FINITE ELEMENT SIMULATION & DESIGN OF HOT TENSILE TEST ON SUPERPLASTIC ALUMINIUM 6063
S.Nakkeeran, Asst.Professor, Bharath University, Chennai-73.

Abstract — Finite element Method is a numerical procedure for analyzing a wide range of problems that are too complicated to be solved satisfactorily by classical analytical methods. The finite element method (FEM) models a structure as an assemblage of small parts (elements). Each element is of simple geometry and therefore is much easier to analyze the actual structure. ABAQUS, a general purpose finite element program is used to carry out the Non Linear analysis of Superplastic Forming. It is a suite of powerful engineering simulation programs based on the finite element method. An experiment will be conducted by using the Universal Tensile Testing Machine on Superplastically formed component by varying process parameters such as Stress, Temperature and Strain rate. The effect of the Strain-rate Sensitivity, Stress and Temperature of the alloy on the rate of superplastic forming can be enabled. By using ABAQUS Software, we can find the optimum parameters to get good Superplastic forming in AL6063 Alloy.

Keywords: Finite element method, Non-Linear Analysis, Superplastic forming.

1. Introduction

Superplasticity is a state in which solid crystalline material is deformed well beyond its usual breaking point, usually over about 200% during tensile deformation. Such a state is usually achieved at high homologous temperature, typically half the absolute melting point. Examples of Superplastic materials are some fine-grained metals and ceramics. The low flow stresses and high sensitivity of flow stress to strain rate are the main aspects of superplastic deformation.

Superplastically deformed material gets thinner in a very uniform manner, rather than forming a “neck” (a local narrowing) which leads to fracture. Also, the formation of internal cavities, which is another cause of early fracture, is inhibited. Those materials which meet these parameters must still have a strain rate sensitivity (a measurement of the way the stress on a material reacts to changes in strain rate) of >0.3 to be considered superplastic. The low flow stresses and high sensitivity of flow stress to strain rate are the main aspects of superplastic deformation. Fine and equiaxed grain size, forming temperature greater than half the absolute melting temperature of the subject material, and controlled strain rate, are the main requirements for superplasticity. The optimum value of strain rate varies with the type of material, but is usually very low, e.g. 1.0E-3 – 1.0E-5/s.

Superplastic forming (SPF) is a near net-shape forming process which offers many advantages over conventional forming operations including low forming pressure due to low flow stress, lower die cost, greater design flexibility, and the ability to shape hard metals and form complex shapes. However, low production rate due to slow forming process and limited predictive capabilities due to lack of accurate constitutive models for superplastic deformation, are the main obstacles to the widespread use of SPF. This factor has restricted the growth of applications of Superplastic alloy to low volume production industries like the aerospace industry. Aluminum alloys, Titanium alloys and magnesium alloys are typical examples of metallic superplastic materials that have been developed and are increasingly being used to produce complex shapes. Some composites and ceramics are also known to behave Superplastically.

The forming temperature is just an important variable in superplastic forming as the strain rate. Temperature variation in a forming die is a primary source of localized thinning. Characterization of material behavior should therefore include not only determination of the optimum superplastic temperature but also the sensitivity of flow stress and elongation to temperature. A large temperature sensitivity of flow stress is not desirable, because local hot spots will lead to severe strain localization. The modes of failure in superplastic forming are strain localization and necking. Therefore fracture occurs in most superplastic materials of engineering application.

1.1 Superplastic Forming

Superplastic deformation can be used in high temperature forming of complex shapes for which forging operations, with their higher strain rates, are not suitable. Microstructure plays a role in superplastic deformation. A fine-grained equiaxed microstructure is required for superplastic behavior to be manifested. Moreover, the structure must be resistant to grain growth at the temperatures and time duration of superplastic deformation. Another microstructural feature of superplastic deformation is that grain shape is essentially preserved during the deformation.

1.2 Strain-Rate Sensitivity
Superplastic behavior correlates with a high strain-rate sensitivity exponent in constitutive equation

\[ \sigma = K \dot{\varepsilon}^m \]

\( \dot{\varepsilon} \) is effective flow stress (N/m²)

\( \dot{\varepsilon} \) is strain rate, (s⁻¹)

\( K \) is material constant depends upon temperature and grain size

\( m \) is strain rate sensitivity index

Superplastic alloys are characterized by \( 0.3 < m < 1 \). Actual values of \( m \) depend on temperature, strain rate and grain size. By virtue of high values of \( m \) (compared to conventional metals) superplastic metals have a large capacity for stretching.

2. Finite Element Modelling

The Finite Element Method (FEM) has developed into a key, indispensable technology in the modelling and simulation of advanced engineering systems in various fields like housing, transportation, communications, and so on. In building such advanced engineering systems, engineers and designers go through a sophisticated process of modelling, simulation, visualization, analysis, designing, prototyping, testing, and lastly, fabrication. Note that much work is involved before the fabrication of the final product or system. This is to ensure the workability of the finished product, as well as for cost effectiveness.

2.1 FINITE ELEMENT ANALYSIS

A typical finite element analysis on a software system requires the following information:
1. Nodal point spatial locations (geometry)
2. Elements connecting the nodal points
3. Mass properties
4. Boundary conditions or restraints
5. Loading or forcing function details
6. Analysis options

The FE simulation has been performed using a commercial FE package, ABACUS. This FE code includes direct implicit integration, which is chosen for superplastic analysis since it enables a full static solution of deformation problems with convergence control. In addition, the time increment size can be defined with practical limits. In explicit dynamic software, on the other hand, the stability requirement forces the time step to be very small, and the total number of steps for complete analysis can run into thousands of steps. Thus, for the superplastic forming process, which is usually slow, static implicit software is the best choice compared with explicit dynamic software.

2.2 FEM Solution process

1. Divide structure into pieces (elements with nodes) (discretization/meshing)
2. Connect (assemble) the elements at the nodes to form an approximate system of equations for the whole structure (forming element matrices)
3. Solve the system of equations involving unknown quantities at the nodes (e.g., displacements)
4. Calculate desired quantities (e.g., strains and stresses) at selected elements.

3. Superplastic Forming Simulation (ABAQUS)

Superplastic forming rectangular shape component simulation of Aluminium 6063 alloy, with follow the standard ABAQUS procedure as shown the figure 1.0

![Figure 1.0 Abaqus module flow chart](http://www.ijpttjournal.org)

3.1 Material parameter Input
As Superplastic deformation was dominated by large plastic deformation, the elastic deformation was neglected. Strain hardening was ignored by giving a very low value of n, even n=0 could be used. The large deformation in Superplasticity occurred in the area after uniform elongation. Therefore, the flow stress was affected only by the strain rate. The detailed value of each parameters were given as follows:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density of material</td>
<td>2.7 X10^3 Kg/m³</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>0.33</td>
</tr>
<tr>
<td>Elastic Modulus</td>
<td>68.9 Gpa</td>
</tr>
<tr>
<td>Material Constant</td>
<td>6.762 Gpa</td>
</tr>
<tr>
<td>Strain hardening exponent</td>
<td>0.1</td>
</tr>
<tr>
<td>Strain rate sensitivity</td>
<td>0.5</td>
</tr>
<tr>
<td>Initial Strain rate</td>
<td>1X10^2</td>
</tr>
<tr>
<td>Viscoplastic</td>
<td>Active</td>
</tr>
</tbody>
</table>

As Superplasticity stipulated homogeneous micro grains, as matter of fact, grain growth could not be avoided due to exposing in recrystallisation temperature. Tensile test results indicated the occurrence of sigmoidal curve but finite element came out with a linear relationship logσ and logε.

### 3.2 Finite Element Simulation

3D Deformable models were used to determine the influences of various factors on the Elongation of the model is shown in figure 2.0 deformable view is shown in the figure 4.0, Meshing and visualization view as shown the figure 3.0 and 5.0

### 4.0 Experimental Procedure

Table 1 gives the chemical composition of the Superplastic Al-alloy 6063-t6

<table>
<thead>
<tr>
<th>Si</th>
<th>Fe</th>
<th>Cu</th>
<th>Mn</th>
<th>Mg</th>
<th>Cr</th>
<th>Zn</th>
<th>Ti</th>
<th>Al</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### 4.1 Tensile step strain rate test

A series of superplastic step strain rate tests were conducted to examine the relationship between the stresses and strain rates. Fig 6.0 shows the specimen geometry of tensile test. The tests were performed at the Superplastic temperature of 600°C. All the specimens were coated by boron nitride to avoid the oxidation at high temperatures. A tensile test specimen was deformed at constant cross head velocity until a steady load is registered. An abrupt change of crosshead velocity was performed during the test until a new steady state is registered. Jump tests for Al-alloy were conducted to obtain the superplastic characteristics in the strain rate range from $5 \times 10^{-4}$ to $1 \times 10^{-2}$ s$^{-1}$. The m value was calculated using the following equation:

$$m = \frac{\log(P_2/P_1)}{\log(V_2/V_1)}$$

where $P$ is the tension load and $V$ is the crosshead speed.

### Table 2 Superplastic material parameters used in FEM simulation

<table>
<thead>
<tr>
<th>Material</th>
<th>Specimen No.</th>
<th>Tensile test m</th>
<th>k /MPa. s$^m$</th>
</tr>
</thead>
<tbody>
<tr>
<td>AL6063</td>
<td>1</td>
<td>0.499</td>
<td>193</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>0.585</td>
<td>385</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>0.621</td>
<td>527</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>0.627</td>
<td>578</td>
</tr>
</tbody>
</table>

### 5. Result And Discussion

#### 5.1 FEM (ABAQUS) Result

- FE Simulation Mainly Depends On the accuracy of representation of actual material behavior during forming
- FE Simulation (ABAQUS) many no's of experiment generated and Visualization, to Varying the Process Parameter Such as Strain-rate, Temperature and Pressure.
- FE simulation carried out using Visco plastic material property at temperature using 600°C.

#### 5.2 Experimental Result

FEM (ABAQUS) result and literature survey based on conducted experiment work
6.0 Conclusion

A detailed FE analysis of superplastic forming was carried out for different temperatures, strain rate and pressure conditions. The experimental test were conducted for the different temperatures 560°C, 580°C and 600°C. The obtained values were used in ABAQUS simulation. Different load and strain rate superplastic tests were conducted in ABAQUS. The optimum value was obtained at 2 X 10−3 at 580°C.

REFERENCES

[8] Non-Linear Finite Element Modeling of the Titanium Briquettes Hot Extrusion Process - by Alexey I.Borovkov, Denis V. Shevchenko