Proceedings of the 19th International Symposium on the
Packaging and Transportation of Radioactive Materials
PATRAM 2019
August 4-9, 2019, New Orleans, LA, USA

IMPACT ANALYSES OF A NEW SAFKEG

Chi-Fung Tso
Arup

Mark Johnson
Croft Associates Ltd

David McLennan
Arup

Adam McConaghy
Croft Associates Ltd

ABSTRACT

A new SAFKEG has been developed by Croft Associates Ltd for the shipment of materials between UK licensed sites. It has been designed to satisfy all the requirements of a B(M)F package as specified in the IAEA Transport Regulations.

Impact performance of the package has been demonstrated by a combination of physical drop testing and finite element analyses. The finite element analyses have been used to demonstrate the performance of the package in normal and accident conditions of transport impacts and under temperature and pressure conditions which were not addressed in the physical drop tests.

Two series of drop tests were carried out. In each series of tests, a sequence of tests in the centre of gravity over lid edge orientation followed by a sequence of tests in the axis horizontal orientation were carried out.

A state-of-the-art detailed finite element model was developed. It was validated against the drop tests and then used in the detailed analyses and evaluation of eight separate impact cases.

This paper presents a summary of the finite element analysis - modelling, validation, analysis and evaluation.

INTRODUCTION

A new SAFKEG has been developed by Croft Associates Ltd (hereafter Croft) for the shipment of materials between UK licensed sites. It has been designed to satisfy all the requirements of a B(M)F package as specified in the IAEA Transport Regulations [1].

Arup was appointed by Croft to demonstrate the impact performance of the package, in the impact scenarios and temperature conditions that were not covered by drop tests, with the aid of finite element simulation.

DESCRIPTION OF THE SAFKEG

The SAFKEG consists of a Keg Assembly, which carries within it a Containment Vessel (CV), enveloped within an Aluminium Spacer around the side and the bottom, and cork packing around the top. The layout and the components of the SAFKEG are illustrated in Figure 1.

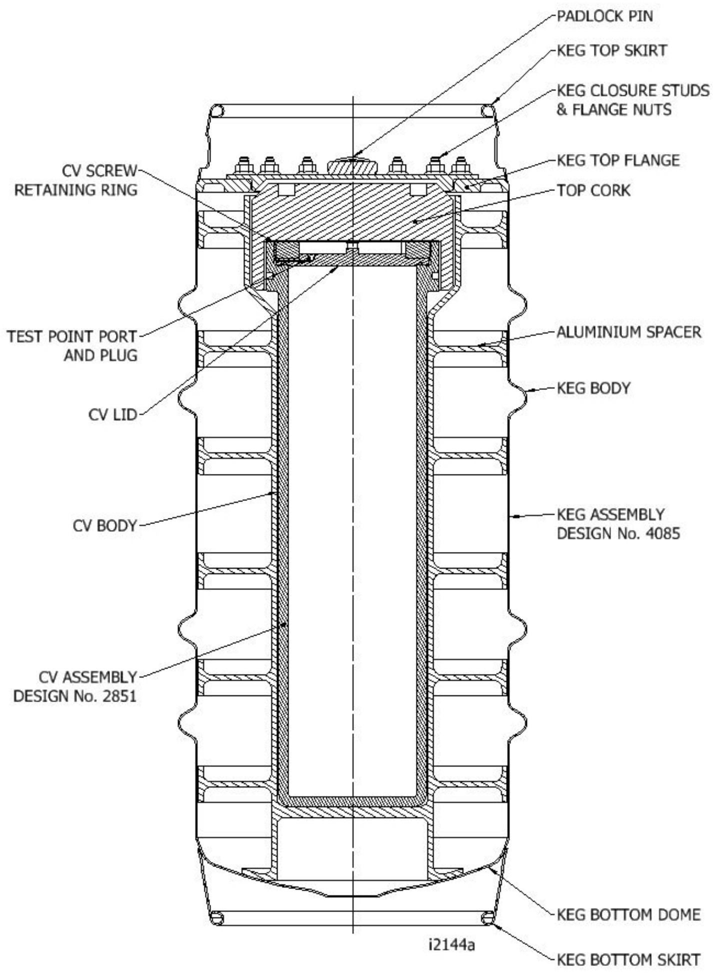

Figure 1. Components and layout of the SAFKEG

The Keg Assembly consists of a cylindrical body, with a bottom dome welded to the bottom end and a flange welded to the top end. Closure is achieved by a lid which is attached to the flange via thirteen studs and hex nuts. A padlock arrangement pin at the lid-flange closure is used to prevent unauthorised lid removal. To allow the SAFKEG to stand vertically, a bottom skirt is welded to the bottom end of the Keg Assembly. And for handling purposes, a top skirt is welded to the top of the Keg Assembly. The Keg Assembly is manufactured entirely from grade 1.4307 stainless steel, except for the studs which are made from Nitronic 60 and the nuts which are made from A2-70 grade stainless steel.

The CV consists of a body and a lid which are made from 1.4307 stainless steel, and a screw retaining ring which is made from Nitronic 60. The screw retaining ring is engaged with the body of the CV via a pair of screw threads, with one set machined onto the perimeter and over the height of the screw retaining ring and the other set machined onto the inner perimeter of the body upstand. The CV lid has central feature in the lid incorporating a bow shackle for lifting the assembled CV. The CV is fitted with two O-ring seals at the lid-body interface. The outer seal resides in a groove on the body side, and the inner seal resides in a groove on the lid side. The containment boundary consists of the body, the lid and the inner O-ring seal.

The aluminium spacer is machined from a single bar of grade 6082 T6 aluminium. It has been designed to provide structural rigidity to the whole package, to minimise heat ingress during the accident conditions of transport (ACT) thermal test and to provide minimum impedence from heat release during the normal conditions of transport (NCT).

STRATEGY TO DEMONSTRATE IMPACT PERFORMANCE

The impact performance of the package has been demonstrated by a combination of drop tests and finite element analysis.

Two series of drop tests have been carried out
- Preparatory Tests, carried out at the Croft Didcot facility on the 23 and 24 May 2017 and
- Regulatory Tests, carried out at the VINCI drop test facility in Leighton Buzzard on the 13 and 14 June 2017.

In both the Preparatory Tests and the Regulatory Tests, one test specimen was used for two sequence of tests
- Sequence 1 - centre of gravity over lid edge orientation
  - 1.2m drop
  - 1m penetration
  - 9m drop
  - 1m drop onto a punch
- Sequence 2 - axis horizontal orientation
  - 1.2m drop
  - 1m penetration
  - 9m drop
  - 1m drop onto a punch

All the tests were carried out at room temperature.

In order to demonstrate the impact performance of the package in impact scenarios, and temperature and pressure conditions that have not been addressed by the drop tests, a set of finite element analyses were carried out. The analysis matrix is as follows:

Table 1. Analysis Matrix

Load Case	A	B	C	D	E	F	G	H
Condition of transport	Normal Conditions of Transport (NCT)			Accident Conditions of Transport (ACT)				
Drop height	1.2m			9m				
Target	Flat unyielding							
Orientation of axis with respect to target	CG over lid edge	Axis parallel to target	Axis vertical top down	CG over lid edge (as Load Case A)	Axis parallel to target (as Load Case B)	Axis vertical top down (as Load Case C)	Lid first slap down at multiple angles	Base first slap down at multiple angles
Hoop orientation	Handles & CV screw retaining ring notch on symmetry plane		Not applicable	Handles & CV screw retaining ring notch on symmetry plane		Not applicable	Handles & CV screw retaining ring notch on symmetry plane	
Content	35kg empty dummy model representing a 4088CV.							
Package temperature	Uniform -30°C throughout; Minimum properties throughout.							
Package condition	Pristine			After Load Case A	After Load Case B	After Load Case C	After Load Case B	After Load Case B
Pressure	40 bar gauge inside CV							

MODELLING

The model consists of the entire package, in that no advantage has been taken of symmetry in the package in any of the impact scenarios. It consists of 787,708 elements.

All the analyses were carried out as explicit transient analyses using the finite element software LS-Dyna [2]. The model is shown in the following figures.

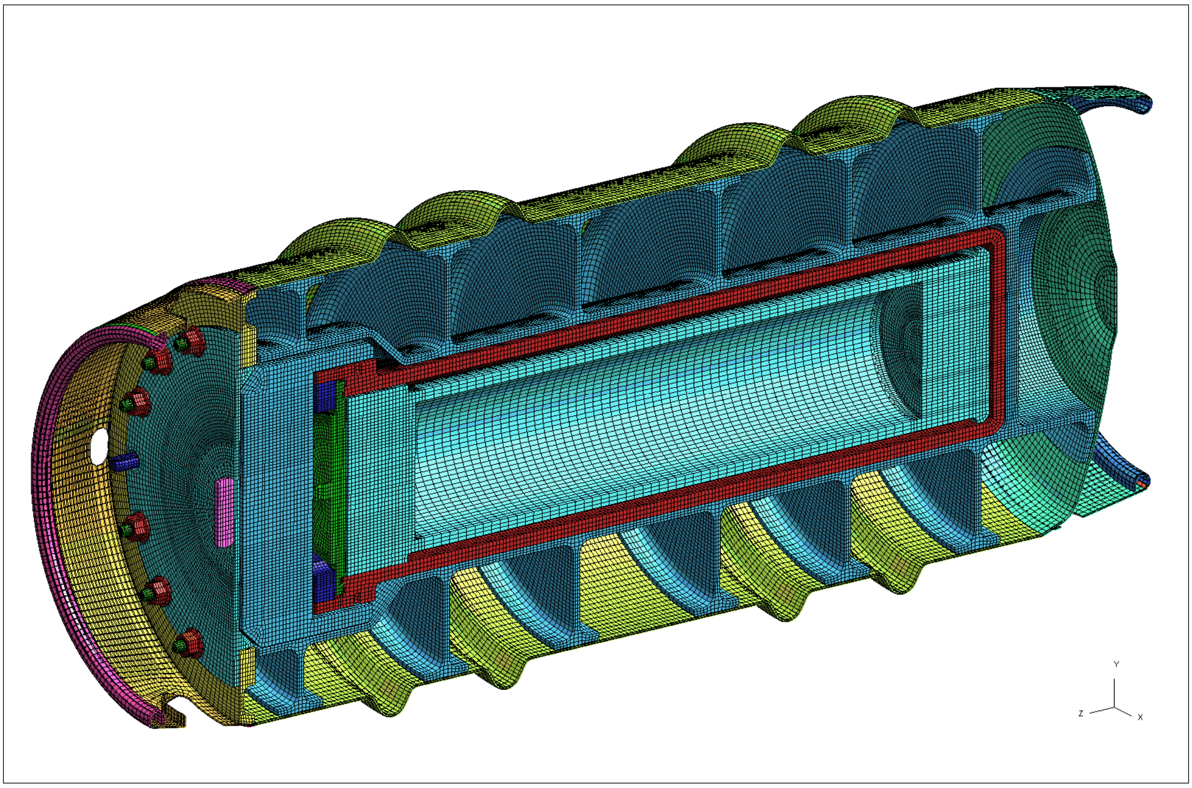

Figure 2. Section through the model

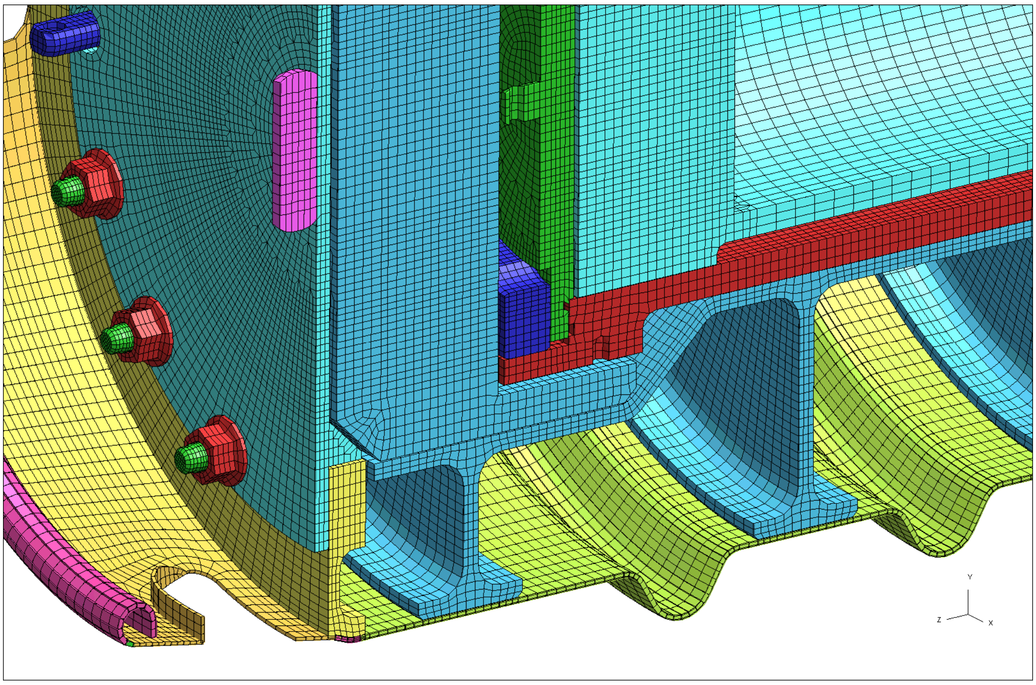

Figure 3. Section through the model – close-up on the top of the package

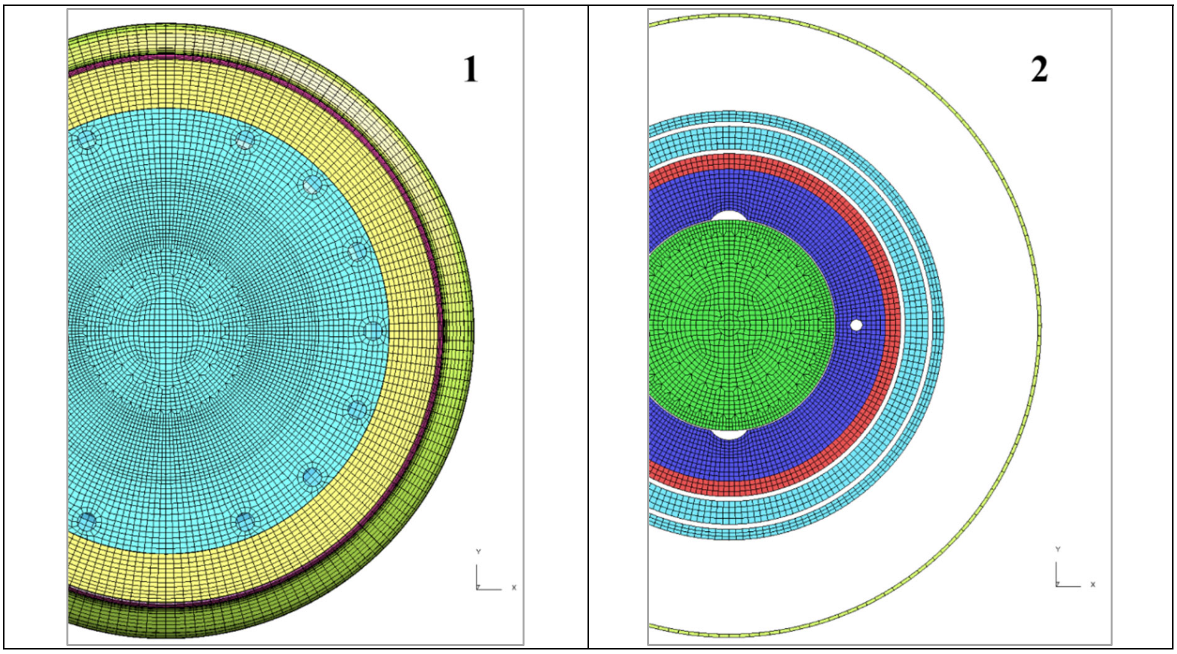

Figure 4. Constant-Z sections through the model just above the keg lid (left) (with the studs not shown) and through the CV screw retaining ring (right)

Modeling principles

The model has been designed based on the following principles:

- Identical mesh for each repeating unit of the geometry.
- Uniform mesh around the hoop direction of the cylindrical part of the geometry for all the components.
- Identical mesh for similar components that undergo large deformations.
- Identical mesh for all stud and nuts.
- Matching mesh between all adjacent cylindrical surfaces, e.g. between keg body and aluminium flange, between aluminium inner and CV surface, between stud shank and lid hole.
- Matching mesh between adjacent straight mesh and adjacent flat surfaces.
- More refined mesh for areas which undergo larger deformations with a refinement that adequately captures the deformations.
- Aspect ratio of elements as close to 1 as possible and not larger than 2.5, except where absolutely unavoidable.
- Element quality adhering to established good practice of LS-Dyna.

Modelling has been based on the good practice as defined in TCSC 1087 [3].

Choice of type of element

Metallic components of the package, besides the studs and the nuts, consists predominantly of:

- thin plates with a thickness in the order of 2mm (e.g. keg body), and
- machined parts with thickness from up to about 30mm (CV body upstand), down to 12mm (e.g. CV body), to 6.5mm (e.g. aluminium spacer), to 6mm (e.g. keg lid) and to 5mm (e.g. cut-out in the keg top flange).

Typically there are two option for modelling these components - solid elements or thin shell elements. Thick shell elements in LS-Dyna are difficult to use and the results may not be robust, and hence were not an option.

The advantage of modelling using solid elements is that the interaction between the components can be visualised easily and the connection between the components can be modelled as they have the actual dimensions. The disadvantage, is that a number of elements are required in the through thickness direction. Increasing the number of elements in the through thickness direction reduces element size, and reducing element size reduces analysis timestep and increases analysis run time. Additionally, in order to maintain a reasonable element aspect ratio, increasing the number of elements in the through thickness direction also requires increasing the number of elements in the orthogonal directions in the component itself and in the adjacent components. This is not an issue with thin shell elements.

The main problem with using thin shell elements, is that it would be difficult to model any geometry other than flat plate geometry accurately. For example, if thin shell elements are used to model the aluminium spacers, the radiused transition from flange to web of the aluminium spacers would be difficult to represent. Additionally, interfaces between components cannot be represented as precisely as when solid elements are used. Combining thin shell elements and solid elements in the modelling of a component could also be challenging regarding connections.

Another issue that needs to be taken into consideration is the modelling of the studs and nuts. The only adequate option to model them, and in accordance with current good practice, is to model them with solid elements and with the threaded stud-nut interface and threaded stud-flange interface modelled as meshed-in.

Taking all these into consideration, it was decided that

- keg body, keg bottom dome, keg top skirt and keg bottom skirt should be modelled with thin shell elements as it would be impractical to model them with solids
- The rest of the package should be modelled with solid elements.

All the solid elements were eight-noded brick elements with fully-integrated selectively-reduced formulation. All the thin shell elements were four-noded quad elements, apart from four three-noded elements around two of the holes in the top skirt. All the thin shell elements were fully-integrated elements with five integration points through their thickness.

Choice of number of elements through thickness

Many of the components that were modelled with solid elements have thin sections, in some locations, down to 5mm. A key decision in the modelling of these components is the number of elements through the thickness of the thin areas.

In order to inform the decision, extensive mesh sensitivity studies were carried out. Different mesh refinements of components of the package were analysed under loading and extent of deformations similar to those to be encountered in the impact scenarios. Based on the results of these studies, it was decided that three elements through the thickness was adequate. It was found that for the specific structures and the extent of loading and deformations, increasing the number of elements to four would not bring a marked benefit, and increasing further to five or more is not practical considering the resulting reduction in element size and increase in the number of elements.

Choice of mesh in hoop-wise and axial directions

One of the principles of the mesh design is that the mesh between adjacent cylindrical surfaces should match. The reason for this is to minimise the possibility of spurious stresses that might occur when the edge of elements come into contact with the middle of adjacent elements. Bearing in mind that there is no space in the geometry for transitioning from larger numbers of elements to few elements progressing inwards in the model, the model needs to have the same number of elements hoop-wise from the outside of the keg body, via the keg flange and the aluminium spacer into the CV body. Taking into account the size of the elements in the through thickness directions, as discussed above, and the requirement to achieve acceptable element aspect ratios, it has been decided that the number of elements around the circumference should be 192. This number of elements, evenly spaced around the circumference, will be employed in the keg body, keg flange, keg lid and spigot, aluminium spacer, CV screw retaining ring, CV lid, CV body and dummy CV.

MODEL VALIDATION

Validation of the model against drop tests is not strictly necessary, since:

- well established element types and material types have been employed in the model
- it has been designed following established good practice and drawing on the experience gained in similar packages subjected to similar impacts
- sensitivity studies have been carried out to justify mesh refinement in potentially sensitive areas
- the model employs similar mesh refinement as and in many areas a higher refinement than the impact analyses of a similar earlier design of SAFKEG.

However, in order to confirm the adequacy of the model, it has been benchmarked against four of the drop tests in the Preparatory Test series of drop tests – a 1.2m drop followed by a 9m drop in a centre of gravity over lid edge orientation; a 1.2m drop followed by a 9m drop in an axis horizontal orientation.

The 1m penetration tests were omitted from the benchmarking as the damage was too slight and the damage would not affect the behaviour in the subsequent 9m drop.

The 1m drop onto a punch tests were omitted from the benchmarking for two reasons:

- as the performance of the package in the 1m drop onto a punch scenario have already been demonstrated adequately in the drop tests and it is therefore not included in the analysis matrix for the finite element analysis matrix, benchmarking of the model against this scenario is superfluous, and
- The 1m drop onto a punch drop tests were carried out after the 9m drop, hence not analysing them would not affect the results of the analyses of the 9m drops.

The benchmarking analyses were carried out on the basis that the model and initial/boundary conditions should, as far as possible, be like-for-like with the test package and the drop tests.

Material properties of all the steel components (except for the A2-70 material) and the aluminium component were based on mechanical properties as stated in their material certificates. Realistic stress strain curves for steels were derived using the method in ASME Boiler and Pressure Vessel Code Section VIII Division 2 Annex 3-D [4]. Realistic stress strain curve for aluminium was derived using the Ramberg Osgood method from Section E.2.2 of Eurocode 9 [5]. For the cork, stress-strain behaviour was derived from material tests carried out by Croft.

For each of the four analysis, results from the analysis and the drop test have been compared in terms of deformed geometry and static measurements of damage. Deformation from the analyses and the tests are shown side by side in the following figures.

Figure 5. 1.2m lid edge drop – deformation in the test (left) vs deformation in the analysis (right)

Figure 6. 9m lid edge drop – deformation in the test (left) vs deformation in the analysis (right)

Figure 7. 1.2m side drop – deformation in the test (left) vs deformation in the analysis (right)

Figure 8. 9m side drop – deformation in the test (left) vs deformation in the analysis (right)

Very good correlation has been achieved in all four analysis, which included two different impact orientations and two sets of sequential analysis, with all the features of the deformed test package replicated in the analysis in each case. The benchmarking confirms that the mesh, in terms of its design, geometry, refinement and choice of element type, is adequate to replicate the deformation behaviour of the test package, and the model is adequate to be used for the analyses to demonstrate the impact performance of the package.

ANALYSES AND EVALUATION

Following the benchmarking, the analyses defined in the analysis matrix were carried out. From each analysis, the behaviour of the package was studied and the performance of the package assessed.

The integrity of thin shell elements were evaluated against failure plastic strains. The integrity of the steel and aluminium components that have been modelled with solid elements has been assessed using a material damage model, where material damage is measured as a cumulative parameter within each solid element based on the plastic strain increment divided by the current limiting triaxial strain, which is based on the current stress triaxiality factor. The stress triaxiality factor is defined as the mean stress divided by the Von Mises stress. The potential that material failure has occurred is indicated when this damage parameter, A, exceeds a value of 1. This approach is based on the methodology given in ASME Boiler and Pressure Vessel Code Section VIII Division 2 [4].

Opening at the inner seal between the CV lid and CV body has also been obtained by obtaining the time history of vertical displacement between the CV lid and CV body at the seal location.

The evaluation shows that:

- material failure is not expected in any of the components modelled with thin shells in any of the impact scenarios, except for local areas around the handle in the top skirt in the 9m CG over lid edge drop.
- material failure is not expected in any of the components of the Keg Assembly that were modelled with solid elements or any of the CV components in any of the impact scenarios.
- material failure is not expected in the aluminium spacer in any of the impact scenarios except for very local areas in the lid-first slap down drop and the base-first slap down drop, but such local material failures do not have a significant effect on the overall behaviour of the package.
- Transient opening at the containment seal of the CV is negligible in all impact scenarios.

CONCLUSIONS

A set of explicit transient finite element analysis have been carried out to demonstrate the impact performance of a new SAFKEG package in the impact scenarios and at the temperature condition at which its performance has not already been demonstrated by drop tests.

The model was carefully designed, based on good practice as defined in TCSC 1087 and with the aid of mesh sensitivity studies. Its adequacy was demonstrated by benchmarking against four drop tests, including two sequential drops. It was then used in a comprehensive matrix of analysis to demonstrate impact performance of the package in scenarios and temperature conditions in which its performance has not been demonstrated by drop tests.

The analyses and evaluation show that the package satisfies the impact performance requirements of the IAEA Transport Regulations for a Type B(M)F package.

REFERENCES

[1] IAEA, Regulations for the Safe Transport of Radioactive Material - IAEA Safety Standards Series No SSR-6, 2012.
[2] LSTC, LS-DYNA Keyword User's Manual, Revision 5471, May 2014
[3] Transport Container Standardisation Committee, Good Practice Guide - The Application of Finite Element Analysis to Demonstrate Impact Performance of Transport Package Designs, TCSC 1087, 2017
[4] American Society of Mechanical Engineers, ASME Boiler and Pressure Vessel Code Section VIII Rules for Construction of Pressure Vessels Division 2 Alternative Rules, 2015
[5] British Standard Institution, Eurocode 9 - Design of aluminium structures – Part 1-1: General Structural Rules, BS EN 1999-1-1:2007, 2007