*Department of Construction Sciences*
Solid Mechanics

1 ISRN LUTFD2/TFHF-16/5213-SE(1-67)

# Improved methodology for postulated fuel handling accident

Master's Dissertation by

Max Fagerberg

Supervisors: Per Ståhle, LTH

Markus Ståhlbom, Westinghouse Electric Sweden AB

Examiner:Ralf Denzer, LTH



Printed by Media-Tryck AB, Lund, Sweden For information, address:
Sölvegatan 14, Lund

Division of Solid Mechanics, Lund University, Box 118, SE-221 00 Lund,
Sweden Webpage: www.solid.lth.se

# Improved methodology for postulated fuel handling accident

Max Fagerberg

LUND UNIVERSITY

# Abstract

This report presents a suggestion of an improved method for evaluation of a fuel handling accidents. Using a 3D FE-model and ANSYS LS-DYNA EXT the structural analysis is carried out. Importing geometry from SpaceClaim and simplifying it creates the geometry used in this analysis. By modifying the weight of the parts to resemble the real world product and coupling it with Ramberg-Osgood used as the material model the computational model will be similar to the real assemblies. Four different load cases are suggested to be examined. Together they analyse the effect of tilt angle and distribution of energy across multiple assemblies.

The results show that higher strain might occur in the top of the impacted assembly than previously thought. It also shows tendencies for the falling assembly to glance of the impacted assembly and continue until it affects another assembly.

Several improvements on this model can be done. The implementation of birth and death and combining the split assemblies into one singe calculation.

**Keywords:** ANSYS LS-DYNA, FEM, nuclear fuel, solid mechanics.

# Acknowledgments

I wish to thank both Markus Ståhlbom as well as Per Ståhle for giving excellent advise and for helping to create this thesis. Also a great thank you to Ralf Denzer for his endless patience and generosity.

# Table of contents

# List of acronyms and abbreviations

| | |
|---|---|
| BWR | boiling water reactor |
| FE | finite element |
| LC | Load case |
| PWR | pressure water reactor |
| RSM | remote solve manager |
| WB | Workbench |
| Westinghouse | Westinghouse Electric Sweden AB |

# 1 Introduction

## 1.1 Goals

The overall goal of this document is to present the method and approach used when simulating handling accidents on TRITON11™ for Westinghouse Electric Sweden AB (Westinghouse). The method should be applicable on several other drop tests when nuclear fuel is involved.

The report is structured so as to provide the background information needed in order for the reader to get an understanding of the problem. After that the solution is presented in the chronological order that it was done during the project, for instance the input data is presented before the simulation data and so on.

The ultimate results of the report are to present the number of fuel rods that will be damaged during an accident were a fuel assembly is dropped from a certain height onto another assembly. Along the way several other critical questions will also be provided the answers to, since these questions need to be answered first in order to complete the project.

The analysis will be done using ANSYS WB ACT Extension for LS-Dyna 16.1.

## 1.2 Underlying reasons

The reason a finite element (FE) analysis is needed is because, at the moment, the methodology is to use hand calculations to approximate the number of fuel rods damaged in an accident. This means that changes in the model will create the need to calculate the results again by hand. This is an inaccurate method and time consuming work that can be avoided if a FE-analysis is used instead. This also has the benefits of, being able to simulate different load cases almost as quickly as just one, produce a time dependent animation to graphically describe what happens over the course of the crash, and better estimates.

## 1.3 Brief company history

Westinghouse is a company that provides components and equipment to the nuclear industry. Westinghouse also provides fuel and maintenance to several Swedish reactors, as well as developing new fuel types. It was founded in 1962 but back then it was called ASEA atomkraftavdelning. In 1968 ASEA ATOM is founded by a merger between the state run Atomenergi and ASEA AB. The company is contacted by several nuclear power providers and tasked with creating their reactors. From 1968 to 1988 ASEA ATOM builds more than 10 reactors to Swedish and Finnish parties. 1988 ASEA ATOM and BBS Brown Boveri merge and becomes ABB Atom. The company is then acquired by Westinghouse in 2000 and is renamed Westinghouse Atom. And finally Westinghouse Atom changes name to Westinghouse Electric Sweden AB in 2006. Westinghouse Electric Company, which is the parent company, is now owned by the Japanese corporation Toshiba via a majority of the stocks, 78 %.

# 2 Background

## 2.1 A brief history of radioactivity and nuclear power

The start of history for nuclear energy begins in 1789 with the discovery of uranium by a man named Martin Klaproth. It would take more than a century until the next major discovery in the field of radiation was discovered. In 1895 Wilhelm Rontgen discovered ionizing radiation and produced continuous X-rays. Just a year later Henri Becquerel managed to demonstrate that beta radiation and alpha radiation was two types of radiation. The same year Villard proved the existence of a third kind of radiation, gamma radiation. The phenomenon however was not called radiation until Pierre and Marie Curie gave it the name 'radioactivity'.

Several discoveries followed that deepened our understanding of the atom and furthered science but it  was not until 1938 that Otto Hahn and Fritz Strassmann managed to demonstrate that atomic fission occurred in these radioactive materials. In order to utilise the energy released during fission a self-sustainable chain reaction was necessary. Bohr came to America in 1939 and shared the discoveries made by Hahn and Strassmann with Albert Einstein. He also met Enrico Fermi, an Italian scientist that showed that neutrons could split atoms, at a conference and they started discussing the fact that a self-sustaining reaction could release large amounts of energy.

In 1942 Fermi led a group of scientists at the University of Chicago and started to work on developing their theories. By late 1942 they were ready to begin construction on the world's first reactor. On December 2, 1942 at 3:25 p.m., the nuclear reaction became self-sustaining.

Because of the war most of nuclear development was centred on producing a weapon however some scientists worked on it for peaceful use. Since they were so few it took until 1951 to produce a reactor that managed to produce electricity, see Figure 2-1. 6 years later in 1957 the first large-scale commercial nuclear power plant was finished.

**Figure 2-1 Experimental Breeder Reactor I, first electricity produced**

## 2.2 Current state of nuclear power

The International Atomic Energy Agency publishes each year a booklet of nuclear power information about its member states [1]. Among those membership states France leads the statistics by producing 77.8 % of its electricity with nuclear power. Other notable countries are Slovakia with 57 % and Hungary with 54 %

There exists today several different kinds of reactors that each has its positive sides and negative sides. Since TRITON11™ will be used for a certain kind of reactor, a boiling water reactor (BWR) it will be described a bit more in-depth.

### 2.2.1 **Boiling Water Reactor**

Westinghouse provides equipment for boiling water reactors as well as for pressure water reactor (PWR). A BWR uses the energy released from the nuclear fission directly by boiling water with it while in PWR the steam is produced using heat exchangers. Westinghouse in Sweden develops and researches fuel for BWR but the company as a whole makes fuel for several different reactor types. The TRITON11™ fuel will be used exclusively for BWR.

In order to start the process of extracting energy out of nuclear fuel fission needs to occur. This process begins by firing a neutron into the core. When this neutron hits an atom it cleaves the centre of it and releases more neutrons and high amount of energy. Those neutron then crashes into other atoms and the process repeat itself until the control rods are moved up to absorb the neutrons. The energy released takes it form in heat and with that heat the power plants boil large amounts of water. The steam then enters a steam turbine that starts to spin and generate electricity. The following Figure 2-2 is an example of how a BWR can look like.

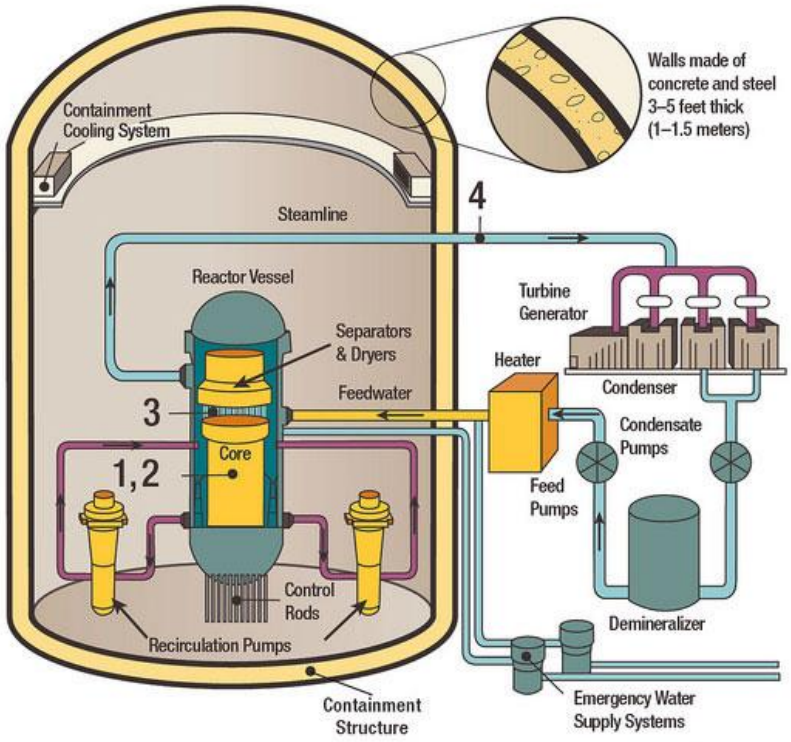



**Figure 2-2 Boiling water reactor (U.S NRC)**

## 2.3 Related work

To get a clear understanding of the problem a literature study was made. Internal documents for the old fuel types was read and analysed. These made it clear how the estimations have been done in the past. From these reports a failure criterion was shown. However this failure criterion is an in-house practice and not the proper failure criterion based on testing this is however the criterion used in this analysis.

In order to be able to have something to compare the results of the analysis with the company is working on getting their calculations finished in time for the FE-analysis. These results will show whether or not the model and the assumptions made are good approximations. This will in not show the exact results and should only be an estimate of whether the results from the FE-analysis are in the right order of magnitude.

## 2.4 FE procedure

When a FE-analysis is created several things are needed before a simulation can be run. First of all a model of the object being tested needs to be created. Then if the model is a design drawing it might be overly complex for the program to be able to produce a valid result in time. Therefore several assumptions and simplifications are made to ease the computational burden. After that input data needs to be researched and decided upon, things like Young's modulus, the yield stress to name a few. When that is finished the size of the mesh grid needs to be decided, this is a weighing between time and accuracy, and the distribution of elements closer examined so as to decide where the mesh needs to be more detailed and where it might not need to be. Later the boundary conditions and the loads have to be defined and decided. Once all of these are finished the simulation can be done. The process is however not done when the simulation is finished. The results have to be verified, validated, and compared to other results to see whether or not they are reasonable.

## 2.5 Problem formulations

In order to create a FE-analysis several obstacles must first be overcome. In no particular order the boundary conditions must be evaluated and examined to determine whether or not they are applicable to the situation, the model and how it is attached to its individual parts must be analysed so no major damage is caused because of the connections, the load cases must be applied and analysed for the same reasons since it has a profound impact on the solution etc.

A couple of questions will be answered in this report to make it clear how the results are achieved.

- What material models are used?
- What are the boundary conditions?
- What are the load cases?
- What is the failure criterion?
- How is the model connected?

The main question however that needs to be answered is:

- How many fuel rods will fail during the impact?

In order to get a good understanding of what the fuel assemblies looks like a picture of the design for SVEA-96 Optima3 (Optima3) will be used since the overall look is similar. The Optima3 fuel assembly design is in use presently and is being produced by Westinghouse. For the general design of Optima3 see Figure 2-3

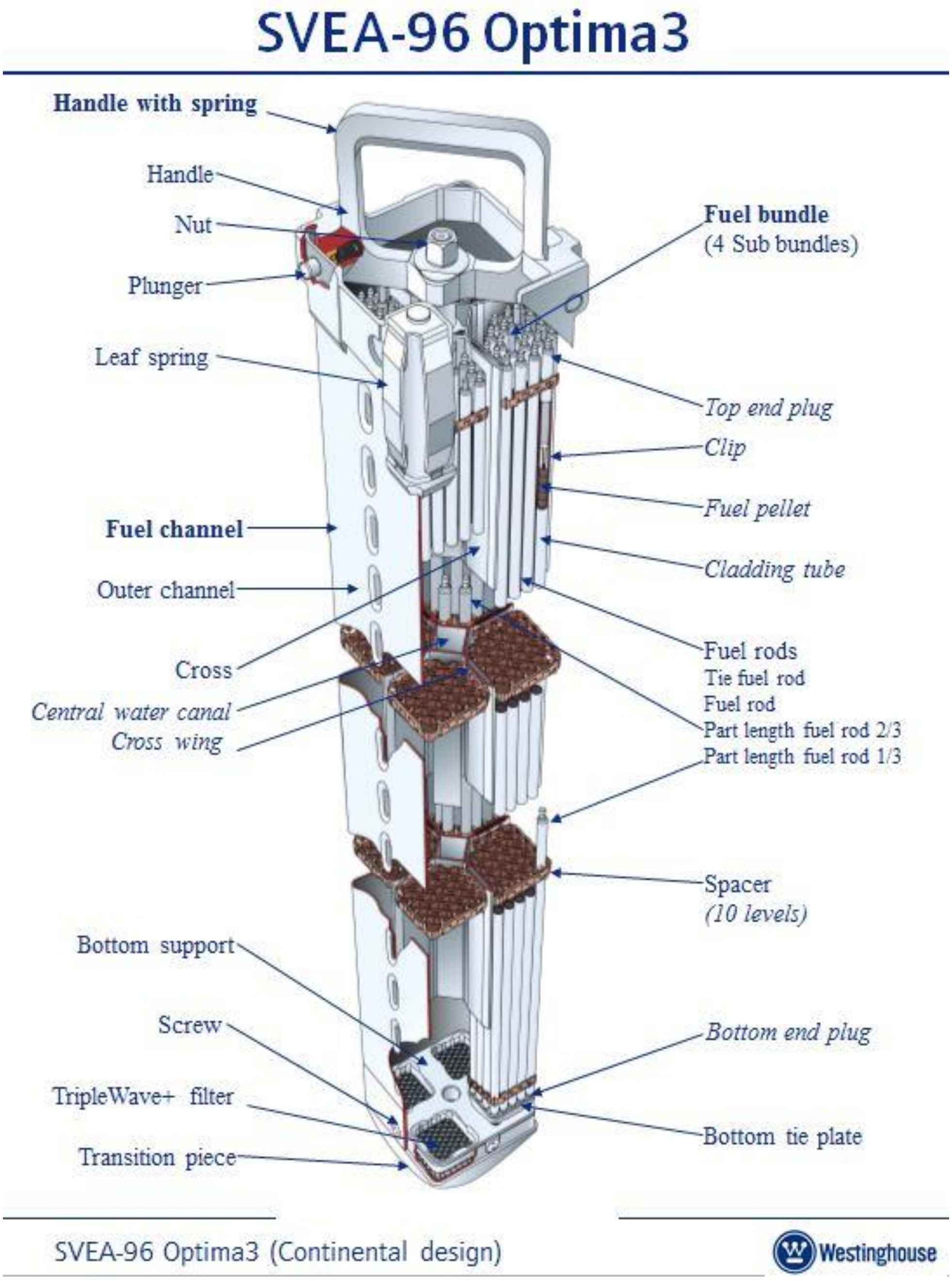



**Figure 2-3 Design of SVEA-96 Optima3 continental design**

### 2.5.1 **Event description**

The postulated fuel handling accident can occur because of several reasons, for example improper grappling, breakage of the fuel assembly handle, or breakage of the fuel grapple to name a few. When the fuel assembly is dropped it gains energy. This energy is divided between the dropped assembly and the assembly that it hits.

After the initial collision the dropped assembly will then fall sideways. The way this assembly falls will be important to the number of fuel rods so several different cases will be examined. Beyond that it is important to distinguish between where the impacted assembly is struck. If it is hit right on the handle it might prevent damage to the impacted assembly, however if struck at other places more excessive damage might occur. The falling assembly is assumed to strike at a small angle and might therefore be subject to bending modes of failure. Since TRITON11™ has its handle connected to the fuel cladding it is more resistant to drop damage compared to many other types of fuel.

## 2.6 Literature recap

### 2.6.1 **Material property degradation**

In metals, research has demonstrated high degradation of mechanical properties due to exposure to high energy particle irradiation. The main points of radiation damage on mechanical behaviours, that is important to this project will be summarised as:

- At low to moderate doses an increase in yield stress usually followed by reduced ductility, and strain hardening capacity
- An instability that leads to loss of ductility, and premature failure

The increase in yield stress means that the material becomes more resistant to plastic deformation and can therefore be loaded more heavily than otherwise. This means that the mechanical properties of the fuel assemblies actually increases and is beneficial for the fuel assemblies.

These changes in mechanical properties occur because of changes in the material microstructure. So in order to predict and understand the property changes of the material a detailed understanding of the link between microstructure and mechanical properties and how the microstructural changes are affected by the radiation is necessary [2].

### 2.6.2 **Failure criteria**

The following subsection is taken from [3]. In order to create something to perform a specific function a clear understanding of the modes of failure needs to exist. Failure criteria can then be established that accurately predict these modes of failure. Normally to be able to determine these modes of failure an extensive knowledge about the response to loads that a system has is necessary. Specifically a detailed stress analysis needs to be made. The response of the system is heavily influenced by the material and therefore the modes of failure are heavily dependent on the material used. Furthermore the load application history is also of critical importance, static or dynamic load, compression or tension, fast application or slow, all affect the modes of failure to a certain degree.

Since there are so many different failure criteria based on the loading of the system only a couple of failure modes will be of interest in this thesis. Those given by fatigue and slow application are not examined since it is an impact that will happen and that is a fast application that only happens once and not under several cycles. An impact has fast application and is dynamic in the sense that it varies in compression during the impact.

There are several different ways with which a structural member can fail. These failures are:

- Failure by elastic deflection
- Failure by yielding
- Failure by fracture

These criteria each have a couple of subsections depending on how and why these failure criteria occur.

#### *2.6.2.1 Failure by Elastic Deflection*

Failure by Elastic Deflection means that the maximum load applied without the structure ceasing to function in its intended way can be limited by the maximum allowed stress or strain. Failure by Elastic Deformation can occur during the following conditions:

a) Deflection under condition of stable equilibrium such as the tension on a stretched beam.

b) Deflection under conditions of unstable equilibrium such as buckling.

c) Elastic deformation as a result of vibrations such as parts colliding because of the violent shaking of the system.

#### *2.6.2.2 Failure by Yielding*

Failure by yielding is when a member fails by being subjected to inelastic (plastic) deformation on a significant portion of the member, as opposed to the small localized deformation such as in the chapter above. This sort of failure criterion is more commonly seen in simple geometries such as beams and pipes. The criterion can be initiated by everything from torsion to high compression of the member.

Failure by yielding is often divided into two categories, namely:

a) Extensive Yielding at Ordinary Temperatures

b) Extensive Yielding at Elevated Temperatures, Creep

#### *2.6.2.3 Failure by Fracture*

Failure by fracture means that some members fracture before they experience failure by the other failure modes. This mode is divided into four rather different categories:

a) Sudden fracture of brittle material

b) Fracture of cracked or flawed members

c) Progressive fracture (fatigue fracture)

d) Fracture at time with elevated temperature

### 2.6.3 **Explicit vs. Implicit solver**

In order to understand what it is LS-DYNA does an understanding of what an explicit solver does is needed which means it is necessary to elaborate on the differences between and implicit and an explicit solver. An implicit solver uses the previous time step to calculate the results of the next load step. While doing so it has to calculate the inverse of the stiffness matrix which is a very demanding computational task and therefore it takes a long time to do. The advantage of using this method is that since it uses a so called Euler Time Integration Scheme the solution is always stable.

An explicit solver does functions by not calculating the inverse of the stiffness matrix but can instead utilize the inverse of the mass matrix. This matrix is usually a diagonal matrix since the explicit solvers prefer to use lower order elements and thus the inversion is just a single step for the computer to do. This means that the calculation steps are very easily done. However since an Euler Time Integration

Scheme is not used the solution isn't inherently stable and thus must be compensated for by using smaller time steps.

Since LS-DYNA will be used to solve these problems an explicit solver has already been chosen.

### 2.6.4 **Nonlinear modelling**

When working with FE one has to decide whether it is worth the computational power to use a nonlinear simulation instead of a linear one. A linear FE program is made to simulate what happens when the stress-strain curve is within its linear elastic part, in other words when no permanent deformations occur. However once larger deformations happen and plastic deformation is a fact a nonlinear environment is necessary to model these phenomena. This of course requires greater computational power but since no accurate results will be acquired without it, it is a drawback that has to be accepted.

### 2.6.5 **Solver analysis**

LS-DYNA is capable of solving numerous engineering problems since Livermore Software Technology Corporation (LSTC) was founded to work solely with developing LS-DYNA. The code's origins lie in highly nonlinear, transient dynamic finite element analysis using explicit time integration [4].

"Nonlinear" means at least one (and sometimes all) of the following complications:

- Changing boundary conditions (such as contact between parts that changes over time)
- Large deformations (for example the crumpling of sheet metal parts)
- Nonlinear materials that do not exhibit ideally elastic behaviour (for example thermoplastic polymers)

"Transient dynamic" means analysing events that are time dependent and where inertial forces are important. Typical uses include:

- Automotive crash (deformation of chassis, airbag inflation, seatbelt tensioning)
- Explosions (underwater Naval mine, shaped charges)
- Manufacturing (sheet metal stamping)

For the purpose of this thesis the fact that it can be used in crash testing of automobiles makes the software capable of simulating the processes and events of the drop test that will be performed.

In order to get an estimation of how big LS-DYNA is on the market the number of hits given when searching for “LS-DYNA” in both google scholars database and in the Lund university library database will be recorded. This will then be compared to other solvers such as the like of “ABAQUS/Explicit” and “ANSYS Explicit”. Using this phrase generated about 36,600 hits on google scholar and 3,241 hits in the university library. The ABAQUS search generated 14,100 hits on google scholar and 1,336 hits on the university library. The ANSYS search generated 18,500 hits on google scholar and 223 hits on the university library. With this comparison it can be seen that LS-DYNA is one of the biggest and most extensive explicit solvers.

# 3 State of the art

Most of the reports and literature dealing with solid mechanical analysis of nuclear fuel is focused on the vibrational problem that occurs when the water flows between the rods. However some address accidents during handling outside of the reactor. These accidents usually deal with fully irradiated material but there are still useful in comparing results as well as verifying if the methods used within the thesis are applicable at all and

In [5] the fuel rods are encapsulated in a transport case that minimizes the damage from a drop. It however shows the tendencies of the case to deform at the bottom in a way that might be applicable for this thesis. The fuel rod damage is not evaluated in [5] but an experimental setup is created and utilized to confirm the results of the FE-analysis. The experimental setup is simplified by making it shorter. In order to still have the weight be somewhat close to real assembly a weight is attached to the top of the experimental assembly. The drop height utilized is that of 13 metres. In [5] shell elements are used to simplify the rods. This simplification seems to be acceptable since both the experimental setup and the computation model produce similar results. Both the FE-analysis and the experimental results show that the connection between the bottom piece and the support plate for the rods will get moved closer together and that the grid separating them will buckle outward. Fuel rod damage is not analysed in [5], however the strain is. The strain result show that the maximum strain achieved in the experiment is 0.7 % strain inside a container. It also shows that vibrations occur due to the impact which is to be expected. There are discrepancies between the residual strains in the model compared to the experiment. A possible explanation is the shell elements, or that the mesh has to be further refined.

[6] deals with the different kinds of stress that the rods experience in a PWR. While not entirely the same as a BWR the results might still be applicable in this thesis. [6] also uses one experimental setup and one FE model to validate the results obtained from the model. In order to save computational time [6] uses half symmetry to lessen the number of nodes

and elements. In [6] the simplification of the rods is also shell elements. The main bulk of elements used in this FE-analysis is shell181, unsurprising since the model is half of a fuel assembly without a channel meaning it is mostly the rods that are modelled and thus shell181 is suitable.

[7] shows how $UO_2$ fuel ruptures in a nuclear reactor. It is not entirely the case in this thesis but it is useful to know how rupture and happens in irradiated fuel. There are two different samples used, one notched and one smooth. The crack models used are based on two different main parameters, the critical stress and the surface energy. In order to model the crack propagation a simplified smeared crack model is used in finite element simulations. Another type of crack model is also explored: the cohesive zone model. In [7] an alternative analytical model is used as well to compare to the FE-analysis, this however is not of great use for this thesis. The way that [7] makes the crack propagate is through a bending test which is could have relevance since bending occurs during the fuel handling accident described later. One of the clear conclusions of [7] is that the critical stress is dependent on, the surface energy, the rupture stress and the thickness of the sample. Using all of these assumptions as well as the criterion used within [7] the scope of this thesis can be expanded upon should one wish and need. These tests are performed on fully irradiated materials so the material parameters will differ between [7] and this thesis. One experimental rig is also set up to verify the results. To obtain all the rupture parameters used in the crack models, an evaluation of the critical stress from Vickers indentation tests is essential. During the bending test, after a stable propagation of damage, the crack due to indentation becomes unstable. It is also concluded that the critical stress intensity factor is the main parameter differentiating between fresh and irradiated fuel

[8] mentions that the majority of stress or strain tests performed on fuel assembly claddings are performed with irradiated materials. This radiation does create some differences in material parameters as previously discussed. [8] utilizes only an experimental model and no FE-analysis at all. It does however broach the subject of how testing on the fuel cladding might be performed. [8] is not examining a full assembly with channels and spacers but is rather an examination of just the cladding for one single fuel rod. The experiment is done by using a driver tube to simulate the expansion of a fuel pellet and thus seeing how the cladding material might behave. [8] mentions the temperatures used and shows that the ones used in

this report is reasonable. It also shows that the cladding is highly loading dependent and if the impact lasts longer or shorter than described in the parameters there will be differences. [8] mentions how the cladding seems to be hydrogen concentration independent while below the brittle to ductile transition temperatures

# 4 Method

## 4.1 Significant Assumptions

In order to make sure that the results of this report can be replicated the assumptions made will be outlined.

The material is assumed to follow a Ramberg-Osgood material model for stress-strain, this affects the material model and might ultimately affect how much stress and strain the fuel assembly can absorb.

The temperature of the surrounding water and thus also the working temperature will be 60°C.

## 4.2 Limitations

The model that is used is the design model for TRITON11™ finished in December of 2015. This limitation means that it is not the exact geometry that will be finished however there will be no need to update the computational model each time a change is made in the TRITON11™ model.

The goal of the report is to examine the number of fuel rods damaged during a drop test and therefore mechanics and solid mechanics are the focus of the report. The report does not describe the consequences of possible release of nuclear material within the reactor pressure vessel.

This report will be limited to simulate and evaluate only one of the identified load cases discussed in chapter 4.

## 4.3 Governing Equations

When a FE-analysis is made governing equations needs to be stated. These are the equations that calculations program will solve. They range from heat flux to strains to stresses and more. In this analysis several different governing equations are important. They are:

- Conservation of mass
- Conservation of momentum
- Conservation of energy
- A constitutive equation
- Momentum equation

Most of these are automatically used by LS-DYNA when you start a simulation. The conservation of mass, momentum, and energy are almost universally used on all forms of FE-analysis, however the constitutive equations needs to be user defined [9]. The following text and equations in this subsection is taken from [9]

The equation for the conservation of mass for a Lagrangian formulation is stated as:

$$\rho J = \rho_0 J_0 \tag{4.1}$$

Where J is the Jacobian, $J_0$ is the initial Jacobian which is assumed to be unity, ρ is the density and $\rho_0$ is the initial denisty. This equation states that the mass in the beginning of the event has to be the same during the entire event.

The conservation of momentum means that the momentum in a closed system, which in FE-analysis most systems are assumed to be, is constant. The equation is stated as:

$$(A_0 P)_{,X} + \rho_0 A_0 b = \rho_0 A_0 \ddot{u} \tag{4.2}$$

Where:

$$(A_0 P)_{,X} = \frac{\partial (A_0 P)}{\partial X} \tag{4.3}$$

And $A_0$ is the cross-sectional area at coordinate X, b is the body force and the superposed dots means the second time derivative.

Conservation of energy means that the energy in the system needs to be constant over time. Meaning no new energy is added or removed during the event. The equation looks like:

$$\rho_0 \dot{w}^{int} = \dot{F} P \tag{4.4}$$

Where $\dot{w}^{int}$ is the rate of the internal work, $\dot{F}$ is the rate-of-deformation gradient, and $P$ is the nominal stress.

The constitutive equations are meant to give the stresses that result from the deformations. These equations are specific to each material used in the model.

The momentum equation is given in [9] as:

$$\rho \frac{D\boldsymbol{v}}{Dt} = \nabla * \boldsymbol{\sigma} + \rho \boldsymbol{b} \equiv div\boldsymbol{\sigma} + \rho \boldsymbol{b} \qquad \textbf{(4.5)}$$

The term on the left side represents change in momentum due to it containing acceleration and density; it is also called the kinetic or inertial term.

## 4.4 FE-formulation

When it comes to the implementation of the finite element equations, two approaches are popular [9]. Either the indicial expressions are directly treated as matrix equations or Voigt notation is used and the square stress and strain matrices are converted to column matrices.

Here follows an algorithm for Lagrangian formulation.

Equations of motion:

$$M_{ijIJ}\dot{v}_{jJ} + f_{iI}^{int} = f_{iI}^{ext} \text{ for (I,i) not on the edge of } v_i \qquad (4.6)$$

Internal nodal forces:

$$f_{iI}^{int} = \int_{\Omega} B_{iJ}\sigma_{ji} d\Omega = \int_{\Omega} \frac{dN_I}{dx_j}\sigma_{ji} d\Omega \text{ or } (\boldsymbol{f}_I^{int})^T = \int_{\Omega} B_I^T \boldsymbol{\sigma} d\Omega \qquad (4,7)$$

$$\boldsymbol{f}_{\boldsymbol{I}}^{int} = \int_{\Omega} \boldsymbol{B}_I^{\boldsymbol{T}}\{\boldsymbol{\sigma}\} d\Omega \text{ in Voigt notation.} \qquad (4.8)$$

External nodal forces:

$$f_{iI}^{ext} = \int_{\Omega} N_I \rho b_i d\Omega + \int_{\Gamma_{t_i}} N_I \bar{t}_{\ i} d\Gamma \text{ or} \qquad (4.9)$$

$$\boldsymbol{f}_I^{ext} = \int_{\Omega} N_{I\rho} \boldsymbol{b} d\Omega + \int_{\Gamma_{t_i}} N_I \boldsymbol{e}_i * \bar{t} d\Gamma \qquad (4.10)$$

Mass matrix (updated Lagrangian):

$$M_{ijIJ} = \delta_{ij} \int_{\Omega_0} \rho_0 N_I N_J d\Omega_0 = \delta_{ij} \int_{\Box} \rho_0 N_I N_J \boldsymbol{J}_\zeta^0 d\,\Box \qquad (4.11)$$

$$\boldsymbol{M}_{IJ} = \boldsymbol{I}\bar{M}_{IJ} = \boldsymbol{I} \int_{\Omega_0} \rho_0 N_I N_J d\Omega_0 \qquad (4.12)$$

Internal nodal force computation for element

1. $\boldsymbol{f}^{int} = 0$
2. For all quadrature points $\zeta_Q$

   a. Compute $[B_{Ij}] = \frac{\partial N_I(\xi_Q)}{\partial x_J}$ for all I

   b. $\boldsymbol{L} = [L_{ij}] = [v_{iI} B_{Ij}] = \boldsymbol{v_I} \boldsymbol{B_I^T}; L_{ij} = \frac{\partial N_I}{\partial x_j} v_{iI}$

   c. $\boldsymbol{D} = \frac{1}{2}(\boldsymbol{L}^T + \boldsymbol{L})$

   d. If needed compute **F** and **E**

   e. compute Couchy stress **σ** or PK2 stress **S** by constitutive equation

   f. if S computed, compute **σ by** $\boldsymbol{\sigma} = J^{-1}\boldsymbol{FSF}^T$

   g. $\boldsymbol{f}_I^{int} \leftarrow \boldsymbol{f}_I^{int} + \boldsymbol{B_I^T} \sigma \boldsymbol{J}_\zeta \bar{w}_Q$ for all nodes *I*

   End loop

   $\bar{w}_Q$ are quadrature weights

In order for the computer to be able to solve FE problem, the equations first needs to be stated in a form that it can understand and solve. So in order to construct these equations they need to be written in a FE-formulation. This means that the equations, whether that be heat flux, stresses or deformations, are written on matrix forms. This means that the combined equation for all of the nodes is put together into one matrix. This matrix is either the stiffness matrix or the mass matrix. Since an explicit solver will be used it is the mass matrix that will be used to solve the equations.

This is done by first creating a strong formulation of whatever problem is needed to solve and applying it to either one- two- or three-dimensional space. This form of the equation is the traditional one and is easier for humans to solve. The next step is to transform the strong form into the weak form. The weak form is divided into segments according to where on the body the equation is solved e.g. if on the boundary it is a boundary integral with special conditions.

Down below is a flowchart for explicit time integration provided by [9]

1. Initial conditions an initialization:
   set $\boldsymbol{v}^0$, $\boldsymbol{\sigma}^0$, and initial values of other material state variables;
   $\boldsymbol{d}^0 = \mathbf{0}$, $n = 0$, $t = 0$; compute $\mathbf{M}$
2. *getforce*
3. Compute accelerations $\boldsymbol{a}^n = \boldsymbol{M}^{-1}\left(\boldsymbol{f}^n - \boldsymbol{C}^{damp}\boldsymbol{v}^{n-1/2}\right)$
4. Time update: $t^{n+1} = t^n + \Delta t^{n+1/2}$, $t^{n+1/2} = \frac{1}{2}(t^n + t^{n+1})$
5. First partial update nodal velocities: $\boldsymbol{v}^{n+1/2} = \boldsymbol{v}^n + (t^{n+1/2} - t^n)\boldsymbol{a}^n$
6. Enforce velocity boundary conditions
   If node $I$ on $\Gamma_{v_i}$: $v_{iI}^{n+1/2} = \overline{v_i}(\boldsymbol{x}_I, t^{n*1/2})$
7. Update nodal displacements: $\boldsymbol{d}^{n+1} = \boldsymbol{d}^n + \Delta t^{n+\frac{1}{2}}\boldsymbol{v}^{n+1/2}$
8. *getforce*
9. Compute $\boldsymbol{a}^{n+1}$
10. Second partial update nodal velocities: $\boldsymbol{v}^{n+1} = \boldsymbol{v}^{n+1/2} + (t^{n+1} - t^{n+1/2})\boldsymbol{a}^{n+1}$
11. Check energy balance at time step *n*+1
12. Update counter: n←n+1
13. Output; if simulation not complete, go to 4

Subroutine *getforce*

1) Initiation: $\boldsymbol{f}^n = 0, \quad \Delta t_{crit} = \infty$
2) Compute global external nodal forces $\boldsymbol{f}^n_{ext}$
3) Loop over element $e$
   a) GATHER element nodal displacements and velocities
   b) $\boldsymbol{f}^{int,n}_e = \boldsymbol{0}$
   c) Loop over quadrature points $\boldsymbol{\xi}_Q$
      i) If $n$=0, go to 4
      ii) Compute measures of deformation: $\boldsymbol{D}^{n-\frac{1}{2}}(\boldsymbol{\xi}_Q), \boldsymbol{F}^n(\boldsymbol{\xi}_Q), \boldsymbol{E}^n(\boldsymbol{\xi}_Q)$
      iii) Compute stress $\boldsymbol{\sigma}^n(\boldsymbol{\xi}_Q)$ by constitutive equation
      iv) $\boldsymbol{f}^{int,n}_e \leftarrow \boldsymbol{f}^{int,n}_e + \boldsymbol{B}^T \boldsymbol{\sigma}^n \bar{w}_Q J|_{\xi_Q}$

      END quadrature point loop
   d) Compute external nodal forces on element, $\boldsymbol{f}^{ext,n}_e$
   e) $\boldsymbol{f}^n_e = \boldsymbol{f}^{ext,n}_e - \boldsymbol{f}^{int,n}_e$
   f) Compute $\Delta t^e_{crit}$ , if $\Delta t^e_{crit} < \Delta t_{crit}$ then $\Delta t_{crit} = \Delta t^e_{crit}$
   g) SCATTER $\boldsymbol{f}^n_e$ to global $\boldsymbol{f}^n$
4) END loop over elements
5) $\Delta t = \alpha \Delta t_{crit}$

## 4.5 Hydrogen Embrittlement

The process with which the assembly’s mechanical properties of the fuel rods changes is called hydrogen embrittlement. Hydrogen embrittlement is when, during operating life, hydrogen slowly diffusions into the material and the material properties slowly change because several per cent of the object now consists of hydrogen. The reason that this process is of such importance in radioactivity is that at higher temperatures the hydrogen diffusion increases. This happens because the solubility of the hydrogen increases at higher temperatures. The benefits of this embrittlement are that some of the mechanical properties such as the yield strength are increased so that it is harder to plastically deform it. However this process also makes the rods brittle as hinted to by the name of the process, meaning that the rods becomes sensitive to bending. This is a process that comes into effect during the life of the rods and is therefore not something to consider at BOL but rather at EOL.

## 4.6 Failure Criterion

The failure condition that Westinghouse uses in the hand calculations is a cross-section with a total strain of 1%. This criterion is based on the failure by elastic deflection, namely the first of the categories, failure under stable equilibrium. It should be noted that this criterion means that if it is below this limit the rods can still be used in the reactor without any further actions.

## 4.7 ANSYS and LS-DYNA

By first simplifying the given geometry in SpaceClaim it can be imported into the ANSYS environment. In ANSYS the different parts were assigned their respective materials, contact conditions and connections. They were then meshed and the simulation started. Since the LS-DYNA extension cannot be interrupted the Restart WB LS-DYNA was used to get results without restarting the simulation.

## 4.8 Modified Ramberg-Osgood

In order to describe the relation between stresses and strains when a material is near its yield point, a modified version of the Ramberg-Osgood equation will be used. The equation models the plastic strain and creates the nonlinear part of the typical stress-strain curve. The original equations were presented in [10] but are modified for use in the LS-DYNA solver as per [11].

# 5 Computational Input

## 5.1 Geometry

The overall look of TRITON11™ as mentioned before looks like the Optima3, see Figure 2-1. See Figure 5-1 for a simplified geometry.

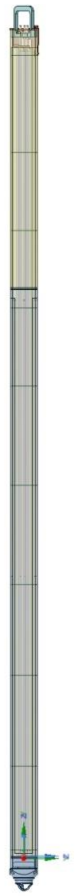


**Figure 5-1 Simplified Triton geometry**

Inside however the design is different. Instead of being organised in a pattern of four smaller bundles TRITON11™ uses an 11x11 grid on its spacers. This pattern is broken up by three water rods that each take up the space of four fuel rods for a total of 109 rods, see Figure 5-2.

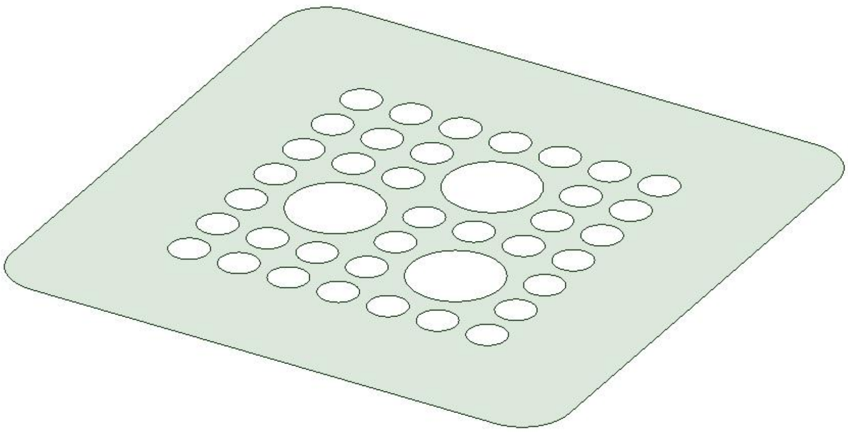


**Figure 5-2 Spacer design sketch**

## 5.2 Load Cases

It is important to distinguish between the two different striking regions of the impacted assembly. If it is struck on top of the handle some damage to the impacted assembly might be prevented, however if struck on the channel directly more excessive damage might occur. The dropped assembly can be assumed to strike at a small angle and might therefore be subject to bending modes of failure. Since the fuel rods at EOL are long brittle beams they are not very resistant to the bending moment that might be applied and therefore it has always been assumed that the rods in the dropped assembly all fail. This however does not necessarily have to be the case.

The temperature of the surrounding water and thus also the working temperature will be 60°C.

In order to determine which parameters are important to the results it is first necessary to define what solid mechanics phenomena that might introduce failure to the fuel rods.

The load cases presented in this chapter are the cases that are suggested to be studied.

### 5.2.1 **Fuel Rods of Dropped Assembly**

There are several factors that determine whether or not the fuel rods of the dropped assembly fail:

- Bending moment
- Compression stress
- Impact time
- Stage of life

The bending moment if sufficiently high can cause the rods to fail because the hydrogen embrittlement makes the rods a lot more brittle at EOL. Bending moment can be adjusted depending on for example the angle with which the dropped assembly is dropped i.e. if the angle is higher the force created from the impact will bend the rods rather than compress.

Impact time affects the rods by either increasing or decreasing the load applied. A fast impact time usually results in higher acceleration however if the impact time resembles the natural frequency of the object the impact is more damaging even if it happens over a longer time period.

The stage of life is influential because of the hydrogen embrittlement that occurs over the life of the assembly. The hydrogen diffuses into the material making it more brittle than at the start. However this same process increases the yield strength by a factor between 1.5 and 2. This means that at EOL the assembly has more favourable properties for not breaking. In this analysis BOL condition will apply

### 5.2.2 **Fuel Rods of Impacted Assemblies**

The same factors affecting the dropped assembly also affect the impacted assemblies. These however have the extra conditions of where on the assembly they are struck.

The area which the impacted assemblies are struck by the dropped assembly possibly has major consequences to the damage caused to the impacted assemblies. If it is struck on the handle it will absorb part of the energy needed to deform the assembly and might therefore prevent some of the rods from failing. If however it is struck on the channel, the deformed channel might hit the rods sooner than if the handle would be struck.

### 5.2.3 **Assumptions**

If the dropped assembly is dropped while almost being held stationary the angle of tilt it is dropped with is assumed to be 0°. If the assembly is dropped while in motion a larger tilt can occur. This tilt is chosen to be 10°. This tilt is quite high since it will only fall for 6 meters. It is meant to examine how tilt affects the damage and therefore a large tilt is chosen.

Since the worst case scenario is the basis the amount of assemblies struck will be tested. In the current methodology it is assumed that the dropped assembly only strikes one other assembly in the reactor. Because of the fact that the energy is then split between these two bodies that means that this scenario is assumed to be the worst case. In this report, cases where several assemblies are struck are suggested to be investigated.

The assemblies are assumed to be either struck on the channel or the handle, in order to see which case is the most damaging.

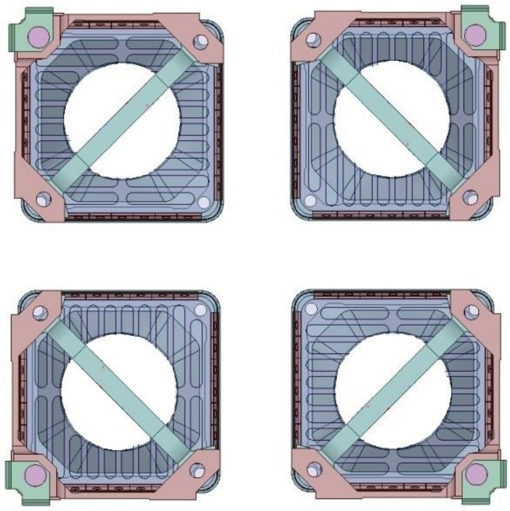


**Figure 5-3 Super cell configuration (distance between not to scale)**

### 5.2.4 **Load Case 1**

Load case (LC) 1 will largely be the same as the LC used in the current methodology. Since the end results of this report will be compared to the results of the current methodology it is logical to have the first LC be the one that is the most similar to the one used today. In the current methodology the falling assembly is assumed to strike at a small angle. In LC1 this small angle is simplified to 0°. It will fall onto only a single other assembly so as to try maximising the damage on impact. This LC is divided into two simulations. The first simulation strikes only the handle of the impacted assembly in order to see the effects on the falling assembly. The other will have only a transition piece with modified weight falling onto an assembly to simulate what happens to the impacted assembly. This means less complexity and less ways for failure to occur. The striking area is the top of the handle. This is the only LC that will be tested in this thesis partly because of the fact that it is the LC that most resembles the current methodology and partly because of time constraint.

### 5.2.5 **Load Case 2**

LC2 will have slightly higher degree of tilt 10° but will strike the assembly in the same way. The purpose of this LC is to investigate the effect of the angle on the damage caused.

### 5.2.6 **Load Case 3**

LC3 will fall with the tilt of 10° but this time strike two assemblies to see how much effect that will have on the damage of the fuel rods. Since it will strike in the middle of two assemblies it will most likely strike the handle as can be seen in Figure 5-3

### 5.2.7 **Load Case 4**

LC4 will strike with 10° tilt but will strike in the middle of the super cell and will therefore strike all four of the assemblies on the channel itself. The purpose of this LC is to see how much damage will occur if all the assemblies are affected by the accident. The configuration of the assemblies that will be struck makes it virtually impossible to strike three assemblies since the fourth almost always will be stuck as well as seen in Figure 5-3.

**Table 5-1 Load Cases**

| Load case | Θ (tilting angle) | Assemblies hit | Striking region |
|---:|---:|---:|---:|
| 1 | 0° | 1 | handle |
| 2 | 10° | 1 | handle |
| 3 | 10° | 2 | handle |
| 4 | 10° | 4 | channel |

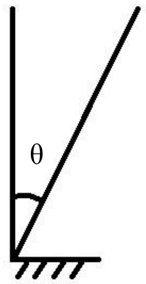



**Figure 5-4 Tilt angle θ**

# 6 Simulation preparation

## 6.1 Geometry

To be able to proceed with the FE-analysis, simplifications of the geometry are made. These simplifications are necessary to create a geometry that is solved within an acceptable timeframe.

Since this report only focuses on structural integrity all the components that do not contribute in a noticeable way to it can be safely neglected and its weight can be redistributed to the surrounding material.

One of the simplifications will be to remove all the screws of the geometry and instead add the mass to the surfaces being held together. The surfaces will, in the model, be bonded together so as to create a contact condition. The screws structural integrity adds very little to the model and can therefore be safely neglected. However in order to not change the weight of the assembly the mass of the screws has to be reallocated to the surfaces they held together. The same reasoning can be applied to all of the nuts of the geometry.

The flow filter has an important task but is structurally insignificant for the geometry and can therefore be disregarded in this capacity, thus simplifying the model considerably. The filter however has mass that will be transferred to surrounding objects.

In order to mesh springs properly a lot of elements and nodes have to be used. This means they are a significant drain on the resources while only contributing in a miniscule way. This means that they will be ignored in the FE geometry.

## 6.2 Mesh

In order to get a mesh that is functional in both size and accuracy a number of constraints had to be put onto the mesh. The exact number of nodes and elements is presented further down.

Using ANSYS multizone method to create the handle and using solid elements results in a mesh that follows the contour of the handle very well and manages to create very good elements. Using a sizing constraint saying that the elements should be 8mm, the result is shown in Figure 6-1.

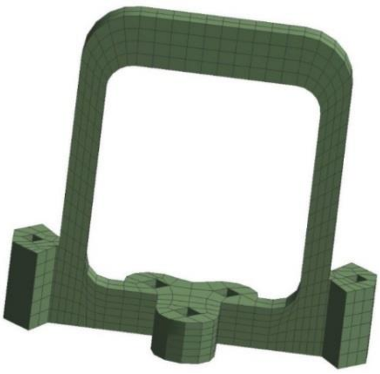


**Figure 6-1 Handle mesh**

The channel fastener is one of four parts of the model that needs solid elements. The only constraint on the fastener is that the size of the elements should be 9mm. See Figure 6-2.

**Figure 6-2 Channel fastener mesh**

All the rods are meshed using shell elements and with using face meshing and size constraints since they are long thin bodies. Using facemesh the length of the rods is divided into 20 parts and the size constraint divides the circular cross-section into a hexagonal shape. The rod mesh is shown in Figure 6-3 and Figure 6-4.

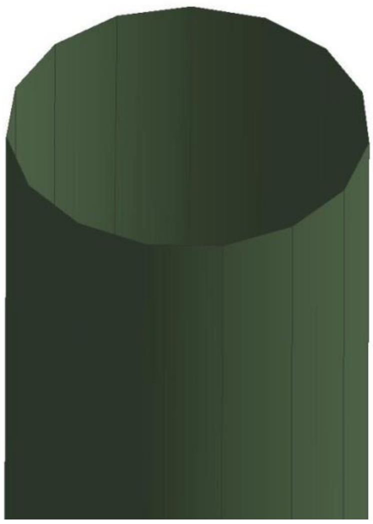


**Figure 6-3 Fuel rod division at top level**

**Figure 6-4 Rod side mesh**

The spacers are the parts that make up the largest amount of elements. This partially due to the fact that there are 10 of them on every assembly and partly due to the fact that the geometry is so complex it requires small elements. The sizing constraint on the spacers is 6 mm. See Figure 6-5.

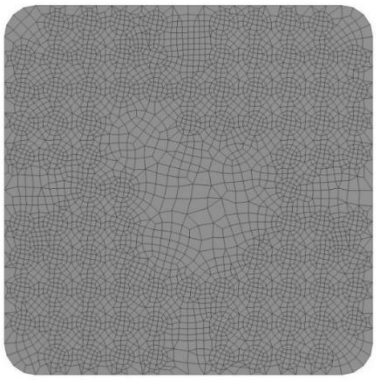


**Figure 6-5 Spacer mesh**

The top of the channel is very coarsely meshed because it does not have a lot of interfering geometry and no parts needing a good estimation of its curvature. The sizing constraint on the elements is 10mm which creates large blocks as shown in Figure 6-6. The channel is meshed using face meshing to try and provide a structured mesh. This is however coupled with a sizing constraint that creates elements with the size of 10 mm. The result is a mesh that contains large quadrilateral elements that create a nice structured mesh containing small irregularities.

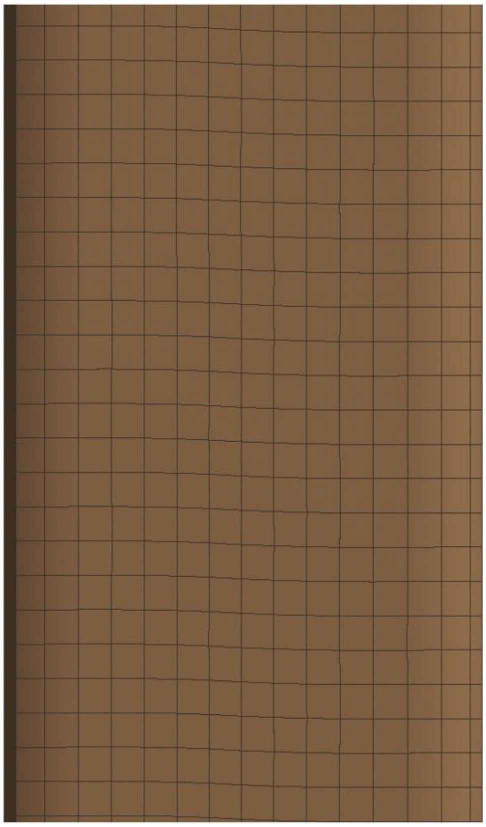


**Figure 6-6 Top channel mesh**

The bottom plate is meshed using solid elements and with a size constraint of 8.5mm. See Figure 6-7.

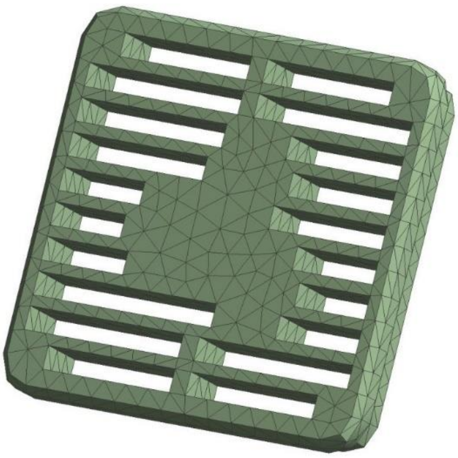


**Figure 6-7 Bottom plate mesh**

The transition piece is created using solid elements since the geometry does not allow anything else. Because of this it is one of the objects with the greatest amount of elements. The fact that the geometry of the piece is complex also means that tetrahedral elements have to be used, the result of which is shown in Figure 6-8.

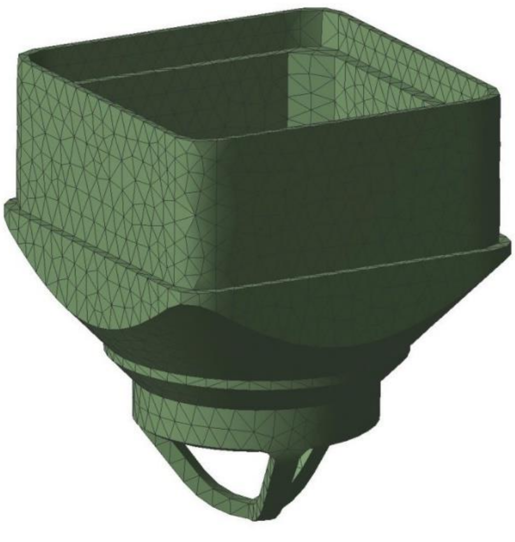


**Figure 6-8 Transition piece mesh**

The spacers and fuel rods vary in size and therefore have different amounts of elements and node between them. The other parts are all constant and a summary of all the elements and nodes can be seen in Table 6-1.

**Table 6-1 Element and node description**

| | **Upper Assembly** | | **Lower Assembly** | |
|---|---|---|---|---|
| | Elements | Nodes | Elements | Nodes |
| Handle | 1160 | 2008 | 970 | 1716 |
| Fastener | 2150 | 841 | 10841 | 3281 |
| Rods | 195-435 | 210-450 | 150-300 | 135-315 |
| Water rods | 522 | 540 | 360 | 378 |
| Spacers | 4564-5064 | 4330-4790 | 4593-5066 | 4358-4801 |
| Channel | 24423 | 24476 | 24748 | 24799 |
| Transition piece | 8570 | 2545 | 10562 | 3054 |
| Bottom support plate | 4474 | 1602 | 4460 | 1590 |
| **Total** | 135829 | 110761 | 109681 | 97782 |

## 6.3 Contact & Connections

The fastener will be in contact with the handle using bonded contact at the on the surface where the screws are keeping them together, see Figure 6-9.

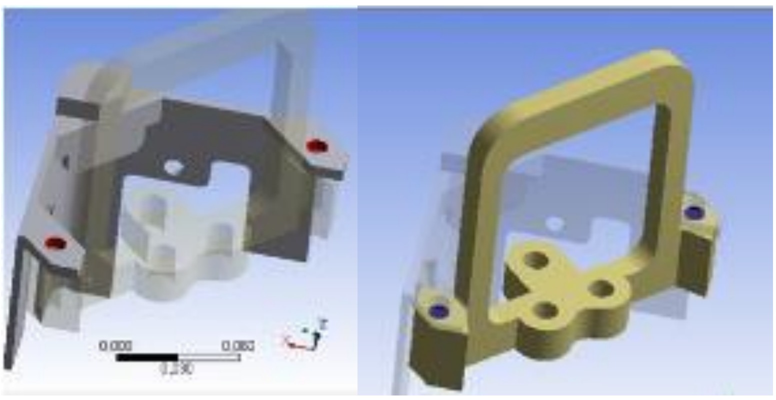


**Figure 6-9 Bonded contact between handle and fastener**

There exists one other bonded contact between the water rods and the bottom plate since they are held into place beneath it. LS-DYNA is however unable to create a contact between an edge and a surface and therefore it is not shown in the results.

The spacers and the rods are meshed together creating a bonded contact between them, shown in Figure 6-10.

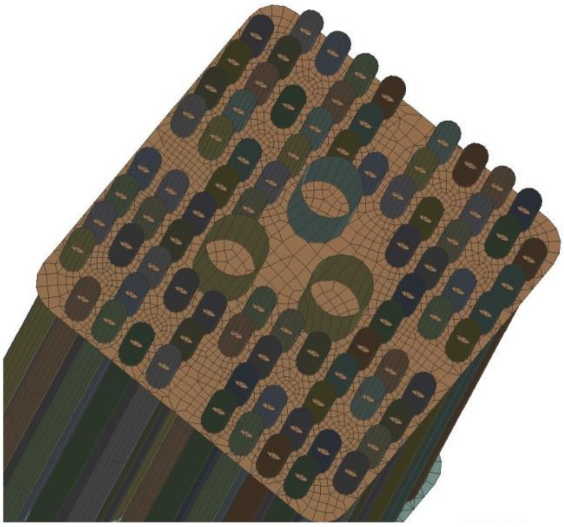


**Figure 6-10 Spacers and rods meshed together**

The bottom plate and the transition piece are meshed together as a single entity thus creating an equivalent bonded contact as seen in Figure 6-11.

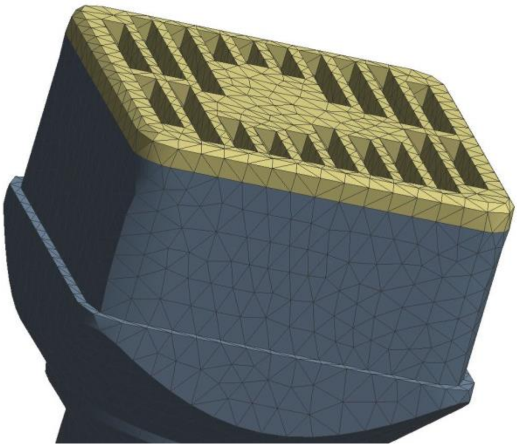


**Figure 6-11 Transition piece and bottom plate meshed together**

For the rest of geometry simple body interaction will be the main connection. Body interaction means that no friction applies when the bodies meet they will however be able to impact one another, such as deforming and bouncing of each other and is a LS-DYNA specific contact.

## 6.4 Boundary Conditions

The boundary conditions state how the simulation will behave. A couple of things will affect the chosen boundary conditions such as:

- How much of the impacted assemblies will be modeled.
- The surrounding medium.

The simulation can be carried out in one of two ways. Either the entire geometry of the impacted assemblies is modelled or only parts of it. This means the simulation can be carried out in one step or two. The advantage of the two step method is that it is not as computationally demanding as the other one. It is however less accurate since the impacted assemblies have to be simulated later on by transferring the forces from the first step creating a less accurate simulation overall. If the one step method is used the simulation will take longer time but will be more accurate since the entire geometry of the impacted assemblies is used. The boundary conditions are affected since it is important which nodes will be locked. Another consequence of the step choice is the fact that the stiffness matrix affects the impact time which in turn decides the size of the forces acting on the assemblies. Using the two step approach means that the stiffness will be significantly higher in the simulation than in a real life environment. This is because the bottom handle is the only part deforming. The stiffness of the bottom assembly is therefore increased with this method, this is however an acceptable trade off in this report and therefore the two step method will be used.

The model is set up so that the geometry can be duplicated and thus be used several times during the same simulation. This means that the same model can be used to create both the falling assembly as well as the impacted assemblies.

Part of the transition piece of the impacted assemblies is fixed in all degrees of freedom, see Figure 6-12.

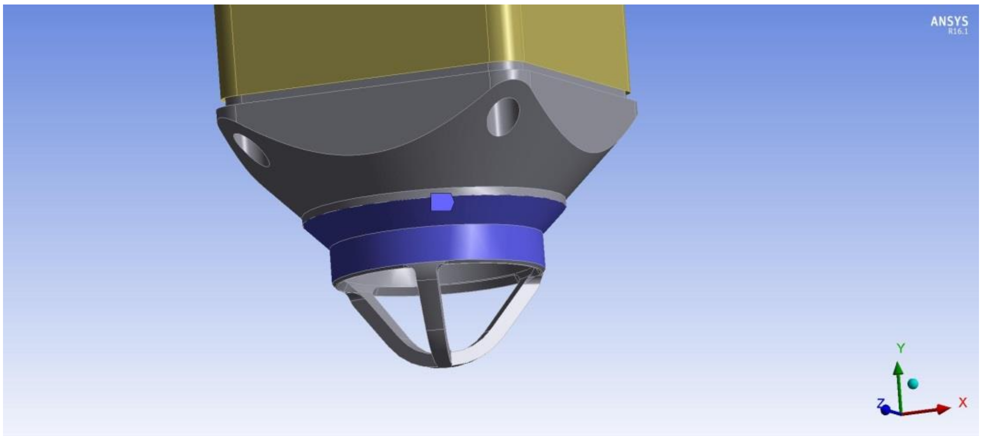



**Figure 6-12 Fixed support area**

Since the falling assembly is in free fall no boundary conditions will act on it.

Since the reactor surrounding the assemblies will not be modelled no boundary conditions for it applies.

## 6.5 Load Application

The water inside the reactor has a small albeit noticeable effect on the velocity. The water will act as a dampener and will therefore slow the dropped assembly down. This lowers the initial velocity given to the dropped assembly by a small amount.

The simulation will begin with the dropped assembly being only just above the other assemblies. This saves time and computational efforts, since a free fall is easily calculated by hand. Since the water will dampen the acceleration somewhat, the initial velocity just above impact will be calculated. Assuming that the assemblies are 4 meters high and that the dropped assembly’s bottom is at ten meters the falling distance will be $\approx$6m.

Using Archimedes principle which states that:

$$\rho g V = F \tag{5.1}$$

Where V=0.0136 $m^3$ the volume of the assembly, ρ=997 $kg/m^3$ for water, and g=9.81 $m/s^2$ the earth's gravitational acceleration. This gives us a buoyancy force of:

$$\rho g V = 133.602\ N \tag{5.2}$$

This force is directly countering the gravitational force which is:

$$\text{ma} = \text{F} = 2982.24\ \text{N} \tag{5.3}$$

where $m = 304\ kg$ is the approximated mass according to [10] and $a = 9.81\ m/s^2$.

The resulting acceleration is therefore calculated as:

$$2982.24 - 133.602 = 304 * a \tag{5.4}$$

Which leads to $a = 9.371$. Using the relation of constant accelerated motion which states that:

$$a = \frac{v^2 - v_0^2}{2s} \tag{5.5}$$

Where $v_0$is the initial velocity which is zero, and $s \approx 6\ m$ is the distance, this result in a velocity that will be the load applied:

$$v = 10.595 m/s \tag{5.6}$$

This velocity is not significantly different than if the surrounding medium would have been air, which would be:

$$v = \sqrt{2 * a * s} = 10.84\ m/s \tag{5.7}$$

## 6.6 Contact & Connections

The obstacle to be able to create a functioning and fast model is the contact conditions. If the original geometry was to be directly imported a lot of unnecessary contact conditions and geometry features would have followed along. This is never how a FE-analysis is done. The model would take too long to import into ANSYS and any simulation would also take too long. Therefore several of the contact conditions will be changed. And since a lot of the components will be removed or simplified there will be a lot fewer contact regions thereby reducing the computational time.

The fastener will be in contact with the handle using bonded contact at the on the surface where the screws are keeping them together.

At the bottom the transition piece will be in contact with the channel using frictionless contact. The bottom plate and the transition piece are meshed together as a single entity thus creating an equivalent bonded contact.

For the rest of geometry simple body interaction will be the main connection.

## 6.7 Simulation procedure and errors

In order to see potential errors and problems before the simulation have been going for a long time, the simulation is implemented in parts. This means that each part is first simulated and then the others added to it. In this simulation the transition piece was the first object tested. When this was error free the channel was added and then the handle and so on. The very last components added would be the spacers. The ANSYS LS-DYNA interface was unable to retrieve stresses from the beams. In order to get around this the choice was made to change the fuel rods and water rods into shell elements instead. This means that a lot of new elements was added which slowed down the simulation. However this means that the stresses and the strains will be able to be retrieved and therefore the failure criteria can be analysed and evaluated.

It was discovered that the lower support plate was a problem. The program would be unable to handle the elements of the plate and delete or suppress nodes and elements outright. Other times the elements would deform to such a degree so as to be a meter long which is not possible. The solution to this problem was that instead of using shell elements to represent the plate using solid elements

The interaction between the spacers and the channel proved to be bothersome since no contact between them was allowed or else the region where they were in contact created strains that were large enough to be impossible. This problem occurred because the top part of the channel was too coarse. The solution was a more refined mesh.

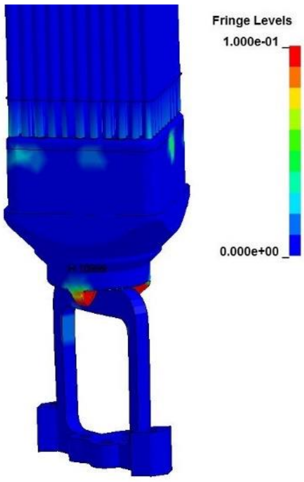
# 7 Results

## 7.1 LC1 Upper Assembly

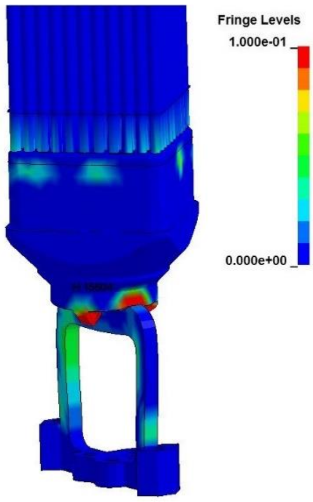
Using the setup given in section 6, and 48 processors, it took 171 hours to complete the calculation.

In the computation of the upper assembly the channel, fastener and handle does not have the velocity they should. For unknown reason they do not get the initial velocity and therefore does not interact with the rods, transition piece or the bottom plate. They are therefore neglected below.

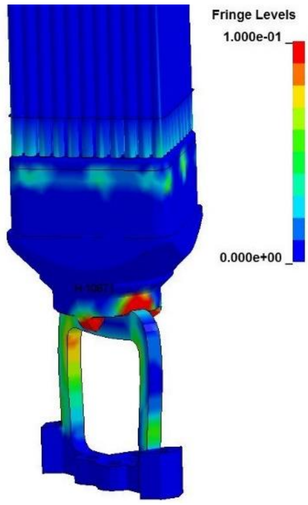
In the computation for the upper assembly the impact starts with a contact between the transition piece and the handle and the guide cross of the transition begins to deform which is continued until it is totally collapsed at about 13 ms into the accident. This deformation is expected and is favourable since some of the energy is absorbed. Until the handle reaches the “rim” of the transition piece the fuel and water rods are barely affected by the accident. Once the “rim” comes into contact with the handle, the rods starts to experience stresses and strains, as seen in Figure 7-1.



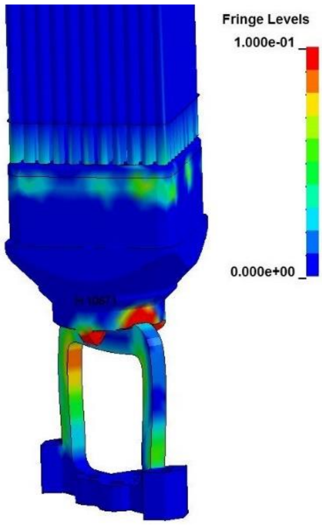

*13 milliseconds into the simulation*



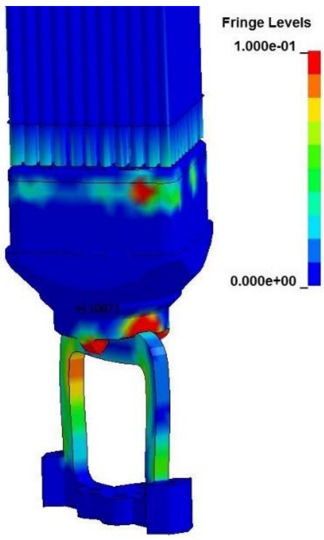

*17 milliseconds into the simulation*




*21 milliseconds into the simulation*




*25 milliseconds into the simulation*




*30 milliseconds into the simulation*

**Figure 7-1 Deformation of transition piece**

During the course of the impact the guiding cross of the transition piece experience large amount of strain. The maximum amount is over 180% shown in Figure 7-2 which is not feasible thus showing that the computational model is stronger than it should be. Had birth and death been used a better approximation would have been obtained.

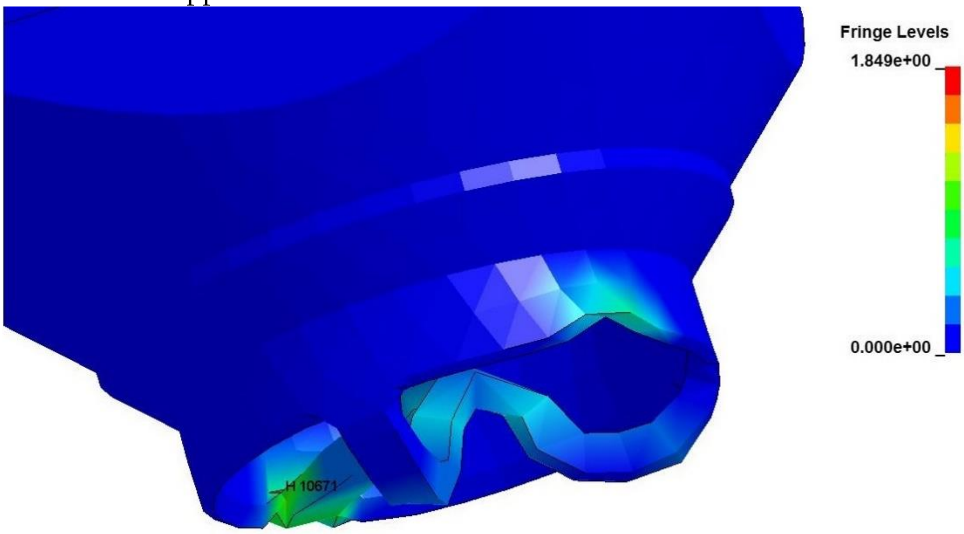



**Figure 7-2 Max deformation of the transition piece**

It is interesting to note that where the transition piece and bottom plate interact there are large strains. This occurs when the bottom plate deforms it starts pushing outwards and thus pushing on the transition piece as seen in Figure 7-3.

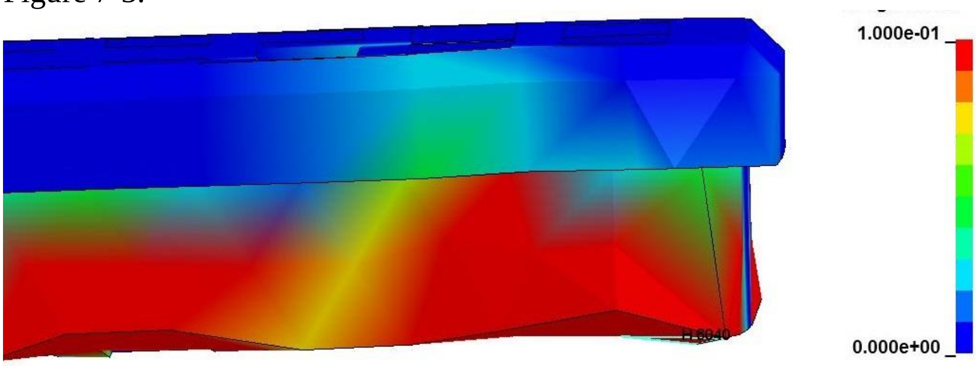



**Figure 7-3 Deformation of the bottom plate**

Using Figure 7-4 it can be noted that the accident lasts for about 22.5 milliseconds before the fuel bundle starts to bounce upwards again.

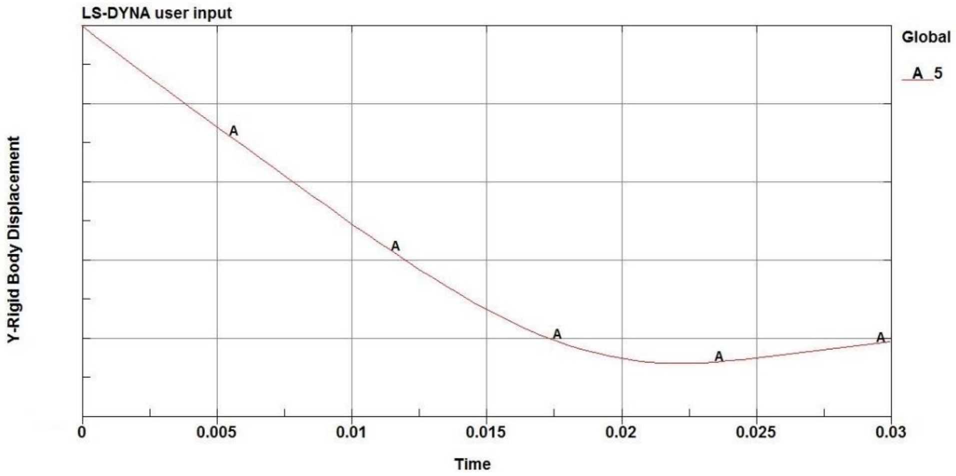



**Figure 7-4 Global displacement in Y-direction over time**

The highest strains that occur in the fuel rods are just over 5.5%, as seen in Figure 7-5 and Figure 7-6. The strains occur at the bottom of the falling assembly which is expected since it is the first part experiencing the impact.

Using Figure 7-5 it can be seen that in most of the rods the strain stays below 3.5 %. Only the rods at the furthest from the centre experience high strain while the rods closer to the middle stay between 0% and 2%.

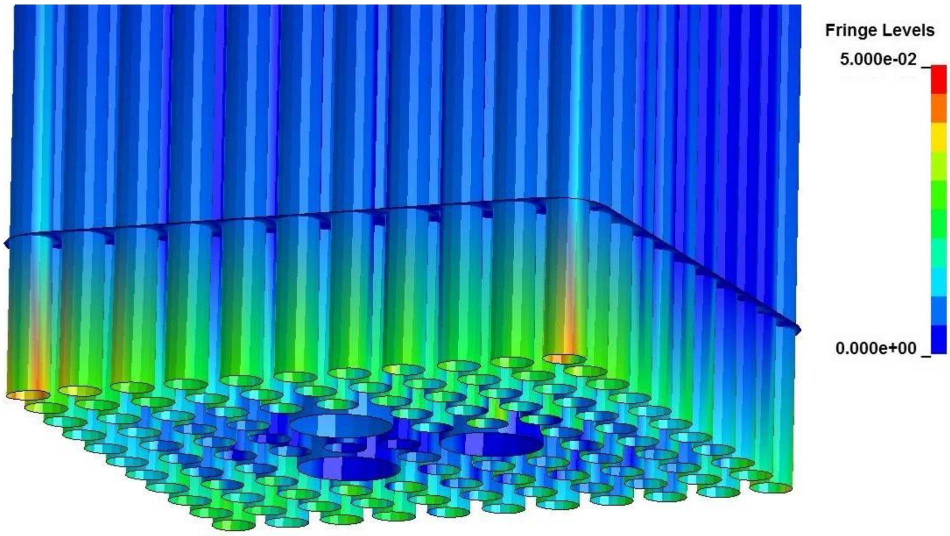



**Figure 7-5 Plastic strain in the bottom of the fuel and water rods**

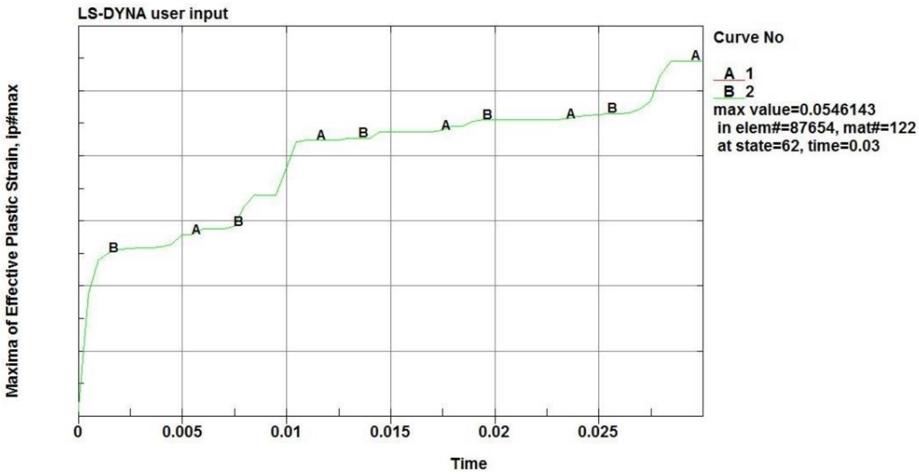



**Figure 7-6 Plastic strain over time in the fuel rod element with highest strain**

The plastic strains are large in the bottom part of upper assembly's fuel rods most probably because it is the part firstly affected by the impact and not because of the weight associated with them, see Figure 7-7. Had it been the weight a more even distribution of the strain would have occurred.

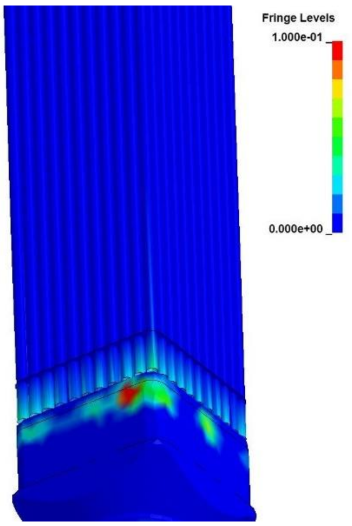



**Figure 7-7 Bigger part of the bottom shown to highlight the lack of strains distributed in the fuel rods**

The pressure acting on the bottom of the handle can be used as verification that the strains are consistent and not an anomaly. The pressure for two elements is presented in Figure 7-8. It can be seen that the increase in pressure at the handle corresponds roughly to an increase of strain in the rods, thus confirming that the result is consistent.

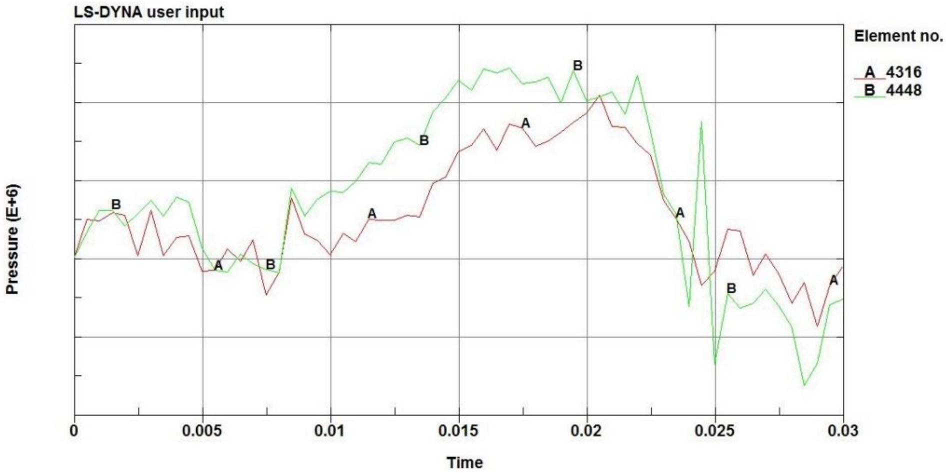
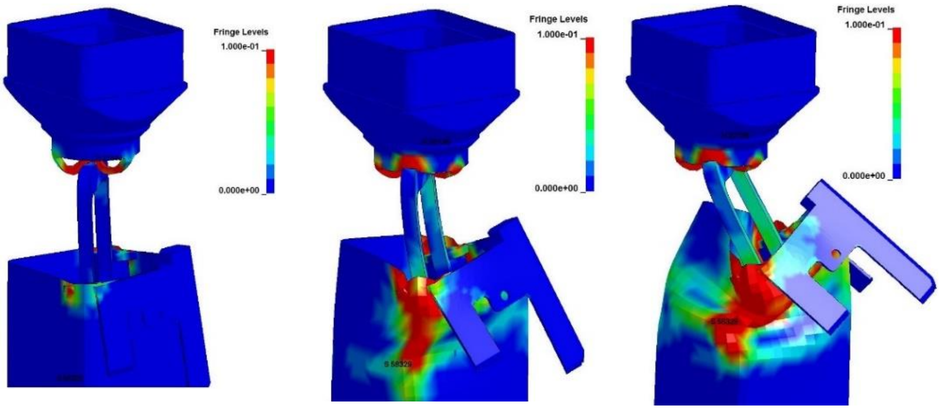



**Figure 7-8 Pressure-time plot of two elements at the boundary condition of the handle**

## 7.2 LC2 Lower Assembly

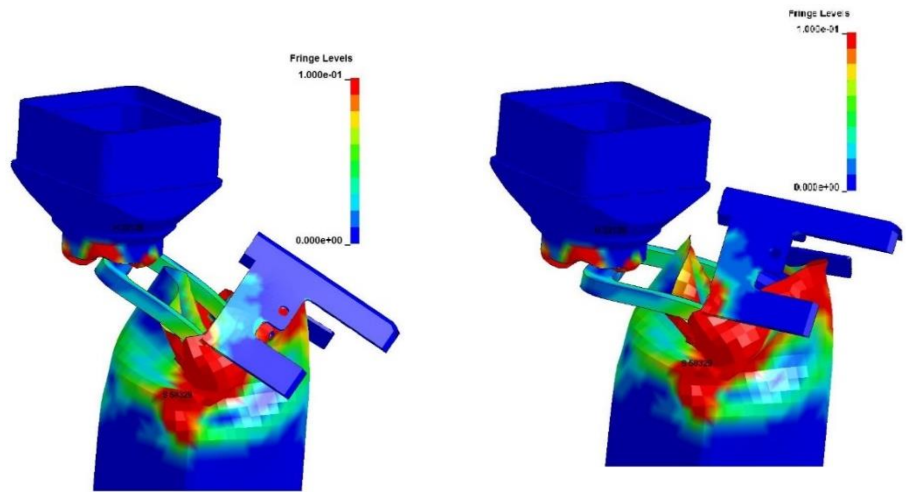
In the second simulation the large deformation of the lower assembly can be seen. What happens is that the fastener begins to rotate and thus deforms the channel with local buckling as can be seen in Figure 7-9. This deformation is severe enough that the handle makes contact with the water rods and creates high strains.




*4 milliseconds into the computation*

*8 milliseconds into the computation*

*12 milliseconds into the computation*




*16 milliseconds into the computation*

*20 milliseconds into the computation*

**Figure 7-9 Deformation of the top for the impacted assembly**

In the case of the lower assembly the maximum amount of strain experienced in the rods is over 100% as seen in Figure 7-10. The deformation of these elements cannot look like that without a numerical error. Figure 7-11 is a much better representation of what happens over the impact. It deforms in a way that is the result of the impact and is therefore a better approximation of the strains that occur. It can also be seen that there are some rods that has yet to be affected by the accident.

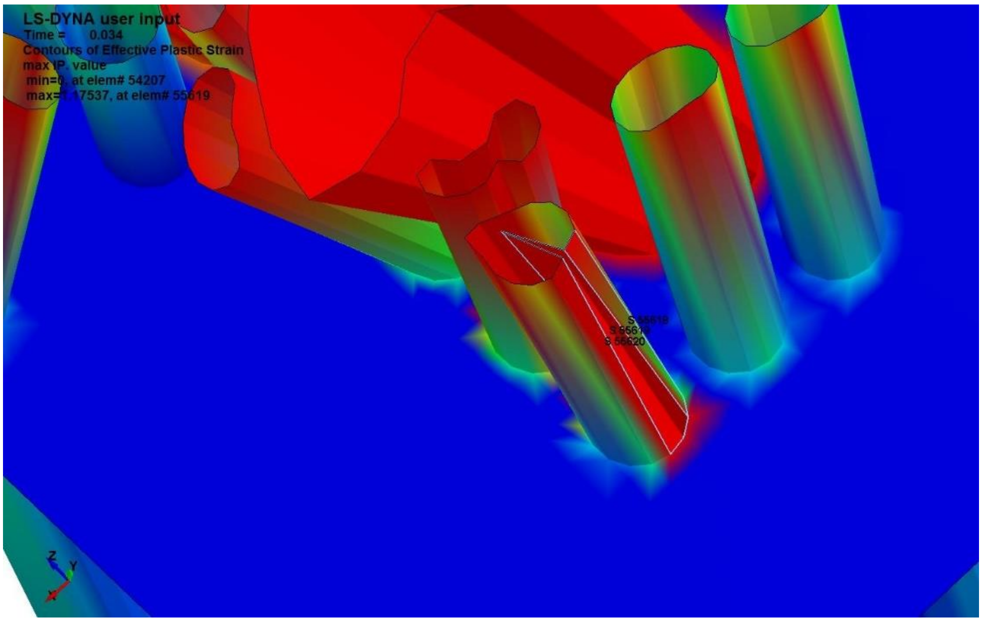



**Figure 7-10 Elements with max strain**

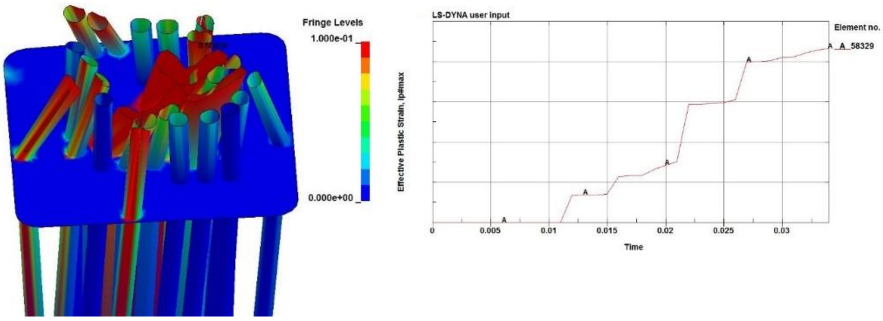



**Figure 7-11 Plastic strain over time for the marked element with a maximum of 5.5 %**

In the analysis of the lower assembly the pressure-time plot cannot be used in the same way but is still interesting to see in that the first increase in pressure corresponds to the increase in strain as well, see Figure 7-12.

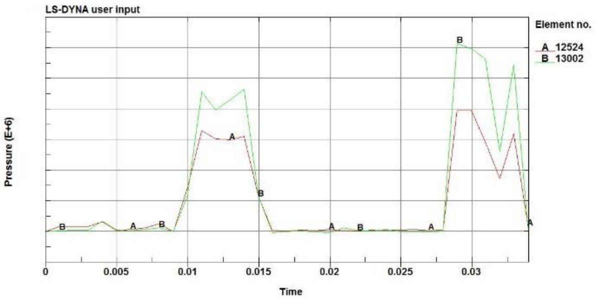
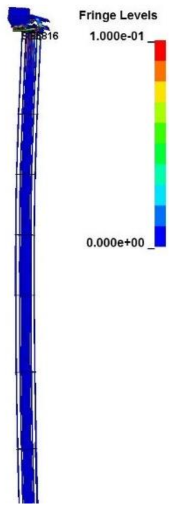
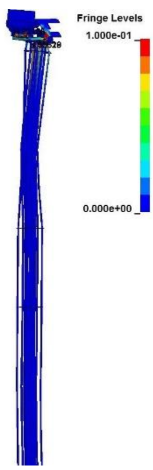



**Figure 7-12 Pressure-time plot for two elements on the bottom plate of the lower assembly**

The spacers have a positive effect on the fuel rods in that they prevent a lot of the buckling that would otherwise occur. With fewer spacers reinforcing the assembly more buckling would occur, as seen in Figure 7-13.




*Computation with all spacers*




*Computation with some spacers removed and zoomed in to show the localized buckling*

**Figure 7-13 The effect of the spacers at 23 milliseconds. The left computation uses all spacers and is only present in this figure**

# 8 Conclusion

## 8.1 Improvements

With this 3D FE-model there are notable improvements over the current methodology. It is capable of showing events over time and producing visual aids so as to closer examine the results. It is also capable of showing several different results such as strains, deformation and stress. It can also show what happens to the assembly as a whole or to its individual parts. This means it is superior in showing the results it acquires. This however comes at the drawback of being a complex analysis that takes a long time to complete.

## 8.2 Event Description

The results clearly show that it is the guard cross that takes the brunt of the damage initially. Since it is has less material it is weaker and cannot withstand the strains and deformation put to it.

It also shows that in the lower assembly a large deformation occur at the top. It also indicates that the channel and rods might be subjected to bending. This bending might occur because of the speed of the impact.

## 8.3 Fuel Rod Damage

In the current methodology the upper assembly is assumed to almost stop after impact and then fall sideways. However this analysis shows tendencies for it to glance of the lower assembly and continue to fall. This glance would change its course and it might therefore impact the assemblies next to the original supercell or might even impact another assembly in the same supercell.

This report also shows tendencies for the lower assembly to be affected more by the impact itself than previously thought. In the current methodology it is assumed that because of the bending that the upper assembly is subjected to all of the rods there fail and that half of the kinetic energy is transferred to the lower assembly. But in this report the lower assembly is experiencing a higher degree of strain than the upper one.

The spacers are an important part in preventing the buckling and thus decreasing the damage on the rods. This effect is exaggerated in this report due to it being modelled as an entirely solid plate.

# 9 Discussion

## 9.1 Failure Condition

If the failure condition that Westinghouse uses would have been translated to 1% strain it can be seen that a majority of the rods will fail. However this failure condition cannot be directly translated to such a degree. It has been discussed during the duration of this project that the failure condition for an analysis like this might not be until the strain reaches levels of up to 3% to take into account the fact that this analysis shows localised strain and not only global strain.

## 9.2 Accuracy

Several things can be done to increase the accuracy of the results. Refining the mesh at several places might give a more accurate result but a change like that is only marginal and will slow down the simulation considerably.

Birth and death handles the destruction of elements during a computation. If used this effect can simulate the destruction of material. Since birth and death is not used in these calculations the model is a lot stronger than its real life counterpart. This is one area of the project that should be improved if studied further.

Time should be spent on creating LC1 as a single computation to avoid the problem with the large increase of stiffness for the upper assembly.

In the computation of the upper assembly the channel, fastener and handle are not given their proper velocity. While it being an error is not believed to have a significant effect on the outcome of the first calculation. When proceeding with combining the two assemblies into one computation it will have a larger effect.

The transition piece that falls into the lower assembly only has its weight modified and the author was unable to manipulate the inertia moments in the different directions to change the falling behaviour. Over the course of several seconds this might impact the results but during the short duration that these calculations are made seems to not affect it in a large way.

## 9.3 Suitability of Ansys Ls-dyna Ext

During the calculation process and the pre-processing many problems arose due to the fact that the integration of LS-DYNA into the workbench code is done in less than a perfect way. First problems arose with the fact that beam results are not saved in a way that Workbench (WB) can post process which meant abandoning beam elements in favour of shell elements. This puts an unnecessary computational load on the computer. Then, for reasons yet to be determined there were problems with retrieving results from the RSM.

As of ANSYS WB 16.1, using the LS-DYNA extension creates barriers and obstacles that do not warrant the use of the code until several of these problems have been solved and it slows down the engineer when wanting to simulate results.

In order to get the calculations to function one has to make changes inside the k-file that cannot be made in the ANSYS environment creating another step that has to be made. Since the change in the k-file has been made, consequently ANSYS is unable to display the results and another program such as lsprepost or hypermesh has to be used.

LS-DYNA is ultimately capable of running a simulation such as this but it requires dedicated time and is prone to, with the version used during this project, fail give errors that are not properly explained thus leading to more work.

# 10 References


[1] Nuclear Power Reactors in the World, Reference Data Series No.2, 2015 edition, Vienna, Austria

[2] B.D. Wirth, M.J. Caturala, T. Diaz de la Rubia, T. Khraishi, H.Zbib (2001). Mechanical property degradation in irradiated materials: A multiscale modeling approach. *Nuclear Instruments and Methods in Physics Research B, 180*, 23-31. http://dx.doi.org/10.1016/S0168-583X(01)00392-5

[3] Arthur P.Boresi, Omar M. Sidebottom. (1985). Advanced Mechanics of Materials, (4.ed), Chicago, Illinois: JOHN WILEY & SONS

[4] LS-DYNA (2011) Retrieved February 29, 2016 from: http://www.lstc.com/products/ls-dyna

[5] P. Petkevich, V. Abramov, V. Yuremenko, V. Piminov, V. Makarov, A. Afanasiev (2013). Simulation of the nuclear fuel assembly drop test with LS-Dyna. *Nuclear Engineering and Design, Volume 269, 136-141.* https://doi.org/10.1016/j.nucengdes.2013.08.019

[6] Waseem, G. Murtaza, N. Elahi (2014). Structural integrity assessment and stress measurement of CHASNUPP-1 fuel assembly. *Nuclear Engineering and Design*, *Volume 28, 130-136.* https://doi.org/10.1016/j.nucengdes.2014.10.010

[7] J.-M. Gatt.J. Sercombe,I. Aubrun, J.-C. Ménard (2014). Experimental and numerical study of fracture mechanisms in $UO_2$ nuclear fuel. *Engineering Failure Analysis*, *Volume 47,299-311.* https://doi.org/10.1016/j.engfailanal.2014.07.019

[8] K. Yueh, J. Karlsson, J. Stjärnsäter, D. Schrire, G. Ledergerber, C. Munoz-Reja, L. Hallstadius (2015). *Fuel cladding behavior under rapid loading conditions, Journal of Nuclear Materials, Volume 469, 177-186.* https://doi.org/10.1016/j.jnucmat.2015.11.032

[9] T. Belytschko, W. Kam Liu, B. Moran, K.I. Elkhodary (2014). Nonlinear Finite Elements for Continua and Structures (2.ed): John Wiley & Sons. Ltd

[10] W. Ramberg, W.R. Osgood (1943). Technical note 902 *Description of stress-strain curves by three paramaters,* Washington. https://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/19930081614.pdf

[11] John O. Hallquist (2006) LS-DYNA Theory manual, California ISBN: 0-9778540-0-0 http://www.dynasupport.com/manuals/additional/ls-dyna-theory-manual-2005-beta

[12] Internal documents