

Chapter 2

Rigid-Plastic Finite Element Method and FE Simulation

2.1 Introduction

The finite element method (FEM) is one of the most powerful and efficient numerical methods for solving governing equations, which are generally partial differential equations, of a complex system over a continuous physical domain. The complex system over the continuous physical domain is discretized into simple geometric shapes called elements for solving the governing equations. This discretization process is called meshing. For a FEM mesh, the points in which the elements are connected are called nodes. A FEM mesh thus consists of all the elements and nodes. FEM can be used for solving field physical variables of concern in different domains including physical, mechanical, and thermal domains represented by governing equations, which can be simplified as a set of simultaneous algebraic linear equations after linearization in the following:

$$[K]\{u\} = \{F\} \quad (2.1)$$

where $[K]$ is the property-related matrix; $\{u\}$ is the vector of physical variables to be solved and represents the working behaviors of the system of concern; and $\{F\}$ is an external loading. The whole system is thus represented by the relationship of $[K]$, $\{u\}$, and $\{F\}$ under a specific physical domain with a detailed physical meaning. Therefore, Eq. (2.1) is a general equation, which represents the working relationship of a complex system in any physical domain. Taking an object deformation as an instance, Eq. (2.1) represents the property and behavior relationship of the system under an external loading $\{F\}$. For this deformation case, $[K]$ is called stiffness matrix, representing the material and geometry properties of the system, and $\{u\}$ is the displacement or velocity matrix, representing the

displacements or deformation velocities of nodes. $\{F\}$ is the external loading, which causes the deformation represented by the displacements or deformation velocities of the nodes, but determined by the properties of the system. If Eq. (2.1) models a thermal system, then $[K]$ represents the conductivity matrix of the system, while $\{u\}$ denotes the temperatures at the nodes; and $\{F\}$ is the heat source. If the equation represents a fluid system, $[K]$ is the viscosity matrix of the system, while $\{u\}$ is the velocity vector of the nodes; and $\{F\}$ represents the body force of the liquid system. Therefore, for any domain of concern, the nonlinear governing equations of a complex working system can be linearized by mathematical methods such as Newton–Raphson method to get the general linear formulation designated by Eq. (2.1).

Regarding the FEM applications in metal-forming PDD, various types of material models for representing different deformation behaviors of materials have been developed and used for the analysis and simulation of different deformation phenomena and behaviors of a single deformation body or the whole forming system. There thus exist different types of FEM by using different material models for solving different problems and meeting different needs. Elastic FEM uses the elastic material model, which assumes the constitutive relationship between strain and stress is linear. The elastic FEM is used to analyze the elastic deformation of deformation bodies in such a way to address the elastic deformation of tooling and further the dimensional accuracy of the deformed parts. For tooling design and service life prediction, the elastic FEM is usually used for strength-based design and estimation of tooling service life via determining the working stress and the accumulated deformation strain in deformation process. In addition, if the material model is elastic–plastic or rigid-plastic, the FEM with this type of material model would be the elastic–plastic or rigid-plastic FEM. The elastic–plastic FEM treats the workpiece as an elastic–plastic body. Under this material model, the material undergoes elastic deformation first and then plastic deformation once the effective stress of the material exceeds its flow stress or yield stress. For this kind of deformation body, the elastic recovery or springback of deformation body can be determined by using this material model such that the elastic deformation of tooling and workpiece can be considered in the design of metal-deformed parts and tooling to meet the geometry accuracy driven requirement. If the elastic deformation of workpiece can be ignored, the rigid-plastic FEM can be used. For the rigid-plastic FEM, the material model is rigid-plastic and the large-scale plastic deformation of the deformation body can be analyzed by the rigid-plastic FEM. In hot working process or superplastic deformation of materials, the material deformation is very sensitive to strain rate in the deformation process. The viscoplastic FEM can thus be used. For this type of FEM, the material model is viscoplastic model by which the strain rate sensitive behavior of materials can be modeled and analyzed. For most hot working processes in which the strain rate must be considered, the viscoplastic FEM should be used.

From application perspective, FEM is widely used in different deformation scenarios for analysis of different physical phenomena and behaviors, addressing design and engineering issues, and eventually supporting design and production

activities. This chapter aims at summarizing the concepts of modeling and simulation, the rigid-plastic FEM formulation, and the FEM applications-related issues in metal-forming PDD.

2.2 Modeling and Simulation

Modeling and simulation are two imitating approaches to representing the real systems with the concerned physical behaviors and phenomena by virtual or physical models on the basis of the associated physical principles. Modeling and simulation display or demonstrate the system to be investigated virtually or physically to reveal the phenomena, behaviors, functions, or performances of the systems in such a way to obtain the needed information for decision-making in research, design, and production or to explore the unknown phenomena and behaviors of the systems. The goal of modeling and simulation is to reduce the physical experiment, time, and cost in realization of the above-mentioned activities and explorations. Most importantly, modeling and simulation can reveal the hidden phenomena and behaviors or the instantaneous phenomena and performances of the system, which is difficult, if not impossible, to be identified and revealed by the traditionally physical experiments.

In modeling and simulation, models are generally developed first. A model of a real system represents the system physically or virtually from the perspective of the phenomena or behaviors of the real system to be explored and investigated. A model is similar to but generally simpler than the real system it represents, while abstracting and approximating most of the same salient features of the real system as close as possible. A good model is a judicious trade-off between realism and simplicity [1]. A key feature of a model is manipulability [1]. A model can be a physical model, which should tangibly represent the system to be modeled by scaling down or up the real geometry and dimension of the system using the same or replaceable materials with similar properties and can demonstrate the functions and performances of the system. A virtual model is a digital mock-up which represents the real system mathematically and physically. Similarly, it can represent and demonstrate the functions or performances of the system of concern. Modeling is an action process to build model. The models can be classified static and dynamic models. The most important difference in-between the static and dynamic models is the run-time representation. The static modeling is a time-independent representation of the system, while dynamic modeling is the run-time model of the system. Therefore, the dynamic models keep changing with reference to time, whereas the static models are at equilibrium in a steady state.

Simulation is a process of using models to reveal the behaviors, phenomena, functions, or performance of a real or theoretical system via identification of the values and variations of the related physical variables of concern. In simulation process, models are used to represent the characteristics of a system to be simulated. The purpose of a simulation is to represent the characteristics of the real system by

manipulating variables that cannot be controlled in a real system. Simulation evaluates the model of the system such that the design of the system or the performance of the system can be optimized. In addition, simulations are useful to study the real-life systems that would be difficult to access such as too complex, too large or small, too fast or slow, not accessible, too dangerous or unacceptable to engage. While a model aims to be true to the system it represents, a simulation can use a model to explore the states that would not be possible in the original system [1].

2.3 Rigid-Plastic Finite Element Method

As mentioned before, FEM uses different material models to simulate different deformation behaviors and phenomena of materials, and thus different types of FEM have been developed and used in metal-forming PDD. From the deformation perspective of billet materials, the elastic deformation is generally ignored as it is quite small compared with the large plastic deformation the materials undergo and thus the rigid-plastic FEM is widely used to handle this type of deformation scenario. To conveniently represent the formulation of FEM, some basic concepts of tensor are introduced in the following.

2.3.1 Cartesian Tensor Representation

There is the following relationship if a Cartesian coordinate (x, y, z) is transformed into the other (x', y', z') :

$$\left. \begin{array}{c|ccc} & x & y & z \\ \hline x' & l_1 & m_1 & n_1 \\ y' & l_2 & m_2 & n_2 \\ z' & l_3 & m_3 & n_3 \end{array} \right\} \begin{array}{l} x' = l_1x + m_1y + n_1z \\ y' = l_2x + m_2y + n_2z \\ z' = l_3x + m_3y + n_3z \end{array} \quad (2.2)$$

where $l_1 = \cos(x', x)$, $m_1 = \cos(x', y)$... $l_2 = \cos(y', x)$...

Using x_j and x'_i ($i, j = 1, 2, 3$) to represent x, y, z and x', y', z' , respectively, the transformation formula of Eq. (2.2) can be designated as:

$$x'_i = \sum_{j=1}^3 l_{ij}x_j \quad (i = 1, 2, 3) \quad (2.3)$$

In the above, there are two types of suffixes:

- (1) One appears precisely once, e.g., i , which is known as free suffix;
- (2) The other appears precisely twice, e.g., j , which is known as dummy suffix.

In tensor representation and calculation, the dummy suffix can be used to represent the summation in the following:

$$x'_i = \sum_{j=1}^3 l_{ij}x_j = l_{ij}x_j \quad (2.4)$$

where j equals to 1, 2, and 3. The dummy suffix j represents the summation of the three terms from $j = 1$ to 3. This is also called the rules of summation convention.

On the other hand, u_i and v_j are two vectors in a Cartesian coordinate (x, y, z) . After they are transformed to (x', y', z') coordinate, they are:

$$u'_k = l_{ki}u_i \quad v'_k = l_{kj}v_j \quad (2.5)$$

If the following matrix T_{ij} is defined:

$$T_{ij} = \begin{bmatrix} u_1v_1 & u_1v_2 & u_1v_3 \\ u_2v_1 & u_2v_2 & u_2v_3 \\ u_3v_1 & u_3v_2 & u_3v_3 \end{bmatrix} \quad (2.6)$$

The tensor format can thus be designated as:

$$T_{ij} = u_iv_j \quad (2.7)$$

After coordinate transformation:

$$T'_{kh} = u'_kv'_h = l_{ki}u_il_{hj}v_j = l_{ki}l_{hj}T_{ij} \quad (2.8)$$

The tensor-like T_{ij} with nine components is called second-order Cartesian tensor. Obviously, the stress and strain are both the second-order tensors. Similarly, a tensor with 27 components is called third-order tensor. To transform a three-order tensor, following equation can be derived which is similar to the vector's and the second-order tensor's transformation.

$$S'_{mnp} = l_{mi}l_{nj}l_{pk}S_{ijk} \quad (2.9)$$

In addition to the above basic concepts of Cartesian tensor representation, it is necessary to introduce the following basic operations of the second-order tensor as they are needed in the representation of the formulations of rigid-plastic FEM and the computation and derivation of those equations.

- (A) Addition: Each element of one tensor is added to the corresponding element of the other one, respectively.

$$C_{ij} = A_{ij} \pm B_{ij} \quad (2.10)$$

- (B) Multiplication: There are two kinds of multiplication of tensors, viz. outer and inner product.

- *Outer product* If the subscripts of the two second-order tensors are different, their outer product is a fourth-order tensor:

$$C_{ijkh} = A_{ij}B_{kh} \quad (2.11)$$

- *Inner product*: If there are the subscripts of the two second-order tensors, a second-order tensor will be obtained due to the rules of summation convention.

$$C_{ik} = A_{ij}B_{jk} \quad (2.12)$$

(C) Differentiation: The differentiation of a tensor is represented in the following.

$$\frac{\partial u_i}{\partial x_j} = u_{i,j} \quad \frac{\partial T_{ij}}{\partial x_i} = T_{ij,i} \quad (2.13)$$

(D) Substitution: The Kronecker Delta is introduced as follows:

$$\delta_{ij} = \begin{cases} 1, & i = j \\ 0, & i \neq j \end{cases} \quad (2.14)$$

With the Kronecker Delta, there is the following relationship:

$$A_i = \delta_{ij}A_j = \delta_{i1}A_1 + \delta_{i2}A_2 + \delta_{i3}A_3 \quad (2.15)$$

In fact, the Kronecker Delta is:

$$\delta_{ij} = \begin{pmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{pmatrix} \quad (2.16)$$

In addition, the deviatoric stress is denoted as:

$$\begin{aligned} S_x &= \sigma_x - \sigma_m & S_{xy} &= \tau_{xy} \\ S_y &= \sigma_y - \sigma_m & S_{yz} &= \tau_{yz} \\ S_z &= \sigma_z - \sigma_m & S_{xz} &= \tau_{xz} \end{aligned} \quad (2.17)$$

Using the Kronecker Delta, Eq. (2.17) can be represented as:

$$S_{ij} = \sigma_{ij} - \sigma_m \delta_{ij} = \sigma_{ij} - \frac{1}{3} \sigma_{kk} \delta_{ij} \quad (2.18)$$

(E) Symmetric and anti-symmetric tensor:

If $T_{ij} = T_{ji}$, T_{ij} is a symmetric tensor.

If $T_{ij} = -T_{ji}$, T_{ij} is an *anti-symmetric* tensor, the diagonal elements of the *anti-symmetric* tensor are equal to zero.

Any tensor can be written as the sum of symmetric and anti-symmetric tensors in the following.

$$T_{ij} = \overset{\text{Symmetric}}{\frac{1}{2}(T_{ij} + T_{ji})} + \overset{\text{Anti-symmetric}}{\frac{1}{2}(T_{ij} - T_{ji})} \quad (2.19)$$

For concise representation of the rigid-plastic FEM, the following basic equations used in the rigid-plastic FEM can be represented by tensor and its representation.

(1) Equilibrium equation:

$$\sigma_{ij,j} = 0 \quad (2.20)$$

which can have the detailed formulations in the following:

$$\begin{aligned} \frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} &= 0 \\ \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} &= 0 \\ \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_z}{\partial z} &= 0 \end{aligned}$$

(2) Flow rule:

The Levy–Mises equations can be denoted in tensor form as:

$$\dot{\epsilon}_{ij} = \dot{\lambda} S_{ij} \quad (2.21)$$

where $\dot{\lambda} = \frac{3\dot{\epsilon}}{2\bar{\sigma}}$. The full formulations of the above Levy–Mises rule are:

$$\begin{aligned} \dot{\epsilon}_x &= \frac{3\dot{\epsilon}}{2\bar{\sigma}} S_x & \dot{\epsilon}_{xy} &= \frac{3\dot{\epsilon}}{2\bar{\sigma}} \tau_{xy} \\ \dot{\epsilon}_y &= \frac{3\dot{\epsilon}}{2\bar{\sigma}} S_y & \dot{\epsilon}_{yz} &= \frac{3\dot{\epsilon}}{2\bar{\sigma}} \tau_{yz} \\ \dot{\epsilon}_z &= \frac{3\dot{\epsilon}}{2\bar{\sigma}} S_z & \dot{\epsilon}_{xz} &= \frac{3\dot{\epsilon}}{2\bar{\sigma}} \tau_{xz} \end{aligned}$$

(3) Geometric equations:

The relationship between deformation velocity and strain rate can be designated as:

$$\dot{\epsilon}_{ij} = \frac{1}{2} (v_{i,j} + v_{j,i}) \quad (2.22)$$

This is the so-called geometry equation and can be extendedly formulated as:

$$\begin{aligned} \dot{\epsilon}_x &= \frac{\partial v_x}{\partial x} & \dot{\epsilon}_{xy} &= \frac{1}{2} \left(\frac{\partial v_x}{\partial y} + \frac{\partial v_y}{\partial x} \right) \\ \dot{\epsilon}_y &= \frac{\partial v_y}{\partial y} & \dot{\epsilon}_{yz} &= \frac{1}{2} \left(\frac{\partial v_z}{\partial y} + \frac{\partial v_y}{\partial z} \right) \\ \dot{\epsilon}_z &= \frac{\partial v_z}{\partial z} & \dot{\epsilon}_{xz} &= \frac{1}{2} \left(\frac{\partial v_x}{\partial z} + \frac{\partial v_z}{\partial x} \right) \end{aligned}$$

(4) Yield criterion:

The Mises yield criterion is:

$$J_2 = -(S_x S_y + S_y S_z + S_x S_z) + (S_{xy}^2 + S_{yz}^2 + S_{xz}^2) \quad (2.23)$$

When J_2 reaches the critical value k^2 of materials (k is the shear strength limit of materials), the yielding of material starts. The deviatoric stresses thus follow the following equations:

$$(S_x + S_y + S_z)^2 = S_x^2 + S_y^2 + S_z^2 + 2S_x S_y + 2S_y S_z + 2S_x S_z = 0$$

Hence:

$$J_2 = \frac{1}{2} (S_x^2 + S_y^2 + S_z^2) + S_{xy}^2 + S_{yz}^2 + S_{xz}^2 = \frac{1}{2} S_{ij} S_{ij}$$

The yield condition can be formulated as:

$$\frac{1}{2} S_{ij} S_{ij} = k^2 \quad (2.24)$$

Furthermore, according to the flow rule $\dot{\epsilon}_{ij} = \dot{\lambda} S_{ij}$, there is the following relationship:

$$\dot{\epsilon}_{ij} \dot{\epsilon}_{ij} = (\dot{\lambda})^2 S_{ij} S_{ij} \Rightarrow \dot{\lambda} = \frac{\sqrt{\dot{\epsilon}_{ij} \dot{\epsilon}_{ij}}}{\sqrt{2}k}$$

Thus:

$$S_{ij} = \frac{\sqrt{2}k}{\sqrt{\dot{\epsilon}_{ij}\dot{\epsilon}_{ij}}} \dot{\epsilon}_{ij} \quad (2.25)$$

(5) Incompressibility:

In deformation of materials, if the elastic deformation is not considered, the whole deformation can then be considered as a rigid-plastic deformation. The volume of materials thus remains unchanged in the deformation process. This is the so-called volume constancy condition of materials in plastic deformation process and can be designated as:

$$\dot{\epsilon}_x + \dot{\epsilon}_y + \dot{\epsilon}_z = 0$$

It can further be denoted as:

$$\dot{\epsilon}_{ij}\delta_{ij} = 0 \quad (2.26)$$

(6) Boundary condition:

The loading at the boundaries of deformation body must satisfy the following conditions:

$$\begin{aligned} \sigma_x l + \tau_{xy} m + \tau_{xz} n &= \bar{p}_x \\ \tau_{xy} l + \sigma_y m + \tau_{yz} n &= \bar{p}_y \\ \tau_{xz} l + \tau_{yz} m + \sigma_z n &= \bar{p}_z \end{aligned}$$

which can be concisely designated as:

$$\sigma_{ij} n_j = \bar{p}_i \quad (2.27)$$

2.3.2 Basic of Rigid-Plastic Finite Element Method

The Markov's variational principle is the fundamental of rigid-plastic FEM, which can be stated as follows:

Among all the kinematically admissible and incompressible fields $\dot{\epsilon}_{ij}$ and v_i , which satisfy the stationary point of the following variational function will be the actual solutions:

$$\begin{aligned}
\phi^* = & \sqrt{2k} \iiint_V \sqrt{\dot{\epsilon}_{ij}\dot{\epsilon}_{ij}} dV - \iint_{S_p} \bar{p}_i v_i dS - \iiint_V \alpha_{ij} \left[\dot{\epsilon}_{ij} - \frac{1}{2} (v_{i,j} + v_{j,i}) \right] dV + \iiint_V \lambda \dot{\epsilon}_{ij} \delta_{ij} dV \\
& - \iint_{S_v} \mu_i (v_i - \bar{v}_i) dS
\end{aligned} \tag{2.28}$$

where α_{ij} , μ_i , and λ are the Lagrangian multipliers, which introduce the constraints of velocity–strain rate relation, incompressibility or volume constancy of materials and velocity boundary condition into the variational equation of Eq. (2.28) in such a way to ensure they are satisfied.

The variation in Eq. (2.28) can further be formulated as:

$$\begin{aligned}
\delta\phi^* = & \sqrt{2k} \iiint_V \frac{2\dot{\epsilon}_{ij}}{v2\sqrt{\dot{\epsilon}_{ij}\dot{\epsilon}_{ij}}} \delta\dot{\epsilon}_{ij} dV - \iint_{S_p} \bar{p}_i \delta v_i dS - \iiint_V \delta\alpha_{ij} \left[\dot{\epsilon}_{ij} - \frac{1}{2} (v_{i,j} + v_{j,i}) \right] dV \\
& - \iiint_V \alpha_{ij} \left[\delta\dot{\epsilon}_{ij} - \frac{1}{2} (\delta v_{i,j} + \delta v_{j,i}) \right] dV + \iiint_V \delta\lambda \dot{\epsilon}_{ij} \delta_{ij} dV + \iiint_V \lambda \delta\dot{\epsilon}_{ij} \delta_{ij} dV \\
& - \iint_{S_v} \delta\mu_i (v_i - \bar{v}_i) dS - \iint_{S_v} \mu_i \delta v_i dS
\end{aligned} \tag{2.29}$$

Since $S_{ij} = \frac{\sqrt{2k}}{\sqrt{\dot{\epsilon}_{ij}\dot{\epsilon}_{ij}}} \dot{\epsilon}_{ij}$, $S_{ij} = \sigma_{ij} - \sigma_m \delta_{ij}$ and $\frac{1}{2} (\delta v_{i,j} + \delta v_{j,i}) = \delta v_{i,j}$, the following thus exists:

$$\begin{aligned}
\delta\phi^* = & \iiint_V (\sigma_{ij} - \sigma_m \delta_{ij}) \delta\dot{\epsilon}_{ij} dV - \iint_{S_p} \bar{p}_i \delta v_i dS - \iiint_V \left[\dot{\epsilon}_{ij} - \frac{1}{2} (v_{i,j} + v_{j,i}) \right] \delta\alpha_{ij} dV \\
& - \iiint_V \alpha_{ij} \delta\dot{\epsilon}_{ij} dV + \iiint_V \alpha_{ij} \delta v_{ij} dV + \iiint_V \delta\lambda \dot{\epsilon}_{ij} \delta_{ij} dV + \iiint_V \lambda \delta\dot{\epsilon}_{ij} \delta_{ij} dV \\
& - \iint_{S_v} \delta\mu_i (v_i - \bar{v}_i) dS - \iint_{S_v} v_i \delta v_i dS
\end{aligned} \tag{2.30}$$

By using integration by parts and Gauss formula, there is the following equation

$$\begin{aligned}
\iiint_V \alpha_{ij} \delta v_{i,j} dV &= \iiint_V (\alpha_{ij} \delta v_i)_j dV - \iiint_V \alpha_{ij,j} \delta v_i dV \\
&= \iint_{S_p + S_v} \alpha_{ij} \delta v_i n_j dS - \iiint_V \alpha_{ij,j} \delta v_i dV
\end{aligned}$$

Substitute the above equation into Eq. (2.30), the following is obtained:

$$\begin{aligned}
\delta \phi^* &= \iiint_V (\sigma_{ij} - \alpha_{ij}) \delta \dot{\epsilon}_{ij} dV + \iiint_V (\lambda - \sigma_m) \delta_{ij} \delta \dot{\epsilon}_{ij} dV + \iiint_{S_p} (\alpha_{ij} n_j - \bar{p}_i) \delta v_i dS \\
&+ \iiint_V \left[\dot{\epsilon}_{ij} - \frac{1}{2} (v_{i,j} + v_{j,i}) \right] \delta \alpha_{ij} dV + \iint_{S_v} (\alpha_{ij} n_j - \mu_i) \delta v_i dS - \iiint_V \sigma_{ij,j} \delta v_i dV \\
&+ \iiint_V \dot{\epsilon}_{ij} \delta_{i,j} \delta \lambda dV - \iint_{S_v} (v_i - \bar{v}_i) \delta \mu_i dS
\end{aligned} \tag{2.31}$$

Let $\delta \phi^* = 0$ and taking into account the randomness of the variations in the above equation, the following can be obtained in the whole deformation body V :

$$\begin{aligned}
\sigma_{ij} - \alpha_{ij} &= 0 \Rightarrow \sigma_{ij} = \alpha_{ij} \\
\sigma_{ij,j} &= 0, \quad \dot{\epsilon}_{ij} - \frac{1}{2} (v_{i,j} + v_{j,i}) = 0 \\
\lambda - \sigma_m &= 0, \quad \dot{\epsilon}_{ij} \delta_{ij} = 0 \\
\text{In } S_p: \alpha_{ij} n_j - \bar{p}_i &= 0 \\
\text{In } S_v: v_i - \bar{v}_i &= 0, \alpha_{ij} n_j - \mu_i = 0
\end{aligned}$$

The above equations show as long as the deformation velocity v_i and strain rate $\dot{\epsilon}_{ij}$ are the solutions of $\delta \phi^* = 0$, they will be the true solutions as they satisfy all the basic equations listed above. Since the velocity-strain rate relation and velocity boundary condition are easily satisfied, the constraint of incompressibility or the volume constancy of materials is kept in the variational function and the following Eq. (2.32) is obtained.

$$\phi = \sqrt{2k} \iiint_V \sqrt{\{\dot{\epsilon}\}^T \{\dot{\epsilon}\}} dV - \iint_{S_p} \{v\}^T \{p\} dS + \iiint_V \lambda \{\dot{\epsilon}\}^T \{C\} dV \tag{2.32}$$

where $\{\dot{\epsilon}\}$ is the strain rate matrix:

$$\{\dot{\epsilon}\} = \begin{bmatrix} \dot{\epsilon}_x & \dot{\epsilon}_y & \dot{\epsilon}_z & \frac{1}{\sqrt{2}}\dot{\gamma}_{xy} & \frac{1}{\sqrt{2}}\dot{\gamma}_{yz} & \frac{1}{\sqrt{2}}\dot{\gamma}_{zx} \end{bmatrix}^T \quad (2.33)$$

$\{p\}$ is the surface force matrix applied on the boundary S_p :

$$\{c\} = [1 \quad 1 \quad 1 \quad 0 \quad 0 \quad 0]^T \quad (2.34)$$

Let the deformation body be discretized into m elements with n nodes. For a given shape function $[N]$, we have

$$\{v\}^e = [N] \cdot \{\mu\}^e \quad (2.35)$$

where $\{v\}^e$ is the velocity at any arbitrary point inside the element and $\{\mu\}^e$ is the nodal velocity of the element. Therefore, there is

$$\{\dot{\epsilon}\}^e = [B]\{\mu\}^e \quad (2.36)$$

Thus:

$$\dot{\bar{\epsilon}}^e = \left(\frac{2}{3} \dot{\epsilon}_{ij}^e \dot{\epsilon}_{ij}^e \right)^{\frac{1}{2}} = \left(\frac{2}{3} \{\mu\}^{eT} [B]^T [B] \{\mu\}^e \right)^{\frac{1}{2}} = \left(\frac{2}{3} \{\mu\}^{eT} [K] \{\mu\}^e \right)^{\frac{1}{2}} \quad (2.37)$$

Hence, we have the following variational function of element e :

$$\begin{aligned} \phi^e &= \sqrt{3}k \iiint_{V_e} \left(\frac{2}{3} \{\mu\}^{eT} [K] \{\mu\}^e \right)^{\frac{1}{2}} dV \\ &\quad - \iint_{S_p} \{\mu\}^{eT} [N]^T \{p\} ds + \iiint_{V_e} \{\mu\}^{eT} \lambda^e [B]^T \{C\} dV \\ \phi^e &= \phi^e(\{\mu\}^e, \lambda^e) \\ \Phi &\approx \sum_{e=1}^m \phi^e(\{\mu\}^e, \lambda^e) \approx \Phi(\mu_1, \mu_2, \dots, \mu_{3n}, \lambda_1, \lambda_2, \dots, \lambda_m) \end{aligned} \quad (2.38)$$

As a result, the variation in Φ can be written as:

$$\delta\Phi = \sum_{e=1}^m \left(\frac{\partial \phi^e}{\partial \mu_i} \delta\mu_i + \frac{\partial \phi^e}{\partial \lambda_j} \delta\lambda_j \right) = 0 \quad (2.39)$$

Hence:

$$\begin{aligned}\sum_{e=1}^m \frac{\partial \phi^e}{\partial \mu_i} &= 0 (i = 1, 2, \dots, 3n) \\ \sum_{e=1}^m \frac{\partial \phi^e}{\partial \lambda_j} &= 0 (j = 1, 2, \dots, 3n)\end{aligned}\quad (2.40)$$

Just considering element e , we know:

$$\frac{\partial \phi^e}{\partial \{\mu\}^e} = \sqrt{3}k \iiint_{V_e} \frac{\frac{2}{3}[K]\{\mu\}^e}{\dot{\tilde{e}}^e} dV - \iint_{S_p} [N]^T \{p\} dS + \lambda^e \iiint_{V_e} [B]^T \{C\} dV = 0 \quad (2.41)$$

$$\frac{\partial \phi^e}{\partial \lambda^e} = \{\mu\}^{eT} \iiint_{V_e} [B]^T \{C\} dV = \iiint_{V_e} \{C\}^T [B] \{\mu\}^e dV = 0 \quad (2.42)$$

Let $\varphi_1(\{\mu\}^e, \lambda^e) = \frac{\partial \phi^e}{\partial \{\mu\}^e}$ and $\varphi_2(\{\mu\}^e) = \frac{\partial \phi^e}{\partial \lambda^e}$. By using Newton–Raphson approach and Taylor series, we have the following equation by neglecting the terms with the order of two or above. The linear equations in the following are thus obtained. This process is the so-called linearization.

$$\varphi_1(\{\mu\}^e, \lambda^e) = \varphi_1(\{\mu\}_{n-1}^e, \lambda^e) + \left(\frac{\partial \varphi_1}{\partial \{\mu\}^e} \right)_{\{\mu\}_{n-1}^e} \cdot \Delta \{\mu\}_n^e = 0 \quad (2.43)$$

$$\varphi_2(\{\mu\}^e) = \varphi_2(\{\mu\}_{n-1}^e) + \left(\frac{\partial \varphi_2}{\partial \{\mu\}^e} \right)_{\{\mu\}_{n-1}^e} \cdot \Delta \{\mu\}_n^e = 0 \quad (2.44)$$

The above equations can be further designated as:

$$\begin{aligned}& \sqrt{3}k \iiint_{V_e} \frac{\frac{2}{3}[K]\{\mu\}_{n-1}^e}{\dot{\tilde{e}}_{n-1}^e} dV - \iint_{S_p} [N]^T \{p\} dS + \lambda^e \iiint_{V_e} [B]^T \{C\} dV \\ & + \left(\sqrt{3}k \iiint_{V_e} \left[\frac{2}{3}[K]\dot{\tilde{e}}_{n-1}^e - \frac{\frac{2}{3}[K]\{\mu\}_{n-1}^{eT} \cdot \frac{2}{3}[K]^T}{\dot{\tilde{e}}_{n-1}^e} \right] / \dot{\tilde{e}}_{n-1}^e \cdot dV \right) \cdot \Delta \{\mu\}_n^e = 0\end{aligned}\quad (2.45)$$

$$\left(\iiint_{V_e} \{C\}^T \cdot [B] dV \right) \cdot \{\mu\}_{n-1}^e + \left(\iiint_{V_e} \{C\}^T \cdot [B] dV \right) \cdot \Delta \{\mu\}_n^e = 0 \quad (2.46)$$

Thus:

$$\begin{aligned}
 & \left[\begin{array}{cc} \frac{2}{\sqrt{3}}k \iiint_{V_e} \left(\frac{[K]}{\dot{\bar{\epsilon}}_{n-1}^e} - \frac{2}{3} \cdot \frac{[K]\{\mu\}_{n-1}^e \cdot \{\mu\}_{n-1}^{eT} [K]^T}{\dot{\bar{\epsilon}}_{n-1}^{e^3}} \right) dV & \iiint_{V_e} [B]^T \{C\} dV \\ \iiint_{V_e} \{C\}^T [B] dV & 0 \end{array} \right] \cdot \left\{ \begin{array}{c} \Delta\{\mu\}_n^e \\ \lambda^e \end{array} \right\} \\
 &= \left\{ \begin{array}{c} \iint_{S_p} [N]^T \{p\} ds - \frac{2}{\sqrt{3}}k \iiint_{V_e} \frac{[K]\{\mu\}_{n-1}^e}{\dot{\bar{\epsilon}}_{n-1}^e} dV \\ - \left(\iiint_{V_e} \{C\}^T [B] dV \right) \cdot \{\mu\}_{n-1}^e \end{array} \right\}
 \end{aligned} \tag{2.47}$$

Equation (2.47) can be simply represented as follows:

$$[S]_{n-1} \cdot \left\{ \begin{array}{c} \Delta\{\mu\}_n \\ \lambda \end{array} \right\}_n = \{R\}_{n-1} \tag{2.48}$$

To solve Eq. (2.48), the iterative approach is employed. The n th iteration is conducted based on the solution of $(n-1)$ th iteration. If following condition is satisfied, it is assumed the iterative solution is the desirable one.

$$\|\Delta\{\mu\}_n\| / \|\{\mu\}_{n-1}\| < \delta \tag{2.49}$$

where δ is a very small positive number and could be 10^{-5} or even smaller. In addition, the detailed representation of the norms in Eq. (2.49) is:

$$\|\Delta\{\mu\}_n\| = \sqrt{\sum_{i=1}^{3n} \Delta\mu_{in}^2} \quad \|\{\mu\}_{n-1}\| = \sqrt{\sum_{i=1}^{3n} \mu_{in-1}^2} \tag{2.50}$$

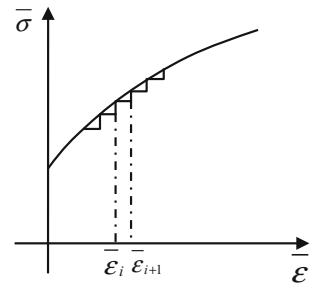
In the iteration process of solution generation, the initial velocity for the next step iteration is determined based on the solution of the previously iterative step. In detail, it can be determined based on the following equation:

$$\{\mu\}_n = \{\mu\}_{n-1} + \beta \cdot \Delta\{\mu\}_n \tag{2.51}$$

where β is chosen between 0 and 1, and β is called the acceleration parameter for convergence.

To handle the material properties and working hardening phenomenon in iteration process, it is generally assumed that the flow stress of materials is constant in each iteration step. As shown in Fig. 2.1, the flow stress of materials is supposed to

Fig. 2.1 Processing of material property and working hardening in FE simulation process



be constant in iteration step i , and a small flat line is used to substitute the small slope curve segment in the stress–strain curve with the working hardening effect. In such a way, the flow stress-related parameter K in Eq. (2.47) can be considered as a constant in all the calculations in the iteration step i ; therefore, it can be shifted out of the integration sign in the above equations to simplify and facilitate the calculation.

Similarly, in processing of time-dependent deformation scenarios, such as viscoplastic deformation in hot working or superplastic forming, and in each time step i with a very small time increment interval, all the time-dependent parameters are considered to be constant in the given time step in such a way to realize the time-dependent simulation.

2.3.3 Finite Element Simulation

Finite Element (FE) simulation is an efficient approach to analyzing the behavior and performance of complex objects or systems in working conditions using the models generated based on the FEM theory. Generally, the models cover physical, mathematical, and computational models. The physical model abstracts and idealizes the real systems of concern to comply with certain physical theory with assumptions. It simplifies the real systems to be feasible enough to model and simulate. The mathematical model specifies the mathematical equations such as the so-called governing equations the physical model should follow. It also details the boundary and initial conditions and constraints. The computational model describes the element type, mesh density, and solution parameters. The solution parameters further provide the detailed incremental interval of iteration, calculation tolerance, error bound, iteration control, and convergence criterion.

In FE simulation, it involves the numerical discretization of the system to be analyzed into a number of small units, which are termed as elements, often a very large number, but limited or the so-called finite to calculate the physical variables,

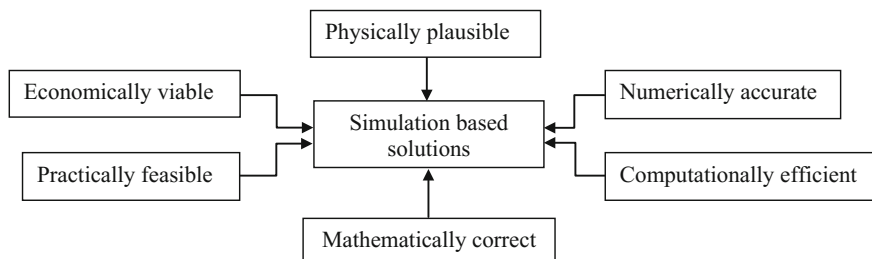


Fig. 2.2 Requirements for generation of simulation-based design solution

such as temperature, stress, strain, and velocity of each individual element such that an approximate governing equation can be generated for each individual element. The solution for the overall system can thus be obtained via solving the assembled governing equations from each individual element. The assembled governing equations represent the physical behaviors of the whole system from the perspective of the interest concerned.

FE simulation can be used in different physical domains to get the needed solutions of the field variables of real systems. The domains can cover diverse areas such as mechanical, thermal, electrical, electronic, environment, civil, aeronautical, aerospace, and manufacturing. The real systems to be analyzed could be difficult to access by human beings for different reasons such as too complex, dangerous, small, fast, or difficult to be physically realized or non-accessible. FE simulation can thus be used to physically and mathematically model and represent these systems in computer and virtually realize the phenomena and behaviors of the systems to be explored and studied. The FEM-based simulation is not only for cutting experimental cost, shortening research and development, effort and time, the most important is that it makes the impossible possible and the unrealistic realistic.

From engineering perspective, the final goal of FE simulation is to help generate feasible and practical solutions for the design and development of products and systems or provide useful information for decision-making and solution generation in different application scenarios. It is also used to reveal the instantaneous behaviors, phenomena, and performances of the system. The requirements for simulation-based solution generation are shown in Fig. 2.2. The requirements can be classified into two categories. The first category of requirements is more on how the FE simulation is established and then implemented. First of all, the simulation must be physically plausible from its physical model construction perspective. The physical model should be reasonably abstract and idealize the real systems with reasonable assumptions and comply with certain physical principles and rationales. From mathematical point of view, the mathematical model of FE simulation such as governing equations should be fully correct. From numerical implementation point of view, the computational model, which deals with meshing, element type, mesh size, calculation error, iteration, and convergence control, should be reasonable and accurate. Regarding the computation efficiency, it should be efficient and not too

much time-consuming as designers and engineers usually need to patiently wait for the completion of simulation and then they can extract the useful information to support decision-making and solution generation in design stage.

The second category of requirements is more focused on solution generation based on the simulation results. The generated solution must be economically viable. In other words, the solution generation and solution implementation should not increase the design and development lead time and cost. Eventually, it should reduce tryout and streamline the design process. On the other hand, the generated solution should be practically feasible such that it can be implemented in industries. Therefore, how to translate the FE simulation results into design solution generation is a non-trivial issue as it needs sufficient domain knowledge to interpret simulation results first and then come out with scientific- and domain-specific data, information, and knowledge to support design.

The applications of FE simulation in metal forming include metal-formed part design, forming process determination, die design, defect prediction, product quality control, and assurance, covering dimensional accuracy, surface quality control, and mechanical properties assurance. In addition, prediction and analysis of microstructure evolution of deforming materials is also a promising application. These applications are based on the revealed physical variables and the identified phenomena and behaviors in the deformation process of a forming system under a certain working condition by FE simulation. In detail, it reveals and discloses flow pattern, material flow velocity, and its distribution, stress, strain, and temperature and their distribution in the entire deformation body. Furthermore, flowline, filling status, deformation load, and upfront material flow can also be revealed. With all of these different categories of information, the rationality of the deformed part design can be explored, verified, and validated in design stage. From forming process design perspective, FE simulation helps figure out the design of forming process, operation step, and sequence. It aids the design of the shape, geometry, volume, and dimension of billet and preform. In addition, it also helps process route determination and process parameter configuration, and further verification and optimization of the designs.

From die design point of view, the elastic deformation of die components is usually considered. FE simulation is used to reveal the deformation and deflection of die components during the deformation process and further to determine the stress and strain of die components and their distribution within the components. By employing certain criteria such as the minimum stress and the least stress concentration and further the information of physical variables identified and revealed by simulation, the die structure and assembly relationship and components are determined. This design paradigm is aimed at reducing the tryout in die design and shortening the design and development lead times via offline FE simulation. The final goal is to realize the development paradigm of “design right the first time.” On the other hand, when the die design is aided by FE simulation, the coupling simulation of the plastic deformation of workpiece and the elastic or elastic-plastic deformation of die needs to be done simultaneously. The simultaneous simulation

of the deformation of workpiece and die is termed as “coupled simulation,” while separated simulation of the deformation of workpiece and die sequentially is called “uncoupled simulation” in which the deformation of workpiece is simulated first and the stage with the maximum deformation load is identified. The boundary condition between the workpiece and die in the maximum deformation load stage is then extracted out and transferred to the boundary condition in the simulation of die deformation under the maximum deformation loading. The details will be discussed in Chap. 4.

From die service life design perspective, FE simulation can be used to design die fatigue life under certain forming process and conditions. First of all, FE simulation is used to determine the cyclic working stress of die under a certain forming process configuration, die structure and assembly, and the materials’ property configuration of die and workpiece. It then identifies the deformation stage at which the maximum deformation load occurs in the whole deformation cycle, the die component with the most potential fatigue failure and fatigue location in the die component, the maximum working stress at the peak of deformation load. With all of these information and knowledge, die designers can identify, reveal, and analyze the potential die fatigue mode and the possible root causes of fatigue. Based on the revealed working stress, fatigue mode, and root causes, the methods for die service life improvement can be proposed and implemented eventually in die design to prolong die service life. Similar to die design, the coupled or uncoupled simulation of the deformations of workpiece and die structure can be conducted for die service life enhancement.

From product quality assurance and control perspectives, FE simulation can be used to help reveal the formation mechanisms and root causes of defects generated in different forming processes. The defects can be classified into flow-induced, stress-induced, dimension-related, microstructure-related, and property-related defects, and their formation mechanisms and root causes can also be totally different. Through figuring out the mechanisms and root causes, the methods to avoid the defects can be proposed and developed, and further used in defect-free metal-forming PDD.

2.4 FE Simulation of Metal-Forming Systems

As described in Chap. 1, the elements in a metal-forming system can be classified into tangible and intangible ones. The tangible one refers to the physical items, while the intangible one covers the virtual items imbedded in the forming system. The physical items further refer to the materials used in the system, billet, and preform including their shape, geometry and dimensions, feeding mechanism, forming and ejection structures, and the lubrication condition in-between the workpiece and tooling. The virtual items include process and process chain, process parameters and process parameter configuration, which are generally imbedded in

the design of the related physical elements in the forming systems. From FE simulation perspective, all the needed information related to the above physical and virtual items needs to be input into the FE simulation system via different models or predefined in the simulation system.

2.4.1 Modeling of Die and Workpiece

From modeling point of view, all the die components of concern need to be modeled from geometry perspective. The geometry models of die components are usually created in a computer-aided design (CAD) system and then exported into a FE simulation system for simulation as the latter generally does not have a strong geometry modeling capacity to model the complex and nonlinear geometries of die components. For the CAD models of die components generated in a CAD system, they are generally converted into a data exchange format such as initial graphics exchange specification (IGES), standard for the exchange of product model data (STEP) or stereo lithography (STL), which can be recognized by FE simulation systems and exported into a FE simulation system for preprocessing of simulation. In addition, the CAD model of workpiece or preform is also needed to be created in a CAD system and then followed by exporting to a FE simulation system via data exchange conversion. From deformation point of view, the die components with different materials can be treated as different deformation models including rigid body, elastic, and elastic-plastic models. For the workpiece, on the other hand, the deformation behaviors of workpiece materials can be modeled as an elastic-plastic, rigid-plastic, or visco-plastic model. In detail, they are described in the following.

When a die component is treated as a rigid body, it means that the component is assumed to not have any deformation in the simulation process. The real life, however, is not so, and this is an idealization as elastic deformation, no matter how small it is, exists ubiquitously in any deformation process. The function of this model is only the representation of component geometries. In other words, an object under the rigid body model is non-deformable, but the model represents the geometry profile of the object in a three-dimensional space and the geometry constraint imposed by the model in the 3D space of die assembly. When the rigid body model is used to represent a die component, the elastic deformation of die component is not critical and can thus be neglected. For this model, the movement speed of die component is specified initially, and there is no information related to the displacement, stress, strain, and thermal expansion of the die component.

For the elastic deformation model, it is usually used for modeling of die component for strength design, fatigue analysis, design optimization, and dimensional accuracy analysis. From material properties perspective, the main elastic material properties include Young's modulus and Poisson's ratio. In the scenarios of using this model, the information of stress, strain, and deflection of die components is available and can be used in the above-mentioned design activities. In simulation process, when the working stress of die component exceeds the yield stress of die materials, the stress,

strain, and displacement determined by simulation will be incorrect as the real deformation of die component is no longer in elastic deformation scope. A good practice in elastic deformation simulation is to check the effective stress in simulation iteration process frequently to ensure this situation would not happen.

Regarding the plastic deformation model, it generally includes rigid-plastic and rigid-viscoplastic models, which are basically used for modeling of the deformation of workpiece materials. For the rigid-plastic deformation model, it assumes the deformation of workpiece has only plastic deformation and does not consider any elastic deformation involved. In addition, if the deformation is rate-independent, viz. the deformation behavior and flow stress of materials are independent of deformation velocity; there is thus no strain rate included in the constitutive model of materials. In these cases, where the strain rate sensitivity is not important and can thus be ignored, the rigid-plastic deformation model is used. The real-life example is cold forming processes in which the elastic deformation of workpiece can be ignored and the effect of strain rate can also be neglected. For the rigid-viscoplastic deformation model, on the other hand, the elastic deformation of materials is also not considered. The deformation is rate-dependent and the strain rate sensitivity plays an important role as it affects the deformation behavior and flow stress of materials in deformation process. The rigid-viscoplastic deformation model is generally used for analysis and modeling of warm and hot forming processes. The superplastic forming and isothermal forging are also good examples, which can be accurately simulated by using the rigid-viscoplastic deformation model.

For elastic-plastic deformation model, the material represented by this model is treated as an elastic deformation body when the effective stress is less than the yield stress of material, while as a plastic deformation object when the yielding condition of material is met. In elastic-plastic deformation of materials, the total strain is the combination of elastic and plastic strains. This model can be used for modeling of the elastic and plastic deformations of workpiece and die components. It can also be used for analysis of elastic recovery such as the springback in sheet metal forming and the strain induced by thermal expansion in warm and hot working processes. The model can further be used for analysis of residual stress in bulk cold forming. Since the strain rate sensitivity is not considered in this model, the hot working processes with large plastic deformation cannot be handled efficiently by this model. Furthermore, this model takes a longer calculation time compared with the time needed by using the rigid-plastic deformation model. And sometimes, it may have difficulty with convergence by using the model.

For viscoplastic deformation model, it represents the rate-dependent plastic deformation well. For viscoplastic deformation scenario, the deformation of materials depends on the rate at which the external loads are applied, or more accurately, the deformation velocity applied. The viscoplastic deformation materials exhibit not only the permanent deformation after the application of loads, but also the creep flow with time under the applied loads [2]. In metal forming, the creep flow is generally ignored, but the rate-dependent plastic deformation is the point to be considered.

2.4.2 Modeling of Frictional Behaviors

In metal forming, the friction behaviors and conditions in-between the die and workpiece interfaces significantly affect deformation and flow behaviors, deformation texture and flowlines, formation of new surfaces, and the surface quality of the deformed parts. In addition, friction also affects deformation load, energy consumption, occurrence of surface and internal defects, die stress and die service life, and process parameters configuration. In FE simulation, friction behaviors need to be quantitatively represented and modeled such that it can be fully embedded into the FE simulation as an input.

To quantitatively model and represent the frictional behaviors, the relationship between the relative movement of workpiece and die and the material properties of workpiece need to be quantitatively determined. There are three widely used models in metal-forming arena listed in following [3, 9].

A: Coulomb model

The Coulomb model represents the frictional stress and the compressive normal stress in the interface in-between the die and workpiece in the following equation.

$$f_s = up \quad (2.52)$$

where u is the friction coefficient and p is the compressive normal stress.

B: Shear stress model

Frictional stress in the interface of the die and workpiece is designated as follows:

$$f_s = mk \quad (2.53)$$

where m is the friction factor, which has the value from 0 to 1, and k is the shear strength of workpiece material.

C: Modified shear stress model

In addition, the frictional stress can also be represented as the following formulation:

$$f_s = mk[(2/\pi) \tan(v_r/a|v_d|)]t \quad (2.54)$$

where m is the friction factor, k is the shear strength of workpiece material. v_r is the magnitude of relative velocity between die and workpiece. a is a small positive constant such as 10^{-5} . $|v_d|$ is the absolute value of die velocity and t is a unit tangential vector. In this model, the relative velocity in between the die and workpiece is considered and the unit tangential vector t can be used to specify the direction.

For various forming conditions, the values of m vary, and the following provide a reference for different materials in different working processes [3]:

$m = 0.05\text{--}0.15$ in cold forming of steels, aluminum alloys, copper, using conventional phosphate–soap lubricants or oils.

$m = 0.2\text{--}0.4$ in hot forming of steel, copper, and aluminum alloys with graphite-based (graphite–water or graphite–oil) lubricants.

$m = 0.1\text{--}0.3$ in hot forming of titanium and high-temperature alloys with glass lubricants.

$m = 0.7\text{--}1.0$ when no lubricant is used, e.g., in hot rolling of plates or slabs and in non-lubricated extrusion of aluminum alloys.

2.5 Geometric Symmetry in FE Simulation

Symmetry is an attribute of a shape or relation. It refers to the correspondence in size, form, and arrangement of parts on opposite sides of a plane, line, or point. It can also deal with the regularity of form or arrangement in terms of like, reciprocal, or corresponding parts. Geometry symmetry deals with the symmetry of geometric shapes or objects. An object is geometry symmetric if it can be divided into two or more identical pieces that are arranged in an organized fashion [4]. This means that an object is symmetric from geometry perspective if there is a transformation that moves individual pieces of the object but does not change the overall shape. The type of symmetry is determined by the way the pieces are organized, or by the type of transformation.

Translation shifts a pattern some distance from its original in a specific direction and leaves the pattern appearing unchanged. In this process, the orientation is kept the same. Figure 2.3 shows the translational symmetry, and the symmetry is referring to the symmetry plane.

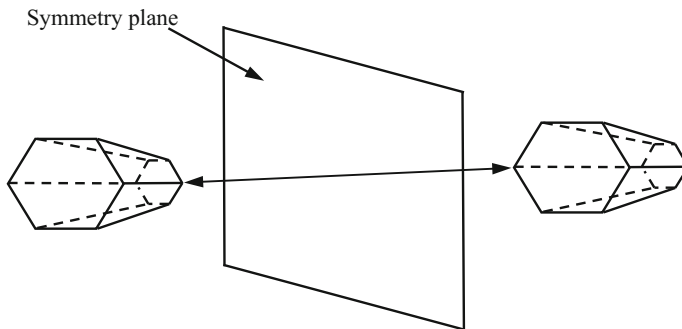


Fig. 2.3 Translation symmetry of an object

Reflection is a transformation in which the direction of one axis is reversed. An object has reflectional symmetry if there is a symmetry plane which divides it into two mirror images of each other, as shown in Fig. 2.4.

An object has rotational symmetry if it remains the same after a certain amount of rotation and without changing the overall shape. In this rotation process, the relative relationship of the geometry entities of the object keeps the same, but the orientation of the whole object changes with the rotation, as shown in Fig. 2.5.

In addition, scale symmetry is also important in FE simulation. In multiscale modeling and simulation, there is a size effect phenomenon from macroscale to mesoscale and further to microscale. In the subsequent chapters, these different size scales are often referred in different size-scaled metal-forming domains and handling the size effects in different size-scaled domains. Scale symmetry can be employed for scale-based modeling and simulation. An object has scale symmetry if it does not change shape when it is expanded or contracted or scaled up or down. From geometry point of view, a fractal is a shape made up of parts that are the same shape as itself and are of smaller and smaller sizes. Fractals also exhibit a form of scale symmetry, where small portions of the fractal are similar in shape to large portions. In FE simulation, scale symmetry is more referring to the geometrical similarity, ratio, and proportion of the system to be modeled. From the physical behavior of real-life perspective, there are two scenarios: One is the same physical behaviors and performances of the size-scaled system to be modeled, and the other

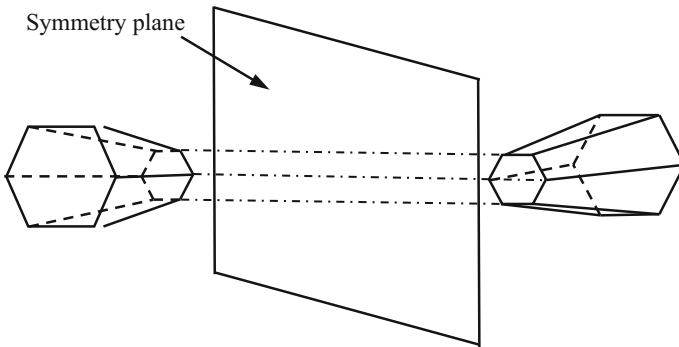
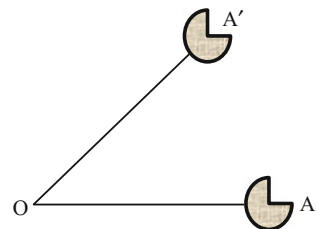


Fig. 2.4 Reflectional symmetry of an object

Fig. 2.5 Rotational symmetry of an object



is the different physical behaviors and performances of the size-scaled systems due to size effect. In reality and practice, the scale-based physical behaviors and phenomena are of importance to be explored and investigated by using the scaled-down size model to reduce the sample size and further the capacity of experimental devices in such a way to cut experimental cost or even to make the impossible possible due to the downsizing of the scale, which becomes feasible to be implemented and realized. From FE simulation perspective, the scale geometries of the real and complex systems can also be employed to downsize the systems to be modeled and simulated in such a way to reduce the meshing size, the memory, and computational capacities of computers and further to enhance the computational efficiency. Figure 2.6 shows the objects with different scale factors from geometry proportional perspective.

In addition, axial symmetry is of importance in simulation as the computation can be significantly reduced by using the axisymmetric characteristic of the systems to be simulated. The axial symmetry refers to the symmetry of systems or objects around an axis. In other words, the appearance of the axisymmetric systems or objects would not be changed if they are rotated around an axis. For any cross section of an axisymmetric object with the symmetrical axis passing through, the cross section is divided into two reflectional geometries by the symmetrical axis. Figure 2.7 shows a cylinder with axial symmetry. In the figure, the cross section of the cylinder is divided into two reflectional parts around the symmetrical axis. From geometry perspective, one reflectional geometry can represent the other. In simulation, the axisymmetric systems or objects can be simplified as a 2D problem or can be represented by part of the axisymmetric systems. In engineering, there are many axisymmetric products or systems, which can be analyzed and simulated by using the characteristics of the axisymmetric characteristics.

In FE simulation, geometric symmetry of the systems to be modeled or simulated should fully be exploited through simulating part of the systems such as a half or one-quarter of the systems to reduce preprocessing job, shorten computation time, save storage and memory of computer, and increase the efficiency and cut the cost of simulation. By using the geometric symmetry of the systems to be modeled, the following conditions must be met:

Fig. 2.6 Scale symmetry of an object

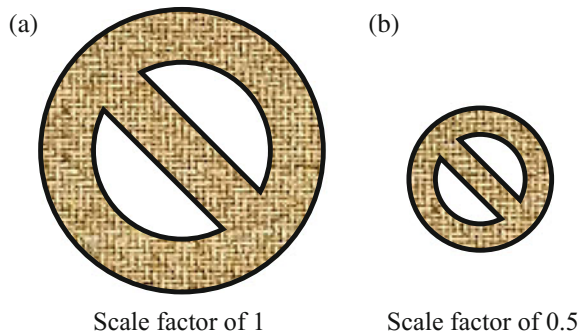
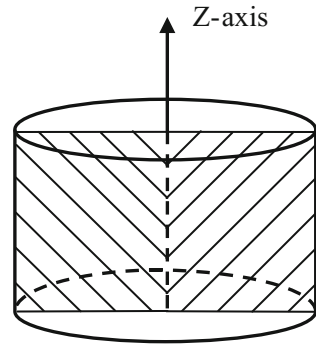


Fig. 2.7 Axial symmetry of an object



- The system to be simulated is of geometrical symmetry.
- The boundary conditions of the system are also symmetry from both physics and geometry perspectives.
- The constraints of the system are also symmetrical and can be applied symmetrically.
- The physical behaviors and performance of the systems must be the same symmetrically.

In this way, part of the system to be modeled can fully represent the whole system to be modeled.

In addition, there are some products or systems, which have pattern-based symmetry. Figure 2.8a shows an example of this type of symmetry case with pumpkin shape. From geometry modeling perspective, it can be generated via copying the basic unit, such as Fig. 2.8b, around the symmetrical axis. Assuming that the object undergoes compression deformation and the loading is uniformly applied and distributed on the top surface of the object in Z-direction as shown in the figure. The loading, boundary condition and constraints are symmetrically the same referring to the basic pattern unit. The pumpkin-shaped part can thus be approximately simulated based on the characteristics of pattern-based symmetry if the effect of the friction on the top surface is not so significant. In this way, one-eighth of the part is simulated as it can approximately represent the deformation of the whole system, as shown in Fig. 2.8b. In addition, Fig. 2.8c shows its boundary condition. On the symmetrical planes OS_1 and OS_2 , which pass through the symmetrical axis OZ , the deformation is assumed to only happen on the symmetry planes, viz. the materials cannot move away from the symmetry planes OS_1 and OS_2 . In other words, the original OS_1 and OS_2 are planes, and they are also plane after deformation. Therefore, the constraints on the two symmetry planes are defined and shown in Fig. 2.8c.

In this case, the deformation load determined based on the one-eighth of the part just represents the deformation force needed for deformation of this one-eighth of the part. For the whole system, the deformation load is the eight times the deformation load of the one-eighth of the part. For other field variables such as stress,

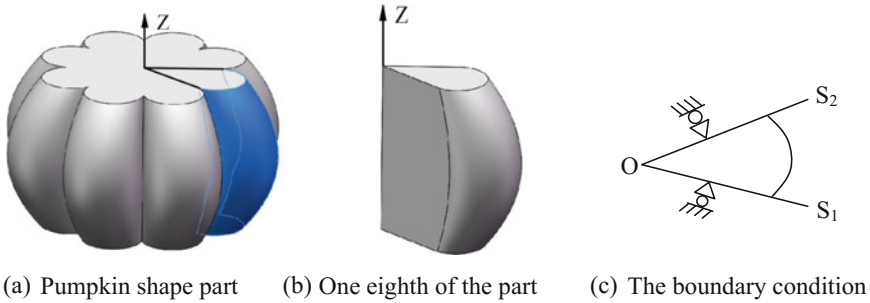


Fig. 2.8 Example of axial symmetry part

strain, and displacement, they can represent the whole part from symmetry point of view. In other words, the values of physical field variables determined based on the one-eighth of the part at a specific location are the same as those in the symmetrical location in the corresponding reflectional place in the part or the whole system. In the real application scenarios, the simulation can be done in this way to shorten the time for execution of simulation, reduce the data storage space for storing the simulation results, and increase the simulation efficiency.

2.6 Validation and Verification of FE Simulation

FE simulation has been widely used in many areas in engineering and science and is being heralded by many as one of the most important developments in the advance of engineering and science in modern history. FE simulation has become an enormously important tool to support decision-making via prediction and revealing of the behaviors and performances of engineering systems or physical events. However, the reliability of such predictions has been an issue with growing concern for a long while. Therefore, validation and verification of FE simulation has become a non-trivial issue, which perplexes people in simulation arena for decision-making by using the data and information provided via FE simulation as the results of FE simulation have been rendered as a legitimate basis in decision-making process. This is also understandable as there are a plethora of factors affecting the validity of simulation models and the reliability and accuracy of simulation results [5–8].

The simulation models defined in this book include physical, mathematical, and computational models generated for modeling and simulation of real systems or physical events from the perspective of physical behavior and functional performance of systems to be simulated. The physical model idealizes the problems or behaviors of interest of the system to be studied and abstracts them to comply with certain physical theory and assumptions. Taking the deformation of a beam as an instance, the physical model of the beam needs to comply with the continuum

mechanics and assumptions describing the physical characteristics of the beam, such as static, three-dimensional, elastic type of material. The mathematical model specifies the mathematical equations such as the differential equations in FEM analysis the physical model should follow. The detailed formulation includes the governing equation of the systems and provides abstract representation of the systems consistent with a scientific theory. It also details the boundary and initial conditions and constraints. The computational model describes the element types, mesh density, and solution parameters. The solution parameters further provide detailed calculation tolerances, error bounds, iteration specifications, and convergence criteria. For the tailor-made commercial simulation systems for domain-specific applications, the common parts of these models are generally built into the FEM systems. Other model-related data and information of the systems to be modeled need to be input by users using different approaches.

Simulation model is an approximate imitation of real-world systems, but never exactly emulates the real-world ones. The validity of the models thus needs to be assessed. If the simulation models are not valid and accurate, the simulation and further the results arising from the simulation are of virtually no value. On the other hand, if the models are fully valid and reasonable, the simulation and the results may not be fully correct and accurate as the correct implementation of the simulation models is another issue. Therefore, the simulation models and the implementation of the models need to be validated and verified, respectively, to ensure the models are correct, and the results arising from simulation can be used as a legitimate basis for decision-making.

Validation and verification of simulation models and results are two important steps in FE simulation. It is generally believed that validation is the process to ascertain the generation of “right model.” It is the process of determining whether the simulation models can accurately represent the real-world systems from physics perspective. Model validation determines the physical theories and assumptions underlying the models are correct, and the model’s representation of the real system’s behaviors being modeled is reasonable for the intended purpose of the model [5]. Verification, on the other hand, is concerned with building the “model right.” It is the process by which it is ascertained that the simulation models can be accurately implemented and the implementation of simulation models can accurately represent the intended purposes of the models from mathematic point of view [6]. Figure 2.9 shows the positions of validation and verification in modeling and simulation.

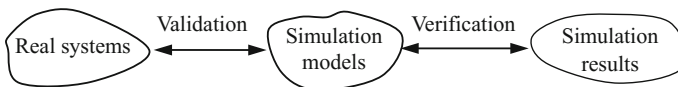


Fig. 2.9 Validation and verification in modeling and simulation

In building of simulation models, assumptions are generally made in establishing the physical, mathematical, and computational models. The assumptions simplify, idealize, and abstract the real systems such that the systems being simulated can be accurately represented by the simulation models. Similarly, assumptions need to be correct to ensure the validity of simulation models. However, it is difficult to validate whether the assumptions are correct alone. Therefore, the assumptions and the built simulation models should be validated together. Following are some practices to ensure the model generation is valid or “right model” and further the simulation results are accurate.

- (A) Cross-checking the correctness of the established models. It is a good practice to discuss with experts on the correctness of the assumptions and further the established simulation models since different people have different understandings of the physical essence of the systems being modeled and cross-checking of the assumptions and the idealized models can figure out the loopholes in assumption making and further in establishment of the models.
- (B) Comparison with the available analytical solutions. If there are available analytical solutions for simple systems, which are similar to the systems being simulated, the assumptions and the established models can be used to simulate the simple systems and then compare the simulation results with the available analytical solutions of the simple systems to validate the validity and correctness of the assumptions and the established models. The deviation in-between can be used to fine-tune the assumptions and the established models.
- (C) Corroboration with the known behaviors or performances of the systems. Simulation is used to explore or identify the concerned behaviors or performances of real systems. The assumptions and the built models can also be used to simulate the similar systems with the known behaviors or performances such that the revealed behaviors or performances can be compared with the known ones of the similar systems.
- (D) Validation by experimental results. The experiments on the systems being simulated in terms of the concerned behaviors can be conducted, and the experimental results can be used to validate the assumptions and the built models. In addition, if the system being simulated is not yet available, the experiments can be done on the scaled-down prototype system with the similar behaviors or performances. The prototype system can be simulated for comparison with the experimental results.

In tandem with validation, verification is the process to ascertain the accuracy of model implementation, as shown in Fig. 2.9. In FE simulation of metal-forming processes or forming systems, the implementation of simulation models is generally realized by coding of the models into the simulation platform or system through the application programming interfaces (APIs) provided by the system or by inputting the model-related data into the simulation system via the input mechanism provided

by the system. The verification of implementation can be done using the following approaches:

- (A) Structured walkthrough: Cross-checking of the implemented codes among peers to evaluate and determine the correctness of the codes if the implementation of the simulation models needs coding. If there is no coding required, input of the model-related data is generally needed. In this way, checking of the correctness of the generated data and its input is usually conducted. This can be considered as a static verification as it does not involve the execution of the coded program to examine its validity.
- (B) Execution of program: Upon implementation of the simulation models by using the coding or direct input of the model-related data, the simulation can then be executed in simulation systems. In the course of execution, the intermediate output can be designed to monitor the simulation and ensure the correct simulation is conducted based on the evaluation and assessment of the intermediate output. In addition, the input and output relationship can be used to examine the correctness of implementation and simulation in the execution process.
- (C) Outcome verification: The outcomes from simulation can be compared with (a) the available analytic solution of the forming processes or systems; (b) the detailed physical variables which are easily measured, such as the deformation load of a forming system; (c) geometry and dimension data such as the diameter and height of the experimental parts as they are easy to be measured or originally designed.

It is impossible to have total and absolute validation and verification of models and model implementation. Validation and verification need to be conducted feasibly and viably. Over validation and verification are not necessary and time-consuming and costly.

2.7 Summary

In this chapter, the basic concepts of modeling and simulation are briefly introduced and the notions of FEM and the FE simulation are presented. Considering the unique characteristics of metal-forming process, the rigid-plastic FEM is extensively described and the formulations are presented. By using the FEM, how the metal-forming system is simulated is articulated via considering the modeling of different elements in the system. Since many metal-forming systems are geometric symmetry, simulation of the whole system can be done by using the symmetry of systems in such a way to cut simulation cost and shorten lead time. Therefore, how to conduct symmetry-based simulation is summarized. Since simulation results provide a legitimate basis for decision-making, how to ensure the model generation and implementation are both correct is a critical issue with growing concern in

simulation-based decision-making. The validation and verification of FE simulation models and model implementation are finally articulated in the end of chapter. All of these lay down theoretical basis for the subsequent chapters on FEM applications metal-forming process and product development.

References

1. Schwendimann B (2010) What is the difference between a simulation and a model. Proto-Knowledge. <http://proto-knowledge.blogspot.hk/2010/12/what-is-difference-between-simulation.html>
2. Perzyna P (1966) Fundamental problems in viscoplasticity. *Adv Appl Mech* 9(2):244–368
3. Altan T, Oh S, Gegel H (1983) Metal forming fundamentals and applications. American Society for Metals, Materials Park
4. Lockwood EH, Macmillan RH (1978) Geometric symmetry. Cambridge Press, Cambridge
5. Pace Dale K (2004) Modeling and simulation verification and validation challenges. *Johns Hopkins APL Techn Digest* 25(2):163–172
6. Sargent RG (2013) Verification and validation of simulation models. *J Simul* 7:12–24
7. Babuska I, Oden JT (2004) Verification and validation in computational engineering and science: basic concepts. *Comput Methods Appl Mech Eng* 193:4057–4066
8. Szabo B, Babuska I (2011) Introduction to finite element analysis: formulation, verification and validation. Wiley, London
9. Scientific forming Technologies Corporation (2014) DEFORMTM 3D Manual, Version 8.11

Design and Development of Metal-Forming Processes
and Products Aided by Finite Element Simulation

Fu, M.W.

2017, XV, 246 p. 167 illus., Hardcover

ISBN: 978-3-319-46462-6