when dealing with multipart geometries is contacts between parts. The contacts modelling must be split into two parts. The first group is the body interactions, which is contacts who are activated when parts are near other parts. These contacts are common for explicit softwares, because the natural usage of explicit softwares in the example of penetration simulations. The second group of contacts is the traditional ones used in AWB implicit. These must be pre-defined or generated for the purpose to be active.

Generally, contacts and body interactions is an advantage when working with explicit software. In an explicit finite element analysis there is no residual convergence check criterions, the time step is very small and the contacts are treated in the force vector.

This three things together makes the analysis very-straight forward in the sense of contacts. Below follows some comments on the contacts in AWBAD:

- Automatic contact generator in AWB works good in the explicit modules

- Some settings combined with contacts give incorrect results. The setting double is the proper setting to avoid incorrect results.

- The Lemur consists of eight parts. The contacts with the proper settings works very well

- Body interactions with the *bonded* options could be used for the Lemur, but when implicit to explicit solution is to be done, then the usual contacts have to be used

4.4   General Pre-Processing

Settings available in AWBAD are presented here. The first three sub-chapters describe the initial conditions, boundary conditions and loads, while the last describes settings for the explicit solver.

4.4.1   Initial conditions

For dynamic simulations initial conditions are important, this is fairly developed in AWBAD, the initial conditions can be seen in the list below and in [1].

- Velocity

- Angular velocity

- Pre-stress

The pre-stress initial condition solves an implicit static or transient analysis and transfers displacements/stresses to the explicit code. The pre-stress option requires that an implicit analysis is connected to the explicit analysis, this is described in [2]. Two different options are available for the pre-stress, the first one is displacement and the other is material state. It is recommended to use material state, which are related elastic stresses and displacements.

4.4.2   Boundary conditions

In the explicit module there are several boundary conditions which are listed below and studied more in [1]. As can be seen the explicit boundary conditions are not as developed as in the implicit module. Some of the boundary conditions from the implicit module would be useful in the explicit one. Examples of this are the compression only and remote displacement, conditions. The available boundary conditions are:

- Fixed support

- Displacement

- Impedance boundary

- Velocity

4.4.3   Load inputs

In the explicit module several types exist which are listed below. A further description of these can be seen in [1]. As can be seen the loads for the explicit module are not as developed as in the implicit module. Load curves can be used for most of the loads. This is an important option for firing simulations. The loading options are:

- Pressure

- Hydrostatic pressure

- Force

- Line pressure

4.4.4   Analysis settings

The analysis settings will refer to the analysis settings in AWBAD. Only those analysis settings which are most important and those which are changed from default will be mentioned here.

4.4.5   Step controls

Resume from cycle can be used if one wants to stop the simulation and change some property and resume the simulation. This control does not work well every time it is used. The user is recommended to control manually that the solution is running if the simulation has been resumed.

Mass scale can be used to increase the density of small elements, a larger time step can be used than.

4.4.6   Solver controls

Precision can be set to single or double, and is the floating point precision used by the solver. Double precision is recommended when contacts or pre-stress are used, otherwise the solution will give false energies.

Hex integration type is an option for volumetric calculations for the elements. This can be set to exact and reduced. This will only affect the volumetric calculations and not the number of gauss points used in the elements. Only the exact options have been examined in this report.

Tet pressure integration will affect the volume calculation for the elements. This can be set to ANP (Avarage Nodal Pressure) and SCP (Single Nodal pressure). This option will affect volumetric locking if the Poisson's ratio is close to 0.5. The ANP formulation should be used if poisons ratio is high, but the mentioned option is recommended for all types of materials if tetrahedrons are used in the model.

4.5 General post-processing

Two types of results are available in AWBAD. The fist ones are the probes for deformation, strain, stress etc. These results are defined exactly as in AWBI. The second ones are those who are defined in the *worksheet* tab. Here are some more options are available compared to the *probes*. Example of this is *mass scaling*, *sound speed* and *bond status*. This *worksheet* tab is important for explicit dynamic post-processing. There is one tab called *User defined results* where the user can combine results from the *worksheet* tab.

Several things which are available in the implicit module are missing in the explicit module. For example the *contact tool*, *reactions forces/moments* and *command objectives*.

5   Lemur simulation and test

Several tests for the Lemur have been performed. This chapter include-descriptions of the Lemur structure, finite element simulations, snap back testing and comparison.

5.1   Short aluminium Lemur

The basic structure of the short aluminum Lemur consists of eight parts. The eight parts are listed below and can be seen in Figure 5-1.

- A Base Plate

- C Foot Plate

- D Ring

- E Right and left Brackets
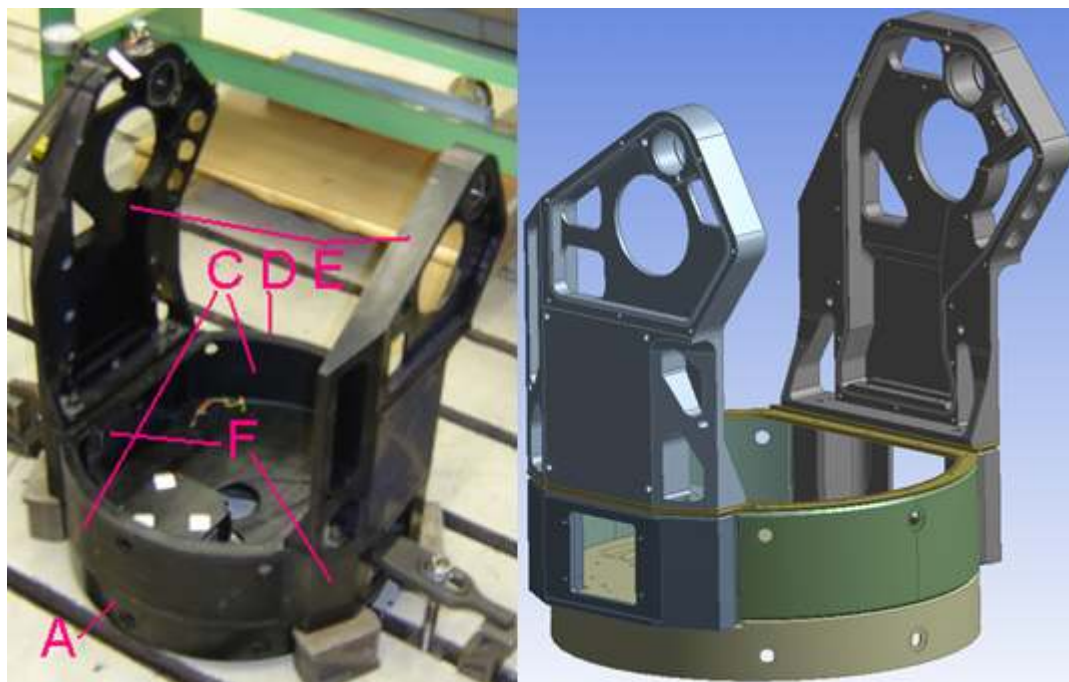
- F Foot Brackets

Table 5.3-1 Part of the Lemur

Figure 5-1 Parts of the Lemur, test and CAD model

5.2   Simulation

Three types of simulations have been carried out for the Lemur structure: implicit static, modal and explicit dynamic. The geometric modelling has been carried out in the same way for the three types of analysis. The red, blue and green colours on the surfaces in Figure 5-5 will be the reference. Default options in AWBAD are used if nothing else is mentioned.

5.2.1 Geometric modelling

Simplification of the model has been done in the AWB Design Modeller. Figure 5-2 shows the input from the CAD software and the geometrically simplified. Figure 5-3 shows the bracket of the Lemur before and after simplification. Removing one of the brackets is done to save computational time.
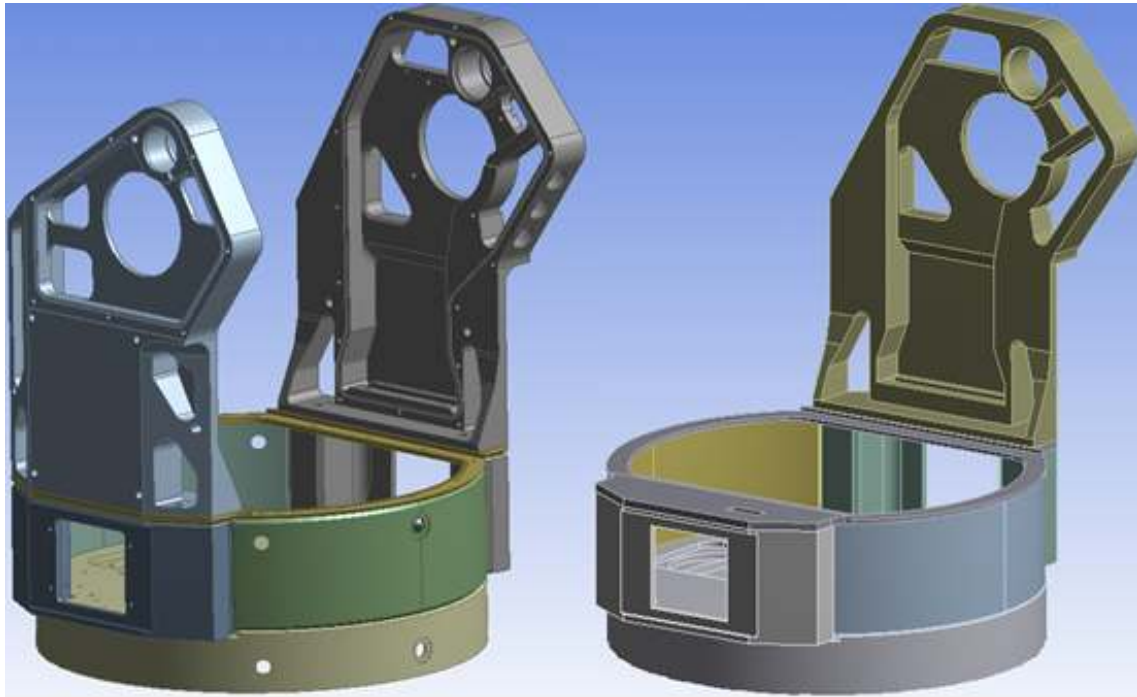

Figure 5-2: CAD geometry and simplified geometry, Lemur
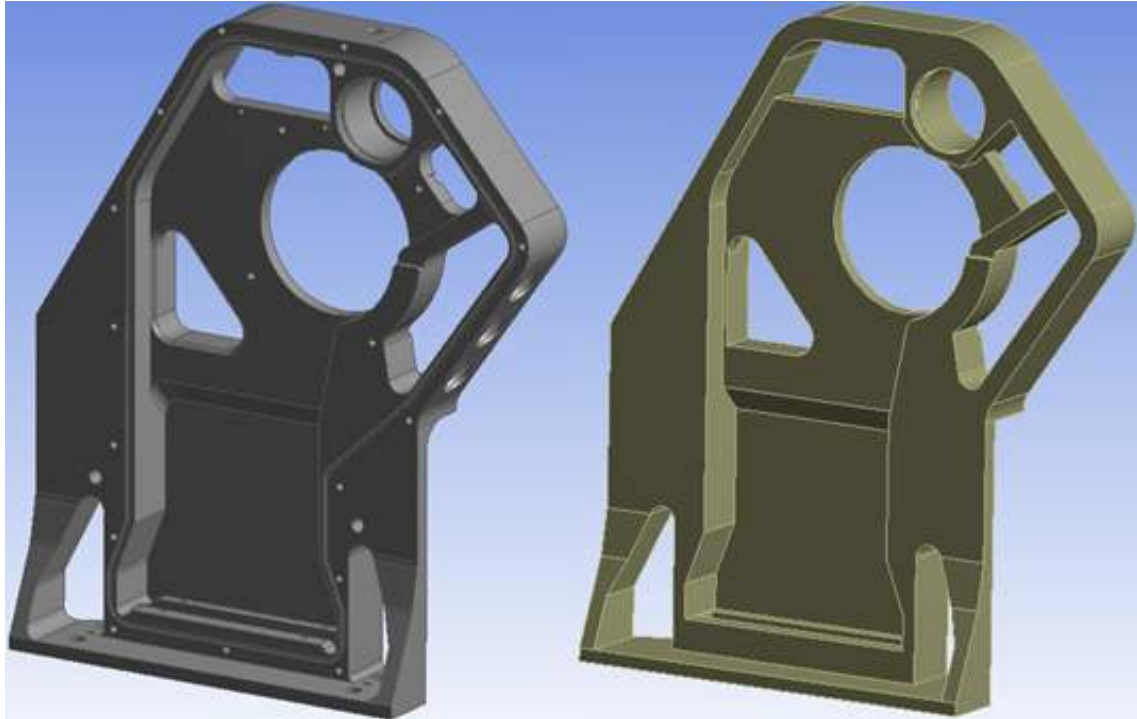
Figure 5-3: CAD geometry and simplified geometry, Lemur Bracket

5.2.2 Mesh

Two meshes has been used. They will be referred to as the coarse and the fine mesh. Both meshes are mainly made of tetrahedrons with the options *fixed* as *advanced* option. The coarse mesh has 15 mm element size, 21 k nodes and 33 k elements. The fine mesh has 10 mm element size, 26 k nodes and 91 k elements, see Figure 5-4. *Midside nodes* are used in the modal and static analysis. In the explicit analyses *midside nodes* are set to *dropped*.
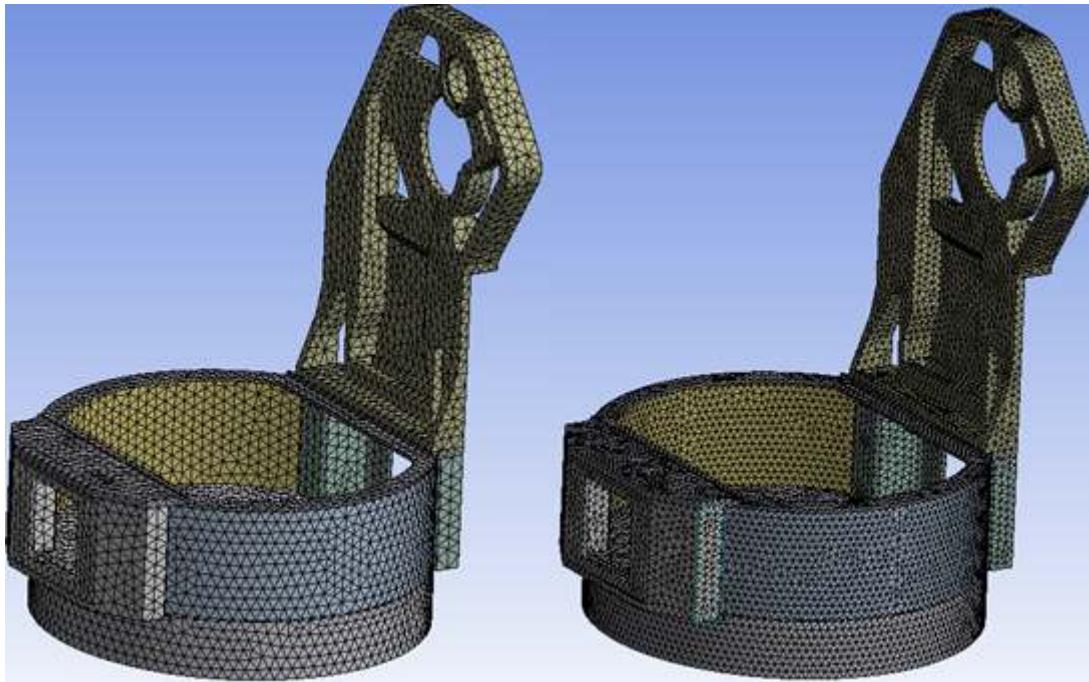
Figure 5-4: Coarse and fine mesh

5.2.3 Static analysis setup

In short, the settings for the static analysis are:

- AWBIS

- Fine mesh

- Red: *Compression Only*

- Blue: *Fixed* in all directions except rotation in x direction

- Green: *Force* 206 N in y direction

- *Bonded* contacts between parts, penalty formulation

5.2.4 Modal analysis 1 setup

In short, the settings for the first modal analysis are:

- AWBM used for this simulation

- Fine mesh

- Red: *Compression Only*

- Blue: *Fixed* in all directions except rotation in x direction

- No force

- *Bonded* contacts between parts, penalty formulation

5.2.5 Modal analysis 2 setup

In short the second setup got modal analysis are:

- AWBM used for this simulation

- Fine mesh

- Free boundary conditions

- No force

- *Bonded* contacts between parts, penalty formulation

5.3 Explicit analysis setup

In short, the setting for the implicit-explicit analysis are:

- AWBAD is used for this simulations

- First an implicit analysis is solved, with the same settings as the static analysis but with fixed supports in the red areas

- Pre-stress, *material state* from the implicit structural

- Coarse and fine mesh used

- Red: *Fixed support*

- Green: Constant *force* 206 N in 0.1 ms

- Precision: Double

- *Mass scale*: both on and off used. The time step was set to 0.15 us in the case of activated mass scaling
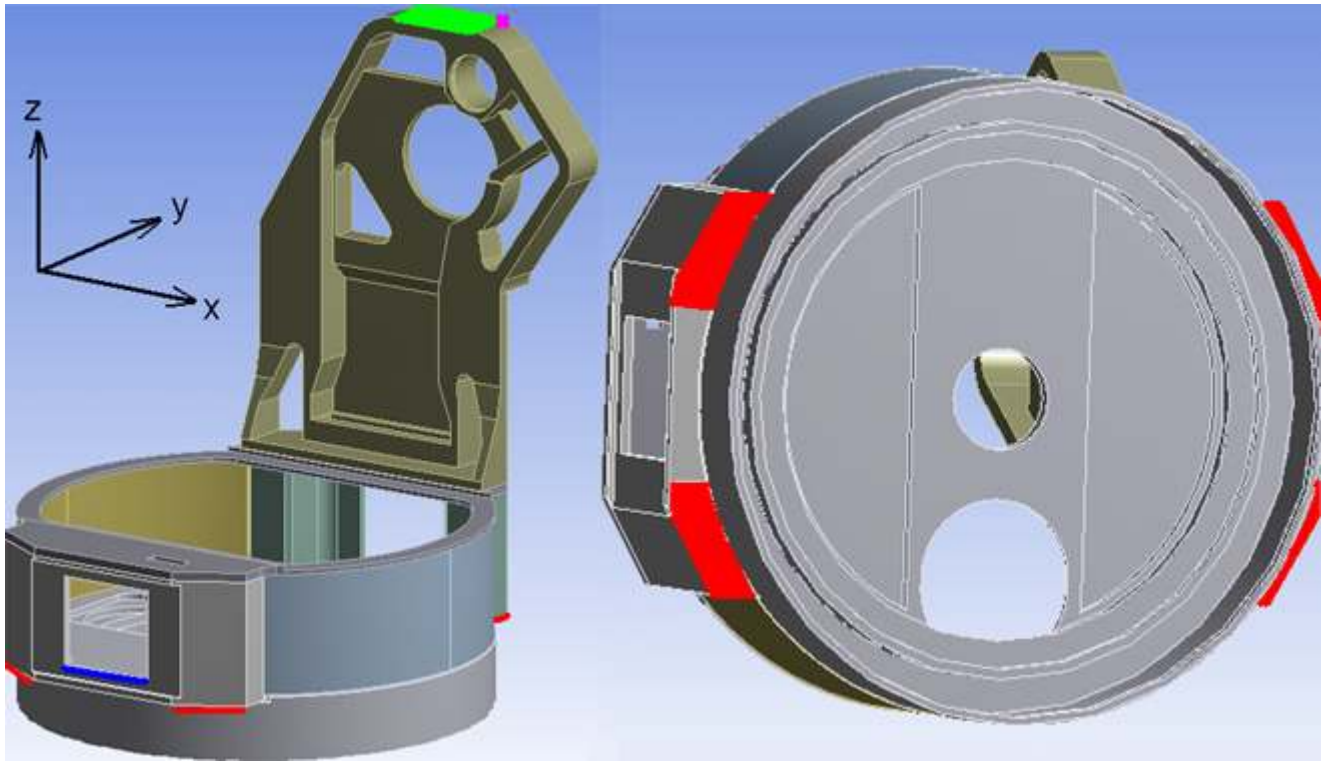
- Visualisation points: 250

Figure 5-5: Boundary conditions and Loads

5.3.1 Simulation results

Displacement results from the static and transient simulations are taken at the pink dot in the y direction, see Figure 5-5. The time history of the displacement can be seen in Figure 5-6. The period of time showed below is calculated from Figure 5-6 by observing when the first and second minimum value occurs in time. Further, the results from the two modal analyses are presented.

- Deformation static 0.21 mm, from pink probe

- Eigen freq mode 1 modal 1: 129.9 Hz

- Eigen freq mode 7 modal 2: 127.5 Hz

- Period of time for explicit simulation coarse fine mesh no mass scale: 7.5 ms

- Period of time for explicit simulation coarse mesh mass scale: 7.5 ms

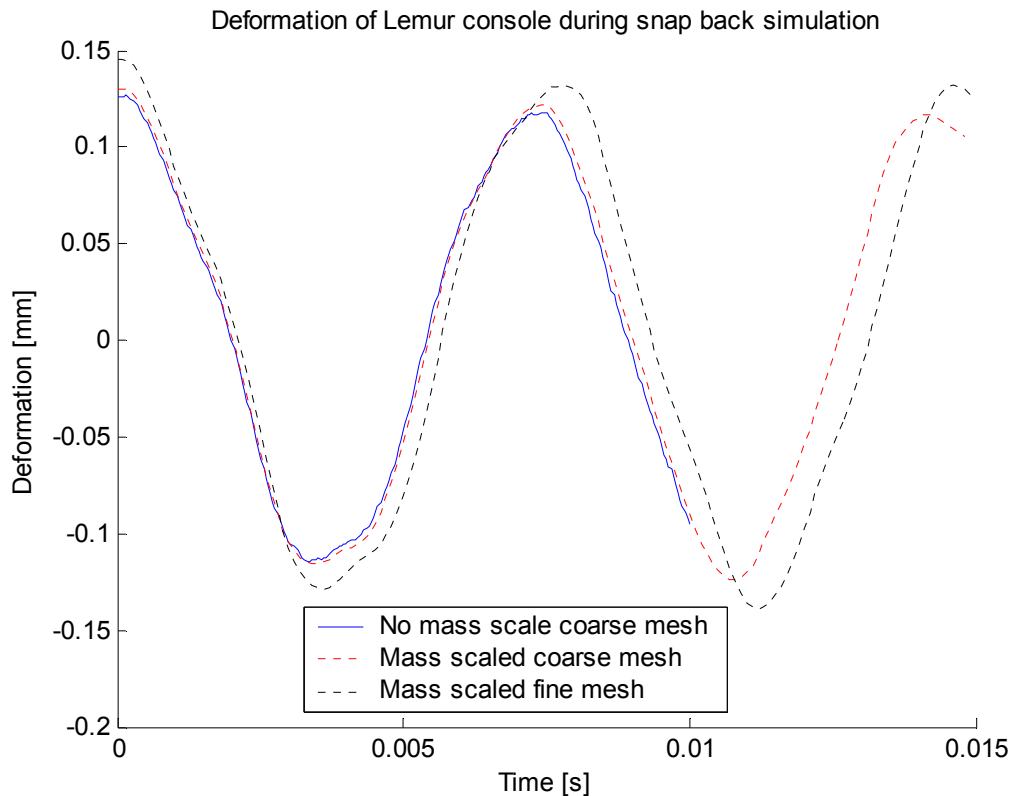- Period of time for explicit simulation fine mesh mass scale: 7.0 ms

Figure 5-6: Displacement for the Lemur simulation

5.4 Snap Back Test

Here follows information for the snap back test and the evaluation methods for the collected data. Description of the test will be arranged in following way:

- Introduction and reason for doing the test.

- How the structure is fixed to the ground.

- How to control force and deformation.

- How to make the structure oscillate.

- Measurement equipment used.

- Results and transformations of results.

- The tests are made on the Lemur 008 (FAK 40)

Four test sessions was performed. The only thing which was different between the tests sessions was the force applied to the console. The aim was to release the pretension when the force was 200 N, but it could not be done exactly every time. The structure mainly consists of eight parts and several bolt joints for joining the consoles with the lower part of the structure. The parts are listed in table 5.3-1. The configuration can be seen in Figur 5-7.

- B: Bolt Joints

- C: Support blocks

- D: Lemur Consoles

- E: Bearing

- F: Bolt for the wire

- G: Composite wire

- H: Deformation measure clock

- J: Three axis accelerometer

The picture shows how the Lemur structure is connected to the ground. Only support set C joins the Lemur with the ground, eg the cylindrical structure A does not touch the ground. The accelerometer is bolted to the small aluminum block under the accelerometer. The aluminum block is glued to the console. There is a ball bearing in E which is not totally fixed. In test session 1 and 2 there was no tape support, so the bearing could move freely. In test 3 and 4 there was a black tape attached that restrained the motion of parts of the bearing. The black tape did not fix the bearing completely.

5.4.1   Static test

The structure is deformed by applying a force on the bolt F by stretching the wire G with a turnbuckle. The force is measured in the wire by a force indicator. The deformation of the structure is measured with the measure clock H.

5.4.2   Free vibration Dynamic test

The settings of the dynamic test are the same as for the static tests. In the first step the structure is pre-stressed by a force of approximately 200 N. After the pre-stress the wire G is burned off and the structure can oscillate freely. The measure clock H does not touch the Console D in the free vibration dynamic test. The measure clock H was only used to measure the initial static deflection.
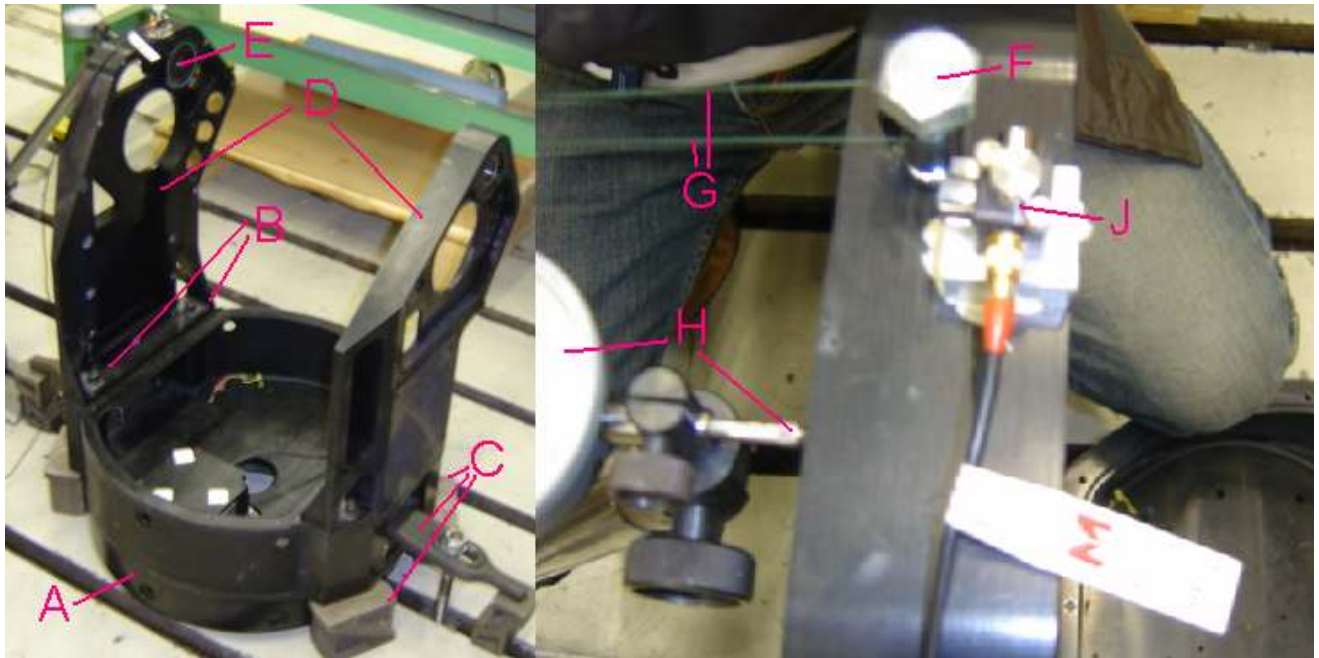
Figur 5-7: Test setup

5.4.3 Measuring Equipment

The force indicator had a measure interval of 0-500 N. The chosen sample frequency for the accelerometer was 10 kHz.

5.4.4 Results and transformations

Four test sessions were performed. In the appendix the measured accelerations are showed for the cases. The difference between the maximum accelerations for the test sessions varies between 23.12-40.69g. Fast Fourier Transform has been carried out on the acceleration data for all test sessions. The results can be seen in Figure 5-9. The resolution of the FFT is 1024 discrete frequencies.
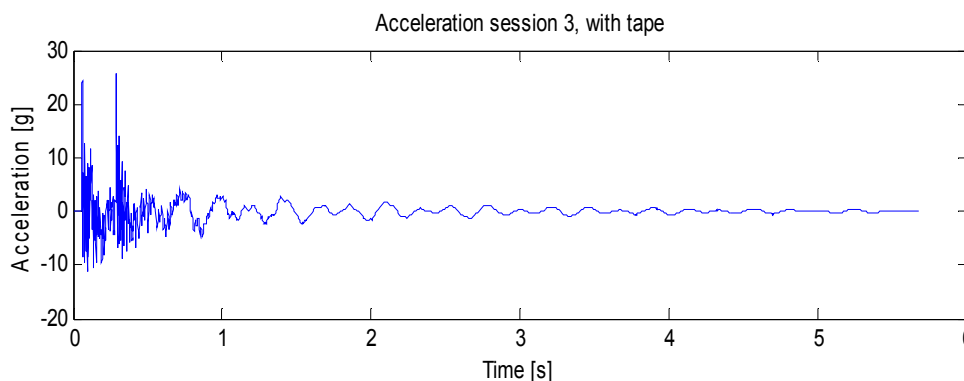

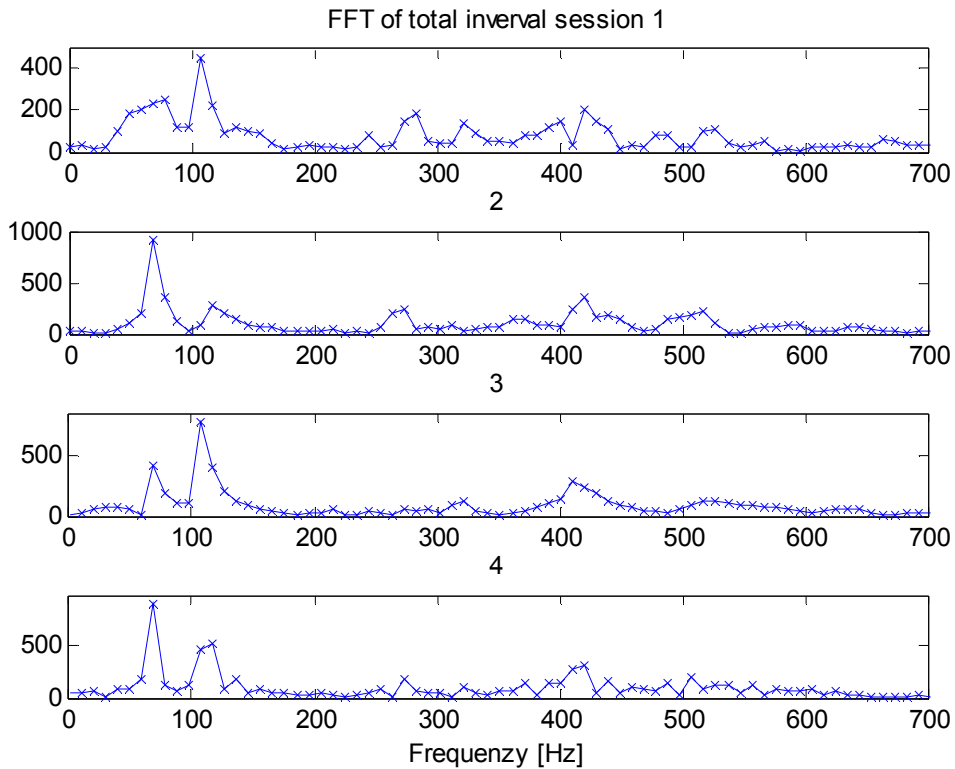**Figure 5-8: Measured acceleration in the weakest direction**

Figure 5-9: FFT for all test sessions with 1024 resolution points

5.4.5 Discussion of results

There are mainly two frequencies which are notable from the FFT plots. The first frequency is located around 67 Hz. The other frequency is located around 117 Hz. The first two plot sequences do not have tape around the bearing and they have different maximum FFT value for the two frequencies. The second two plot sequences do have tape around the bearing and they have different maximum FFT value for the two frequencies.

5.5 Correlation of test and simulation results

If the results from the snap back test, modal analysis and explicit simulation are summarized to following

117 Hz Snap back test

127.5 Hz Modal 1

138 Hz Explicit Transient fine mesh

The difference between the snap back test and explicit simulations is about 20 Hz. In the snap back test the bearing will increase the mass of the console. The bolt joint between the consol and the ring will decrease the stiffness of the structure. The bearing and the bolt joint is not modelled in the explicit simulations or in the modal

analysis, therefore the higher frequency can be expected from the simulations compared to the snap back test.

5.6   Summary of the workflow for the Lemur

There are several ways working with a geometry and model for an explicit dynamic analysis in different softwares. A recommended way of working with AWBAD and structures as the Lemur is:

- Time for the geometry for being simplified from the CAD model to the actual computational model: 2-3 hours

- Time for the geometry for being simplified form the CAD model to a "perfect model" without any small edges: 10-12 hours

- If contacts are used instead of mesh all together less time have to be spent on the geometric modelling

- Large geometric changes will affect the mass properties more than using mass scaling on the model. In addition the stiffness properties will be changed by changing the geometry

- The main purpose of using AWBAD is to receive results fast. This can be obtained by not spending too much time on geometrical modelling and by using contacts and mass scaling.

6 Discussion

In this chapter alternatives to AWBAD are presented, the task is summarized and recommended further work is presented.

6.1 Alternatives to AWBAD

Several alternatives can be used and combined with AWBAD in different steps of the pre-, post- and solving process.

6.1.1 Geometrical modelling

For geometrical modelling the design modeller is good enough if the geometry is not too complicated due to short edges and small geometrical features. If difficult geometries exists these may be simplified with external CAD software.

6.1.2 Meshing

Meshing can be done in some external mesh generator or in ACL. This can be done after simplifying the geometry in the Design Modeller.

6.1.3 Pre-processing

Several alternatives to AWBAD pre-processing are available. One is to do the pre-processing in AWBLSD, but not much is gained by using that optioned if only AWB is used. The second alternative is to transfer the model, mesh and some of the pre-processed part from AWBAD to SAD if certain requirements for the mesh are fulfilled. A third option is to transfer the geometry and/or the mesh to ACLLSD and pre-process. Several of other combinations can be used but are not mentioned here.

6.1.4 Post-processing

The post-processing part is very dependent of which pre-pre-processing part is used. More post-processing features are available in ACLLSD and SAD than in AWBAD. If AWBLSD is used LS PrePost is used as post-processing tool, and this tool was found better than AWBAD post-processing part.

6.2 Conclusion

In this section conclusions are made from the task in chapter 1. The task was:

- The main task is to evaluate if an explicit solver can be used for simulating transient dynamic response for shooting systems.

- Advantages and disadvantages for an explicit solver

- The response in the platform from the explicit transient simulations can be compared to old calculation methods used at BAE Systems Bofors, to measured data, or to implicit transient simulations.

6.2.1 Cases where the explicit method is usable

Depending on the nature of the dynamic system, explicit methods can be used. If the simulation time is reasonably short so not unphysical energies will occur in the model, explicit methods are preferable. As long as force is applied to the structure the quotient between the hourglass energy and the total energy are constant or decreasing. If constant energy error is obtained the explicit simulation should be physical sound.

The main advantage for the explicit method in general is the contact modelling, specially compared to implicit methods. The contacts do not get convergence problems which are the case for implicit methods.

A disadvantage with the explicit method is that it uses low order elements. If a good resolution of the stress field is wanted more elements have to be used when using the explicit method compared to the implicit method. Another disadvantage is that zero energy modes can be generated due to the reduced integrations. This disadvantage can be suppressed with other softwares, example ACLLSD. Long simulation time without external forces on the structure is not recommended for explicit simulations due to the energy error.

6.2.2 Advantages and disadvantages for explicit software

In the AWB environment both AutoDyn and LS-Dyna has limitations. The limitations were showed in both the pre-and post-processing part for AWBAD, mentioned in chapter 4. If certain features which limits the modelling is not needed AWBAD can be used for modelling and simulation. Some of the limitations can be avoided if AWBLSD is used and combined with ACLLSD or LS PrePost.

6.2.3 Verification

The explicit softwares can be said to fit good to the implicit simulations and to the test, especially regarding frequencies. It is hard to do conclusions from the beam simulation results and the Lemur simulation results because free vibration is not the most beneficial for explicit softwares the energy error may grow with any control.

6.3 Further work

In the modelling and simulation process a lot of questions have been raised but not all of them has been investigated or solved. Here follows some recommended tasks for further investigations.

6.3.1 Beam and shells

Investigate how beams and shells work with contacts

6.3.2 Stresses

Investigate stresses for explicit simulation compared to implicit simulations

Compare the stresses from the explicit method with the old static implicit methods

Investigate if sub modelling is available in AWBAD

6.3.3 Plasticity

Plasticity when chock loads are applied to the structure and plastic deformation is generated.

6.3.4 Softwares

Evaluate if pre-processing can be done in AWBLSD and then import the setup file (.k file) to LS PrePost and in that software add the features which are missing, example springs and point masses.

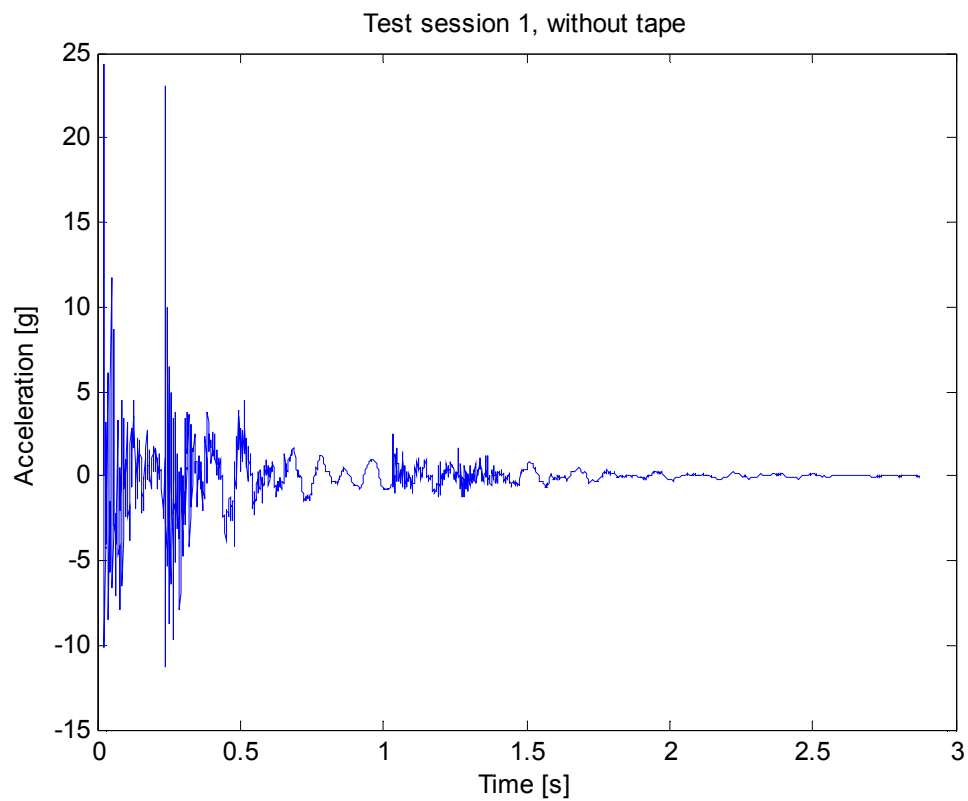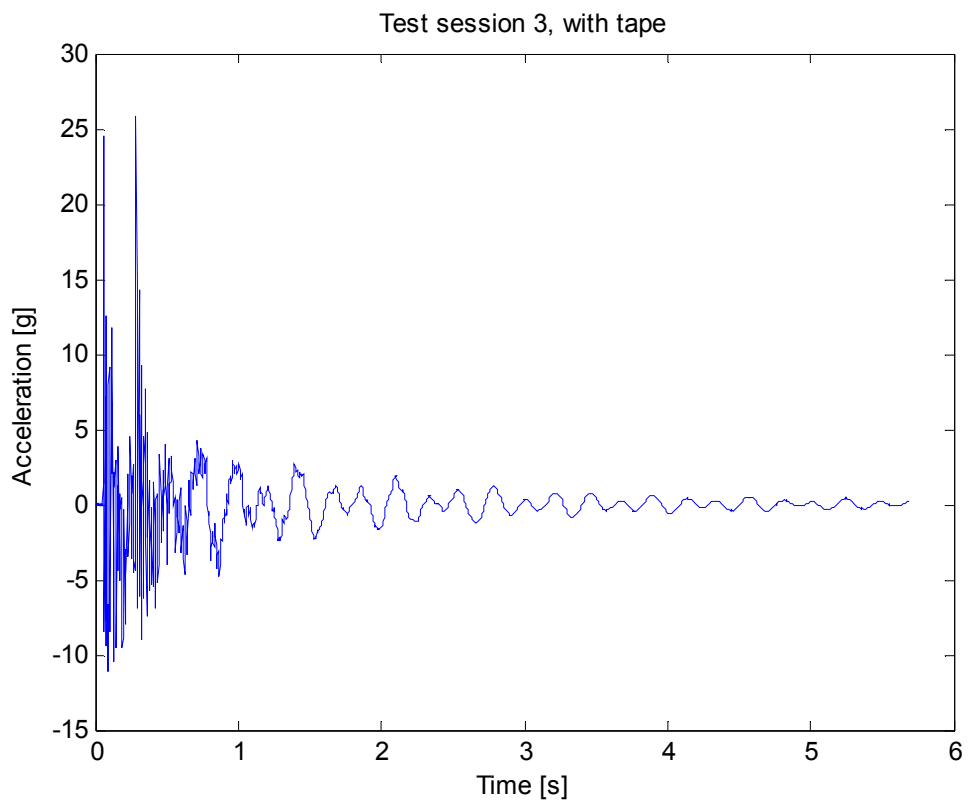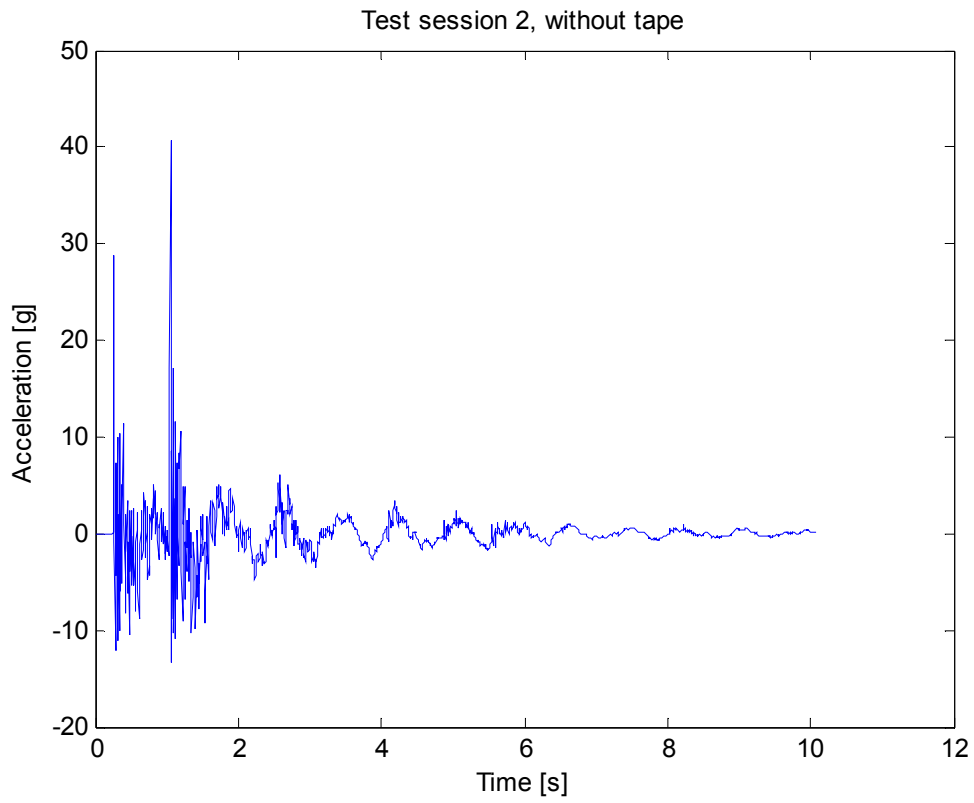Solver time comparison between the softwares

References

[1] Ansys v12.1 Manual, // Mechanical (formerly Simulation) // The Mechanical Application Approach // Analysis Types // Explicit Dynamics Analysis

[2] Mikael Björkmon, Presentation: Utvärdering av FE-verktyg för transientdynamiska förlopp för skjutande system, 02897939_Ed01, BAE Systems Bofors AB

[3] Dynamisk belastning, Aktiebolaget Bofors

[4] Lars Olovsson, Kjell Simonsson, Mattias Unosson. Selective mass scaling for explicit finite element analyses. International Journal of Numerical Methods in Engineering 2005; 63:1436-1445

[5] Lars Olovsson, Kjell Simonsson, Mattias Unosson. Selective mass scaling for thin walled structures modelled with tri-linear solid elements. DOI 10.1007/soo466-004-0560-6

[6] Grand Roman Joldes, Adam Wittek. Karol Miller, Improved Linear Tetrahedral Elemenet for Surgical Simulation, Intelligent Systems for Medicine Lab., The University of Western Australia

[7] J. Bonet, J. Burton . A simple average nodal pressure tetrahedral element for incompressible and nearly incompressible dynamic explicit applications. Communications in Numerical Methods in Engineering 14; 437-449 (1998)

[8] J. Bonet, H. Marriott, J. Burton. An averaged nodal deformation gradient linear tetrahedral element for large strain explicit dynamic applications. Communications in Numerical Methods in Engineering; 17:551-561

[9] F. M. Andrade Pires, E. A. de Souza Neto, J. L. de la Cuesta Padilla. An assessment of the avarage nodal volume formulation for the analysis of nearly incompressible solids under finite strains .Communications in Numerical Methods in Engineering; 20:569-583
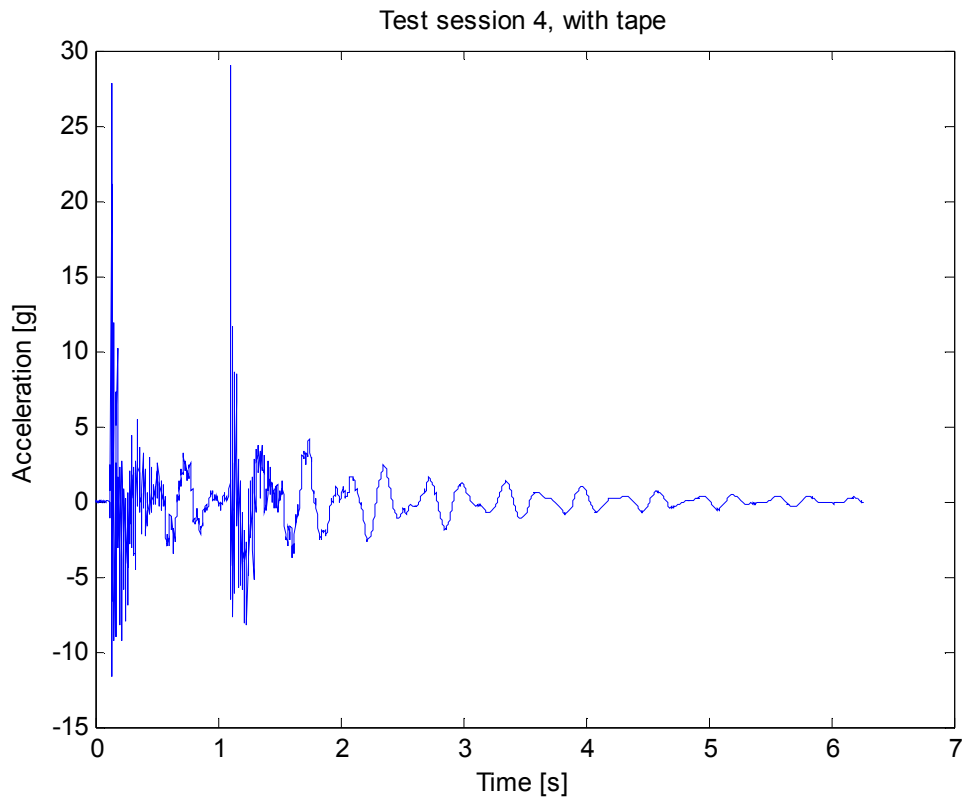
37

Content Appendix:

- Acceleration plots from the test

- Script for beam simulation ACLI

- Script for beam simulation ACLLSD

Acceleration plots

Test session 1, without tape

Test session 2, without tape

Test session 3, with tape

Test session 4, with tape

Script for beam simulation ACLI

```
finish
/clear
/TITLE, Dynamic Analysis
/FILNAME,Dynamic,0            ! This sets the jobname to 'Dynamic'
/PREP7                       ! Enter preprocessor
K,1,0,0                      ! Keypoints
K,2,1,0
L,1,2                        ! Connect keypoints with line
ET,1,BEAM3                   ! Element type
R,1,0.0001,8.33e-10,0.01     ! Real constants
MP,EX,1,2.000e11             ! Young's modulus
MP,PRXY,1,0.33               ! Poisson's ratio
MP,DENS,1,7800               ! Density
LESIZE,ALL,,,10              ! Element size
LMESH,1                      ! Mesh the line
FINISH
/SOLU                        ! Enter solution phase
ANTYPE,TRANS                 ! Transient analysis
DELTIM,0.001                 ! Specifies the time step sizes
!BETAD,0.01                  ! Styvhetsdämpning 0.01
D,1,ALL                      ! Constrain left end
F,2,FY,-100                  ! Load right end
!
!*
!http://www.mece.ualberta.ca/tutorials/ansys/CL/CIT/Transient/Print.h
tml
!Copyright 2003 - University of Alberta
!At time equals 0.001s
TIME,0.001                   ! Sets time to 0.001 seconds
KBC,1                        ! Ramped load step
TIMINT,ON
LSWRITE,1
!
FDELE,2,FY                   ! Delete the load at the end
!
!
                             ! At time equals 1s
TIME,0.2                     ! Sets time to 1 second
KBC,1                        ! Ramped load step
LSWRITE,2

!*
OUTRES,ALL,ALL
ALLS
LSSOLVE,1,2,1 ! solve multiple load steps
FINISH
/POST26                      ! Enter time history
FILE,'Dynamic','rst','.'     ! Calls the dynamic file
NSOL,2,2,U,Y, UY_2           ! Calls data for UY deflection at node
2
STORE,MERGE                  ! Stores the data
PLVAR,2,                     ! Plots vs. time
!
/eof
SET,LAST
PLNS,U,Y
ANTIME,10,0.5, ,1,0,0,0
!
```

```
/eof
```

Script for beam simulation ACLLSD

```
finish
/clear
/TITLE, Dynamic Analysis
/FILNAME,Dynamic,0               ! This sets the jobname to 'Dynamic'
/PREP7                           ! Enter preprocessor
BLOCK,0,0.25,0,0.01,0,0.01
BLOCK,0.25,0.50,0,0.01,0,0.01
ET,1,SOLID168                    ! Element type
MP,EX,1,2.000e11         ! Young's modulus
MP,PRXY,1,0.33                   ! Poisson's ratio
MP,DENS,1,7850                   ! Density
!
MP,EX,2,2.000e11         ! Young's modulus
MP,PRXY,2,0.33                   ! Poisson's ratio
MP,DENS,2,7850                   ! Density
!
!
!LESI,3,,,4
!LESI,6,,,4
!LESI,10,,,4
!LESI,11,,,4
!LESI,2,,,20
!lesi,4,,,20
!lesi,7,,,20
!lesi,5,,,20
!LESI,1,,,4
!LESI,8,,,4
!LESI,9,,,4
!LESI,12,,,4
!
SMRT,8
MSHAPE,1,3D
MSHKEY,0
!
MAT,1
VMESH,1
!
MAT,2
VMESH,2
!
!
FINISH
/SOLU                    ! Enter solution phase
!
nsel,s,loc,x,-1e-4,1e-4
cm,fixsu,node
alls
!
!
nsel,s,loc,x,0.499,0.501         !
cm,last,node
*get,ant_nod,node,0,count        !Räknar antalet noder
alls
!
ksel,s,,,11
nslk
*get,res_nod,node,0,num,min      !nodnumret för vilken vi vill veta
resultatet
```

```
cm,res_nod,node
alls
!
edpart,create
!EDCGEN,AG,,,0,0,0,0,0, , , , ,0,10000000,0,0
EDCGEN,TDSS,1,2,0,0,0,0,0, , , , ,0,10000000,0,0
!EDCGEN,TDSS,2,1,0,0,0,0,0, , , , ,0,10000000,0,0
!
curve_n=1
! Define ANSYS arrays
*DIM,t_%curve_n%,ARRAY,2,1,1,,,        ! Time
*SET,T_%curve_n%(2,1,1),1
*DIM,s_%curve_n%,ARRAY,2,1,1,,,        ! Value
*SET,s_%curve_n%(1,1,1),0
*SET,s_%curve_n%(2,1,1),0
EDCURVE,ADD,%curve_n%,T_%curve_n%,S_%curve_n%
EDLOAD,ADD,UY,,fixsu,,,,%curve_n%,1
EDLOAD,ADD,UX,,fixsu,,,,%curve_n%,1
EDLOAD,ADD,UZ,,fixsu,,,,%curve_n%,1
!
curve_n=4
!
last_n=100/ant_nod
!
! Define ANSYS arrays
*DIM,t4_,ARRAY,4,1,1,,, ! Time
*SET,t4_(1,1,1)  , 0
*SET,t4_(2,1,1)  , 0.001
*SET,t4_(3,1,1)  , 0.0011
*SET,t4_(4,1,1)  , 0.5
*DIM,s4_,ARRAY,4,1,1,,, ! Value
*SET,s4_(1,1,1)  , -last_n
*SET,s4_(2,1,1)  , -last_n
*SET,s4_(3,1,1)  , 0
*SET,s4_(4,1,1)  , 0
! Define a LS_Dyna curve
EDCURVE,ADD,%curve_n%,t4_,s4_
EDLOAD,ADD,fy,,last,,,,%curve_n%,1
!1
alls
save,last,db
!
/SOLU
TIME,0.100                ! Stop time
EDRST,,1e-3        ! Plot frequency
!EDHTIME,,0.5e-4         ! TH frequency
EDHIST,res_nod
!EDDUMP,1           ! Dump freq
!EDOPT,ADD,,lsdyna
!EDCTS,-1e-7               ! Min time step 1e-7 sec      1e-7 => ~0.5 mm
element length of steel, may be raised to 1.4e-7
!
edout,glstat
edout,matsum
!
!/eof
!
!EDWRITE,LSDYNA,in,k
solve
!
finish
```

```
/POST26
FILE,'Dynamic','his','.'
KEYW,LSDYNA_H,1
/UI,COLL,1
NUMVAR,200
SOLU,191,NCMIT
STORE,MERGE
!*
NSOL,2,res_nod,U,Y,UY_2
XVAR,1
PLVAR,2
ALLSEL,ALL
SAVE

/eof
```