PLEASE SCROLL DOWN FOR ARTICLE
Three-Dimensional Nonlinear Finite Element Analysis of Hot Radial Forging Process for Large Diameter Tubes

J. Chen1, K. Chandrashekhara1, V. L. Richards2, and S. N. Lekakh2

1Department of Mechanical and Aerospace Engineering, Missouri University of Science and Technology, Rolla, Missouri, USA
2Department of Materials Science and Engineering, Missouri University of Science and Technology, Rolla, Missouri, USA

A nonlinear coupled finite element model is developed to predict the behavior of large diameter tubes subjected to mechanical and thermal loadings during hot radial forging process. The model is formulated in a three-dimensional (3D) framework to account for both axial and circumferential effects. This model considers both material and geometric nonlinearities. A rate-dependent material model is used to describe the viscoplastic behavior of the workpiece subjected to high temperature and large strain. A tubular workpiece with the mandrel inside and four forging dies outside is modeled in commercial finite element code. A subroutine is developed and implemented to simplify the modeling process for radial forging simulation. Numerical results presented include residual stress, plastic strain, and temperature distribution along the axial and hoop directions in the deformed workpiece. Results are also presented for contact force to evaluate the performance of the die in the forging process. Finite element model predictions are compared with experimental and two-dimensional (2D) axisymmetric simulation results available in literature. A variety of case studies are conducted for hot radial forging process using the developed 3D model.

Keywords Large diameter tube; Mandrel; Radial forging; Three-dimensional finite element model.

INTRODUCTION

Hot radial forging is an open die metal forging process in which the tubular workpiece is deformed between the mandrel and two pairs of simple shaped dies with a series of compressive strokes. Due to the smooth surface finish, considerable material savings, minimum notch effect, preferred fiber structure, and increased material properties, radial forging process is an ideal manufacturing process to produce large cannon barrels, automobile axles and shafts, and other tubular components [1]. For many years, the design of forging process was based on trial-and-error methods. However, factors such as deformation, metal flow, friction between the dies and workpieces, and heat transfer and generation, which are crucial to obtain quality parts, cannot be predicted by trial-and-error techniques. Assessment of process parameters, therefore, are necessary for better understanding the radial forging process and quantitative design and optimization of the process.

Many researchers have studied the radial forging process. Using the slab method, Lahoti and Altman [2] analyzed the mechanical behavior of radial forging of tubes with compound-angle dies. Ghaei et al. [3] studied the effects of die geometry on deformation of a workpiece in the radial forging process. Using the upper bound method, Ghaei et al. [4] and Sanjari et al. [5] predicted the effects of process parameters and the maximum required forging load in the radial forging process. However, the highly simplified slab and upper bound methods offer only a rough means of investigation and cannot accurately predict process parameters. Recently, the finite element method has proved to be a powerful computational tool to analyze material forming processes. Many numerical studies have been performed on the radial forging process. Domblesky et al. [6] and Altan et al. [7] used a two-dimensional (2D) axisymmetric finite element model to investigate the mechanical and thermal behavior of the process. These models do not fully account for temperature-dependent material properties and circumferential effects such as rotational feed and clearance between dies. Jang and Liou [8] studied the residual stress during the radial forging process using a three-dimensional (3D) nonlinear finite element model without considering circumferential motion of the workpiece. Also, the material model considered does not include rate-dependent plasticity behavior and thermal effect.

In the present work, a nonlinear dynamic finite element model has been developed to simulate the hot radial forging process. Circumferential motion of the workpiece and thermal effects on material properties are considered. The elasto-viscoplastic material model is used to account for strain rate dependent plastic behavior. The model uses fully coupled thermal-stress technique to account for stress analysis and heat transfer during the forging process. It is formulated in a 3D coordinate system that accounts for both the axial and circumferential effects. Thermal properties of the material are used to investigate heat transfer during the process. Four hammer dies and...
an inner mandrel are modeled as rigid bodies throughout the simulation, thus reducing the computational cost. The heat generated by plastic deformation, and the friction between hammer dies, inner mandrel, and workpiece, are considered in the model. Heat dissipation due to radiation and convection are also considered in the analysis. The model is developed and implemented in the commercial finite element code ABAQUS, and accounts for contact and material nonlinearities. Due to the lack of experimental data for hot radial forging process of large diameter tubes, the 3D model simulation results for cold radial forging process are verified with the experimental results. Also, simulation results for hot radial forging process are compared with available 2D axisymmetric model results in the literature.

RADIAL FORGING PROCESS

A typical schematic of the radial forging process is shown in Fig. 1. The tubular workpiece is clamped on the machine by grippers. The deformation of the workpiece is formed by the short-stroke of four hammer dies arranged radially around the workpiece. During forging, the rotation of the workpiece is intermittent and synergic with the die motion to prevent the workpiece from twisting [9]. When the hammers are in contact with the workpiece, the rotation is stopped. When the hammers move out of contact with the workpiece, the workpiece rotates by a specific angle to obtain a good surface finish. After each blow of the hammers, the tubular workpiece is fed axially towards the die inlet at a specified rate. Consequently, at each stroke only a small portion of the workpiece is subjected to plastic deformation, thereby a fairly low deformation load is required [10]. This process is repeated till the whole part is manufactured.

The deformation of the workpiece (shown in Fig. 2) can be divided into three typical zones: the sinking zone, the forging zone, and the sizing zone. The diameter of the tubular workpiece is reduced in the sinking zone. The deformation takes place mainly in the forging zone. The sizing zone creates the inner product shape and a good surface finish [11].

MODELING OF HOT RADIAL FORGING PROCESS

Material Model

Because the hot radial forging process produces large deformations and high temperatures, both the mechanical and thermal properties of the tubular workpiece should be considered in the analysis. AISI 4337 steel is used as the material for the tubular workpiece, and this material shows elasto-viscoplastic behavior during the hot forging process. Since the viscoplastic material has a time-dependent behavior, the effect of the strain rate cannot be ignored.

Shida’s formula [12] is based on experimental data obtained from compression tests at high temperatures and high strain rates. This formulation is specifically suited for flow stress measurement [13] and can be applied to carbon steel for a carbon content range of 0.07–1.2%, a temperature range of 700–1200°C, strains up to 0.7, and strain rates up to 100 s⁻¹ [14]. The application of Shida’s formula is convenient in hot radial forging process because experimental test data for plastic behavior of carbon steel at different strain rates and temperatures is not commonly available. For the present study, Shida’s formula is used to describe the flow stress (σ) of AISI 4337 steel as a function of strain (ε), strain rate (ε̇), and temperature (T), and can be expressed as

$$
\sigma = \sigma_d \cdot f_w(\varepsilon) \cdot f_r(\varepsilon) 
$$

where \( f_w(\varepsilon) \) and \( f_r(\varepsilon) \) are functions dependent on strain and strain rate, respectively; \( \sigma_d \) is the deformation resistance function and is expressed as

$$
\sigma_d = 0.28 \exp \left( \frac{5.0}{T} - \frac{0.01}{C_{eq} + 0.05} \right) \quad \text{for } T > T_p 
$$

or

$$
\sigma_d = 0.28 g(C_{eq}, T) \exp \left( \frac{C_{eq} + 0.32}{0.19(C_{eq} + 0.41)} - \frac{0.01}{C_{eq} + 0.05} \right) 
$$

for \( T \leq T_p \),

where

- \( f_w(\varepsilon) = \frac{28}{\varepsilon^2} \exp \left( \frac{5.0}{T} - \frac{0.01}{C_{eq} + 0.05} \right) \)
- \( f_r(\varepsilon) = \exp \left( \frac{C_{eq} + 0.32}{0.19(C_{eq} + 0.41)} - \frac{0.01}{C_{eq} + 0.05} \right) \)
- \( g(C_{eq}, T) \) is a function of carbon equivalent and temperature.
where
\[
C_{eq} = C + \frac{\text{Mn}}{6} + \frac{\text{Cu} + \text{Ni}}{15} + \frac{\text{Cr} + \text{Mo} + \text{V}}{5},
\]
\[
T(K) = \frac{T(\degree C) + 273}{1000},
\]
\[
T_p = 0.95 \frac{C_{eq} + 0.41}{C_{eq} + 0.32}.
\]

and
\[
g(C_{eq}, T) = 30.0(C_{eq} + 0.9) \left( T_{eq} - 0.95 \frac{C_{eq} + 0.49}{C_{eq} + 0.42} \right)^2
\]
\[+ \frac{C_{eq} + 0.06}{C_{eq} + 0.09}.
\]

The strain hardening function \( f_w(\varepsilon) \) is expressed as follows:
\[
f_w(\varepsilon) = 1.3 \left( \frac{\varepsilon}{0.2} \right)^n - 0.3 \left( \frac{\varepsilon}{0.2} \right),
\]
where \( n = 0.41 - 0.07 C_{eq} \).

The strain hardening function \( f_r(\dot{\varepsilon}) \) is expressed as follows:
\[
f_r(\dot{\varepsilon}) = \left( \frac{\dot{\varepsilon}}{10} \right)^m,
\]
where
\[
m = (-0.019C_{eq} + 0.126)T + (0.0176C_{eq} - 0.05)
\]
for \( T > T_p \)

and
\[
m = (0.081C_{eq} - 0.154)T + (-0.019C_{eq} + 0.207)
\]
\[+ \frac{0.027}{C_{eq} + 0.32} \text{ for } T \leq T_p.
\]

Heat conductivity among workpiece, hammer dies, and mandrel, as well as convection and radiation between the hot workpiece and the environment are considered in the simulation. Also, the heat generated by the plastic forging and friction among workpiece, hammer dies, and mandrel is considered in the analysis. Thermal conductivity, thermal expansion coefficient, specific heat, film coefficient, and inelastic heat fraction are required in the material model to investigate the thermal behavior. The stress-strain curves of AISI 4337 at different temperatures based on Shida’s formula are shown in Fig. 3 and the material parameters used in the simulation are listed in Table 1.

**Table 1.—Material parameters for hot radial forging process.**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (kg/m³)</td>
<td>7850</td>
</tr>
<tr>
<td>Young’s modulus (GPa)</td>
<td>200</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.3</td>
</tr>
<tr>
<td>Inelastic heat fraction</td>
<td>0.9</td>
</tr>
<tr>
<td>Conductivity (W/m°C)</td>
<td>15</td>
</tr>
<tr>
<td>Thermal expansion coefficient (1/°C)</td>
<td>1.2 x 10⁻⁵</td>
</tr>
<tr>
<td>Specific heat (J/kg°C)</td>
<td>750</td>
</tr>
</tbody>
</table>
Finite Element Model

Hot radial forging process includes significant heat generation due to the large plastic deformation of the material which, in turn, changes material properties. In addition, the friction and contact conditions generate more heat which depends on the pressure between the surfaces. Hence, a fully coupled temperature-displacement model must be used to solve the thermal and mechanical solutions simultaneously.

Explicit method is used to solve the hot radial forging model. Compared to the implicit method which is more efficient for smooth nonlinear problems, the explicit method is the clear choice for hot radial forging problem which includes large deformation, thermal loading, contact, and material complexity. Also, the explicit solution procedure is suitable for large models involving discontinuous loading steps.

The 3D formulation for dynamic analysis can be written as

\[
[M^e]\{\dot{\Delta}^e\} + [K^e]\{\Delta^e\} = \{F^e\},
\]

where

\[
[M^e] = \int_V \rho[N]^T[N]dV, \quad [K^e] = \int_V [B]^T[C][B]dV,
\]

\[
\{\Delta^e\} = \{u, v, w\}^T,
\]

\[
[M^e] \text{ is the mass matrix, } [K^e] \text{ is the stiffness matrix, } \{F^e\} \text{ is mechanical loading, } N \text{ is shape function, } B \text{ is strain-displacement function, } C \text{ is elasticity matrix, } \rho \text{ is the density, and } \{u, v, w\}^T \text{ are displacement components in a rectangular coordinate system.}
\]

The heat conduction can be expressed as

\[
[C^e]^T[\dot{\theta^e}] + [K^e]^T[\theta^e] = \{Q^e\},
\]

where

\[
[C^e] = \int_V c_p[N]^T[N]dV, \quad [K^e] = \int_V N^T k N dV,
\]

\[
\{Q^e\} = \int_S N^T q dS + \int_V N^T q dV,
\]

\[
[C^e] \text{ is the heat capacitance matrix, } [K^e] \text{ is the conductivity matrix, } \{Q^e\} \text{ is the external flux vector. } c_p \text{ is the specific heat of the material, } k \text{ is the thermal conductivity, } q \text{ is the surface heat flux, and } r \text{ is the body heat flux generated by plastic deformation.}
\]

Combining Eqs. (2) and (3), the coupled thermal-stress equation can be written as

\[
\begin{bmatrix}
M & 0 & 0 \\
0 & C_T & 0 \\
0 & 0 & C_T
\end{bmatrix}
\begin{bmatrix}
\{\dot{\Delta}^e\} \\
\{\theta^e\}
\end{bmatrix}
+ \begin{bmatrix}
K^e \\
K_T
\end{bmatrix}
\begin{bmatrix}
\{\Delta^e\} \\
\{\theta^e\}
\end{bmatrix}
= \begin{bmatrix}
\{F^e\} \\
\{Q^e\}
\end{bmatrix}.
\]

The equations of motion are integrated using the explicit central-difference integration rule

\[
\{\dot{\Delta^e}\}_{(i)} = (M^e)^{-1}((F^e)_{(i)} - (I^e)_{(i)})
\]

\[
\{\Delta^e\}_{(i+\frac{1}{2})} = \{\Delta^e\}_{(i-\frac{1}{2})} + \frac{\Delta t_{(i+1)} + \Delta t_{(i)}}{2} \{\dot{\Delta^e}\}_{(i)}
\]

\[
\{\Delta^e\}_{(i+1)} = \{\Delta^e\}_{(i)} + \Delta t_{(i+1)} \{\dot{\Delta^e}\}_{(i+\frac{1}{2})},
\]

where \(I^e\) is the internal force vector and is given by \((I^e)_{(i)} = [K^e]\{\Delta^e\}_{(i)}\).

The subscript \(i\) refers to the increment number in an explicit dynamics step. A lumped mass matrix is used because its inverse is simple to compute.

The heat transfer equations are integrated using the explicit forward-difference time integration rule

\[
\{\dot{\theta^e}\}_{(i)} = [C^e]^T((Q^e)_{(i)} - (I^e)_{(i)}T)
\]

\[
\{\theta^e\}_{(i+1)} = \{\theta^e\}_{(i)} + \Delta t_{(i+1)} \{\dot{\theta^e}\}_{(i)},
\]

where \((I^e)_{(i)}\) is the internal heat flux vector, and is given by \((I^e)_{(i)} = [K_T^e]\{\theta^e\}_{(i)}\).

Hard contact is used to describe the normal contact to the surfaces. The friction of friction work converted to heat is defined in the contact model. A Coulomb friction model is used to describe the tangential interaction of contacting surfaces and is given by

\[
\tau_{crit} = \mu p,
\]

where \(\tau_{crit}\) is the critical friction force, \(\mu\) is the coefficient of friction, and \(p\) is the contact pressure between the two surfaces.

Numerical simulation

A 3D thermomechanical finite element model is developed using ABAQUS/CAE. The dimensions of the tube and mandrel used in the present study are taken from Domblesky et al. [6]. A circular die shape with multiple die angles is used to reduce the forging force and make the process more cost-effective [4]. The geometric parameters are summarized in Table 2. During the forging process, the rotation of the tubular workpiece is intermittent and synergic with the die motion. The stroking rate of the hammer dies is 200 per minute, and the stroking velocity is based on

<table>
<thead>
<tr>
<th>Table 2.—Geometry of workpiece, mandrel and die.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Out diameter of the original workpiece (mm)</td>
</tr>
<tr>
<td>Inner diameter of the original workpiece (mm)</td>
</tr>
<tr>
<td>Out diameter of the forged part (mm)</td>
</tr>
<tr>
<td>Inner diameter of the forged part (mm)</td>
</tr>
<tr>
<td>Out diameter of mandrel (mm)</td>
</tr>
<tr>
<td>Die total length (mm)</td>
</tr>
<tr>
<td>Die land length (mm)</td>
</tr>
<tr>
<td>Die inlet angle</td>
</tr>
<tr>
<td>Die outlet angle</td>
</tr>
</tbody>
</table>
Table 3.—Parameters for the motion of simulation.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stroking rate (strokes per minute)</td>
<td>200</td>
</tr>
<tr>
<td>Rotational feed</td>
<td>15°</td>
</tr>
<tr>
<td>Number of strokes performed before an axial feed</td>
<td>12</td>
</tr>
<tr>
<td>Axial feed rate after each circle stroke (mm)</td>
<td>20</td>
</tr>
<tr>
<td>Number of axial feeds</td>
<td>3</td>
</tr>
</tbody>
</table>

The sinusoidal function. The chuck-heads remain stationary when the dies are in contact with the tube during the stroke. After each stroke, the tube is automatically rotated by 15°. The strokes continue until a good surface finish is obtained in the circumferential direction. The tube is fed 30 mm axially towards the die inlet at a specified feed-rate to start a new stroke cycle. Parameters of the motion are shown in Table 3. The contact parameters are listed in Table 4. The 8-node temperature-displacement coupled brick element is used to mesh the tubular workpiece. To avoid shear-locking associated with this element, a reduced integration strategy with hourglass control is used. The die and mandrel are meshed using rigid elements since both undergo very little deformation compared to the deforming workpiece. The number of elements for each part is shown in Table 5. The mesh of different parts is shown in Fig. 4.

Table 4.—Contact parameters.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Friction factor between workpiece and die</td>
<td>0.6</td>
</tr>
<tr>
<td>Friction factor between workpiece and mandrel</td>
<td>0.5</td>
</tr>
<tr>
<td>Fraction of dissipated energy caused by friction</td>
<td>0.9</td>
</tr>
<tr>
<td>Friction of converted heat distributed to slave surface</td>
<td>0.5</td>
</tr>
</tbody>
</table>

Table 5.—Type and number of elements used for different parts.

<table>
<thead>
<tr>
<th>Type (ABAQUS)</th>
<th>Number of elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tubular workpiece</td>
<td>C3D8RT (8-node thermally coupled brick, trilinear displacement and temperature, reduced integration, hourglass control)</td>
</tr>
<tr>
<td>Die</td>
<td>R3D4 (4-node 3D bilinear rigid quadrilateral)</td>
</tr>
<tr>
<td>Mandrel</td>
<td>R3D4 (4-node 3D bilinear rigid quadrilateral)</td>
</tr>
</tbody>
</table>

The implementation procedure of the developed model is summarized as shown in Fig. 5. As the die stroke, workpiece rotation, and workpiece axial feed are alternate in the radial forging process, many simulation steps are necessary to simulate the process. Also, the boundary conditions change among the various simulation steps resulting in extensive modeling work. To simplify the modeling procedure, a subroutine has been developed in MATLAB to generate the input file for ABAQUS. This technique significantly reduces modeling time thus making the modeling process more efficient.

RESULTS AND DISCUSSION

Residual stress distribution is an important parameter because it directly affects the fatigue life of the workpiece and its dimensional stability. A large residual stress field
can result in propagation of a crack, causing the workpiece to fail. Figure 6 shows the maximum principal stress distribution after three axial feeds (36 strokes). The tensile residual stress on the inner surface and the portion of the outer surface that are in contact with the hammer dies and the mandrel, can help the propagation of possible crack and hence reduce the product life. The plastic strain distributions shown in Fig. 7 are mainly at the forging and sizing zones. No plastic deformation is observed away from the forging zone.

**Comparison with Experimental Results of Cold Radial Forging Process**

The current 3D finite element model is verified with experimental results of cold radial forging process presented by Uhlig [15]. This problem is also studied by Ameli [10]. The elastoplastic material properties for workpiece used are $E = 203 \text{ GPa}$, $\nu = 0.29$, $\sigma_Y = 200 \text{ MPa}$. A power law is used for strain hardening, and the parameters for the power law are $k = 750 \text{ MPa}$, $n = 0.2$. Geometry of workpiece and the contact force from experimental measurement are listed in Table 6. The contact forces obtained from the current finite element model are compared with experimental results in Fig. 8. The simulation results agree well with experimental findings.

**Comparison with 2D Axisymmetric Results for Hot Radial Forging Process**

Due to the lack of available experimental data on hot radial forging process, the accuracy of the present 3D model is compared with the available 2D axisymmetric results. Table 7 shows comparison of 3D model results with the 2D axisymmetric results for hot radial forging. The maximum forging temperature based on the current model is higher than those predicted by the 2D axisymmetric model. This is because the heat generated by the friction between mandrel...
Three-Dimensional Nonlinear Finite Element Analysis

Table 6.—Geometry of workpiece and contact force from experiment in the cold radial forging process [15].

<table>
<thead>
<tr>
<th>Sample No.</th>
<th>Outer radius of preform (mm)</th>
<th>Inner radius of preform (mm)</th>
<th>Outer radius of product (mm)</th>
<th>Inner radius of product (mm)</th>
<th>Contact force (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>15.97</td>
<td>5.8</td>
<td>13.18</td>
<td>3.915</td>
<td>172.00</td>
</tr>
<tr>
<td>2</td>
<td>15.97</td>
<td>5.8</td>
<td>13.25</td>
<td>3.915</td>
<td>167.00</td>
</tr>
<tr>
<td>3</td>
<td>15.03</td>
<td>5.8</td>
<td>13.11</td>
<td>3.915</td>
<td>124.00</td>
</tr>
<tr>
<td>4</td>
<td>13.99</td>
<td>5.8</td>
<td>13.03</td>
<td>3.915</td>
<td>74.00</td>
</tr>
</tbody>
</table>

Axial Deformation

After the first axial feed, the plastic strain distribution along the axial direction in the inner, middle, and outer areas of the workpiece is shown in Fig. 9. Three deformation zones (sinking, forging, and sizing) exist during the forging process. Only a small amount of deformation occurs in the sinking zone. In the forging zone, the deformation increases, and it reaches the maximum value in the sizing zone. Compared to the middle surface, the end of the tube on the outer and inner surfaces has significant deformation.

Figure 10 shows the maximum principal stress distribution of the inner, middle, and outer surfaces along the axial direction after the first axial feed. The outer and inner surfaces show higher residual stress than the middle surfaces. The largest residual stress occurs in the outer surface. There is no significant variation of residual stress along the axial direction in the middle surface. The comparison of residual stresses among the three surfaces indicates that the outer surface may pull the material, causing it to crack and fail first.

Circumferential Deformation

A 3D finite element model is required to simulate the circumferential deformation of hot radial forging process. The cross-section of the deformed tubular workpiece at different strokes is shown in Fig. 11. After the first stroke, the deformation is mainly concentrated at forging area of the tube where it is in contact with the hammer dies. Moreover, there are small gaps between the mandrel and the inner surface of the tube. After the fourth stroke, no gap is found between the mandrel and the inner surface of the tube, and a good inner surface has been formed. However, some protuberances are found at the outer surface. After the twelfth stroke, no gap is found, and a round outer surface is obtained. Both inner and outer surfaces are well finished.

Figure 12 shows the plastic strain distribution on outer surface along the hoop direction at various specific strokes.
It is obvious that the plastic strain increases as the number of strokes increases. Initially, the strain curve plotted against the hoop angle shows variation due to the deformation along the circumference of the tube caused by the four hammers. As the number of strokes increases, the variation reduces significantly. After the ninth stroke, there is less variation in the strain distribution, resulting in a good finish on the outer surface.
The residual maximum principal stress distributions on the various surfaces after the first stroke are shown in Fig. 13. The variation of the curve indicates the changes in deformation force in the circumference direction. This conforms to the variation of strain curve at the first stroke. In comparison to the middle and inner surfaces, the outer surface undergoes the highest residual stress. The maximum residual stress occurs at the area in contact with the die on the outer surface after the first stroke. Figure 14 shows the contact force between the die and the tubular workpiece for the first twelve strokes before the first axial feed. The contact force during the hot radial forging process is in the range of 2600 to 3200 KN.

**Temperature Distribution**

The temperature distribution caused by plastic deformation and friction after three axial feeds is shown in Fig. 15. The temperature on the outer surface does not change too much. This is because the heat generated on outer surface by plastic forging and friction between the tube and hammer dies is quickly transferred to the environment by convection and radiation. On the inner surface, however, the temperature increases to about 250°C, because the heat generated by friction between the mandrel and the workpiece does not dissipate easily inside the tube.

Hot radial forging experiments are expensive and time consuming. Simulation results using a comprehensive 3D model is a cost-effective tool to fully understand the various aspects of hot radial forging process. The prediction of residual stress distribution indicates that the outer surface of the tube is most prone to fatigue failure. Also, the axial plastic strain distribution shows the existence of three deformation zones during hot radial forging process. Furthermore, the circumferential deformation results from the 3D model can be used to determine the number of stroke for good surface finish. The predicted contact force and temperature distribution can also be used as important parameters for the design of radial forging dies.

**Conclusion**

A comprehensive 3D finite element model has been developed to analyze the behavior of the tubular workpiece subjected to mechanical loading and heat transfer during the hot radial forging process. The model considers both axial and circumferential effects of the forging process. Shida’s constitutive equation is used to describe the flow stress of the workpiece material during hot radial forging deformation. A rate-dependent elasto-viscoplastic material model has been implemented to accurately predict the plastic deformation of the tubular workpiece at high temperatures. The model is formulated in the 3D coordinate system using the commercial software ABAQUS. A subroutine is developed to generate the input file to simplify the modeling work. The explicit method is used to solve the coupled thermo-mechanical...
model. Analysis of a typical case is conducted using the developed model. Residual stress, plastic strain, contact force, and temperature along the axial and hoop directions are reported. Finite element model predictions are in good agreement with experimental and 2D axisymmetric simulation results available in literature. Results for circumferential deformation are presented, which cannot be predicted by using 2D axisymmetric model. The model developed here can be used to accommodate various cases of hot radial forging process. In addition, this model can be expended for the cold radial forging process by changing the mechanical loading, thermal condition, and material properties.

ACKNOWLEDGMENT

We would like to thank U.S. Army Benet Labs for funding this research. The conclusions and opinions expressed are those of the authors and not of Benet Laboratories.

REFERENCES


