**Proceedings of the 17th International Symposium on the Packaging and Transportation of Radioactive Materials**
**PATRAM 2013**
**August 18-23, 2013, San Francisco, CA, USA**

# ASSESSMENT OF DUCTILE CAST IRON FRACTURE MECHANICS ANALYSIS WITHIN LICENSING OF GERMAN TRANSPORT PACKAGES

**S. Komann** **Y. Kiyak** **F. Wille** **U. Zerbst** **M. Weber** **D. Klingbeil**

*BAM Federal Institute for Materials Research and Testing, Berlin, Germany*

## ABSTRACT

In the design approval of transport packages for radioactive materials, the mechanical and thermal safety assessment is carried out in Germany by competent authority BAM. In recent years BAM was involved in several licensing procedures of new spent fuel and HLW package designs, where the cask body is of Ductile Cast Iron (DCI). According to IAEA regulations package designs have to fulfill requirements for specific conditions of transport. Type B(U) packages must withstand the defined accident conditions of transport. The temperature range from -40°C up to the operational temperature has to be considered. For the cask material DCI, it is necessary to determine safety against brittle fracture. The German guideline BAM-GGR 007 defines requirements for fracture mechanics of packagings made of DCI. Due to complex cask body structure and the dynamic loading a fracture mechanical assessment by analytical approaches is not always possible. Experience of recent design approval procedures show that the application of numerical calculations are applicable to determine the stresses and stress intensity factors in the cask body. At the first step a numerical analysis has to be done to identify the loading state at the whole cask body. Secondly an analysis of a detail of the cask body is made considering the displacement boundary conditions of the global model. An artificial flaw is considered in this detailed model to calculate the fracture mechanical loading state. The finite element mesh was strongly refined in the area of the flaw. The size of the artificial flaw is based on the ultrasonic inspection acceptance criteria applied for cask body manufacture.. The applicant (GNS) developed additional analysis tools for calculation of stress intensity factor and/or J-Integral. The assessment approach by BAM led to the decision to develop own tools to the possibility for independent proof of the results.
The paper describes the authority assessment approach for DCI fracture mechanics analysis. The validation procedure incl. the development of own tools is explained. BAM developed a post-processor to determine the fracture mechanical loads. A horizontal 1 m puncture bar drop test is used to give a detailed description of the assessment procedure.

## INTRODUCTION

The Federal Institute for Materials Research and Testing (BAM) is one of the two German competent authorities in charge of design assessment in conformity to the IAEA Regulations for the Safe Transport of Radioactive Material [1]. The BAM responsibility comprises safety assessment for mechanical and thermal safety cases, containment of the radioactive material and the quality assurance program during design manufacturing, operation and maintenance. The work of BAM is performed in close cooperation with the German Competent Authority, the Federal Office for Radiation Protection (BfS). BfS is responsible for shielding and safety assessment. Based on BAM's safety evaluation report, BfS issues the package design approval certificate.

For each design of a transport package for radioactive material, compliance with national and international transport regulations has to be demonstrated. According to IAEA [1] the safety case can base on one of the following approaches or a combination of these:

a) Performance of tests with specimens, with prototypes or samples of the packaging.

b) Reference to previous satisfactory demonstrations of a sufficiently similar nature.

c) Performance of tests with models of appropriate scale.

d) Calculation, or reasoned argument, when the calculation procedures and parameters are generally agreed to be reliable or conservative.

The compliance of safety objectives of the package design is shown with a combination of the above mentioned methods usually. The combination of experimental drop tests with subsequent numerical analyses corresponds to the state-of-the-art safety analysis.

## MECHANICAL DESIGN REQUIREMENTS

According to IAEA transport regulations [1] BAM requires the applicant must present conclusive concepts for the safety analyses to keeping safety objectives of the package under routine, normal and accident transport conditions. This includes the selection of relevant drop test positions and the appropriate drop test sequences (Figure 1). The specimen shall drop onto the target so as to suffer maximum damage in respect of the safety objectives to be tested. The mechanical test for Type B(U)-packages consists of drop tests, e.g. 9m drop test or crush test for light weight packages and 1m pin drop.

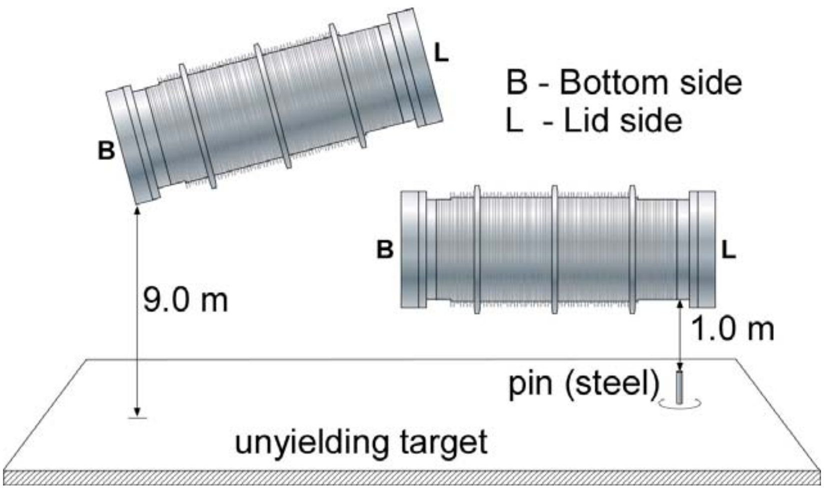



**Figure 1. Example of drop test sequence for new German HLW-cask design**

With regard to accident transport conditions for Type B(U) package approvals, a temperature range from -40°C to operating temperature has to be taken into account [1]. An evaluation of the (local) loading has to be performed with respect to the fracture behavior which e.g. comprises possible plastic deformations at operating temperature as well as local stresses at the lowest temperature of -40°C. If the packages are made of ductile cast iron the correct application of the guideline BAM-GGR 007 "Guideline for Assessment of Ductile Iron Cask Components" [2] is required. The assessment is then based on the resistance of the component against crack initiation for which the crack driving force in the component is compared with the fracture resistance of the material. Crack tip parameters are the K factor or the J-integral. The results of the global analyses for specifying the loading state, in general, have to be verified by experimental drop test results. This shall be illustrated for a 1m pin drop.

## VERFICATION OF A FINITE ELEMENT MODEL FOR A 1 M PIN DROP POSITION

The 1m pin drop onto a steel bar is one of the mechanical test to be considered for accident transport condition of Type B(U) packages according to IAEA regulations [1] (Figure 2). The impact of the steel pin and cask components (e.g. the cask body) could lead to plastic deformations combined with possible high strain rates in the local contact zone. An adequate description of such problems by static analytical approaches is not possible. Therefore the analyses have to be performed by dynamical finite element analyses (including mass terms) with mesh refinement in the highly loaded pin and package component areas.

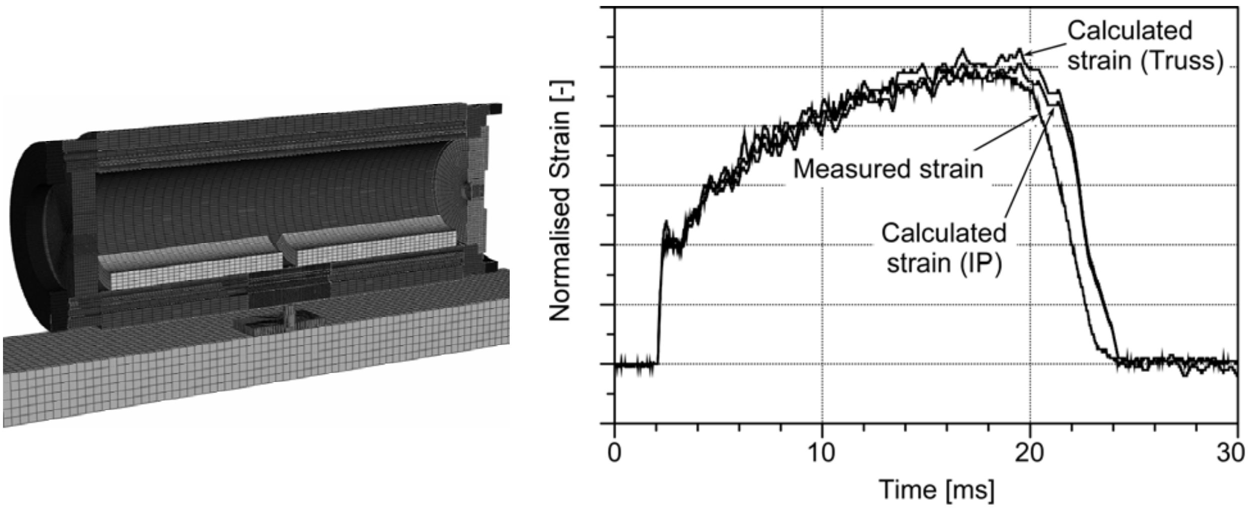
**Figure 2. 1m pin drop with a half-scale drop test model**

As mentioned above, the cask body is made of ductile cast iron. The puncture bar has to be made of mild steel according to the IAEA requirements [1]. The behavior of both materials has been extensively investigated by static and dynamic tensile and compression tests carried out by the applicant. Suitable material models for metal plasticity exist in several finite element codes (e.g. ABAQUS/Explicit, LS-DYNA).

During the approval procedure for the new package design of a German HLW cask (GNS-CASTOR ® HAW28M) BAM performed  a 1m drop of a half-scale model of the cask onto a steel pin (Figure 2). The package was equipped with strain gauges and accelerometers at points of interest. In addition, finite element analyses were carried out by the applicant and the BAM. The BAM modelling provides an independent assessment and to examine the safety analysis presented by the applicant. In the following, the basic steps of the modelling and verification of the finite element model are summarized.

The estimation of the pin deformation is a first indication of the quality of the finite element pin model. In the chosen example, a simplified FE model was generated modeling the cask as a rigid body. The material behavior of the pin has to be modeled as realistic as possible and has to be verified in a first step. The elastic behavior of the pin is described by the Young’s modulus and Poisson's ratio, the plastic behavior by strain rate-dependent stress-strain curves. The density of the rigid body segment is modified in order to match the true mass of the drop test model. In addition to the material law of the steel pin, the friction coefficient between cask and pin had to be investigated in detail.

In a next step, a more detailed finite element model with a refined material description of the cask body was developed. In addition to the control of the global response as reaction force and deformations of the steel pin, the measured and analytically determined accelerations and strains of the cask body had to be verified. In case of the 1m pin drop the experimental maximum strain values determined by strain gauges in the area next to the impact had to be considered. Furthermore the accelerations and strains determined all over the drop test model were important for the assessment of the impact process. Figure 3 shows the curves of measured and calculated strains at the inner surface of the cask, opposite of the impact area. In order to get the strains at the surface of the cask body, truss elements had been joined to the nodes of the element surface segments. The strain-time curves resulting from both integration points (of the solid and truss elements) show a good correlation with the measured curve.




IP-Integration Point

**Figure 3. FE-model and comparison of measured and calculated strains**

If the half-scale model, by comparison with the experimental data, is proven to be suitable, it can be adopted to the full-scale package by modifying all node coordinates in the FE-input file by a scaling factor. In a first step, the initial and boundary conditions remain unchanged. The material laws and densities are identical to the scaled model according to similarity laws which means that strains and stresses should be constant compared to the results of scaled model, but the time is doubled and the forces should be four times greater if the scaling factor is 2.

Finally, the full-scale finite element model has to be specified for the safety analysis of the package design to be approved. For this purpose the unyielding target is modeled as a rigid body. The material laws are adapted to the material specifications. The analyses are carried out using material properties at -40°C and maximum operational temperature. The model can also be used to investigate the package performance under other safety relevant conditions, which cannot be covered by experimental drop tests e.g. other positions of the cask body impact onto the steel pin.

The analyses results can also be applied to the assessment of stresses in sections of the package where measurements during the experimental investigations were impossible, and also to fracture mechanics investigations assuming artificial flaws in the cask body.

## FRACTURE MECHANICS ASSESSMENT

### General Principles and Analysis Steps

The fracture mechanics analyses are performed according to BAM-GGR 007 [2] by the applicant and are reported in the safety analysis report. They have to be assessed by BAM as the competent authority. In the following it shall be explained how this is usually done. The aim of the fracture mechanics analysis, as it presently stands, is the proof that potential cracks assumed at the positions of the highest stresses would not initiate and grow under accident conditions of transport. The analysis steps are summarised in Table 1 and will be explained in more detail in the sections below. Note that the discussion, instead of giving a complete overview, will concentrate on aspects which require attention when cross-checked in the evaluation process.

**Table 1. Analysis steps of fracture mechanics assessment of transport containers (DCI)**

| Step | Analysis step description | Remarks |
|---|---|---|
| 1 | Global finite element analysis of the container | Dynamic analyses adequate for impact loading (container falls down); analyses of the structure without crack (previous section) |
| 2 | Specification of critical positions | Positions where the maximum principal stress ≥ 50% of the yield stress at assessment temperature according to BAM-GGR007 [2] |
| 3 | Specification of the dimensions of the potential flaws | Based on non-destructive inspections (NDI) require-ments (for ultrasonic testing obtained from “equivalent reflectors”); Re-characterisation of embedded flaws as surface cracks (semi-elliptical surface crack according to IAEA [5]) |
| 4 | Reduction of the number of potential crack positions within different categories | The positions for which the highest crack driving force is expected are considered as “covering”. Positions for which the crack driving force potentially is lower do not have to be taken into account. |
| 5 | Specification of finite element sub-models containing cracks | To be large enough such that the influence of the crack on the stress-strain field is subsided at sub-model surface; bonding to the global model based on node displacement. |
| 6 | Determination of dynamic crack driving force | Usually finite element calculation for determining the J-integral as a line integral; formal transfer to dynamic stress intensity factor |
| 7 | Safety assessment | Comparison between crack driving force and fracture resistance; analysis based on crack initiation |

**Analysis step 1: Global finite element analysis of the container**

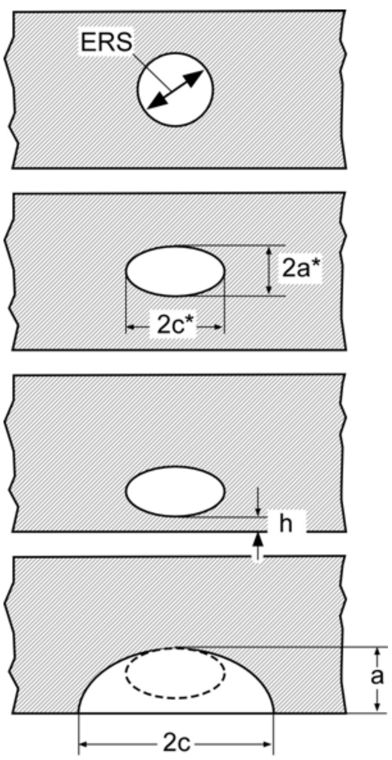
In a first step the stress distribution in the container (e.g. cask body) is determined by a global finite element model without flaws.

**Analysis step 2: Specification of critical positions**

The specification of potential crack positions follows the German guideline BAM-GGR 007 [2]. Positions at which the maximum principal stress exceeds half the yield strength are considered as relevant. For Type B(U) packages, the yield strength has to be considered for a lower bound temperature of -40°C [1]. This is a conservative assumption which does not take into account the possible warming of the container by the nuclear material inside. That the yield strength according to the material specification is used rather than the yield strength of the individual container material is conservative as well, as the material specification gives a lower bound to the real values and it also covers possible scatter in material properties.

**Analysis step 3: Specification of the dimensions of the potential flaws**

The dimensions of the most dangerous potential cracks are based on the requirements of non-destructive inspection (NDI) such that a crack size which is not guaranteed to be reliably detected is assumed as existent. In Germany, inadmissible defect sizes are usually obtained by the so-called disc reflector method (DGS) [3] which specifies an equivalent reflector size (ERS) as a circular disc with its plane assumed to be perpendicular to the ultrasonic beam axle. Although the NDI requirements usually refer to embedded defects the fracture mechanics analysis is based on surface flaws as a conservative limit state.




Inadmissible defect size given as equivalent reflector size (ERS) by the circular disc reflector method (DGS) [3]

Step 1: Defect assumed as embedded crack with its plane normal to the largest principal stress

Step 2: Re-characterisation of the circular crack as a semi-elliptical one of identical area as the ERS; crack geometry: c* = 3a* (following [6])

Step 3: Determination of a*

$$a^* = \frac{k}{\sqrt{12}}\left(ERS\right) \quad k = 2 \quad \text{(empirical factor)}$$

Step 4: Shifting the crack towards the surface until the criterion for re-characterising it as surface crack ($h \leq 0.4 \cdot a^*$ [4]) is fulfilled

Step 5: Determination of the dimensions of the semi-elliptical surface crack for h = 0.4 a*

$$a = 2a^* + h \quad \text{and} \quad 2c = 6 \cdot a \quad [5]$$

**Figure 4. Re-characterising of the ultrasonic equivalent reflector size to semi-elliptical surface cracks for fracture mechanics analyses**

Figure 4 illustrates how this is obtained from the ERS. The basic idea is that the wall break-through of an embedded crack has to be assumed when its distance to the surface, in the figure designated as h, is smaller than a critical distance. The transformation is necessary for conservatism reasons when no information on the location of the defect in wall thickness direction is available.

It is usually assumed that a larger crack size, in terms of fracture mechanics, is more dangerous than a smaller one. In general this statement is correct. Note, however, that there might be cases where the simple rule might be misleading.

**Analysis step 4: Reduction of the number of potential crack positions**

The identification of potential crack positions for a given loading scenario (Analysis step 2) can result in a large number of positions. Because these will not be equally critical it makes sense to reduce this number and to concentrate on the positions with the highest loading which are then considered as "covering" the others in a conservative way.

How to identify these sites? The criterion is not just the highest stress at surface but the highest crack driving force which depends

- on the stress profile in wall thickness as well as in longitudinal direction across the area of the potential crack and
- on the local stiffness of the component in that region or, in fracture mechanics terms, the local constraint.

In Figure 5 three schematic stress profiles are compared for a two-dimensional section. The profile of Position 3 "covers" the two other profiles in that it shows the highest stresses over the whole distance between the surface and the potential crack tip. Because it is that region which is relevant for crack tip loading, Position 1 would be more dangerous than Position 2 although the stress profiles of both intersect further right. Note that all stress profiles refer to the section without crack in the global model.

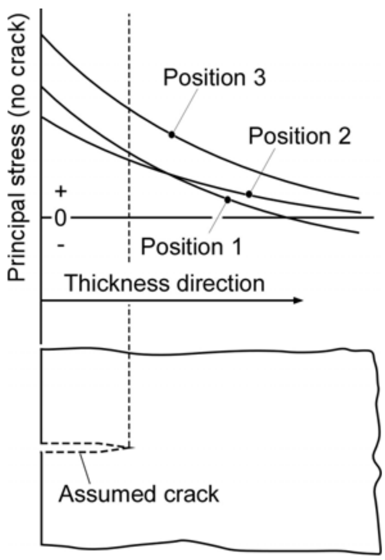



**Figure 5. Schematic stress profiles in thickness direction for various potentially critical sites according to Analysis step 2. The comparison is carried out within one site category only.**

Besides the stress distribution in thickness direction also those in the longitudinal direction parallel to the potential crack length at surface has to be taken into account because it will also have an effect on the crack driving force at the different positions on the crack front. Note, however, that there are many cases where the stress field in longitudinal direction will show almost no gradient.

As mentioned above, it is important to recognise that the criterion of "covering" stress profiles requires similar local stiffness conditions which result in similar constraint when a crack is modelled later on. For identical stress profiles a lower stiffness will cause a higher crack driving force and vice versa. Note further that the stress profiles so far have been obtained by the global model. Due to its coarse mesh this might not correctly describe the local stress in cases of significant stress gradients.

As a consequence it is necessary to assign the various high stress positions obtained by the global model to distinct site categories based on the local component geometry, crack orientation etc. Only within a site category a reduction of the number of critical positions as covered by those of the highest stress profile is permissible.

**Analysis step 5: Specification of finite element sub-models containing cracks**

The global analysis (Step 1) is carried out for structures without cracks. The crack driving force is then determined by sub-models with a much finer mesh including crack. The information transferred from the global to the sub-model is the displacement of the nodes at the outer nodes of the sub-model where it is inserted into the global model. It is important, in that context, that the sub-model has to be large enough such that the local influence of the crack on the stress-strain field is subsided at its boundary. In order to guarantee this, sub-models of different size can be used. The requirement is fulfilled by sub-models which result in identical J-integral or K values.

As a rule, both the global and local finite element analyses should be carried out for elastic-plastic deformation unless it can be shown that the loading is limited to small-scale yielding of the ligament and that the strain at the boundary between global and sub-model is elastic. In all other cases linear elastic sub-model analyses in conjunction with an elastic-plastic global model, in principle, would yield an erroneous prediction of the crack driving force. This is because a potential plastic deformation at the outer nodes taken over from the global model would be interpreted as elastic in the sub-model with the consequence that the corresponding stresses are strongly overestimated. Although this combination might yield a conservative assessment at the end for an individual application it certainly cannot be recommended for use because two uncertainties, the linear elastic instead of an elastic-plastic analysis and the misinterpretation of plastic deformations as elastic, balance each other with consequences which hardly can be foreseen.

**Analysis step 6: Determination of dynamic crack driving force**

Because the applied loading of the container is impact, dynamic global analyses, i.e., analyses considering mass terms have to be performed. Although this is quite clear in general, problems arise in detail. One question is whether the sub-model analysis should be dynamic too. Assume two limit states: (a) When the sub-model is identical with the global model the analysis has clearly to be dynamic. (b) On the other hand, when the dimensions of the sub-model are small, almost the complete dynamic information is provided by the (dynamic) global model. In such a case, performing the sub-model analysis with mass terms could even yield a "vibration characteristics" of its own that would not occur if the global model had been performed with crack. It is hard to decide which sub-model analysis is correct in such a case. An option is to perform both static and dynamic analyses and then to choose the more conservative results.

Another problem of dynamic analyses (including those of the global model) is the specification of the output frequency of the results. Whereas a high frequency is combined with increased effort and a large data volume a small frequency might smooth out vibrations with the risk of removing stress peaks.

Usually the crack driving force is determined in terms of the J-integral which is than formally transferred to a stress intensity factor in order to make it comparable to the fracture toughness of the material. The J-integral is obtained as a line integral in order to guarantee its validity by testing its path independency using different integration paths. Note that because the loading is introduced as impact ABAQUS/Explicit is used which, at its present state, does not allow for the determination of the J-integral. This is provided by an in-house post-processor of the applicant. In order to guarantee an independent evaluation of the results, BAM makes use of an own post-processor software, JINFEM [6], which is regularly used for cross-checking the results.

The safety analysis is finally based on the maximum crack driving force at the most critical position of the component, the most critical position at the crack front and the most critical time increment following the impact.

**Analysis step 7: Safety assessment**

Analyses steps 1 to 6 aimed at providing the crack driving force of a potential crack specified mainly by NDI at the most critical position(s) of the container. In order to perform a safety assessment this crack driving force has still to be compared with the dynamic fracture toughness of the material. Of the various options of safety assessment: (a) assessment for stable or unstable crack initiation, (b) assessment for stable crack extension, (c) assessment for crack arrest after instable crack extension option (a) is chosen at present evaluations [2].

## CONCLUSIONS

This paper presented the fracture mechanics approach, which, at BAM, is followed in compliance with the IAEA regulations [1, 5] for safety analysis reports for packages with components made of DCI for the transport of radioactive material. Due to the complexity of the packages with respect to loading and geometry the use of handbook solutions, e.g, for stress intensity factors is usually not possible. Instead finite element analyses are regularly performed which consist in the two steps: (a) numerical determination of the stresses and strains at the positions of highest loading in conjunction with experimental investigations and (b) Sub-modelling of the most critical sites with assumed surface cracks the size of which is specified by the requirements to non-destructive inspection.

## REFERENCES

[1] International Atomic Energy Agency (IAEA), Regulations for the Safe Transport of Radioactive Material, 2009 Edition, Safety Standard Series No. TS-R-1, Vienna, 2009

[2] BAM Bundesanstalt für Materialforschung und -prüfung, BAM-GGR 007, Leitlinie zur Verwendung von Gusseisen mit Kugelgraphit für Transport- und Lagerbehälter für radioaktive Stoffe, Rev. 0, Juni 2002

[3] European Committee for Standardization (CEN) (2001), EN 583-2:2001 Non-Destructive Testing – Ultrasonic examination – Part 2 Sensitivity and range setting, Section 6.4.

[4] The American Society of Mechanical Engineers (ASME) (1995), ASME Boiler and pressure Vessel Code, Sect. XI: Rules for Inservice Inspection of Nuclear Power Plant Components, New York, U.S.A.

[5] International Atomic Energy Agency (IAEA), Advisory Material for the IAEA Regulations for the Safe Transport of Radioactive Material, Safety Standard Series No. TS-G-1.1, Rev. 1, Vienna, 2008

[6] R. Boddenberg, J. Matzkows, JINFEM, FEM-Postprozessor zur Berechnung des J-Integrals, Handbuch zur Version 5, IWiS-Ingenieurbüro für wissenschaftliche Software GmbH, Berlin, 1987