
ARTICLE

Simulation research on safety verification of ^{60}Co transportation package

Dongyuan Meng^{a*}, Guoqiang Li^a, Jiangang Zhang^a, Dajie Zhuang^a, Renze Wang^a,
Hongchao Sun^a, Shutang Sun^a, Xuexin Wang^a and Zhe Hong^b

^aChina Institute for Radiation Protection, 102 Xuefu Street, Taiyuan, Shanxi, China; ^bNuclear and Radiation Safety Center, MEP, Beijing, China

Radioactive Material Transport safety is an important branch of nuclear safety. In the safety evaluation for radioactive material transport package, the combination of experiment and simulation is often used. This paper studied the finite element simulation of a kind of ^{60}Co source transport container. The situation of the package experiment and the finite element simulation method were introduced. Several different material models were used to simulate and their results were compare with the experimental data. The results showed that all the calculations were conservative and the bilinear hardening model was better for wood while bilinear hardening with failure strain model for steel. The feasibility of finite element simulation and the importance of material model were illustrated. Finally, some direction of the next step for finite element simulation was discussed.

Keywords: *Radioactive Material Transport; finite element; material model*

1. Introduction

Radioactive Material Transport (RMT) Safety is an important branch of nuclear safety. In RMT, radioactive package is a mobile radioactive source, the location of the accident is uncertain, the affected population is uncertain, and the safety evaluation needs special treatment.

According to the provisions of IAEA SSR-6[1] and China's national standard [2], radioactive transport containers need to undergo a series of tests including free fall, accumulation, puncture, heat and other tests to verify their safety performance. In the process of container safety evaluation, the combination of simulation and experiment is often used to determine the various possible test conditions in the most extreme conditions by simulation, thus saving the number of tests, and cost. Through the comparison of the experimental results and the simulation results, the correctness of the simulation could be verified, so as to analyze the difficult or unmeasurable data in the experiment by the simulation results.

The finite element method is a commonly used and effective simulation method for safety verification of transport containers for RMT. ANSYS/ LS-DYNA explicit dynamics software used in this study is a widely used code in package design and optimization [3,4].

While there are many factors that affect the accuracy of simulation, including grid density, material property parameters, material model and calculation model and so

on. How to improve the accuracy of simulation, and how to compare the simulation results with the experimental results to verify the accuracy of simulation is an important task.

In this paper, a 9m free drop test of a ^{60}Co radioactive source container is taken as an example, which is the most important test as regulated in IAEA SSR-6. The experiment and simulation method are introduced. The influence of material models for simulation is discussed by means of finite element explicit simulation. The results are compared with the experimental results.

2. Simulation model

China Institute for Radiation Protection conducted a series of simulation and tests on a kind of container for transporting ^{60}Co sealed radioactive sources for medical use. Through these tests, the safety performance of the container under routine and accidental transport conditions is described and verified. In these experiments, 9m free drop is generally considered to be the most crucial test of the package. In this paper, the 9m free drop accident is used as an example to calculate by finite element and explicit method.

This package is a B(U) type package, whose total weight is 3600kg. The package is made up of the protective container and the source container as **Figure 1** shows. The protective container consists of a stainless steel housing and its wrapped wood, while the source container consist of a stainless steel housing and filled lead.

*Corresponding author. Email: mdy55@hotmail.com

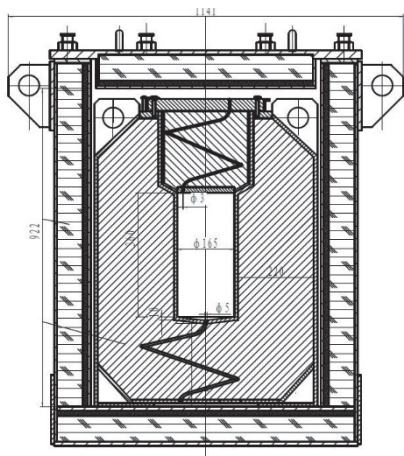


Figure 1. Package.

In the simulation, the model is simplified. The lifting ears, chamfers and so on which are less affected in the impact are neglected. According to the structural characteristics of the container, the semi model of the symmetrical boundary condition is used to calculate. The thinner parts of the shells are expressed in shell elements, and the rests are represented by solid elements. The simplified geometric model and the finite element model of the container are shown in the **Figure 2**.

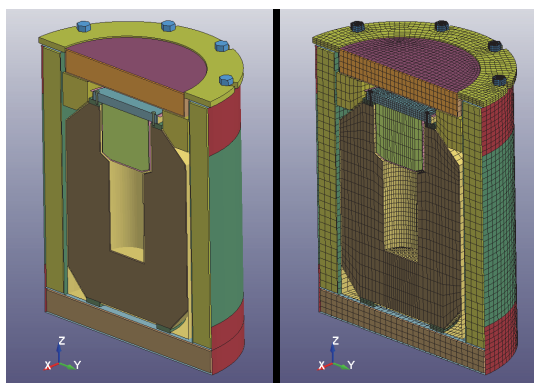


Figure 2. Simplified geometric model vs FE model.

The finite element model is composed of 45993 nodes and 36589 elements, of which 10352 shell elements and 26237 solid elements are used. Using the symmetrical boundary condition, the initial velocity is 9m free fall collision speed of 13.4m/s. Floor is described as rigid body model, while stainless steel and lead with bilinear hardening model, and wood with isotropic elasticity model and bilinear hardening model for testing.

3. Experiment results

The experiment was carried out at the experimental base of the China Institute for Radiation Protection, using the radioactive material transport container mechanics experiment facility. The main part of the facility consists of steel plate, concrete and pile composed of underground parts and lifting towers, hooks and other components above ground. This facility

is the largest radioactive material transport container test platform. The recording system is constituted of strain gauges, accelerometers, high-speed cameras and others which could record the process of falling package. The test facility is shown in **Figure 3**.

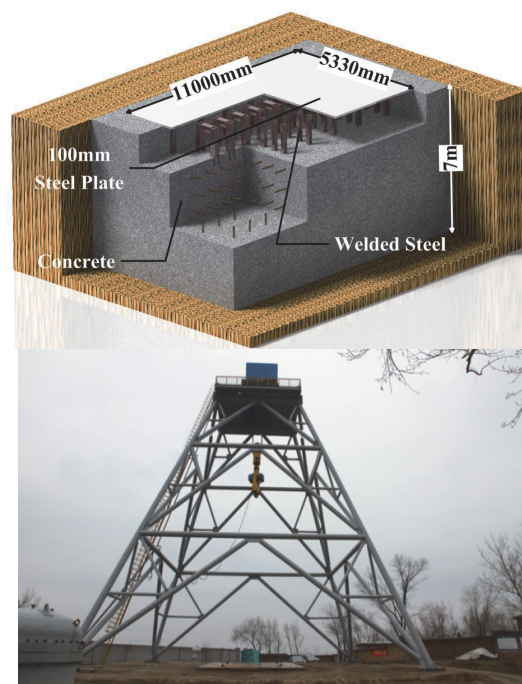


Figure 3. RMT package drop test facility.

According to the simulation before the experiment, the top vertex dropping attitude is considered to be the largest damage to the package. So the experiment is to use the vertex drop posture as shown in **Figure 4**. The strain gauges are arranged on the outer surface of the container to measure the strain changes the container surface during falling impact process. The strain gauges' position is shown in **Figure 5**.

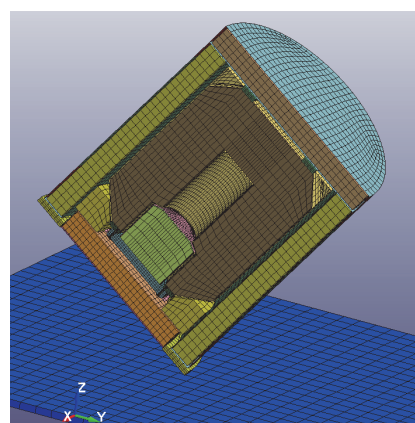


Figure 4. Drop attitude.

According to the measurement results and the container surface material parameters, the strain and stress curve changes with time at each measurement position can be obtained. For example, **Figure 6** shows

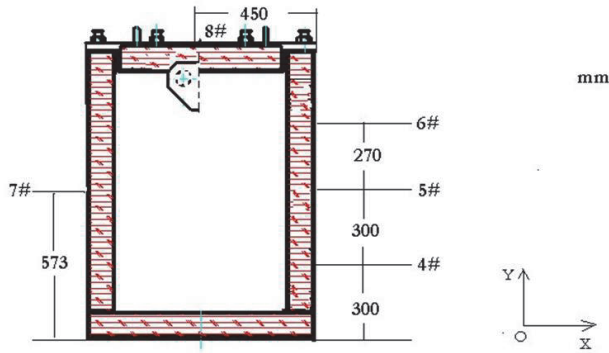


Figure 5. Position of strain gauges.

the principal strain and principal stress changes at 4# points.

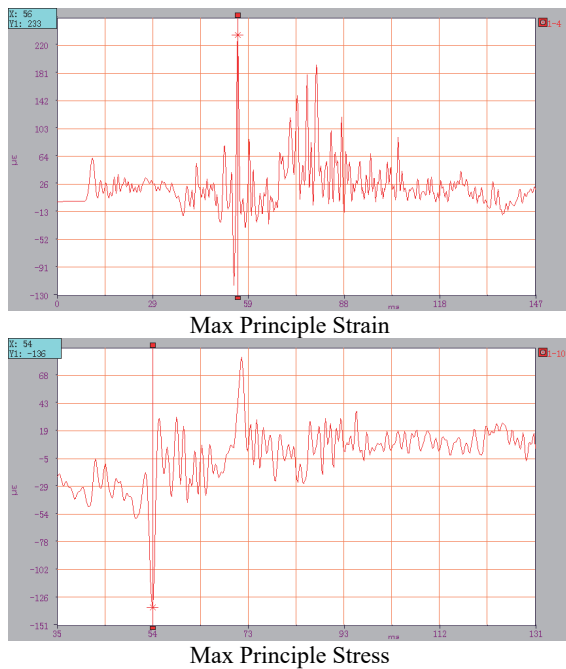


Figure 6. Strain/stress vs time.

4. Calculation results

By using the finite element explicit dynamic method to simulate the falling process of the container, the stress distribution of each part can be obtained. Then the maximum stress of each part in the whole collision process is compared with the design criterion, which could judge whether the stress of each part of the container during the impact exceeds the allowable value, and whether the container meets the design requirements in such circumstances. The use of simulation to evaluate the safety performance of the container is generally performed before experiment. After the experiment, the accuracy of the simulation model is verified by comparing the test results with the simulation results.

In the explicit dynamics simulation, there are many factors that affect the calculation results. Among them, the material model used is an important influencing factor. The isotropic elasticity model and the bilinear

hardening model are two commonly used models. The isotropic elasticity model considers that the material is elastic, and the stress and the strain are always proportional, and then the stress-strain curve is a straight line. When the deformation of the material is small or the yield strength is large, the elastic model can accurately describe the relationship between stress and strain of materials. The bilinear hardening model is consistent with the common elastoplastic material, such as the common metal material. After the elastic phase, the stress and strain ratio changes, but still basically a straight line. Whether or not considering failure is also an important factor of the material parameters in finite element simulation. When the material breaking stress is exceeded, the failure of the structure will absorb a lot of energy, thus reducing the influence of impact energy on the system. The stress-strain curve of the above material model is shown in **Figure 7**.

For this package, the main structural material is stainless steel, the main buffer material is wood. By simulating with different material models for stainless steel and wood respectively, the simulation results are compared with experimental results. Material models used is shown in **Table 1**.

According to the experimental results, three key points of the principle stress and the principle strain were compared, see **Table 2~3**.

Uncertainty is always exist in both experiment and simulation. In this experiment, the main source of uncertainty came from the strain gauge, which is less than 5% for its range as an authorities report for the

Table 1. Material models.

No.	Steel	Wood
1	Bilinear hardening without failure	Isotropic elasticity model
2	Bilinear hardening with failure strain	Isotropic elasticity model
3	Bilinear hardening with failure strain	Bilinear hardening

Table 2. Principle stress (MPa).

Position	Experiment	Calc. 1	Calc. 2	Calc. 3
4#	42	78	88	69
5#	77	108	103	85
8#	98	127	137	110

Table 3. Principle strain(10⁻⁶).

Position	Experiment	Calc. 1	Calc. 2	Calc. 3
4#	233	671	698	439
5#	390	708	644	512
8#	671	15449	7637	962

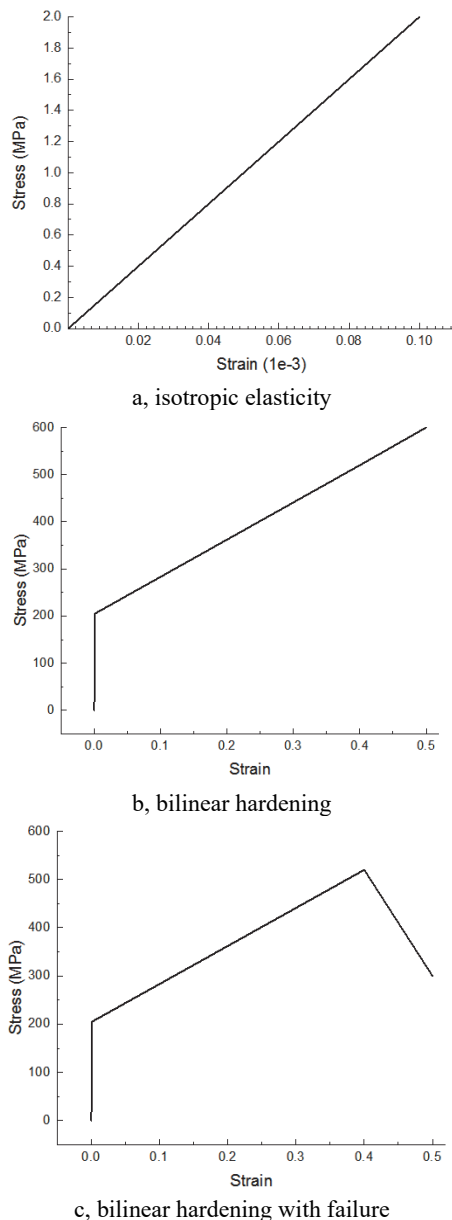


Figure 7. Stress-strain curves.

equipment. Although there were lots of other influence factors during the experiment, the uncertainty of strain gauge is most important for the strain data we used. The uncertainty for simulation is more complex. Several factors would affect the result of simulation. For example, high enough mesh density could produce a relative accurate result. Different element type for different part would produce different results. It is difficult to get an accurate result and even a number of uncertainty. A large amount of works need to be done to determine the uncertainty of a certain model, which could discuss in other works. Generally, it's believed a good finite element simulation could control the error less than 20%. Fortunately, in this problem we discussed, we only changed the material models, so we can assume that the uncertainty for different calculation is nearly the

same. That means, the comparison for different material models could make sense. While comparing the results between experiment and calculation, we should consider the uncertainty of them, separately.

As can be seen from the calculation results, all three calculation models are conservative for the experimental results, even the largest uncertainty is adopted, which is required for safety analysis.

The difference between the calculated results is obvious, indicating that the material model is very important for simulation. The third model, the stainless steel, takes the bilinear hardening model considering failure, and the bilinear hardening model is used in wood, which is in better agreement with the experimental results. Although the 3rd model cannot reproduce the experimental results, it shows the best agreement among them. Possible reasons include, for the structural material of stainless steel, the damage in the impact process will absorb a lot of energy, in the calculation to consider its damage effect more in line with the actual situation; for the buffer material wood, the use of bilinear model compared to the elastic model is softer, closer to the material's true stress-strain curve.

5. Conclusion

The material model is important for the accuracy of finite element simulation. This article only tried some common simplified material models. While the real stress-strain curve is more complex, especially for the wood, which requires a lot of experimental data to support. In addition, the results of the comparison with the experimental results can be considered more parameters, such as acceleration at some locations and other data. Improving the accuracy of simulation and try to reduce the error between experimental results, still involves many aspects of the problem, need more in-depth study.

References

- [1] IAEA, *Regulations for the Safe Transport of Radioactive Material*, SSR-6[S]. Vienna: IAEA, (2012).
- [2] Chinese Standard Publication, GB11806, *Regulations for Radioactive Material Safe Transport [S]*. Beijing: Chinese Standard Publication, (2004).
- [3] R. Li and X. Liu, Properties of polyurethane foam filled in new fuel transport cask, *Packaging Engineering*, 21 (2017), pp.10-14.
- [4] Y. Xu and Y. Ma, Drop gesture analysis of the fuel transport cask, *Packaging Engineering*, 21 (2017), pp.20-24.
- [5] J. Wang and Z. Wang, Dropping analysis and structure optimization of YG-1 cask, *Atomic Energy Science and Technology*, 50 (2016), pp.1853-1858.