Simulation of a Drop onto a Punch of a Transport Container for Fuel Assemblies

Gilles MARCHAUD & Dr Jürgen WERLE

COGEMA LOGISTICS (AREVA Group), Saint-Quentin-en-Yvelines, FR

1. ABSTRACT

International regulations require that a container for the transport of nuclear material has to sustain a 1-meter drop onto a cylindrical punch made of mild steel. It has to be shown that, at all times, the confinement of the radioactive contents is guaranteed. Also, any deformation of the internal arrangement has to be avoided or at least controlled in order to ensure the sub-criticality of the fuel assemblies at all times. LS-DYNA3D models of representative cross sections of two baskets have been developed. The finite element analysis has been validated by successfully calibrating it with an actual drop test. Thus, the most pessimistic configuration of the basket of the package was determined and used for all subsequent safety case studies.

2. INTRODUCTION

The 1-meter drop of a nuclear transport container for fresh fuel assemblies onto a mild steel punch is studied, see figure 1. It has to be shown that, at all times, the confinement of the radioactive contents is guaranteed. The structure needs to rest globally intact and the leakage rate has to stay below an extremely low legal limit. At present, this is demonstrated by the actual drop test. Moreover, any deformation of the basket has to be avoided or at least controlled in order to ensure the sub-criticality of the fuel assemblies at all times.

LS-DYNA3D models of representative cross sections of the baskets have been developed and successfully calibrated with an actual drop test, thereby avoiding an expensive test campaign for the other type of basket. Only the basket, which has been used for the calibration with the drop test, is presented here. The package has been dropped with an angle of 16.5° with respect to the horizontal.

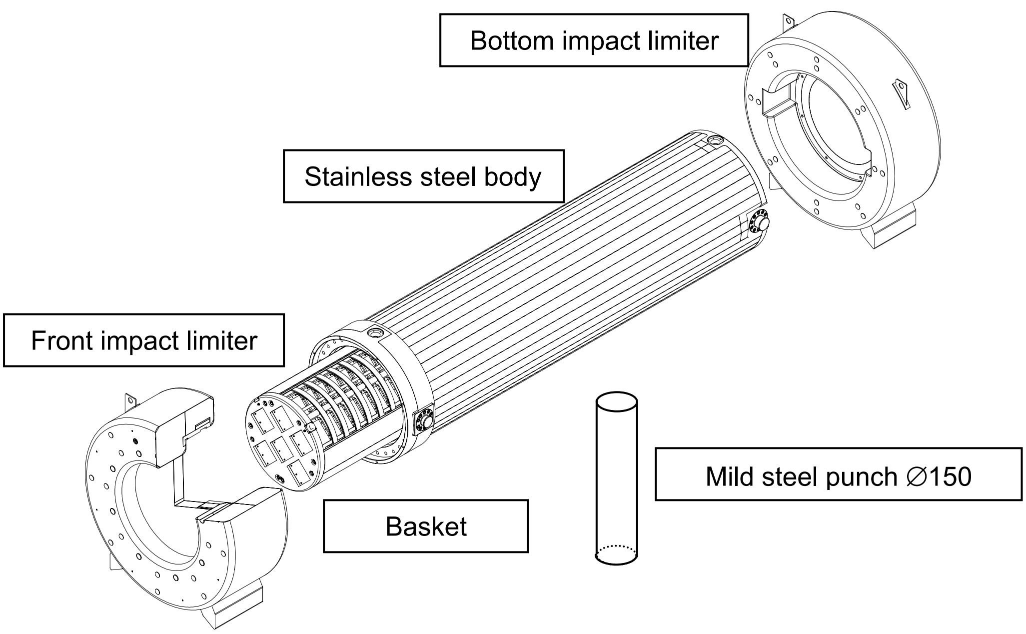


Fig. 1. View of the transport container

3. ASSUMPTIONS

- The half-scale model used during the drop is perfectly representative of the package and the finite element model at scale 1. Conversions are made with the appropriate scale factors.
- The main body consists globally of two cylindrical steel plates connected by stiffeners. A shielding resin fills the void between the two cylinders and aluminium profiles protect the outside. This main body, which is heavily deformed during the impact, is not modeled because it is very difficult to take into account all large strain and fracture and failure mechanisms correctly.
- The main role of the body is to transmit the compressive forces to the basket. Thus, a uniform pressure applied to an angular section of the external surface of the basket structure represents the effect of the impact onto the punch. The pressure and its precise application area are determined by calibrating the deformation obtained from the analysis with the drop test results.

4. MODEL DESCRIPTION

4.1 Geometry

The container (figure 1) consists of a cylindrical steel body, filled with resin for shielding purposes. During transport, two shock absorbers made of wood in a steel shell are attached to the top and the bottom of the main body. There are two pairs of trunnions to allow fixing and handling of the packaging. There are two types of baskets, designed to receive various types of fresh fuel assemblies.

4.2 Meshing

The mesh, shown in figure 2, is created for a representative portion of the basket structure and single lodgment, only. It consists of standard brick elements (8 nodes, 3 dof per node, 1 integration point).

4.3 Materials

The table below summarizes the materials of the main components:

Component	Material	Type LS-DYNA : MAT_…	N° type
Slice of basket	Aluminium	PIECEWISE_LINEAR_PLASTICITY	24
lodgment	Stainless steel	PIECEWISE_LINEAR_PLASTICITY	24

A constitutive law with one hardening slope is used for both materials.

4.4 Boundary Conditions

The boundary conditions are illustrated on figure 3. A uniform pressure is applied to an angular section of the external surface of the basket structure, the size and amplitude of which have been calibrated to give the best result. The finite element model has been clamped at the far end from the load application area. Since only a half-model of the slice is modeled, symmetry boundary conditions are applied.

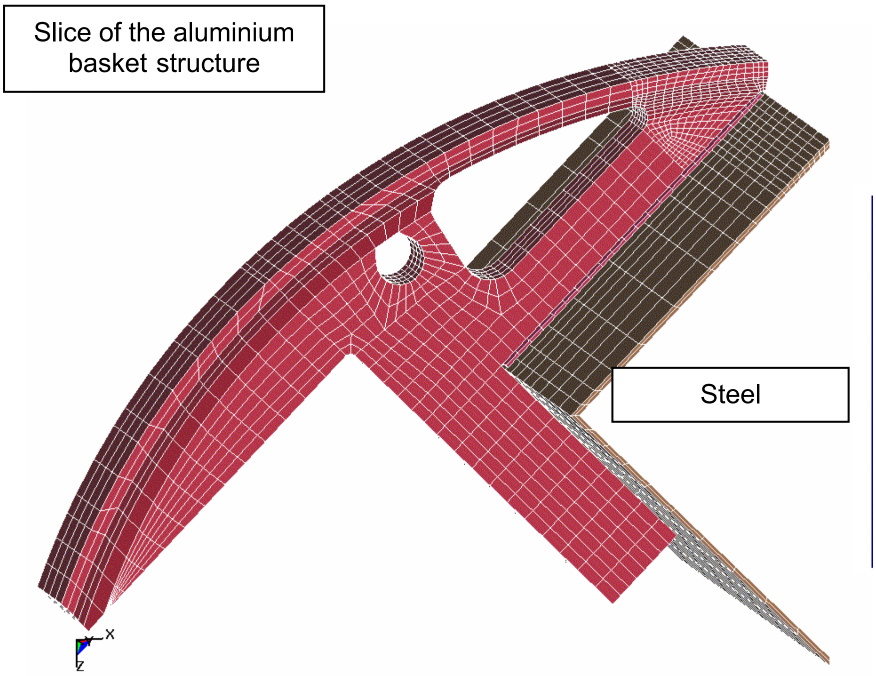


Fig. 2. Finite Element Mesh

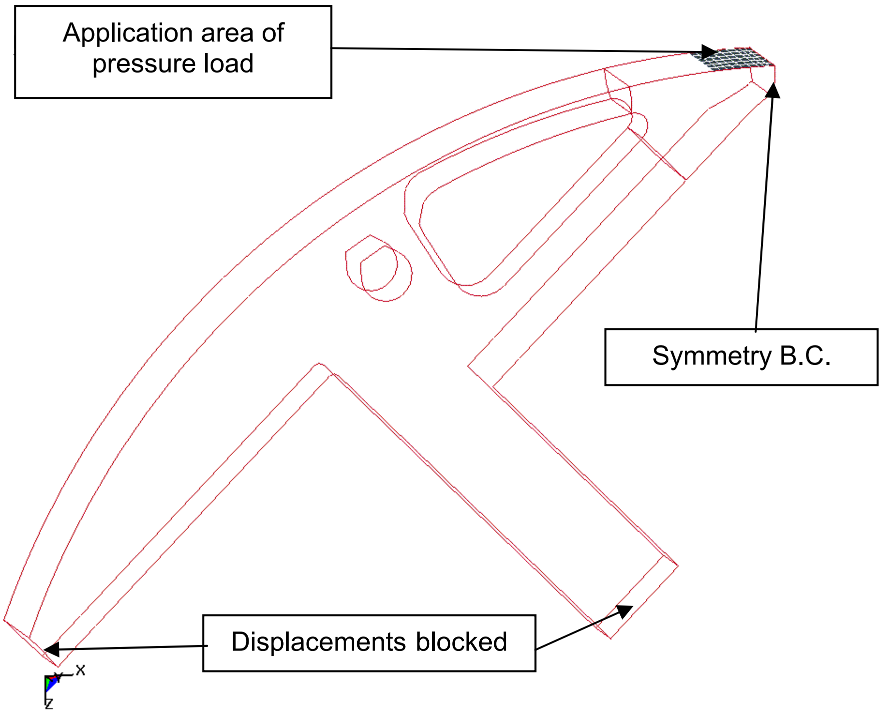


Fig. 3. Pressure load

5 RESULTS

The residual deformation of the finite element analysis coincides almost perfectly with the one observed after the test, see figure 4.

5.1 Calibration

The model has been calibrated by adjusting the applied pressure and its extent.

The success of the calibration is mainly due to:

- The mesh refinement in the heavily deformed region.
- The energy balance given by LS-DYNA3D is correct (very low interface and hourglass energies). This is a direct consequence of a good mesh quality and of limiting the number of contact interfaces.
- The absence of ductile fracture in the basket.

6 CONCLUSIONS

As the LS-DYNA3D model calibrates very well with the punch drop test, it is validated for use in further calculations. The safety case demonstrations for different internal baskets have been made using this model, based on a single drop test, thereby minimising considerably the costs of the drop test campaign.

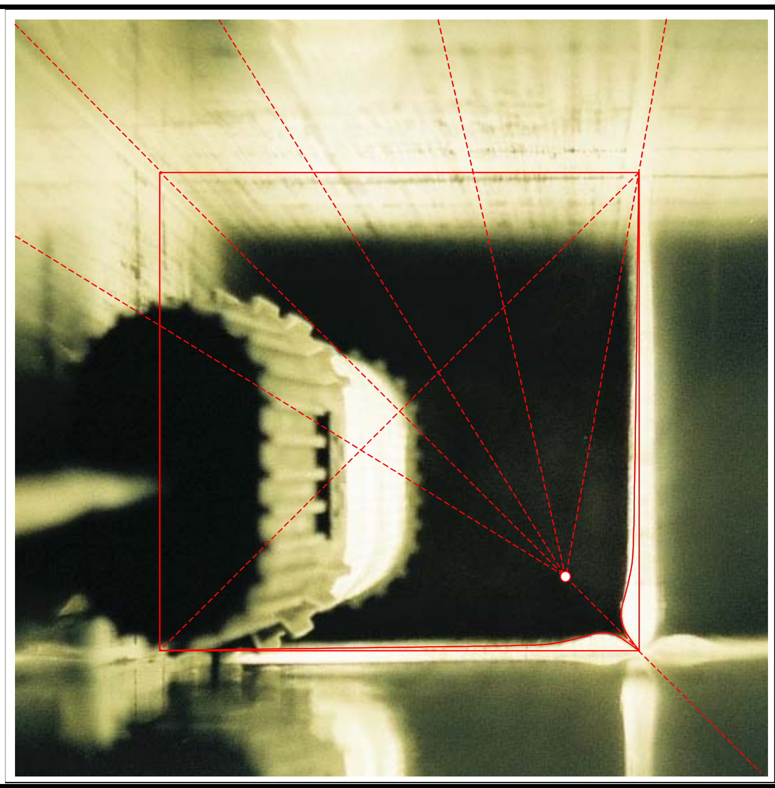
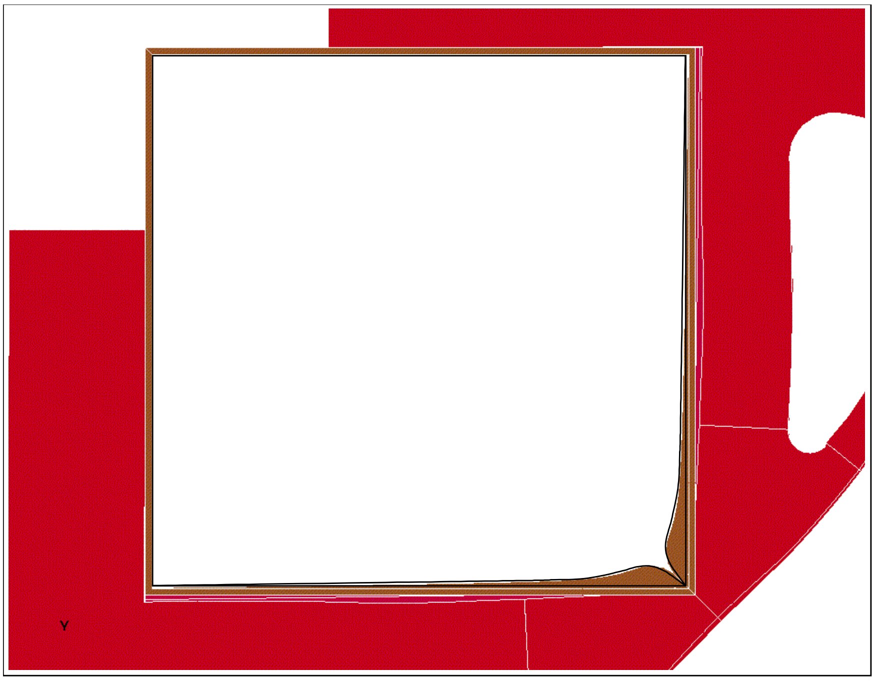
Test	
Calculation	

Fig. 4. Residual Deformations