Impact analysis of casting parts considering shrinkage cavity defect

*Si-Young Kwak[1,2], Jie Cheng[2] and Jeong-Kil Choi[1,2]

(1. Center for e-Design, Korea Institute of Industrial Technology, Incheon 406-800, Korea; 2. Department of Virtual Engineering, School of Engineering, Korea University of Science & Technology, Daejeon, Korea)

Abstract: Shrinkage cavity may be detrimental to mechanical performances of casting parts. As a consequence, design engineers often use overly large safety factors in many designs due to insufficient understanding of quantitative effects of shrinkage cavity defects. In this paper, process of Al alloy wheel impact test was computationally analyzed for both the wheel models with and without shrinkage cavity defects. Based on shrinkage cavity data obtained from industrial CT (Computerized Tomography), the shrinkage cavity defects were modeled with SSM (Shape Simplification Method), which reconstructs shrinkage cavity defects to hollow spheroid primitives. After the impact simulation was conducted, the results show that under impact test condition, the wheel considering shrinkage cavity defects may fracture while the sound-assumed wheel may not.

Key words: casting; shrinkage cavity defect; impact; computational analysis; Al alloy wheel

CLC numbers: TG29/TP391   Document code: A   Article ID: 1672-6421(2011)01-112-05

In recent years, cast aluminum alloy wheels have been used widely in automobile industry. The wheels are critical components in the vehicle, so they must be durable enough to withstand rough loads and harsh environments. To assure this, some wheel tests such as impact test should be performed to test a prototype wheel for various fatigue and durability considerations [1,2]. This design-test-redesign process is a long-time and costly work. However, in modern industry, computational analysis is a good solution to shorten lead time.

A number of studies have been conducted for computational analysis of aluminum alloy wheel impact test [3-6]. However, none of them considered effects of casting defects such as shrinkage cavities.

Shrinkage cavity defects easily occur during flow and solidification processes of fusion metal in castings as shown in Fig. 1. They exist in most of casting parts and may be detrimental to mechanical performance of castings. Despite of this, the quantitative effects of them are not fully understood, and there is no well accepted method to predict the effects of internal shrinkage defects on casting performances. As a consequence, design engineers often do not consider their effects or use overly large safety factors when designing machinery parts[7]. However, neither of them can meet the demands of lighter-
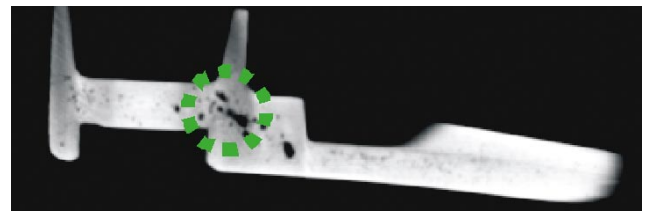

Fig. 1: Shrinkage cavity defects in castings

weight castings nowdays, so it is necessary to find an approach to investigate the effects of shrinkage cavity defects and take them into consideration in engineering design.

The present study extends our recently developed method of modeling the effects of shrinkage cavity defects on mechanical performances of casting parts [8]. By replacing static load in previous study with impact load, effects of shrinkage cavity defects on the results of wheel impact test were observed using a commercial FEM package, ABAQUSTM.

1 Shrinkage cavity modeling

There are three modeling methods for modeling shrinkage cavities for structural analysis: Direct Shape Method (DSM), Material Property Reduction Method (MPRM) and Shape Simplification Method (SSM) [8].

DSM generates the finite element meshes directly from the original shapes of shrinkage cavities in casting parts. This method is easy to use but maybe requires enormous elements to represent the shapes of shrinkage cavities and the aspect ratio of elements maybe very bad, thus leads to high cost in analysis time and maybe as well as many numerical errors in results.

*Si-Young Kwak

Male, born in 1969, Korea, principal researcher. His research interests mainly focus on the structural analysis considering casting defects, software development for FDM/FEM thermal stress module of casting and heat treatment process package (Z-CAST) and its industrial applications.

E-mail: vlvwlw@kitech.re.kr

The concept of MPRM is based on the following theory: modulus of elasticity of materials in the lowest range of porosity can be described by a linear dependence on porosity, derived with the assumption that voids do not interact [9]. There are a couple of correlating equations about MPRM for shrinkage defects of casting [6-9]. However, MPRM can not show the effect of stress concentration raised by shrinkage cavities and its application is limited to elastic range of material.

SSM models shrinkage cavity defects to the shape of hollow spheroid. It is the method making minimum spheroid that encloses STL(STereoLithography) format of shrinkage cavity which is acquired and restored from industrial CT (Computerized Tomography). SSM can reflect stress concentration factor of shrinkage cavity while requires fewer elements than DSM. Details of SSM procedure and performance can be referred to the author's paper [8].

In this study, SSM was applied to investigate the effects of shrinkage cavity defects on AC4C cast aluminum alloy undergoing impact load.

2 Numerical modeling

2.1 Dynamic strain concentration factor

Stress concentration factor is used widely to evaluate the quantitative effect of stress riser in an object. Here we introduce dynamic strain concentration factor (DSCF) [10] for the impact simulation. The dynamic strain concentration factor is defined as following:

$$K_{(t)} = \frac{|\varepsilon_{max}|_{(t)}}{|\varepsilon_{nom}|_{(t)}}$$

where, ε is a function of time t, ε_{max} and ε_{nom} the maximum strain and nominal strain, respectively.

In case of object with sphere shrinkage cavity as shown in Fig. 2, when under static load, the analytical value of $K(t)$ can be evaluated as 2.05 by the reference data as shown in Fig. 2 [11]. According to D. R. Weaver's conclusion [10], the authors took the value of 2.05 as the analytical value to study the convergence of result depended on element type and size in this paper.
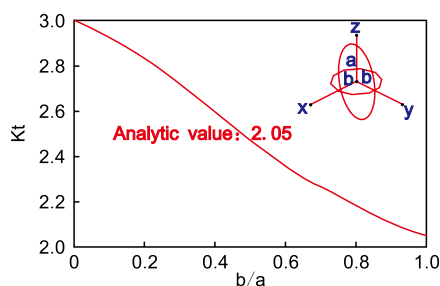

Fig. 2: Stress concentration factor

(b is the equatorial radius along the x and y axes and a is the polar radius along the z-axis)

2.2 Element type selection and element size determination

To investigate the convergence and accuracy of the impact

simulation, benchmark analysis was performed by using a rectangular cuboid bar with a sphere shrinkage cavity at its center. Figure 3 gives the symmetric half model of the bar and the conditions for numerical simulation.
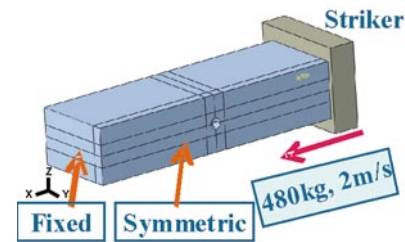
Fig.3: Simple bar model for simulation

The results of impact simulation are very sensitive to both the element type and size. In general, brick element is preferred to tetrahedron element in impact simulation in terms of accuracy, but it is difficult to generate mesh on real casting products with complicated shapes. Therefore, tetrahedron elements are adopted in this study since it is much more universally applicable than brick elements.

As to element size, three cases of impact simulation were carried out. Elements size seeded on the shrinkage cavity surface became finer and finer from case 1 to case 3. Here, the element size is defined as the fraction of the radius (minor radius for ellipsoid), r, of simplified shrinkage cavity. From case 1 to case 3, element sizes are 0.5, 0.25 and 0.125, respectively. Figure 4 plots the results of the maximum longitudinal true strains depend on element size. As mesh became finer and finer, the DSCF obtained by FEM were more closely converged to analytical solution. However, regular element formulation requires enormous elements and thus computational cost to exactly converge to 2.05. An alternative is to apply fine meshes around the area of shrinkage cavity rather than only surface of shrinkage cavity, but this mesh generating work may be not easy for engineers when model shape is complicated. Therefore, the authors recommend a first order tetrahedron element size of 0.125 on shrinkage cavity surface in this paper.
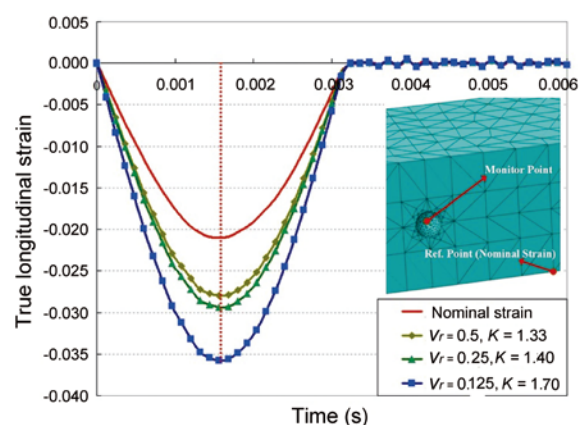
Fig. 4: True strains depend on element size

3 Application

3.1 Wheel impact test

Figure 5 shows the setup used for wheel impact test. It consists of a striker of 480 kg with the contact surface dimension of 375 mm × 125 mm. The wheel is mounted at 13° angle degree to the horizontal plane so that its highest point is presented to the vertically acting striker. The dropping height of the strike weight is 230 mm above the highest part of the rim flange [2].
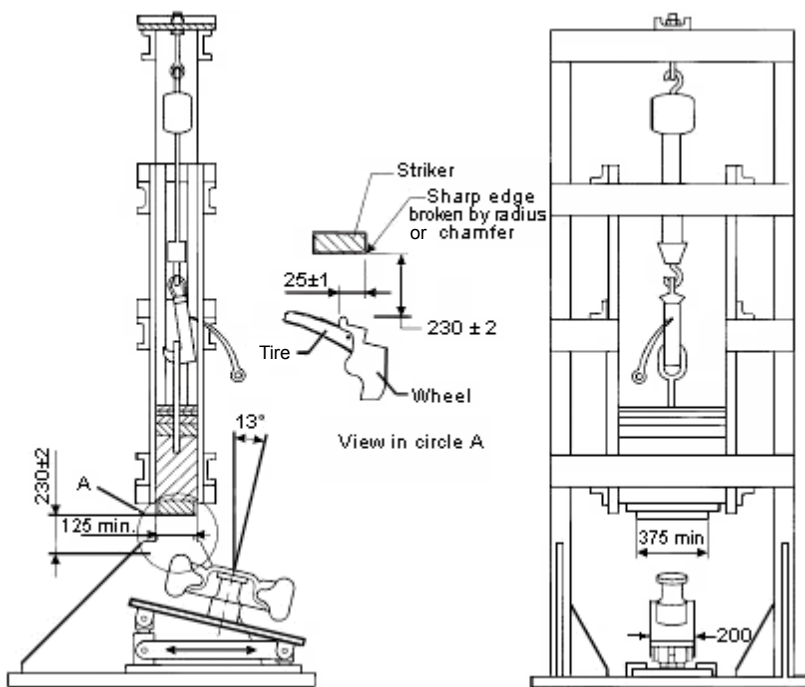

Fig. 5: Impact loading test machine (SAE J175)

In this study, impact simulations for wheels were performed according to SAE J175-Wheels-Impact Test Procedures–Road Vehicles, issued by the Society of Automotive Engineering, Inc. in 2001.

Both the models and setting in impact simulation were simplified as shown in Fig. 6. The analysis includes two cases: one does not consider shrinkage defects while the other considers.
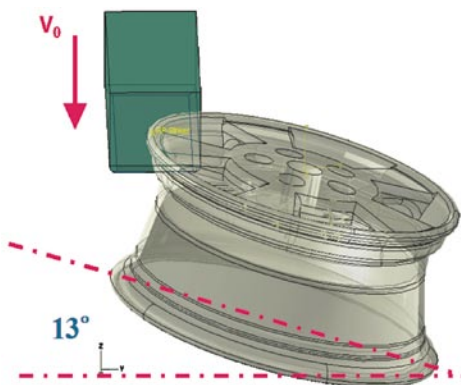

Fig. 6: Models and setting for simulation

3.2 Modeling and conditions

Figure 6 shows the model of wheel and striker. The mass of striker is:

$$Mass: \ D = 0.6W + 180 \approx 480 \text{ kg}$$

where W is the maximum static wheel loading as specified by wheel and/or vehicle manufacturer, expressed in kilograms.

Initial velocity of striker is:

$$Velocity: V_0 = \sqrt{2gh} \approx 2.12 \text{ m/s}$$

where g is the gravitational acceleration and h is dropping height of the strike weight.

Figure 7 depicts the procedures of obtaining shrinkage cavity data from industrial CT and modeling them to the defects with simplified shapes. The main process of this job includes two steps. First step is to acquire the voxel information by CT, and then, restore them to STL.

Because the STL file from CT is extremely fine, so it is uneasy to generate meshes from the STL file that obtained directly from industrial CT without any modification using present mesh generating tools. Therefore, the original CT STL format model should be processed to meet the requirement of mesh generation. To do this, ImageWare™, which is a specific program for inverse engineering, was used in this study.

Finally, the shrinkage cavity models were analyzed and simplified to ellipsoids in CAD format. After Boolean operation, the wheel model with shape simplified shrinkage cavity defects can be obtained to perform the impact simulation. In the case of this study, the shrinkage defects are located at two locations; one (Shrinkage A) is near the interaction area of spoke and rim, and the other (Shrinkage B) is near the interaction area of spoke and hub as shown in Fig. 7.
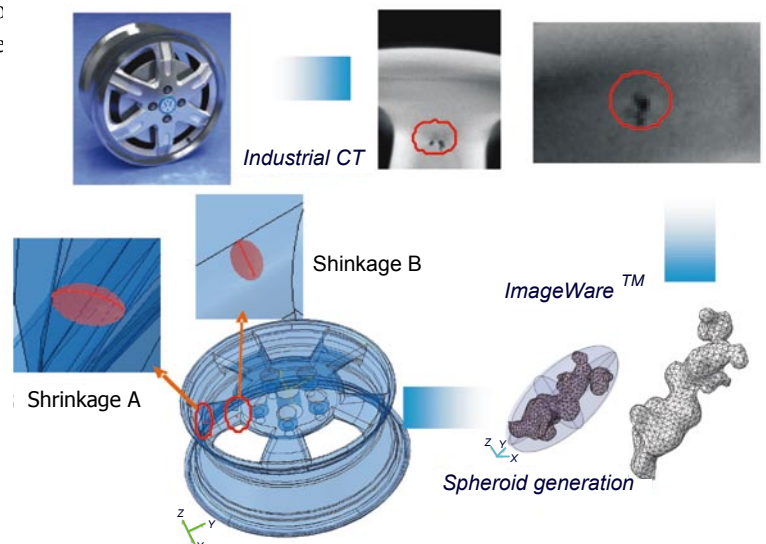

Fig. 7: Procedures of obtaining and modeling shrinkage data using CT

3.3 Failure criteria

There are several criteria of failure in impact load and the criteria are mainly based on the strains [5]. A failure criterion based on a fracture strain criterion was used in this study, that is, if the equivalent plastic strain (PEEQ) ε_p is equal or larger than fracture strain of material ε_f, i.e. $\varepsilon_p \geqslant \varepsilon_f$, the wheel can not pass the impact test. According to tensile test data, the critical true plastic strain (fracture strain) of involved material (AC4C) is 0.059.

4 Results and discussion

During the impact test process, strains in wheel keep raising up before the velocity of striker decreased to zero. The results of simulations showed maximum strain occurs at 0.00087 s after striker getting contact with wheel. This is the critical time point for wheel impact test, so all of following results focus on this time point in this paper.

Deformation of the wheel, or fractures in the area of the rim section contacted by the face plate of the striker do not constitute a failure as noted in SAE J175 [2], so only area out of striker contacted face are of interest in this study.

Contour of equivalent plastic strains in the case of without shrinkage cavity defects is plotted in Fig. 8. The maximum value of equivalent plastic strain (0.048) is less than 0.059, so the wheel is expected to be able to pass the impact test.
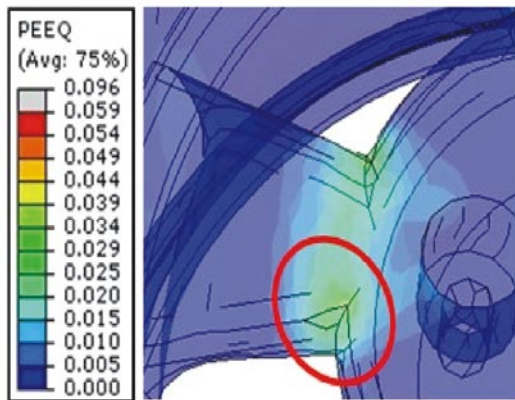

Fig. 8: Contour of equivalent plastic strains (no shrinkage defects)

Figure 9 plots the contour of equivalent plastic strains in the case of with shrinkage defect. For shrinkage cavity A, maximum principal elastic strain on cavity surface is less than 0.001 and no plastic occurred. However for shrinkage cavity B, the maximum equivalent plastic strain (0.077) on cavity surface is over 0.059, so the wheel is expected fracture from shrinkage cavity B.

Even it is difficult to expect whether the fracture will occur or not in the real test by means of computational analysis, but if the analysis considers shrinkage cavity defects, the prediction of wheel impact test performance will be more reliable, focus and realistic. By comparing the wheel impact test results with and without shrinkage cavities, an engineering decision can be made to remedy the shrinkage cavity or not.
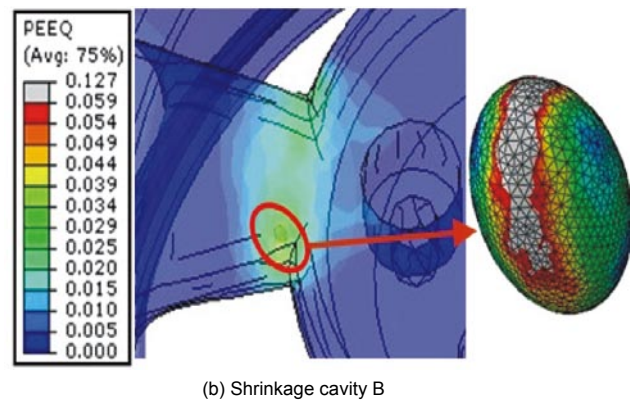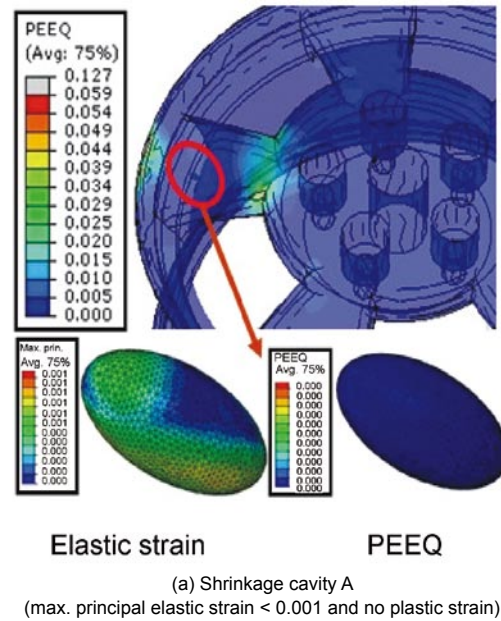

(a) Shrinkage cavity A
(max. principal elastic strain < 0.001 and no plastic strain)

(b) Shrinkage cavity B

Fig. 9: Contour of equivalent plastic strains (with shrinkage defects)

5 Conclusions

This study presents the effects of shrinkage cavities on the performance of casting parts subject to practical impact load. The authors proposed an approach of impact simulation technology considering shrinkage cavity defects in casting parts. The proposed approach used industrial CT scanned shrinkage cavity defects and modeled it to spheroid primitives using SSM.

For impact test analysis of casting Al wheel, the proposed analysis results show that the shrinkage cavity located in high strained area of casting part may be far more detrimental than in low strained area. In conclusion, the proposed approach can reflect strain concentration effect and improve the accuracy of simulation with proper computational cost.

References

[1] SAE J328. Wheels-passenger car and light truck performance requirements and test procedures. Warrendale, (PA): Society of Automotive Engineering, Inc., 2001.

[2] SAE J175. Wheels-impact test procedures-road vehicles. Warrendale, (PA): Society of Automotive Engineering, Inc., 2001.

[3] Li P, Maijer D M, et al. A through process model of the impact of in-service loading, residual stress, and microstructure on the final fatigue life of an A356 automotive wheel. Material Science and Engineering A, 2007, 460-461: 20–30.

[4] Zhu Maotao and Han Bing. Analysis of impact test of aluminum disc wheels based on FEM. SAE International, DOI: 10.4271/2007-01-3648.

[5] Chang Weichan. Modeling for impact test of aluminum wheels. Master Thesis of Taiwan National Central University, July 2008.

[6] Chia-Lung Chang and Shao-Huei Yang. Simulation of wheel impact test using finite element method. Engineering Failure Analysis, 2009, 16: 1711–1719.

[7] Sigl K M, Hardin R A, Stephens R I, and Beckermann C. Fatigue of 8630 cast steel in the presence of porosity. Int'l J. of Cast Metals Research, 2004, 17(3): 130–146.

[8] Kwak S Y, Cheng J, Kim J T and Choi J K. Structural analysis considering shrinkage defect of casting part. Int'l J. of Cast Metal Research, 2008, 21 (1-3): 419–423.

[9] Dewey J M. The elastic constants of materials loaded with non-rigid fillers. Journal of Applied Physics, 1947, 18: 578–581.

[10] Weaver D R. Dynamic strain concentration factor for a notch. Strorming Media, Report 0271076 (Report Date: Apr. 1968).

[11] Pilkey W D. Peterson's Stress Concentration Factors. Second Edition, New York: John Wiley & Sons, Inc., 1997.