CHAPTER 2

FINITE ELEMENT MODELLING AND SIMULATION OF HOT EXTRUSION PROCESS AND EQUAL CHANNEL ANGULAR PRESSING

2.1 Introduction

The goal in product development and manufacturing is to determine the optimum means of producing quality products in a cost effective manner. Metal forming is one of the widely used manufacturing processes in industries. Finite element modeling is the most popular method of simulating the forming processes and is an industry standard practice. As discussed in chapter 1, sections 1.5, Finite Element Modeling (FEM) of metal forming processes have become important tools for designing feasible production processes because of the unique capability of FEM to describe the complex geometry and boundary condition of the metal forming process. FEM simulation has become a practical and efficient tool for prediction of performance in industrial forming operations. Modeling techniques for metal forming have continuously evolved and improved and a considerable effort was devoted for process design and optimization. This progress can be assessed by reviewing the literature [Kob84, 85a, 85b, Doh87, Che92, Deme93, Den94, Doe94, Esp94, Owe95, Fou96c, Fou97, Fou98, Gel98, Bar00, Chou00, Chun03, Bal01, Bor02, Ant02, Ari03, Do04, Bon05a, Bon05b, Flis05, Ach06, Poh07, Bon08, Sch09, Hol10, Ber11]. The finite element codes for metal forming have the following characteristics:

- realistic regarding material constitutive equations;
- complete by taking into account various coupling effects, e.g.: thermo-mechanical coupling;
- user friendly with automatic meshing and re-meshing capabilities;
- reliable by introduction of error estimation and adaptive re-meshing.

Numerical simulation provides extensive information like material flow, tool stresses, plastic strain and elastic spring back effect, information that is difficult to obtain otherwise. It allows process optimization in a user friendly way, for instance to modify the tool geometry in order to decrease the required forging force and to increase the tool life. Also there is great interest to use Finite Element Modeling
(FEM) to analyze microstructure besides mechanical and thermal effects. The block diagram shown in Fig. 2.1 presents all the interactions between various phenomena like stress, strain, temperature and microstructure. FEM programs can predict large deformations and thermally influenced material flow in metal forming. Thus, they can provide detailed information about the forming mechanics. Some recent research and industrial applications are given in the literature [Sha08, Hak09a, Chen10, Kri10, Wei11, Shu11, Shub11].

![Fig. 2.1: Interactions between stress, strain, temperature and microstructure](image)

FE based simulation of the metal forming process on the computer have been used by the forming industry primarily for providing the comprehensive details of following:

1. To establish the kinematic relationships (velocities and strain rates) between the un-deformed part (billet or preform) and the deformed part (product), i.e. predict metal flow during the forming operation.

2. To establish the limits of formability and producibility, i.e. determine whether it is possible to perform the forming operation without causing any surface or internal failures (under fill, lap formation, cracks and folds) in the deforming material.

3. To predict the stresses, forces and the energy necessary to carry out the forming operation. This information is necessary for tool design and for
selecting the appropriate equipment, with adequate force and energy capabilities, to perform the forming operation.

4. To analyze die stress for improving the die life.

5. To model process control (e.g. press speed, forging temperature, lubricant, and heat treatment procedure) for ensuring that the resulting microstructure properties (such as grain size) meet the requirement.

6. Non-steady-state deformation can be treated by FE simulations.

7. To predict temperature increase in the workpiece due to deformation and further predict the heat transfer between the workpiece and the dies. This can help to predict tool wear.

8. FE simulations can be used to conduct heat treatment simulations such as quenching, carburising etc., which change the microstructure and material properties according to the requirements of the finished product.

9. FE simulations are more efficient and cheaper than expensive trial and error experiments to study the effects of various process parameters on the forging process. It is easier to evaluate alternative designs on the computer rather than doing the same experimentally.

10. The FE simulation is used to study metal flow to ensure that the die cavity is completely filled.

2.2 Issues in the application of process simulation

2.2.1 Geometry
The first step in simulating any metal forming process is analyzing the geometry of the workpiece. If the simulation is 2-D then one has to decide whether to simulate it as axisymmetric or plane strain. The geometry can either be drawn in the preprocessor of the FE code or imported from solid modeling software.

2.2.2 Mesh and remesh
The geometries to be studied have to be meshed with a sufficient number of regular elements. There is a direct correlation between the number of elements and the accuracy of the simulation, however a large number of elements will also increase computing time. Hence one needs to reach a compromise between the two. The
number of elements in areas of interest should be high such as areas of high stress concentration, corners etc. It is a general rule that there should be at least three elements at any radius of the geometry. Remeshing is required when large strains are encountered and the elements get highly distorted.

2.2.3 Workpiece and tool material properties
Another preliminary step in setting up a simulation is selecting the material models for the objects involved in the process. The various options are rigid, elastic-plastic, rigid-plastic, viscoplastic etc. The choice of the material model depends on the role of the object to be studied in the actual process e.g. in die stress analysis the dies would be specified as elastic to study the elastic deformation and stresses induced during processing. The flow stress data can either be selected from the material data library or can be user defined. It should be selected depending upon the operating temperature since for hot forming operations the flow stress is dependent on temperature and strain rate. This is a very crucial step since the flow stress defines the material behavior during the forming process.

2.2.4 Boundary conditions
These include the relative positioning of the objects, movement, loading conditions, interface friction etc. Proper constraints need to be specified depending upon the process requirement. The various interface frictions affect the metal flow and should be appropriately specified. Usually in bulk forming operations we use the shear coefficient based on the constant shear stress law.

2.2.5 Characteristics of the code
FE codes can be either implicit or explicit. Implicit codes are slower but more accurate than the faster explicit type.

2.3 Finite Element Modelling
Finite Element Method (FEM) technique can be applied in analysis and simulation of metal forming processes for a large range of boundary value problems with little restriction on workpiece geometry. The ultimate advantage of computer aided design and analysis in metal forming processes is achieved when reasonably accurate and
inexpensive computer software is available for simulating metal flow throughout a particular metal forming process.

In practical metal forming processes, a number of operations (preform steps) are required to transform the initial billet geometry into a complex final geometry while achieving desired tolerances and properties. The desired workpiece shapes are obtained by using dies of various shapes. Therefore, a method of analysis that can treat the boundary conditions of dies is necessary in order to fully utilize the advantage of the finite element method in metal forming analysis.

Numerous commercial FEM codes that are available which are used extensively in simulating metal forming process analysis are specified in table 2.1 metal forming processes. The user is provided with a library of data containing material properties and options for different material models (rigid-plastic, elastic-plastic, viscoplastic etc.).

<table>
<thead>
<tr>
<th><strong>FORGE</strong></th>
<th>Bulk metal forming</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>DEFORM</strong></td>
<td>Bulk metal forming</td>
</tr>
<tr>
<td><strong>LS-DYNA</strong></td>
<td>Metal forming etc.</td>
</tr>
<tr>
<td><strong>SUPERFORGE</strong></td>
<td>Bulk metal forming</td>
</tr>
<tr>
<td><strong>PAM-STAMP</strong></td>
<td>Stamping simulation and analysis</td>
</tr>
</tbody>
</table>

Table 2.1: Commercial FEM Software packages for analysis and simulation of metal forming processes.

Computer implementation of the basic steps in a standard finite element analysis consists of three distinct units (fig. 2.2):

- Preprocessor
- Finite element analysis
- Postprocessor
Fig. 2.2: Implementation of FEA for metal forming analysis.
2.3.1 Preprocessor
This operation precedes the analysis operation. It takes in minimal information from the user (input) to generate all necessary problem parameters (output) required for a finite element analysis.

*Input includes:* Information on the solid model, discretization requirements, material identification and parameters, and boundary conditions.

*Output includes:* Coordinates for nodes, element connectivity and element information, values of material parameters for each element and boundary loading conditions on each node.

In this step, a continuum is divided into a finite number of sub-regions (or elements) of simple geometry (triangles, rectangles etc.). Key points are then selected on elements to serve as nodes where problem equations like equilibrium and compatibility are satisfied.

2.3.2 Finite Element Analysis
This is the main operation in the finite element technique. The output from the preprocessor (nodal coordinates, element connectivity, material parameters and boundary/loading conditions) serves as the input to this module. It establishes a set of algebraic equations, which are to be solved, from the governing equations of the boundary value problem. The governing equations include conservation principles, kinematic relations, and constitutive relations. The main program then solves the set of linear algebraic equations to obtain the state variables at the nodes. It also evaluates the flux quantities inside each element. The following steps are pursued in this operation:

- A suitable interpolation function is assumed for each of the dependent variables in terms of the nodal values.
- Kinematic and constitutive relations are satisfied within each element.
- Using work or energy principles, stiffness matrices and equivalent nodal loads are established.
Equations are solved for nodal values of the dependent variables.

2.3.3 Postprocessor
This operation prints and plots the values of state variables and fluxes in the meshed domain. Reactions may be evaluated. Output may be in the form of data tables or as contour plots.

2.3.4 The FE Method
The procedure of Finite Element (FE) method is described as follows:

- Geometry of the workpiece (and dies in some cases) is meshed into a number of elements. Dies are meshed only if die stress/wear analysis is required.
- Elements store data such as amount of deformation (related to work hardening).
- Nodes are points, which connect elements. Nodes store variables such as temperature and contact between the workpiece and tools. More elements: longer simulation time.
- Boundary conditions (e.g. velocity, force, temperature) are applied to the surface nodes.
- Material models are applied to elements.
- Forming equipment models are applied to dies.

2.3.4.1 Inputs to FE Model
- Workpiece material properties
- Processing conditions (preform & tool geometries, die material, temperature)
- Friction (lubrication)

2.3.4.2 Outputs from FE Model
The outputs are workpiece shape, die fill, forming load and energy, temperature, spring back, mechanical properties, various die stresses, isocontours of stress, strain, strain rate etc.
FEM simulations of metal forming processes can be classified into two categories i.e. elastic-plastic and rigid-plastic analysis. In the elastic-plastic simulation the results are given for not only the plastic deformation but also for elastic deformation like residual stresses and springback. Rigid-plastic simulation assumes material to deform only plastically and in comparison to elastic-plastic simulations, it results in shorter computing time and is suitable for large deformations. Minimizing the time and cost of production trials and design iterations continues to be the goal of FEM development. Input and output parameters for FE simulation of the forging process is shown in Fig. 2.3.

![Fig. 2.3: Input and output parameters for FE Simulation of the forging process](image)

The finite element modelling and simulation of hot extrusion process and equal channel angular pressing is attempted in FORGE-3 environment. It is a software environment which can be used for both 2D and 3D bulk metal forming modelling.

FORGE-3 is domain specific software designed to simulate hot, warm and cold forging. It can perform rigid-viscoplastic analysis which neglect the elastic
deformation of the workpiece and elasto-plastic analysis which include elastic effects. A rigid-viscoplastic analysis is applicable to hot forging where elastic effects are insignificant compared to thermal effects and the large plastic deformations involved. Elasto-plastic analysis is applicable to cold forging. Elasticity is included because elastic effects such as residual stresses and springback can become significant in cold forging. It is designed specifically for the analysis of hot and cold metal forming. This means that key phenomena such as plasticity, viscoplasticity, friction and thermo-mechanical linkage are incorporated in the code. Automatic remeshing with 6 node isoparametric elements for 2D analysis and 10 node isoparametric elements for 3D analysis are incorporated in the software. Remeshing becomes essential to avoid catastrophic element distortion when analysing large scale deformations which occur in industrial closed die hot forging operations.

For rigid-viscoplastic simulations, FORGE-3 uses a viscoplastic formulation based on the Norton-Hoff [Nor29, Hof54] law to model the friction between die and workpiece. Its elasto-plastic solver provides three alternative models of friction, Coulomb, Tresca and the viscoplastic law used in rigid-viscoplastic simulations.

In a FORGE-3 simulation the workpiece is represented by a 3D finite element mesh. The dies, which are assumed to be rigid, are defined by 3D drawings. Mechanical and thermal properties are assigned to the mesh and constants which define thermal and mechanical (i.e. friction) effects are defined at the surface of the workpiece. Once initial temperature conditions are given, the dies are moved in small incremental steps and thermal and mechanical solutions are calculated at each step. Since the mechanical properties of the metal are dependent on its temperature and the thermal solution depends on the geometry and heat generated by friction and plastic work of the material, the two solutions are interdependent. This linkage is managed as: At any given increment the temperature field is known. The mechanical solution is obtained using the mechanical properties for this temperature field. The workpiece is then deformed incrementally according to the strain rates calculated at each node and the thermal analysis to produce a new temperature field is performed simultaneously.
FORGE is used extensively for research and is also being applied to various industrial forging operations [Alv02, Gup02, Tie02, Abd03, Dua04, Gup04a, Sri04a, Pen04, Mil04, Gup05, Gup04b, Gyl05, Hei05, Gup06, Kis06, Mad07, Wal11].

2.4 Mathematical formulation of metal forming processes
The widely used approaches of mathematical formulation of metal forming process are presented here i.e. (i) the elasto-plastic (ii) the elasto-viscoplastic and (iii) the flow formulations. The general finite element discretization and time integration schemes are described. A master and slave discrete formulation for approximate coupling of part and tool with non co-incident meshes is also analysed.

2.4.1 Elasto-plastic and elasto-viscoplastic behavior
For a more accurate description of the material behavior, the elastic effects must be introduced. The additive decomposition of the strain rate tensor can be considered as a satisfactory approximation for usual metallic materials:

\[ \dot{\varepsilon} = \dot{\varepsilon}^e + \dot{\varepsilon}^p \]  

Where, \( \dot{\varepsilon}^e \) is elastic part of the strain rate tensor, \( \dot{\varepsilon}^p \) is the plastic (or viscoplastic) part of the strain rate tensor.

The material behavior is independent of strain rate and elastic deformation is included. This model, shown in figure 2.4, applies to most metals at room temperature.
The elastic contribution obeys a rate form of the Hooke’s Law:

\[
\frac{d\sigma}{dt} = \lambda \text{trace}(\dot{\varepsilon})I + 2\mu \dot{\varepsilon} = D : \dot{\varepsilon}
\]

where \(\lambda\) and \(\mu\) are the usual Lame’s coefficients, \(I\) is the unit (second rank) tensor, \(D\) is the resulting fourth rank elastic tensor. The Juman derivative of the stress tensor in Eqn. 2.2 is defined by:

\[
\frac{d\sigma}{dt} = \dot{\sigma} - \omega \sigma + \sigma \omega
\]

where \(\omega\) is the spin tensor. When the process involves moderate rotations, the simple material derivative can be used in Eqn. 2.2 as a satisfactory approximation.

For an elastoplastic material, the irreversible plastic deformation is computed with the help of the Von Mises plastic yield function, \(f\), defined by:

\[
f(\sigma) = 3 \sum_{ij} \sigma_{ij}'^2 - 2\sigma_0^2(\varepsilon)
\]

Two regimes must be distinguished:

**2.4.1.1 Deformation is purely elastic**

\(\dot{\varepsilon}^p = 0\) if \(f(\sigma) < 0\),

or if \((f(\sigma) = 0\) and \(\frac{\partial f}{\partial \sigma} : \dot{\sigma} < 0\)

---

**Fig. 2.4: Elastic–plastic flow model**

The elastic contribution obeys a rate form of the Hooke’s Law:

\[
\frac{d\sigma}{dt} = \lambda \text{trace}(\dot{\varepsilon})I + 2\mu \dot{\varepsilon} = D : \dot{\varepsilon}
\]

where \(\lambda\) and \(\mu\) are the usual Lame’s coefficients, \(I\) is the unit (second rank) tensor, \(D\) is the resulting fourth rank elastic tensor. The Juman derivative of the stress tensor in Eqn. 2.2 is defined by:

\[
\frac{d\sigma}{dt} = \dot{\sigma} - \omega \sigma + \sigma \omega
\]

where \(\omega\) is the spin tensor. When the process involves moderate rotations, the simple material derivative can be used in Eqn. 2.2 as a satisfactory approximation.

For an elastoplastic material, the irreversible plastic deformation is computed with the help of the Von Mises plastic yield function, \(f\), defined by:

\[
f(\sigma) = 3 \sum_{ij} \sigma_{ij}'^2 - 2\sigma_0^2(\varepsilon)
\]

Two regimes must be distinguished:

**2.4.1.1 Deformation is purely elastic**

\(\dot{\varepsilon}^p = 0\) if \(f(\sigma) < 0\),

or if \((f(\sigma) = 0\) and \(\frac{\partial f}{\partial \sigma} : \dot{\sigma} < 0\)

---

2. 12
2.4.1.2 Deformation is elastoplastic

\[ \dot{\varepsilon}^p = \lambda_p \frac{\partial f}{\partial \sigma} \text{ if } (f(\sigma) = 0 \text{ and } \frac{\partial f}{\partial \sigma}; \sigma \geq 0) \quad \ldots 2.6 \]

where, the plastic multiplier \( \lambda_p \) is positive.

When the stress field depends on the strain rate, the irreversible strain rate is expressed with viscoplastic law:

\[ \dot{\varepsilon}^p = \frac{1}{K} \left( \frac{\sigma_{eq} - R}{K} \right)^{1-m} \sigma' \quad \ldots 2.7 \]

where the bracket function is defined by:

\[ \langle x \rangle = 0 \text{ if } x = 0 \text{ and } \langle x \rangle = x \text{ if } x \geq 0 \quad \ldots 2.8 \]

the equivalent stress by:

\[ \sigma_{eq} = \sqrt{\frac{3}{2} \sum_{ij} \sigma_{ij}^2} \quad \ldots 2.9 \]

and \( R \) is a function of \( \bar{\varepsilon} \).

For materials exhibiting kinematic hardening, Eqn. 2.7 and Eqn. 2.9 must be generalized, they can be replaced respectively by:

\[ \dot{\varepsilon}^p = \frac{1}{K} \left( \frac{\sigma_{eq} - R}{K} \right)^{1-m} (\sigma' - \beta) \quad \ldots 2.10 \]

and

\[ \sigma_{eq} = \sqrt{\frac{3}{2} \sum_{ij} (\sigma_{ij}^2 - \beta_{ij})^2} \quad \ldots 2.11 \]

where, \( \beta \) is the back stress tensor, which is governed by a differential law of hardening in term of the strain rate tensor.

2.4.2 Unilateral contact and friction behavior

We define the velocity difference at the interface \( \partial \Omega_c \) between the part and tools by:

\[ v_s = v - v_{\text{tool}} \quad \ldots 2.12 \]

The contact condition is written:

\[ v_s.n = 0 \text{ if } (\sigma.n).n = 0 \quad \ldots 2.13 \]

while for a loss of contact:

\[ v_s.n = 0 \text{ if } (\sigma.n).n = 0 \quad \ldots 2.14 \]
where, ‘n’ is the normal to the contact surface, which is outside the work piece.

The viscoplastic friction behavior is described by a non-linear relation between the shear stress vector, $\tau$ and the sliding velocity, $v_s$:

$$\tau = -\alpha(\sigma)K|v_s|^{q-1}v_s$$

where,

$\alpha$ = the viscoplastic friction coefficient, can be a function of the normal stress $\sigma_n$

$q$ = the sensitivity parameter to the sliding velocity (which is often taken equal to $m$).

It is easy to convince oneself that this value includes the Coulomb special case.

### 2.4.2.1 Coulomb Friction.

Friction is proportional to the stress normal to the surface of the workpiece. The proportionality is defined by a coefficient of friction which is assumed to be constant throughout the metal forming operation.

$$\tau_f = \mu\sigma_n$$

where,

$\tau_f$ = friction stress tangential to the surface

$\mu$ = coefficient of friction

$\sigma_n$ = compressive stress normal to the surface (contact pressure)

This law is not suitable to bulk metal forming because high contact pressures are involved. At high contact pressures the Coulomb law predicts friction stresses greater than the shear strength of the metal and sticking rather than sliding is modeled at the interface. Coulomb friction is more applicable to sheet metal forming where surface pressures are lower.

### 2.4.2.2 Tresca Friction.

Friction stress at the contact surface is equal to a fraction of the shear yield stress of the workpiece material. It is calculated using a constant friction factor.

$$\tau_f = mk$$

where,

$m$ = friction factor

$k$ = shear yield stress of material
This law is suitable to bulk metal forming because, unlike Coulomb friction, the amount of friction is independent of the normal stress at the surface.

2.4.3 Thermal Analysis

The classical heat equation for deformable bodies is written simply:

$$\rho c \frac{dT}{dt} = \text{div} \left( k \text{grad}(T) \right) + \dot{q}v$$

Where, the heat dissipation represents a fraction $f_w$ (with $0 \leq f_w \leq 1$) of the irreversible mechanical rate of work:

$$\dot{q}v = f_w \sigma : \dot{\varepsilon}$$

and for viscoplastic materials it is:

$$\dot{q}v = f_w K (\sqrt[3]{\dot{\varepsilon}})^{m-1}$$

The constitutive law depends on temperature, for example we put in Eqn.2.3:

$$K = K_0 (\epsilon_0 + \bar{\epsilon})^m \exp(\beta x p), \quad m=m_0 + m_1 T$$

For viscoplastic materials, the incompressibility equation is replaced by the mass conservation condition, taking into account linear thermal dilation with coefficient $\alpha_d$:

$$\text{trace}(\dot{\varepsilon}) = \text{div}(v) = 3 \alpha_d \dot{T}$$

The radiation condition on the free surface $\partial \Omega_s$ is written:

$$-k \frac{\partial T}{\partial n} = \epsilon_r \sigma_r (T^4 - T_0^4)$$

where,

$\epsilon_r$ = emissivity parameter,

$\sigma_r$ = Stephan constant and

$T_0$ = outside temperature.

On the surface of contact $\partial \Omega_c$, due to conduction with the tools with temperature $T_{\text{tool}}$, and friction energy dissipation governed by the effusivity of the part, $b$, and of the tool, $b_{\text{tool}}$, the condition is:

$$-k \frac{\partial T}{\partial n} = h_{cd} (T - T_{\text{tool}}) + \frac{b}{b + b_{\text{tool}}} \alpha_i K |\Delta V|^{n+1}$$

In fact the parameter $h_{cd}$, governing the exchanges between part and tools, is a function of the normal stress, $\sigma_n$, which must be determined experimentally.
2.4.4 Integral formulations

The viscoplastic equations can be transformed into an integral mixed formulation for any virtual velocity field \( v^* \):

\[
\begin{align*}
\mathbf{r}^v(x, \bar{\varepsilon}, v, p) &= \int_{\Omega} 2K(\sqrt{3\varepsilon})^{m-1}\dot{\varepsilon}^* \, dV + \int_{\partial\Omega} \alpha_i K \left| v_i \right|^{p-1} v_i v^* \, dS - \int_{\Omega} p \text{div}(v^*) \, dV = 0 \ldots \text{2.25}
\end{align*}
\]

and for any virtual pressure field \( p^* \):

\[
\begin{align*}
\mathbf{r}^p(x, v, p) &= \int_{\Omega} p^* \text{div}(v) \, dV = 0 \quad \ldots \text{2.26}
\end{align*}
\]

Corresponding integral formulations, in terms of the velocity field, can be derived for elastoplastic or elastic viscoplastic behavior as shown in Chenot [Che98a, b]. But most often a displacement formulation is preferred for simplicity.

2.4.5 Free tool Problem

We suppose that, in addition to the tools with a prescribed velocity \( v_{\text{tool}} \), there is one (or several) free tool with an unknown velocity, \( v_{\text{free}} \), which is determined by the contact condition with the work-piece on the boundary \( \partial\Omega_{\text{free}} \). We must impose the non-penetration condition on \( \partial\Omega_{\text{free}} \):

\[
(v-v_{\text{free}}).n = 0 \quad \ldots \text{2.27}
\]

when there is contact. When a penalty formulation, with penalty factor \( \rho_p \) is chosen for imposing the contact conditions given by Eqn. 2.15 and Eqn. 2.16, the integral equation becomes:

\[
\begin{align*}
\mathbf{r}^v(x, \bar{\varepsilon}, x_{\text{free}}, v, p, v_{\text{free}}) &= \int_{\Omega} 2K(\sqrt{3\varepsilon})^{m-1}\dot{\varepsilon}^* \, dV + \int_{\partial\Omega} \alpha_i K \left| v_i \right|^{p-1} v_i v^* \, dS - \int_{\Omega} p \text{div}(v^*) \, dV \\
&+ \rho_p \int_{\partial\Omega_{\text{free}}} (v - v_{\text{free}}).n)(v^* .n) \, dS = 0 \quad \ldots \text{2.28}
\end{align*}
\]

and for any virtual velocity \( v^*_{\text{free}} \) of the free tool:

\[
\begin{align*}
\mathbf{r}^{v_{\text{free}}}(x, x_{\text{free}}, v, v_{\text{free}}) &= \rho_p \int_{\partial\Omega_{\text{free}}} (v_{\text{free}} - v).n)(v^*_{\text{free}} .n) \, dS = 0 \quad \ldots \text{2.29}
\end{align*}
\]

where, \( x_{\text{free}} \) is the coordinate vector of the centroid of the free tool.
2.4.6 Coupling between work piece and tools

We consider here the case of a contact between two deformable bodies $\Omega^1$, with velocity field $v^1$, and $\Omega^2$ with velocity field $v^2$, through the contact surface $\Gamma_c$ can be expressed as:

\[
\left( v^1 - v^2 \right) \cdot n \leq 0 \text{ if } \sigma_n = 0
\]

\[
\left( v^1 - v^2 \right) \cdot n = 0 \text{ if } \sigma_n \leq 0
\]

which can be imposed practically by a penalty formulation. The mixed formulation is written on the whole domain $\Omega = \Omega^1 \cup \Omega^2$, with a (discontinuous) velocity field $v$:

\[
r^r = \int_{\Omega^1} 2K\sqrt{\varepsilon'} dV - \int_{\Omega^2} p \text{ div}(v^*) dV + \int_{\Omega} (\sigma' + \Delta t \cdot D \varepsilon') dV + \int_{\Gamma_c} a_i K \left| v_s \right|^p v_s v^* dS + \rho \int_{\Gamma_c} \left( (v_1 - v_2) \cdot n \right) v^* n dS = 0
\]

In which the third integral corresponds to the elastic behavior of the tool, and $\varepsilon'$ is the strain rate deviatoric tensor and

\[
r^r(x, v, p) = \int_{\Omega^1} p^* \text{ div}(v) dV + \int_{\Omega^2} p^* (p-\Delta t \cdot \kappa \cdot \text{ div}(v)) dV = 0
\]

2.4.7 Flow formulation and deformation mechanics of metal forming process

The governing laws of deformation process are presented here. In the analysis of metal forming, plastic strains usually outweigh elastic strains, and the idealization of rigid-plastic or rigid-viscoplastic material behavior is quite convenient. The resulting analysis based on this assumption is known as the flow formulation.

2.4.7.1 Viscoplastic flow formulation

The figure 2.5 shows true stress-true strain curves for isothermal conditions generated using a viscoplastic flow model. Stress is sensitive to strain rate and elastic deformation is neglected. Rate sensitivity is applicable to hot metals and the analysis neglects elastic strain because in hot forging elastic strains are small compared to plastic strains.
For hot forming the elastic contribution is often neglected and the Norton-Hoff equation [Nor29, Hof54] is used:

\[
\sigma' = 2K \left( \sqrt[3]{3\dot{\varepsilon}} \right)^{m-1} \dot{\varepsilon} \quad \text{....2.33}
\]

where:
- \(\sigma'\) = deviatoric stress tensor
- \(K\) = material consistency
- \(m\) = strain rate sensitivity index
- \(\dot{\varepsilon}\) = effective strain rate

where effective strain rate

\[
\dot{\varepsilon} = \left( \frac{2}{3} \sum_{ij} \varepsilon_{ij}^2 \right)^{1/2} \quad \text{....2.34}
\]

Dense materials are considered as incompressible:

\[
\text{div}(\mathbf{v}) = 0 \quad \text{....2.35}
\]

Isotropic work hardening is described by putting:

\[
K = K_0 \left( \varepsilon_0 + \bar{\varepsilon} \right)^n \quad \text{....2.36}
\]

where the equivalent strain, \(\bar{\varepsilon}\), is computed by time integration of:

\[
\frac{d\bar{\varepsilon}}{dt} = \dot{\varepsilon} \quad \text{....2.37}
\]
The figure 2.6 shows the influence of the rate sensitivity index, $m$, on the viscoplastic behavior defined by the Norton-Hoff law. The following salient observations are made from the figure:

- With $m = 1$ flow is equivalent for Newtonian fluid with viscosity, $\eta = 3K$.

- With $m = 0$ flow is equivalent to plastic deformation for a material obeying the Von-Mises criterion with flow stress $\bar{\sigma} = K(\sqrt{3})$.

- The general form is with $0 < m < 1$: For hot metals (i.e. when the metal is above its recrystalisation temperature) $m$ will generally lie between 0.1 and 0.2. The consistency, $K$, is a general property used to quantify the resistance of the metal to permanent deformation.

- The dimensions of $K$ depend on the value of $m$ and are $M.L.T^{m-2}$ where $L$ represents a length dimension, $T$ a time dimension and $M$ a mass dimension.

- The viscoplastic property of a metal changes with temperature and deformation.
The FE model used in this work uses these variations by defining consistency to be an explicit function of temperature and strain. All the laws available and their definitions are written in table 2.2.

It is also possible to define more generalized flow behavior by coding any general rule in a user defined subroutine which can then be incorporated into the main program by recompiling the finite element solver with the appropriate modifications.
<table>
<thead>
<tr>
<th>Material Law</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant consistency</td>
<td>( K(T, \varepsilon) = K_0 )</td>
</tr>
<tr>
<td>Exponential Terna Law</td>
<td>( K(T, \varepsilon) = K_0(\varepsilon + \varepsilon_0)^\alpha e^{\beta T} )</td>
</tr>
<tr>
<td>Strain Hardening Power Law</td>
<td>( K(T, \varepsilon) = K_0(\varepsilon + \varepsilon_0)^\beta e^{\beta T} )</td>
</tr>
<tr>
<td>Linear Strain Hardening Law</td>
<td>( K(T, \varepsilon) = K_0(1 + a \varepsilon)^\beta e^{\beta T} )</td>
</tr>
</tbody>
</table>
| Limited Linear Strain Hardening Law              | \( \begin{align*}
K(T, \varepsilon) &= K_0(1 + a \varepsilon)e^{\beta T} \quad \text{if } \varepsilon \leq \varepsilon_{\text{max}} \\
K(T, \varepsilon) &= K_0(1 + a \varepsilon_{\text{max}})e^{\beta T} \quad \text{if } \varepsilon \geq \varepsilon_{\text{max}}
\end{align*} \) |
| Solid Polymer Law                                | \( K(T, \varepsilon) = K_0 e^{\beta T} (1 + e^{w \varepsilon})e^{h \varepsilon^2} \) |
| Interpolation Function                           | \( K(T, \varepsilon) = K_0 k_j(J)(1 - e^{w \varepsilon})e^{h \varepsilon^2} \) |
| Linear Power Strain Hardening Law                | \( K(T, \varepsilon) = K_0(1 + a \varepsilon)^\beta e^{\beta T} \) |

Table 2.2: Material Laws

where,
\( \varepsilon \) = effective strain,
\( K_0 \) = constant term,
\( T \) = Temperature [K],
\( \beta \) = temperature term,
\( n \) = strain hardening index,
\( w \) = strain hardening term,
\( a_i \) = strain hardening term,
\( h \) = strain hardening term,
\( \varepsilon_0 \) = strain hardening regulation term,
\( a \) = exponential temperature term

2.4.8 Finite Element Formulation

2.4.8.1 Space discretization

Utilizing usual isoparametric elements, the velocity field is discretised in terms of the nodal velocity vectors \( V_n \) and the shape function \( N_n \):

\[
v = \sum_n V_n N_n(\xi)
\]

\( \xi \) is the local co-ordinate vector; the mapping with the physical space is defined by:

\[
x = \sum_n X_n N_n(\xi)
\]
and the strain rate tensor is computed with the help of the B linear operator:

\[ \dot{\varepsilon} = \sum_{n} V_n B_n \]  

\[ \text{....2.40} \]

The pressure field is discretized in term of nodal pressure \( P_m \), with compatible shape function \( M_m \):

\[ p = \sum_{m} P_m M_m(\xi) \]  

\[ \text{....2.41} \]

from Eqn. 2.25 and Eqn. 2.26, the discretized mixed formulation for viscoplastic material gives the following set of non linear equations:

\[ R^Y_n = \int_{\Omega} 2K(\sqrt{3\dot{\varepsilon}})^{\text{m}-1}\dot{\varepsilon} : B_n \, dV - \int_{\partial\Omega} \alpha_r K|V|^p-1 \psi N_n \, dS - \int_{\Omega} \text{ptr}(B_n) \, dV = 0 \]  

\[ \text{....2.42} \]

\[ R^p_m = \int_{\Omega} M_m(\text{div}(v)) \, dV=0 \]  

\[ \text{....2.43} \]

which takes the symbolic form:

\[ R(X, \bar{\varepsilon}, V, P) = 0 \]  

\[ \text{....2.44} \]

The time evolution strain is governed by Eqn. 2.2, and co-ordinate vectors by:

\[ \frac{dX}{dt} = V \]  

\[ \text{....2.45} \]

For an elastoplastic or elastic viscoplastic material, the general form of the discretized integral equation is:

\[ R(X, \bar{\varepsilon}, V, P) = 0 \]  

\[ \text{....2.46} \]

the equations of evolution being Eqn. 2.2, Eqn. 2.43 and the stress evolution equation:

\[ \frac{d\sigma}{dt} = \dot{\sigma} \]  

\[ \text{....2.47} \]

2.4.8.2 Time integration

For a viscoplastic materials, the velocity and pressure vectors are obtained after solving Eqn. 2.42, using the well known iterative Newton-Raphson linearization method. Nodal and equivalent strain updates are performed according to Eqn. 2.2 and Eqn. 2.45.

The popular one step Euler scheme gives the following procedure:

- at time \( t \), \( X^t \) is known,
- compute \( V^t \) from Eqn. 2.46 written at time \( t \), i.e.:
\[ R(X^i, \bar{\varepsilon}^i, V^i, P^i) = 0 \] \hspace{1cm} \text{...2.48}

- update co-ordinates vector \( X \) and equivalent strain according to:

\[ X^{i+\Delta t} = X^i + \Delta t V^i, \quad \bar{\varepsilon}^{i+\Delta t} = \bar{\varepsilon}^i + \Delta t \dot{\varepsilon}^i \] \hspace{1cm} \text{...2.49}

and:

\[ \bar{\varepsilon}^{i+\Delta t} = \bar{\varepsilon}^i + \Delta t \dot{\varepsilon}^i \] \hspace{1cm} \text{...2.50}

More accurate schemes can be easily applied, for example a two step Runge and Kutta, Chenot [Che98a, c]. For elastoplastic of elastic viscoplastic materials, more implicit schemes must be used to avoid violation of the yield criterion.

### 2.4.9 Free tools coupling

The discrete analog of Eqn. 2.33 and Eqn. 2.34 re-easily obtained:

\[ R^Y_n(X, \bar{\varepsilon}, x_{\text{free}}, V, P, v_{\text{free}}) = \int_\Omega 2\mathbf{K}(\sqrt{3\bar{\varepsilon}})^{|v|^p} : \mathbf{B} \, dV - \int_{\partial \Omega} \alpha_i \mathbf{K} |v|^{p-1} v_s N_n \, dS - \int_\Omega \text{ptr} (\mathbf{B}) \, dV \]

\[ + \rho \int_{\partial \Omega_{\text{free}}} \langle (v - v_{\text{free}}) \rangle \mathbf{n} \cdot dS = 0 \] \hspace{1cm} \text{...2.51}

and

\[ R^Y_{\text{free}}(X, x_{\text{free}}, V, P, v_{\text{free}}) = \rho \int_{\partial \Omega_{\text{free}}} \langle (v_{\text{free}} - v) \rangle \mathbf{n} \cdot dS \] \hspace{1cm} \text{...2.52}

We obtain a non-linear system of equations, the symbolic form of which will be written:

\[ R(X, \bar{\varepsilon}, x_{\text{free}}, V, P, v_{\text{free}}) = 0 \] \hspace{1cm} \text{...2.53}

The time evolution equations are:

\[ \frac{dx_{\text{free}}}{dt} = v_{\text{free}} \] \hspace{1cm} \text{...2.54}

Remark: if the free tool also possesses rotational degrees of freedom, additional equations and differential equations must be introduced.

### 2.4.10 Master-slave coupling with the tools

We define first the master surface \( \Gamma_c \), which is here the discretized boundary of \( \Omega^2 \) and \( \pi \), the projection operator on \( \Gamma_c \). A gap function \( g \) is also introduced to define the distance of a material point from the master surface:

\[ g(x^i, t) = (x^i - \pi(x^i)) \cdot n^i \] \hspace{1cm} \text{...2.55}

where \( n^i \) is the normal vector to \( \Gamma_c \).
The incremental contact condition is expressed by, Pichelin et al. [Pic01] as:

\[
(v^i(x^i) - v^2(\pi(x^i))).n^i - \frac{g(x^i, t)}{\Delta t} \leq 0
\]

so that the incremental penalty formulation will be obtained by replacing in Eqn. 2.26 the penalty term by the following integral:

\[
\rho_p \int_{\Gamma_c} (v^i(x^i) - v^2(\pi(x^i))).n^i - \frac{g(x^i, t)}{\Delta t} v^*.n^i \ dS
\]

In fact a nodal approximation of Eqn. 54 is preferable in the form:

\[
\rho_p \sum_i \left( (V^i_1 - v^2(\pi(X_i))).n_i - \frac{g_{i_1}}{\Delta t} \right) n_i S_i
\]

where \( S_i \) is a surface area surrounding node number \( I \), such that if \( S_c \) is the area of \( \Gamma_c \) we have the equality:

\[
S_c = \sum_i S_i
\]

In the above section the main mechanical and thermal equations governing metal flow during forming processes and the finite element discretization and time integration procedures are summarized briefly.

### 2.5 FE Simulations of hot extrusion process with finite element model

Hot extrusion is a metal forming process of forcing a heated billet to be reduced in its cross section by forcing it to flow through a shaped die opening under high pressure [ASM88]. During extrusion metal billet is under compression stress state in all three directions and shear forces, fig. 2.7. No tensile force is produced, which makes high deformation possible without tearing the metal. The hot extrusion process is widely used due to its relative low deformation resistance of the metal for production of long straight metal products of constant cross section (such as bars, solid and hollow sections, tubes, wires and strips) from materials that cannot be formed by cold extrusion. Hot extrusion is an attractive process in industry due to its ability to achieve energy and material savings, quality improvement and development of homogeneous properties throughout the component. In spite of these advantages the process is rather complicated as it requires careful control. In the industrial application of the extrusion process, die design and process control are mainly based on empirical knowledge.
Fig. 2.7: Metal flow in steady state extrusion process

In hot extrusion process a preheated billet is loaded into the container and ram presses this billet through a die, producing a profile with a cross-section determined by the shape of the die orifice.

Principal parameters for the hot extrusion are: the extrusion ratio, the working temperature, the speed of deformation, the frictional conditions and lubrication. The extrusion ratio (R) is the ratio of the initial cross-sectional area ($A_0$) of the billet to the final cross-sectional area ($A_1$) after extrusion, $R = A_0/A_1$. This is actually another name for elongation. For temperature selection, the two factors that need to be considered are the temperature at which hot shortness occurs, or, for pure metals, the melting point, temperature rise due to heat generation during metal deformation and friction. This temperature rise is affected by the extrusion ratio, speed, heat transfer at billet-tooling interface and heat conduction within the billet and the tooling. Increasing the ram speed produces a tendency to increase the extrusion pressure; on the other hand, however, low extrusion speeds lead to greater cooling of the billet and thus a tendency to increase the extrusion pressure. The higher the temperature of the billet, the greater the effect of low extrusion speed on the cooling of the billet it has. Therefore, high extrusion speeds are required with materials (e.g., high strength alloys) that need high extrusion temperatures.

The extrusion process is limited by two factors, the maximum extrusion load and the maximum exit temperature, figures 2.8 and 2.9. The maximum extrusion load is either imposed by the strength of the die or by the maximum capacity of the extrusion press. The extrusion load can be lowered by increasing the initial temperature of the billet. However, this is limited by the maximum exit temperature of the material.
When this temperature gets too high, surface defects or even melting of the material can occur. Optimization of the process comes down to choosing an optimum initial temperature. In the design of extrusion dies, a major challenge is to obtain a uniform exit velocity over the entire cross-section of the profile. The objective of FE simulations here is to reduce the problems encountered in extrusion practice. In simulation of extrusion viscoplastic model is used and elasticity of the material is neglected. The reason for this is that the elastic deformations are small compared to the very large plastic deformations that occur during the process. A number of extrusion defects need to be avoided. During extrusion, the center of the billet moves faster than the periphery. After about two-thirds of the billet is extruded, the outer surface of the billet, which was formed as dead zone in the early extrusion stage, moves towards the center and extrudes through the die near the axis of the rod.

Since the surface of the billet often contains an oxidized skin, this type of flow results in internal oxide stringers or internal pipes. With higher friction, and the higher temperature difference between the billet and extrusion container, the tendency for formation of the extrusion defect grows higher. The reason is that they promote faster metal flow in the center part of the billet than in the surface skin.
Fig. 2.9: Extrusion process is limited by two factors, the maximum extrusion load and the maximum exit temperature

Other defects include axial hole (or called funnel), surface cracking, center burst etc. Axial hole is caused by radial metal flow into the die when extrusion is carried to the point at which the length of billet remaining in the container is about one-quarter its diameter. Surface cracking can be produced by longitudinal tensile stresses generated as the extrusion passes through the die. Center burst can occur at low extrusion ratios. One common problem exists in the variation in structure and properties from front to back end of the extrusion in both the longitudinal and transverse directions. Extrusion die geometry, frictional conditions at the die billet interface and thermal gradients within billet greatly influence metal flow in extrusion. The only recourse for modeling this process is to consider FE analysis that can provide insight into the process which cannot easily be obtained in any other way.

The influence of the various process parameters at the die-billet interface on the geometrical accuracy of the extruded part have been investigated in this section for the extrusion process using a finite element analysis in FORGE 3 environment.

A number of finite element simulations are performed for forward hot extrusion of a preform for transmission shaft with various die angles (15°, 30°, 45°, 60° and 75°) at
temperatures varying from 1000°C to 1260°C using the finite element model. This range of operating parameters is often used in industry for hot extrusion as per the ASM specifications, ASM hand book [Cou91]. The dies are kept at constant temperature (350°C). The schematic of die, billet and punch shape are shown in figure 2.10 and the three dimensional models of billet, lower die and upper die (punch) for simulation were developed in Solidworks software. The dies are assumed to be rigid and the billet material is ck-45 steel. The chemical composition of ck-45 steel in weight % and data pertaining to FE simulation for billet is shown below:

<table>
<thead>
<tr>
<th>C</th>
<th>0.46</th>
<th>Si</th>
<th>0.40</th>
<th>Mn</th>
<th>0.65</th>
<th>Cr</th>
<th>0.40 max</th>
<th>Mo</th>
<th>0.1 max</th>
<th>Ni</th>
<th>0.40 max</th>
</tr>
</thead>
</table>

**Chemical composition of ck-45 steel**

No. of Nodes – 1872  
No. of elements - 9943  
Type of meshing - Tetrahedral  

**Finite Element data**

![Diagram of die shape used for simulation of extrusion process.](image)

**Fig. 2.10: Schematic of die shape used for simulation of extrusion process.**  
*(All dimensions are in mm)*

The forging force at 50% reduction in diameter at 203 mm/s die velocity and 1000°C with 0.4 co-efficient of friction are depicted along with extrusion load graph with respect to displacement of upper die in figures 2.11, 2.12, 2.13, 2.14 and 2.15. The figures clearly indicate the evolution of extrusion load with respect to punch travel and effect of die angle on the magnitude of extrusion force.
The graph depicted in figure 2.11 for 15° lower die curvature clearly depicts that extrusion load is in the order of around 270 Tons during the final displacement of the upper die. It decreases as the die approaches to the bottom and is zero when the displacement is maximum at 110mm.

The graph depicted in figure 2.12 for 30° lower die curvature indicates that extrusion load is in the order of around 247 Tons during the final displacement of the upper die. It decreases as the die approaches to the bottom and is zero when the displacement is maximum at 110mm.
Fig. 2.13: Die with 45° curvature w.r.t. horizontal and its corresponding extrusion load curve w.r.t. displacement of upper die.

The graph depicted in figure 2.13 for 45° lower die curvature indicates that extrusion load is decreased in comparison with 15° and 30° during the initial displacement of the upper die. It decreases as the die approaches to the bottom and is zero when the displacement is maximum at 110mm.

Fig. 2.14: Die with 60° curvature w.r.t. horizontal and its corresponding extrusion load curve w.r.t. displacement of upper die.

The graph depicted figure 2.14 for 60° lower die curvature indicates lowest extrusion load in comparison with 15°, 30° and 45° during the final displacement of the upper die and decreases as the die approaches to the bottom.
Fig. 2.15: Die with 75° curvature w.r.t. horizontal and its corresponding extrusion load curve w.r.t. displacement of upper die.

The graph depicted figure 2.15 for 75° lower die curvature suggests increased extrusion load in comparison with 30°, 45° and 60° due to friction encountered in the conical shape pertaining to larger angle of curvature.

Table 2.3 shows the FE simulation results for extrusion load required to extrude a shaft by 50% reduction in its diameter with various die angles under different process conditions at a punch velocity of 203 mm/sec.

The salient conclusions observed from the finite element simulations results as given in table 2.3 are summarized below:

(i) Forging force or extrusion load reduces as the temperature of the billet increases.
(ii) Forging force increases with increase in friction coefficient.
(iii) The forging force decreases with increase in die angle.
(iv) For die angle 75° the forging force increases in comparison with die angles 30°, 45° and 60° due to friction encountered in the conical shape pertaining to larger die angle.
Table 2.3: Finite element simulation results for extrusion load with various die angles under different process conditions.

<table>
<thead>
<tr>
<th>Angle (Degrees)</th>
<th>Co-efficient of Friction</th>
<th>Extrusion load (Tones) for following initial temperatures (°C) of the ck-45 steel billet</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>1000</td>
</tr>
<tr>
<td>15</td>
<td>0.4</td>
<td>270.11</td>
</tr>
<tr>
<td></td>
<td>0.6</td>
<td>273.24</td>
</tr>
<tr>
<td></td>
<td>0.8</td>
<td>275.04</td>
</tr>
<tr>
<td>30</td>
<td>0.4</td>
<td>247.92</td>
</tr>
<tr>
<td></td>
<td>0.6</td>
<td>248.50</td>
</tr>
<tr>
<td></td>
<td>0.8</td>
<td>249.87</td>
</tr>
<tr>
<td>45</td>
<td>0.4</td>
<td>243.18</td>
</tr>
<tr>
<td></td>
<td>0.6</td>
<td>244.60</td>
</tr>
<tr>
<td></td>
<td>0.8</td>
<td>246.80</td>
</tr>
<tr>
<td>60</td>
<td>0.4</td>
<td>234.49</td>
</tr>
<tr>
<td></td>
<td>0.6</td>
<td>236.56</td>
</tr>
<tr>
<td></td>
<td>0.8</td>
<td>238.26</td>
</tr>
<tr>
<td>75</td>
<td>0.4</td>
<td>261.93</td>
</tr>
<tr>
<td></td>
<td>0.6</td>
<td>263.35</td>
</tr>
<tr>
<td></td>
<td>0.8</td>
<td>266.07</td>
</tr>
</tbody>
</table>

(v) The combination of process parameters i.e. temperature, velocity and die angle and friction coefficient for minimum forging force needs an optimization methodology to be applied in tandem with a new process model which can give fast estimates of forging force unlike finite element model which requires considerable time and effort.

The above conclusions can easily be visualized graphically by seeing the graphs in figure 2.16 in which the effect on extrusion load for various die angles, coefficient of friction and billet temperatures are shown.

As is to be expected the minimum extrusion force is for a combination of parameters with maximum temperature, velocity, die angle and minimum friction coefficient. From a practical point of view the high temperature can be obtained by appropriate heating, the die angle can be selected as large as possible, the maximum velocity possible is a property of the forging machine, and the friction coefficient for a billet - die material pair can be altered with suitable lubricant to some extent. However, improper selection of the parameters may cause a folding defect in the billet and inferior or defective parts.
This graph shows the variation of extrusion load for increasing billet temperature at various coefficients of friction for die angle of 15°. Extrusion load is maximum in this case as compared to other die angles and is in the order of 275 tons for 0.8 coefficient of friction at 1000°C billet temperature. The graph also shows decline in the extrusion load as the billet temperature increases.

This graph shows the variation of extrusion load for increasing billet temperature at various coefficients of friction for die angle of 30°. Extrusion load is less in this case as compared to 15° die angle and is in the order of approximately 250 tons for 0.8 coefficient of friction at 1000°C billet temperature.
This graph shows the variation of extrusion load for increasing billet temperature at various coefficients of friction for die angle of 45°. Extrusion load is lesser in this case as compared to 15° and 30° die angles and is in the order of approximately 246 tons for 0.8 coefficient of friction at 1000° C billet temperature.

This graph also shows the variation of extrusion load for increasing billet temperature at various coefficients of friction for die angle of 60°. Extrusion load is least in this case as compared to 15°, 30° and 45° die angles and is in the order of approximately 238 tons for 0.8 coefficient of friction at 1000° C billet temperature.
This graph depicts the variation of extrusion load for increasing billet temperature at various coefficients of friction for die angle of 75°. Extrusion load is more in this case as compared to 30°, 45° and 60° die angles and is in the order of approximately 265 tons for 0.8 coefficient of friction at 1000° C billet temperature. This change is due to the friction encountered in the conical shape pertaining to larger die angle. The graph also shows significant decline in the extrusion load as the billet temperature increases to 1260° C.

Fig. 2.16: Graphs depicting effect on extrusion load for various die angles, coefficient of friction and billet temperatures.

Sample simulations illustrating isocontours of equivalent strain evolved during hot extrusion of transmission shaft for various die angles are shown in figure 2.17 and isocontours of equivalent strain at various increments of extrusion process simulation is shown in figure 2.18.
Isocontour of equivalent strain for 15° die angle

Isocontour of equivalent strain for 30° die angle

Isocontour of equivalent strain for 45° die angle

Isocontour of equivalent strain for 60° die angle
Fig. 2.17: Isocontours of equivalent strain evolved during hot extrusion of transmission shaft for various die angles.

Fig. 2.18: Isocontours of equivalent strain at various increments of extrusion process simulation.
In the above section finite element models of hot extrusion process are developed for various die angles for extruding a transmission shaft of 30 mm diameter from a billet of 60 mm diameter. It has been concluded that extrusion force depends on number of process parameters viz., coefficient of friction, temperature of billet and die angle.

2.6 FE Simulation of Equal Channel Angular Pressing (ECAP)

The processing of bulk metals through the application of severe plastic deformation (SPD) provides an opportunity for achieving significant grain refinement to the submicrometer or even the nanometer level. At the present time, the most attractive and useful processing technique is equal channel angular pressing (ECAP) in which a billet, in the form of a rod of bar, is pressed through a die constrained within a channel which is bent internally through a sharp angle. Simple shear takes place along a diagonal plane at the channel turn, results in the equivalent plastic strain. By repeating this process, a very large (severe) plastic deformation is accumulated in the processed material and its grain structure gets refined. The billet can also be rotated about its axis between consecutive passes (refer section 1.6, chapter 1).

During the last decade, FEM simulations of ECAP process have become quite popular because of the unique capability of FEMs to describe the complex geometry, boundary conditions, evolved equivalent strain and resulting grain refinement during ECAP process [Zuy00, Ros02, Kam03, Ros04, Fuq05, Son06a, Son06b, Tao06, Brie07, Ros07, Maci07, Med08, Seu08, Rob09, Wei09, Kri10, Shu11, Shub11].

The 3D models for the ECAP simulation were developed in Solidworks software with the channel intersection angle $\Phi = 90^\circ$ and angle of curvature $\Psi = 0^\circ$. The dies are assumed to be rigid pieces and the material used is an H13 tool-steel. The dimension of the upper die or punch is 10mm (width) x 10mm (breadth) and 20mm (height). The square shaped three dimensional workpiece (billet) considered has the dimensions of 10 mm (width) x 10 mm (breadth) and 60 mm (height) (refer figure 2.19). The material of the billet is assumed to be Al 6061 aluminum alloy. Finite element simulation is done in FORGE3 environment. It accurately predicts the material flow and evolved equivalent strain during 3D bulk ECAP process. It enables fast simulation of very complex and fully three-dimensional parts.
Fig. 2.19: Geometry of ECAP dies and billet used in FE simulations

The chemical composition of Al 6061 in weight % and data pertaining to FE simulation for billet is shown below:

<table>
<thead>
<tr>
<th>Element</th>
<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>0.4-0.8</td>
<td>0.7</td>
<td>0.15-0.40</td>
<td>0.15</td>
<td>0.8-1.2</td>
<td>0.04-0.35</td>
<td>0.25</td>
<td>0.15</td>
<td>95.85-98.56</td>
</tr>
</tbody>
</table>

Chemical composition of Al 6061

<table>
<thead>
<tr>
<th>Data</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of Nodes –</td>
<td>1405</td>
</tr>
<tr>
<td>No. of elements -</td>
<td>5744</td>
</tr>
<tr>
<td>Type of meshing -</td>
<td>Tetrahedral</td>
</tr>
</tbody>
</table>

Finite element data

The ECAP parameters, viz., amount of deformation shear strain (ε), number of passes (N), rotation angle between each repetitive pressing, the strain rate monitored by movement of punch, and the temperature in process greatly influence the final microstructure and thus the properties of the final product in ECAP. There are four basic processing routes in ECAP and these routes introduce significant differences in the microstructures produced by ECAP. The four different processing routes are summarized schematically in chapter 1, figure 1.22, section 1.6.4. In route A the billet is pressed without rotation, in route BA the billet is rotated by 90° in alternate directions between consecutive passes, in route BC the billet is rotated by 90° in the same sense (either clockwise or counterclockwise) between each pass and in route C the billet is rotated by 180° between passes.
The magnitude of Φ and Ψ along with the number of passes determines the shear strain induced into the billet. The strain increment (ε) that the material undergoes after each pass can be expressed in terms of punch pressure (P) and the flow stress of the material (σₚₚₚ) and depend on the intersection angle (Φ) between two channels as follows:

\[ \Delta \varepsilon = \frac{P}{\sigma_{fp}} = \frac{2}{\sqrt{3}} \cot \Phi \]

The shear strain value greatly depends on the number of passes (N) and the curvature angle at the channel intersection and can be generalized as follows:

\[ \text{Shear Strain}, \ \varepsilon = \frac{N}{\sqrt{3}} \left[ 2 \cot \left( \frac{\Phi + \Psi}{2} \right) + \Phi \csc \left( \frac{\Phi + \Psi}{2} \right) \right] \]

From the above equation it can be inferred that the deformed billet experiences a shear strain value of nearly equal to 1, considering the frequently practiced values of Φ and Ψ.

The strain rate in ECAP depends on the diameter (for round cross-section) or width (for square cross-section) of the billet and the plunger speed during the deformation. As plastic deformation is a dynamic process governed by dislocation mobility, the strain rate in ECAP also affects the properties of final product and can be expressed by the following relationship:

\[ \dot{\varepsilon} = \frac{N}{\sqrt{3}} \left[ 2 \cot \left( \frac{\Phi + \Psi}{2} \right) + \Phi \csc \left( \frac{\Phi + \Psi}{2} \right) \right] \frac{V}{\Phi L} \]

where ‘V’ is the plunger speed and ‘L’ is the width or diameter of the billet. The optimum property evolution by ECAP technique can be envisaged by minimum contact friction, sharp corner channels, and square long or flat billet. The degree of grain refinement in ECAP method depends on various factors like processing parameters, phase composition, and initial microstructure of a material.

During the FE simulations in FORGE-3, the die material is assumed to be homogeneous, isotropic and incompressible. The elastic strains in comparison to visco-plastic ones are considered to be negligible. A fully automated remeshing procedure is incorporated into the analysis. The material behavior is assumed to follow that of Norton-Hoff law written in following tensorial form:

\[ s = 2K(T, \bar{\varepsilon}, \ldots) (\sqrt{3} \bar{\varepsilon})^{m-1} \dot{\varepsilon} \]

where \( s = \) shear stress, \( K = \) material consistency, \( T = \) temperature, \( \bar{\varepsilon} = \) equivalent strain,
\( \dot{\varepsilon} = \) equivalent strain rate and \( \dot{\varepsilon} = \) strain rate.

The flow stress in case of billet material Al 6061 is directly interpolated from the material data file available in FORGE software for various temperatures, strains and strain-rates. The values of \( K \) and \( m \) are not explicitly keyed in by the user in the data file for defining the rheology law.

Generalized coulomb friction law is used in the current analysis given by:

\[
\tau = \mu \sigma_n \quad \text{if} \quad \mu \sigma_n < \frac{m \sigma_0}{\sqrt{3}} \quad \text{and} \quad \tau = m \frac{\sigma_0}{\sqrt{3}} \frac{\Delta V}{\Delta V} \quad \text{if} \quad \mu \sigma_n > \frac{m \sigma_0}{\sqrt{3}}
\]

where \( \tau \) = friction stress tangential to the surface, \( \mu \) = coefficient of friction
\( \sigma_n \) = compressive stress normal to the surface (contact pressure)
\( m \) = Tresca coefficient

In this section the FE modeling of ECAP process using Al 6061 billet is attempted for various combinations of die channel angles (\( \Phi = 90^\circ, 105^\circ \) and \( 120^\circ \)), friction coefficient (\( \mu = 0, 0.1, 0.15, 0.2, 0.25, 0.3, 0.35 \)) and different processing routes, viz., Route A, Route B, and Route C. Effect of these parameters on average equivalent strain in ECAPed billet and forming energy required during ECAP process is studied and the results are obtained by FE analysis.

Sample simulations illustrating isocontours of equivalent strain evolved during ECAP process for channel angles (\( \Phi = 90^\circ, 105^\circ \) and \( 120^\circ \)) after fourth pass of routes A, B, and C are shown in figure 2.20, 2.21 and 2.22. Table 2.4 show results of FE evaluation of average equivalent strain obtained in ECAPed billet and forming energy required during the ECAP process for various channel angles (\( \Phi \)) after fourth pass at various values of \( \mu \). Figure 2.23, 2.24 and 2.25 show graphs between average equivalent strain and various coefficients of friction for different channel angles and passes. Figures 2.26, 2.27 and 2.28 show graphs between forming energy and various coefficients of friction for different channel angles.
Fig. 2.20: Isocontours of equivalent strain for channel angles (Φ = 90°, 105° and 120°) after fourth pass of route A
Fig. 2.21: Isocontours of equivalent strain for channel angles ($\Phi = 90^\circ, 105^\circ$ and $120^\circ$) after fourth pass of route $B_A$. 
Fig. 2.22: Isocontours of equivalent strain for channel angles ($\Phi = 90^\circ$, $105^\circ$ and $120^\circ$) after fourth pass of route C.
<table>
<thead>
<tr>
<th>µ</th>
<th>Channel Angle (Φ) 90°</th>
<th>Channel Angle (Φ) 105°</th>
<th>Channel Angle (Φ) 120°</th>
</tr>
</thead>
<tbody>
<tr>
<td>μ</td>
<td>Average Equi. Strain</td>
<td>Forming Energy(kJ)</td>
<td>Average Equi. Strain</td>
</tr>
<tr>
<td>0.00</td>
<td>6.551</td>
<td>1.192</td>
<td>4.579</td>
</tr>
<tr>
<td>0.10</td>
<td>6.958</td>
<td>1.360</td>
<td>4.667</td>
</tr>
<tr>
<td>0.15</td>
<td>7.198</td>
<td>1.390</td>
<td>4.675</td>
</tr>
<tr>
<td>0.20</td>
<td>7.322</td>
<td>1.404</td>
<td>4.753</td>
</tr>
<tr>
<td>0.25</td>
<td>7.325</td>
<td>1.406</td>
<td>5.126</td>
</tr>
<tr>
<td>0.30</td>
<td>7.328</td>
<td>1.412</td>
<td>5.142</td>
</tr>
<tr>
<td>0.35</td>
<td>7.382</td>
<td>1.416</td>
<td>5.664</td>
</tr>
<tr>
<td>0.40</td>
<td>7.605</td>
<td>1.423</td>
<td>6.118</td>
</tr>
<tr>
<td>Route ‘A’</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.00</td>
<td>7.185</td>
<td>1.215</td>
<td>4.309</td>
</tr>
<tr>
<td>0.10</td>
<td>7.232</td>
<td>1.397</td>
<td>4.623</td>
</tr>
<tr>
<td>0.15</td>
<td>7.381</td>
<td>1.402</td>
<td>5.048</td>
</tr>
<tr>
<td>0.20</td>
<td>7.394</td>
<td>1.423</td>
<td>5.239</td>
</tr>
<tr>
<td>0.25</td>
<td>7.678</td>
<td>1.424</td>
<td>5.569</td>
</tr>
<tr>
<td>0.30</td>
<td>7.721</td>
<td>1.429</td>
<td>5.682</td>
</tr>
<tr>
<td>0.35</td>
<td>7.791</td>
<td>1.463</td>
<td>5.791</td>
</tr>
<tr>
<td>0.40</td>
<td>7.944</td>
<td>1.484</td>
<td>5.916</td>
</tr>
<tr>
<td>Route ‘B_a’</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.00</td>
<td>6.132</td>
<td>1.203</td>
<td>4.468</td>
</tr>
<tr>
<td>0.10</td>
<td>6.235</td>
<td>1.367</td>
<td>4.702</td>
</tr>
<tr>
<td>0.15</td>
<td>6.677</td>
<td>1.413</td>
<td>4.737</td>
</tr>
<tr>
<td>0.20</td>
<td>6.699</td>
<td>1.429</td>
<td>4.562</td>
</tr>
<tr>
<td>0.25</td>
<td>6.926</td>
<td>1.495</td>
<td>4.940</td>
</tr>
<tr>
<td>0.30</td>
<td>7.057</td>
<td>1.463</td>
<td>5.393</td>
</tr>
<tr>
<td>0.35</td>
<td>7.163</td>
<td>1.501</td>
<td>5.628</td>
</tr>
<tr>
<td>0.40</td>
<td>7.188</td>
<td>1.404</td>
<td>5.716</td>
</tr>
</tbody>
</table>

Table 2.4: Finite element simulation results for average equivalent strain and forming energy for ECAP process under different process routes and friction conditions after fourth pass.
Average Equivalent Strain Vs. Coefficient of Friction

(a) Route 'A' Channel Angle 90

(b) Route 'A' Channel Angle 105
Fig. 2.23: Graphs between average equivalent strain and various coefficients of friction for different channel angles and passes for route A.

Graph (a) show variation of average equivalent strain for different values of coefficient of friction and number of passes for route A and channel angle 90°. The average equivalent strain in pass I is minimum and in pass IV is maximum and is in the order of 7.6. The graph also show that average equivalent strain increases with increase in coefficient of friction. Graph (b) show variation of average equivalent strain for different values of coefficient of friction and number of passes for route A and channel angle 105°. The maximum average equivalent strain is induced in pass IV and is less compared to channel angle 90°. Graph (c) also shows variation of average equivalent strain for different values of coefficient of friction and number of passes for route A and channel angle 120°. The maximum average equivalent strain is induced in pass IV and is least compared to channel angle 90° and 105°.
Route 'Ba' Channel Angle 90

(a)

Route 'Ba' Channel Angle 105

(b)
Graph (a) shows variation of average equivalent strain for different values of coefficient of friction and number of passes for route B\textsubscript{A} and channel angle 90°. The average equivalent strain in pass I is minimum and in pass IV is maximum and is in the order of 7.9. The graph also shows that average equivalent strain increases with increase in coefficient of friction. Graph (b) shows variation of average equivalent strain for different values of coefficient of friction and number of passes for route B\textsubscript{A} and channel angle 105°. The maximum average equivalent strain is induced in pass IV and is less compared to channel angle 90°. Graph (c) also shows variation of average equivalent strain for different values of coefficient of friction and number of passes for route B\textsubscript{A} and channel angle 120°. The maximum average equivalent strain is induced in pass IV and is least compared to channel angle 90° and 105°.
Chapter 2

2.50

(a)

Route 'C' Channel Angle 90

(b)

Route 'C' Channel Angle 105
Fig. 2.25: Graphs between average equivalent strain and various coefficients of friction for different channel angles and passes for route C.

Graph (a) show variation of average equivalent strain for different values of coefficient of friction and number of passes for route C and channel angle 90°. It is clear that average equivalent strain in pass I is minimum and in pass IV it is maximum and is in the order of 7.2. The graph also show that average equivalent strain increases with increase in coefficient of friction. Graph (b) show variation of average equivalent strain for different values of coefficient of friction and number of passes for route C and channel angle 105°. The maximum average equivalent strain is induced in pass IV and is less compared to channel angle 90°. Graph (c) also shows variation of average equivalent strain for different values of coefficient of friction and number of passes for route C and channel angle 120°. The maximum average equivalent strain is induced in pass IV and is least compared to channel angle 90° and 105°.
Forming Energy Vs. Coefficient of Friction

Route 'A' Channel Angle 90

Route 'A' Channel Angle 105
Graph (a) show variation of required forming energy for different values of coefficient of friction and number of passes for route A and channel angle 90°. The required forming energy in pass I is maximum and in pass IV is minimum and is in the order of 2.5 kJ. The graph also show that required forming energy increases with increase in coefficient of friction. Graph (b) show variation of required forming energy for different values of coefficient of friction and number of passes for route A and channel angle 105°. The maximum forming energy is required in pass I and is less compared to channel angle 90°.

Graph (c) also show variation of required forming energy for different values of coefficient of friction and number of passes for route A and channel angle 120°. The maximum forming energy is required in pass I and is least compared to channel angle 90° and 105°.
Route 'Ba' Channel Angle 90

Coefficient of friction

(a)

Route 'Ba' Channel Angle 105

Coefficient of friction

(b)
Fig. 2.27: Graphs between forming energy and various coefficients of friction for different channel angles and passes for route B_A.

Graph (a) show variation of required forming energy for different values of coefficient of friction and number of passes for route B_A and channel angle 90°. It is clear that required forming energy in pass I is maximum and in pass IV it is minimum and is in the order of 2.5 kJ. The graph also show that required forming energy increases with increase in coefficient of friction. Graph (b) show variation of required forming energy for different values of coefficient of friction and number of passes for route B_A and channel angle 105°. The maximum forming energy is required in pass I and is less compared to channel angle 90°. Graph (c) also shows variation of required forming energy for different values of coefficient of friction and number of passes for route B_A and channel angle 120°. The maximum forming energy is required in pass I and is least compared to channel angle 90° and 105°.
Fig. 2.28: Graphs between forming energy and various coefficients of friction for different channel angles and passes for route C.

Graph (a) show variation of required forming energy for different values of coefficient of friction and number of passes for route C and channel angle 90°. It is clear that required forming energy in pass I is maximum and in pass IV it is minimum and is again in the order of 2.5 kJ. The graph also show that required forming energy increases with increase in coefficient of friction. Graph (b) show variation of required forming energy for different values of coefficient of friction and number of passes for route C and channel angle 105°. The maximum forming energy is required in pass I and is less compared to channel angle 90°. Graph (c) also shows variation of required forming energy for different values of coefficient of friction and number of passes for route C and channel angle 120°. The maximum forming energy is required in pass I and is least compared to channel angle 90° and 105°.
In this chapter, a brief review of Finite Element Modelling is presented and Finite Element (FE) modelling of one forming process viz. hot extrusion and one severe plastic deformation process (especially ECAP) is accomplished in FORGE 3 environment. Effect of various die angles, friction coefficients and initial temperatures of billet on extrusion load in hot extrusion process and effect of various channel intersection angles (Φ), co-efficient of friction (µ) and different processing routes namely A, B and C for ECAP of Al 6061 preform are studied.

The salient conclusions observed from the finite element simulations results as given in table 2.4 and the graphs shown are summarized below:

i. The average equivalent strain imparted during ECAP is influenced mainly by die channel angle (Φ).

ii. Channel angle of 90° imparts higher average equivalent strain in comparison with those of 105° and 120°

iii. The forming energy required for Φ = 90° is more than that for 105° and 120°.

iv. Route A and B imposes high average equivalent strain for all channel angles in comparison with route C.

v. The average equivalent strain evolved during ECAP process increases with number of passes and is maximum during fourth pass.

vi. The requirement of forming energy decreases with increase in number of passes and is minimum during fourth pass.

**************End of chapter 2**************