Reverse engineering of a hot ring compression test using FEM

Regional Research Laboratory (CSIR), Bhopal 462 026, India

Received 23 June 2003; accepted 16 March 2004

This study is related to reverse engineering solution of friction and yield stress in a hot ring compression test using finite element method (FEM). Ring material is steel and test temperature is 800°C. Load deflection curve and the deformed geometry, obtained from the test, are considered as the benchmark parameters for the reverse engineering. The numerical experimentation has been carried out in an iterative manner. The values of friction are varied to find the final geometry matching with the actual deformed shape. Having established the friction, yields stress is varied to match the tonnage employed in the test. It is observed that FEM can be very helpful in such kind of reverse engineering problems.

IPC Code: Int. Cl. 3 G12B 3/06

Friction conditions at the die material interface greatly influence metal flow, formation of surface and internal defects, stresses arising in the dies and load and energy requirements. Ring compression test is a useful technique for prediction of friction between platen and billet. The standard ring dimensions are 6:3:1 (OD:ID:thickness). Calibration curves of such rings can be calculated using formulae reported elsewhere. This method is quite suitable for rings of considerable thickness. To analyze slender rings, where buckling may be involved, numerical tools like finite element method can be employed.

In this study, elasto-plastic finite element analysis of a slender ring compression at 800°C, has been carried to ascertain friction and yield stress. This particular ring is used as casing in testing of magnetic materials. Deformed geometry and the load deflection curve of the ring were the benchmarks for this. The friction values are varied to obtain desired profile. Once the deformed geometry is matched with the experimental one, yield stress is varied to match with experimental load deflection curve. Non-linear finite element analysis has been carried out using an indigenous software developed at RRL Bhopal. This software is based on total elastic incremental plastic strain (TEIP) algorithm. Such reverse engineering application of finite element method is quite useful in establishing design parameters.

Problem Formulation

Slender rings are used as casing in testing of magnetic materials. As magnetic materials are brittle, they are encased in rings during compression test. Identical rings are compressed separately also. By superposing the two results, behaviour of magnetic materials is calculated. Most of the time, these tests are carried out at high temperature. In this regard, prediction of friction is very necessary. Because slender rings undergo large material rotation, non-linear finite element analysis can be employed for simulation of compression test. But elasto-plastic finite element analysis requires the value of yield stress. The material properties given at higher temperature in hand books are in terms of flow curve and it is very difficult to figure out exact yield stress from that. Hence the prediction of friction, at the test temperature, indirectly involves prediction of yield stress also. These values can be obtained by carrying out finite element analysis (varying the friction coefficient and yield stress) and matching the deformed geometry and load deflection curves with it’s numerical experiment counterparts. In this study friction coefficient and yield stress are obtained by this reverse engineering process.

Experimental Procedure

The deformation experiment was conducted with a servo-hydraulic testing machine (DARTEC Ltd., U.K.) of 100 kN capacity, equipped to provide exponential decay of the actuator speed, such that constant true strain rates in the range of $3 \times 10^4$ to $1 \times 10^2$ s$^{-1}$ could be imposed on the specimen. In order to obtain a constant true strain rate value, the ram speed could be varied instantaneously from a

*For correspondence (E-mail: kkpathak1@rediffmail.com)
minimum of \(3 \times 10^{-3}\) mm s\(^{-1}\) to a maximum of 1.25 \(\times 10^{3}\) mm s\(^{-1}\) by taking into account the change in strain at very small intervals of strain. Isothermal deformation condition was maintained by surrounding the specimen, platen and push rods by a three zone furnace. The temperature was controlled within \(\pm 2^\circ C\). The adiabatic temperature rise was recorded.

**RRLFEM Software**

Using TEIP algorithm, a two-dimensional FEM software, named as RRLFEM, is coded\(^5\). RRLFEM has a very effective pre-processor which generates automatic mesh. There are several triangular and quadrilateral elements of varying order. Application of boundary conditions is very simple. Post-processing of the results can be carried out textually in the output file and graphically using DISPLAY III software\(^6\).

**Contact**

The contact algorithm of the software is based on master and slave surface concept considering friction between the two. When the slave surface approaches the master surface and distance between them at any location falls within the range of ‘tolerance of contact’, the corresponding slave surface node is not allowed to penetrate the master surface at these location. They are allowed to slide on the master surface depending upon the specified frictional conditions.

**Release**

The nodes selected by the identifiers are assigned the node release code. After release elastic strain is deduced from the total strain and remaining stresses are residual stresses.

**Restart**

Software is capable of restart from a previous run job. Restart run can be initiated at any step if the restart dump is available for the previous step. This feature is very essential for jobs, which require long run times.

**Finite Element Modelling**

The dimensions of the ring, considered in this study, are - ID = 7.21 mm, OD = 8.51 mm, wall thickness = 1.3 mm and height of the ring =14.4 mm.

The OD:ID:thickness ratio of this ring is 6.55:5.55:1. Axisymmetric modelling of the ring was carried out considering a quarter section of the ring. The ring was discretized using 4 noded axisymmetric elements. The finite element model of the whole ring is shown in Fig. 1. There were 602 elements and 671 nodes in the quarter model. The compressive load on the ring was applied through platen using master slave concept. In this, nodes lying on plate are masters, whereas nodes on ring surface are slave nodes.

**Material properties**

Material properties of the ring material at 800°C are obtained from the flow curve\(^7\). Values of yield stress is taken as variable and it's initial value is considered as 100 MPa. Similarly friction factor is also considered as variable. Initial properties considered are- Young’s modulus = \(1.1 \times 10^5\) N/mm\(^2\), Poisson’s ratio = 0.3, yield stress = 100 MPa, strain hardening coefficient = 0.125 shear friction factor = 0.0, strain rate= 0.001 and temperature = 800°C.

**Boundary conditions**

Nodes on the bottom section of the quarter model were restricted to move in vertical direction. Load is applied on the top through contact between platen and ring. In this study platen was given a downward displacement of 6.1 mm. The complete displacement was applied in 500 incremental steps.

**Analysis**

Finite element analysis of the ring compression process was carried out in two steps using RRLFEM. First, friction factor is varied to obtain a profile matching with the deformed geometry. After fixing the friction factor, yield stress is varied to match the load deflection curve. In this way friction factor and yield stress, present at the time of experiment, are established.

**Friction factor**

The finite element simulation of the ring compression test was carried out considering various shear friction factor namely 0.0, 0.1, 0.2 and 0.25. Initial yield stress was considered as 100 MPa. It was found that at shear friction value of 0.25, deformed
The co-ordinates of node number 13 and 168 (Fig.1) after upsetting, considering various yield stresses, are given in Table 1. It can be observed that variation of yield stress has negligible effect on the shape of deformed geometry. Hence, it was established that shear friction factor at the time of experiment was 0.25.

**Yield stress**

The yield stress was varied from 100 MPa and onwards with an increment of 10 MPa. At each load, load deflection curve was plotted and matched with the experimental one. Maximum tonnage obtained from the experiment was 18.3 t. The load deflection curves considering different yield stresses are shown in Fig. 2. The closest match which could be obtained between experimental and FEM load deflection curve is for the yield stress value of 145 MPa. It is observed that peaks of the load-deflection curves are at different values of the displacement. This is due to the error in recording of displacement data in the initial stage of the upsetting.

**Results and Discussion**

Stress and strain variation in the specimen, after establishing the friction and yield stress was obtained through finite element analysis. Contour plots of effective stress and strain are shown in Figs 3 and 4. It is observed that the maximum stress is in the range

<table>
<thead>
<tr>
<th>Yield stress (MPa)</th>
<th>x co-ordinates (mm)</th>
<th>y co-ordinates (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>8.60</td>
<td>11.55</td>
</tr>
<tr>
<td>125</td>
<td>8.60</td>
<td>11.55</td>
</tr>
<tr>
<td>150</td>
<td>8.59</td>
<td>11.56</td>
</tr>
<tr>
<td>175</td>
<td>8.59</td>
<td>11.56</td>
</tr>
<tr>
<td>200</td>
<td>8.59</td>
<td>11.56</td>
</tr>
</tbody>
</table>
240.3-331.6 MPa. It indicates that most of the part has yielded. Maximum effective strain is 0.987 which also indicate that most of the part are experiencing large deformation.

Conclusions

In this study finite element simulation of the hot ring compression test was carried out to establish friction and yield stress present at the time of experiment. The guidelines, for them to achieve, were deformed geometry and load deflection curve, obtained from the experiment. Shear friction factor and yield stress in the experiment was 0.25 and 145 MPa respectively. Maximum tonnage in FE analysis and experiment are 18.3 ton.

References
6 DISPLAY III, Pre and Post Processing Program (EMRC, Michigan), 1993.