Necking and Failure Simulation of Lead Material Using ALE and Mesh Free Methods in LS-DYNA®

Sunao Tokura
Tokura Simulation Research

Abstract

After the Fukushima Daiichi nuclear disaster in 2011, the need of experiment to predict failure of the structures including pipes and vessels in nuclear power plant in case of large earthquake or tsunami has been increasing. However it is dangerous and expensive to perform such experiments using real structural material, e.g., 304 stainless steel as very large test facility is needed to cause realistic failure. Alternatively, the idea to use pure lead (100 % Pb) and lead alloy in the experiments has been proposed as alternatives of real materials used in nuclear power plant. Lead is ductile material and lead alloy involving antimony (Sb) is brittle material. So both ductile and brittle failure modes can be reproduced easily in laboratory tests using these materials. For the simulation of failure of the structures, ductility of lead should be modeled accurately. High ductility and large necking are observed in the tensile test of pure lead rod. In this paper, simulation of necking and failure of the lead rod tensile test is tried using ALE and mesh free techniques, i.e., EFG, SPH and SPG implemented in LS-DYNA in addition to the conventional Lagrangian FEM approach, and the results of the simulation are compared and discussed with the experimental result.

Introduction

In general, large-scale test facilities are required to perform the experiments in which the equipment of nuclear power plants is failed under the assumption of severe accidents. In such experiments, the load and structural strength relationship should be investigated. However the experiments using large facilities with real material such as 304 stainless steel need much expense and involve some risk. So easier experiments using soft material instead of the real material is proposed. In such small-scale experiments, lead is used as an alternative of real ductile material since lead is very softer than steel. For example, the pipe structure made of lead can be failed easily using small vibration test device. For this purpose, the exact numerical model is also required to simulate the small failure experiments. A simple tensile test of lead material was executed to get the information of the material properties of lead and the nominal stress-strain curve was measured. In addition, extreme necking was observed in the test. In this paper, the simulation of the tensile test of the lead specimen was tried using mesh based and mesh free modeling techniques implemented in the latest LS-DYNA and compared these results with that of the experiment. The modeling technique using several element formulations includes ALE, SPH, EFG[1], and SPG[2] in addition to the conventional Lagrangian FEM solid model.

Analysis models and conditions

The geometry and the dimensions of the specimen are shown in Fig.1. Quarter models of the specimen for the simulation are created to reduce CPU cost. The mesh size or the interval of the particles in the tensile region are summarized in Table 2. In this paper, pure lead (100% Pb) material is used and the material properties are shown in Fig.2 which were measured in the material test[3]. Gurson model is used for the Lagrangian FEM solid model to consider the damage of the material. The values of the parameters of Gurson model were determined using LS-OPT® to get the closest tensile behavior with the experiment. The parameters are summarized in Table 1. However, since Gurson model cannot be used with ALE and mesh free methods, *MAT_024 is used instead for these formulations. The lower support is fixed using *BOUNDARY_SPC_ SET. *BOUNDARY_PRESCRIBED_MOTION is applied to the node set in the upper support and the specimen is stretched. The termination time is set as 0.01 seconds.

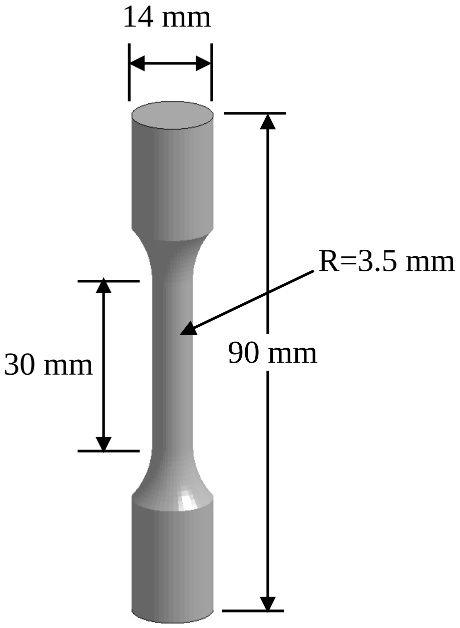


Fig.1 Geometry and dimensions of specimen

Young's modulus = 15250.0 MPa
Poisson's ratio = 0.450
Yield stress = 4.99 (MPa)
Density = 1.133 x 10-8 (ton/mm^3)

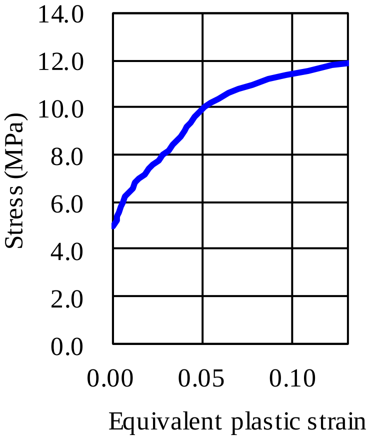


Fig.2 Material properties and stress-strain curve

Table 1 Values of Gurson model determined by parameter identification

Parameter	Value
$q1$	1.90188662
$q2$	1.14535555
f_c	0.074870368
f_0	0.001117206
ε_N	0.309696007
S_N	0.091531935
f_N	0.029673004
f_F	0.168703767

The parameters used for each formulation are defined as follows;

Lagrangian FEM : elform = 1 on *SECTION_SOLID (default)
*MAT_GURSON

ALE : elform = 11 on *SECTION_SOLID (1 point multi material ALE)
*MAT_PIECEWISE_LINEAR_PLASTICITY and *MAT_VACUUM
fail on *MAT_PIECEWISE_ LINEAR_PLASTICITY = 0.0
density of vacuum = 1.0 x 10^{-18} ton/mm^3
*CONTROL_ALE = default settings

SPH : *CONTROL_SPH = default settings
*SECTION_SPH = default settings
fail on *MAT_PIECEWISE_ LINEAR_PLASTICITY = 0.0

EFG : elform = 41 on *SECTION_SOLID_EFG
Settings of *SECTION_SOLID_EFG
- dx,dy,dz = 1.01 (normalized dilation parameters of the kernel function, default)
- ispline = 0 (cubic spline function, default)
- idila = 0 (maximum distance based on the background elements, default)
- iebt = 1 (full transformation method, default)
- idim = 1 (local boundary integration)
- toldef = 0.01 (semi Lagrangian kernel, default)
- ips = 1 (moving least squared pressure recovery)
- stime = 0.0 (time to switch from stabilized EFG to standard EFG, 10^{20}, default)
- iken = 0 (moving least square approximation, default)
- sf = 0.0 (failure strain, default)
- cmid = 0 (cohesive material ID)
- ibr = 2 (branching is allowed)
- ds = 0.01 (normalized support)
- ecut = 0.01 (the minimum distance to the node that crack surface can cut to the edge)

fail on *MAT_PIECEWISE_ LINEAR_PLASTICITY = 0.52

SPG : Settings of *SECTION_SOLID_EFG
- elform = 47 (smoothed particle Galerkin method)
- dx,dy,dz = 1.5 (normalized dilation parameters of the kernel function, default)
- ispline = 2 (cubic spline function with circular shape)
- kernel = 0 (updated Lagrangian kernel)
- lscale = 0.0 (length scale for displacement regularization)
- smstep = 0 (interval of time steps to conduct displacement regularization, default=15)
- swtime = 0.0 (time to switch from updated Lagrangian kernel to Eulerian kernel)
- idam = 0 (continuum damage mechanics, default)
- fs = 0.0 (failure strain if idam = 1, maximum principal strain if idam = 2)
- stretch = blank
- fail on *MAT_PIECEWISE_ LINEAR_PLASTICITY = 0.0

Results and discussion

The nominal stress - nominal strain curves were obtained through the simulation. The stress-strain curves from the results are shown in Fig.3. The deformation of the specimen in the tensile test is shown in Fig.4. It can be seen that large necking is formed before the failure occurs in this figure. The deformed shapes of the specimen for each simulation are also shown in Fig.5. In the real test, the specimen shows large necking caused by material damage and excessive softening is also observed just before the failure of the material. The curve of the tensile test in Fig.3 shows the softening clearly. For the simulation, the Lagrangian case shows slight softening and failure occurs suddenly. The ALE case shows necking and softening (see Fig.3 and 5) similar to the real test. But the necking occurs in later stage than the test. For the SPH case, necking cannot be seen at all and failure happens in very early stage of the deformation in spite of no definition of material failure criteria. So this is the numerical failure or instability. The EFG case shows no softening and failure occurs at the failure strain which is defined in the material input. For SPG case, necking occurs but the timing of the necking delays exceedingly comparing to the test result. The results of the simulation are summarized in Table 2.

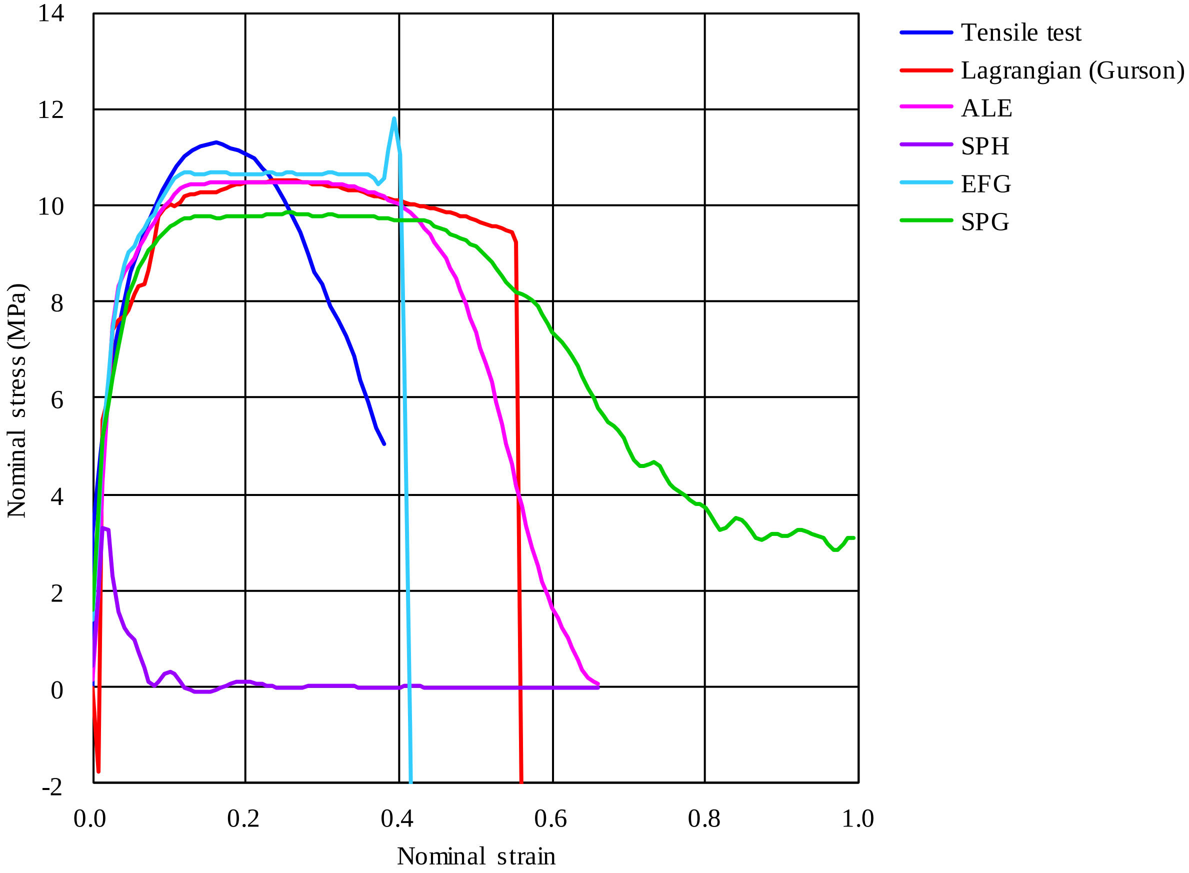
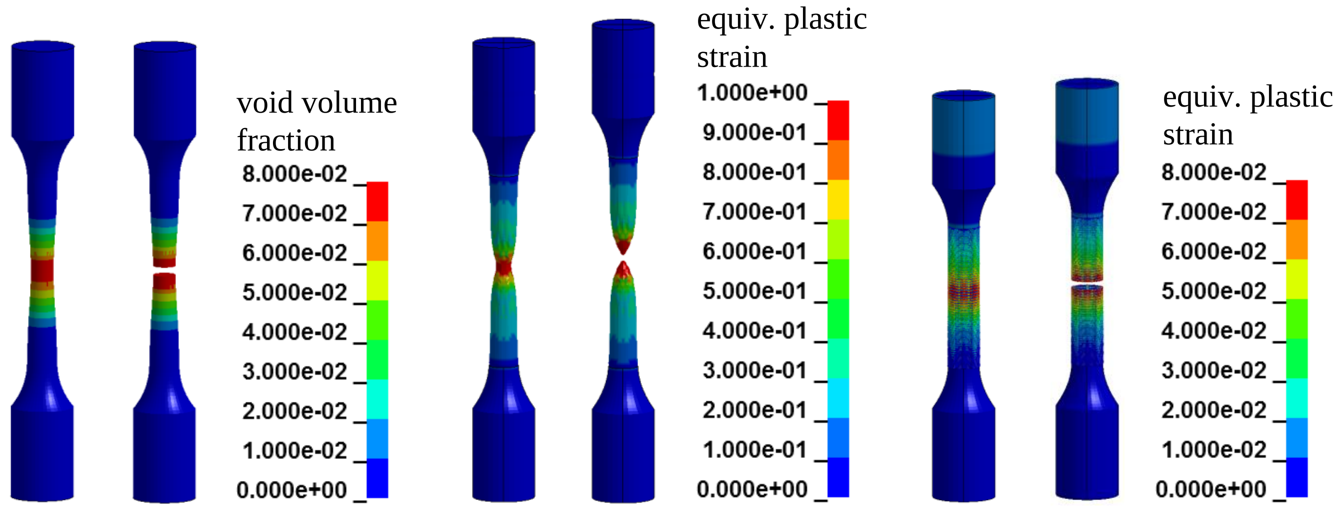
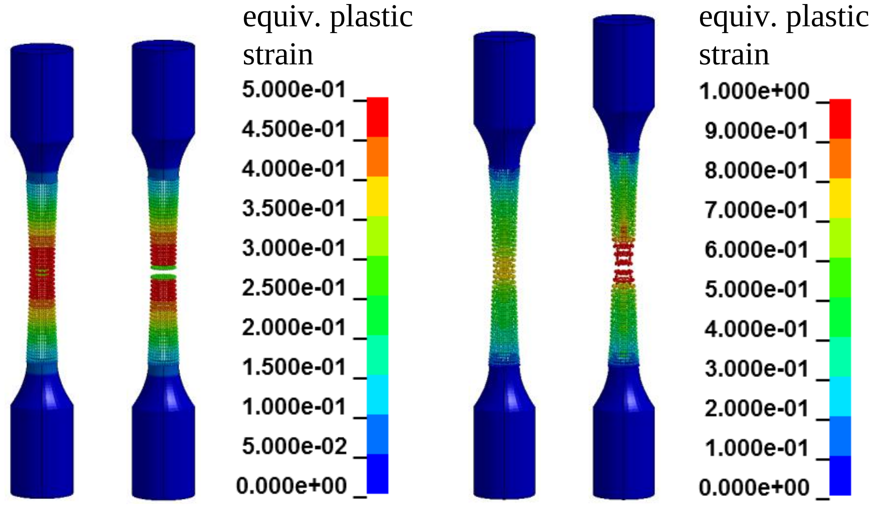


Fig.3 Comparison of nominal stress - nominal strain curves for tensile test and each element formulation

Fig.4 Deformation of specimen in tensile test



(*a*) Lagrangian FEM (Gurson) (*b*) ALE (*c*) SPH



(*d*) EFG (*e*) SPG

Fig.5 Deformed shape of specimen before and after failure for each element formulation

Table 2 Summary of simulation

Element formulation	Interval of nodes/particles	Number of elements in tensile region	CPU time (Lagrangian FEM = 1)	failure status	necking status
Lagrangian FEM	0.5	3615	1.00	physical failure	small
ALE	0.5	2400	1.95	numerical failure	large
SPH	0.25	22022	19.92	numerical failure	none
EFG	0.6	2091	1.00	physical failure	small
SPG	0.6	2091	2.94	no failure	small

Conclusions

The simulation of the necking and failure behavior of lead material in the tensile test was performed using Lagrangian FEM, ALE, SPH, EFG and SPG methods. ALE shows the best agreement with the real test in these cases. However the history of the mesh free methods is still very short and further investigation about many parameters, e.g., interval of particles, failure criteria, shape of kernel function are required.

References

[1] H. Lu and C. T. Wu, An Improves 3D Adaptive EFG Method for Forging and Extrusion Analysis with Thermal Coupling in LS-DYNA, 11th International LS-DYNA Users Conference, 2010

[2] C. T. Wu, Y. Guo and W. Hu, An Introduction to the LS-DYNA Smoothed Particle Galerkin Method for Severe Deformation and failure Analyses in Solids, 13th International LS-DYNA Users Conference, 2014

[3] N. Kasahara, I. Nakamura, H. Machida, H. Nakamura and K. Okamoto, Identification of Failure Modes Under Design Extension Conditions, ASME 2015 Pressure Vessels and Piping Conference Volume 1B: Codes and Standards, 2015