Actual simulation techniques for forming processes

Simulation Techniques in Manufacturing Technology
Lecture 3

Laboratory for Machine Tools and Production Engineering
Chair of Manufacturing Technology

Prof. Dr.-Ing. Dr.-Ing. E.h. Dr. h.c. Dr. h.c. F. Klocke
## Contents

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Introduction</td>
</tr>
<tr>
<td>2</td>
<td>Calculation methods for the process design</td>
</tr>
<tr>
<td>3</td>
<td>Procedure of Finite Element Analysis</td>
</tr>
<tr>
<td>4</td>
<td>Non-linearities in FEM</td>
</tr>
<tr>
<td>5</td>
<td>Summary</td>
</tr>
</tbody>
</table>
Analysis of forming processes enables:

- **Forming processes** design and improvement
  e.g. layout of pre-forms and process stages
- **Workpiece** design
  e.g. definition of sheet metal thickness in metal forming
- **Tool** shape optimization
  e.g. reduction / homogenization of contact stresses

Analysis of forming problems started from the formulation of the analytical theories

Wide industrial application of the analytical theories was enabled by a progress in numerical methods and software

- Simulation result of a FE deep drawing simulation
  - **Punch stroke**
  - **STH [mm]**
  - **t_p = 3.0**
  - **t_p = 2.0**
  - **1,316 0,814 STH – Sheet thickness**

Legend: **t_p** – Process time
Introduction
Calculation methods in plasticity theory

"elementary" plasticity theory

analytical methods
- energy method
- stripe model
- disk model
- tube model
  based on simplifications

graphical-, empirical-, analytical method
- strict solution
- slip field calculation

viscoplasticity

numerical methods
- upper and lower bound method
- error compensation method
- finite element method

based on theory

closed solution

approximate solution
Introduction

Analytical calculation methods with the stripe-, disk- and tube model

stripe model

disk model

tube model
Introduction

Disk model

Example: drawing of rods or wire

- Equilibrium condition in drawing direction:
  \[ \sigma_z + A \frac{d\sigma_z}{dA} + p(1 + \mu \cot \alpha) = 0 \]

- Yield criterion according to Tresca with \( p \) (compressive stress) and \( \sigma_z \) (tensile stress) \( \geq 0 \):
  \[ p + \sigma_z = k_f \]

- Elimination of \( p \) results in a differential equation of 1st order:
  \[ \frac{d\sigma_z}{dA} - \frac{\mu \cot \alpha}{A} \sigma_z + \frac{k_f(1 + \mu \cot \alpha)}{A} = 0 \]

  \[ \Rightarrow \sigma'_z + f(x)\sigma_z + g(x) = 0 \]

Assumptions: \( v_z(r) = \text{const.} \), \( v_z(z) \neq \text{const.} \)

Source: Pawelski, O. (1976)
Introduction

Viscoplasticity in bulk metal forming

Source: Lange, K. (1990)

- Characterization of the velocity field by measurement of the temporary change of markers.
- No information about the stress distribution.

\[ \vec{V}_r = \frac{\Delta S_r}{\Delta t} \quad \text{und} \quad \vec{V}_z = \frac{\Delta S_z}{\Delta t} \]
## Calculation methods for the process design

1. **Finite Element Method (FEM)**
2. **Boundary Element Method (BEM)**
3. **Molecular Dynamics (MD)**

## Procedure of Finite Element Analysis

## Non-linearities in FEM

## Summary
Contents

1 Introduction

2 Calculation methods for the process design
   2.1 Finite Element Method (FEM)
   2.2 Boundary Element Method (BEM)
   2.3 Molecular Dynamics (MD)

3 Procedure of Finite Element Analysis

4 Non-linearities in FEM

5 Summary
Calculation methods for the process design

**Finite Element Method (FEM)**

- Industry standard for the approximate solution of complex real processes is the finite element method (FEM).
- FEM is an approximate method for numerical solutions of continuous field problems.
- By discretizing, a complex problem is transferred into a finite number of simple, interdependent problems.
- Resulting systems of equations can be solved using numerical methods.
- FEM allows statements about the size and distribution of the state variables (stress, elongations, true strains, temperatures etc.)

Discretization
Calculation methods for the process design

Advantages of FEM

- Application of FEM for:
  - Process design
  - Tool design
  - Machine tool design

- FEM enables the analysis of forming processes during the design phase

- Reduction of development times and thus also of time-to-market

- Time for start-ups and setting the process is shortened

- Reduction of material-, labour- and tool-costs and thus the cost of production
Calculation methods for the process design

FEM/BEM-Connection for a time-efficient numerical simulation

Methods for tool design:

1. Tool design based on expertise and standards
2. Manual improvement of the tool design (also using the FEM)
3. Numerical optimization of the tool design using the FEM

Significant reduction of the resource requirements and production costs can be only achieved using optimized tools.

The iterative numerical tool optimization is time intensive because of the high computation time.
Calculation methods for the process design

Application of BEM in elastostatics

Finite Element Method (FEM)

für elasto-plastic problems

Discretization of the volume

Results

Boundary Element Method (BEM)

only for elastic problems

Only discretization of the boundary

Results on the boundary

Results in the inside

Comparison of computation time: FEM vs. BEM

The application of BEM enables a reduction of the computation time at the same accuracy for problems in elastostatics.
Calculation methods for the process design
Modeling of deep drawing using FEM/BEM-coupling

- Domain decomposition of the tool system in large BEM-domains for the ground tool parts
- FE-modeling of the active tool surfaces, which directly interact with the work piece and the work piece itself
- Advantages:
  - No necessity of a complex contact formulation between FEM and BEM
  - Usage of already availably sophisticated FEM contact formulations possible
- High resulting volume ration between FEM and BEM of 1:700
- Definition of the material behavior:
  - Elastic: Young's modulus and Poisson's ratio
  - Plastic: Combined isotropic/kinematic hardening according to Chaboche
  - Anisotropy according to Hill
Calculation methods for the process design

**BEM vs FEM**

Depending on the type of the problem, degree of accuracy required, and the available time to be spent on preparing and interpreting data, engineer should decide whether BE or FE method should be used.

<table>
<thead>
<tr>
<th>BEM</th>
<th>FEM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Suitable and more accurate for linear problems, particularly for three-dimensional problems with rapidly changing variables such as fracture or contact problems</td>
<td>More established and more commercially developed, particularly for complex non-linear problems</td>
</tr>
<tr>
<td>Due to reduced time needed it is suitable for preliminary design analyses, where geometry and loads can be subsequently modified with minimal efforts</td>
<td></td>
</tr>
<tr>
<td>Mesh generators and plotting as well as many load incrementation and iterative routines in non-linear problems developed for FE applications are directly applicable to BE problems</td>
<td></td>
</tr>
</tbody>
</table>
Calculation methods for the process design

Molecular Dynamics

- In the framework of Molecular Dynamics (MD) the material representation is realized by using arrangements of atoms or molecules considering their specific structure.

- MD simulation consists of the numerical, step-by-step, solution of the classical equations of motion, which for simple atomic system can be written as:

\[ m_i \ddot{r}_i = f_i \]

\[ f_i = - \frac{\partial}{\partial r_i} U \]

\( f_i \) are forces acting on the atoms, derived from a potential energy \( U(r^N) \)

\( r^N \) represents the complete set of \( 3N \) atomic coordinates
Advantage: detailed consideration of anisotropic, crystalline properties of the modeled material

Possibilities: investigation of microscopic structure, residual stress, deformations, phase transitions, and damages on the grain and intergranular level

Disadvantage: extremely large computing time required for realistic simulation, even when small domains and time periods are considered

Application:
- cutting
- contact with friction
- forming
- indentation
## Contents

1. Introduction

2. Calculation methods for the process design

3. Procedure of Finite Element Analysis
   - 3.1 Domain discretization
   - 3.2 Solution methods
   - 3.3 Element choice
   - 3.4 Determination of element equations

4. Non-linearities in FEM

5. Summary
Procedure of Finite Element Analysis

Phases of a Finite Element Analysis (FEA)

<table>
<thead>
<tr>
<th>Pre-processing</th>
<th>Calculation</th>
<th>Post-processing</th>
</tr>
</thead>
<tbody>
<tr>
<td>▪ discretizing the continuum</td>
<td>▪ solving the system of</td>
<td>▪ graphical representation</td>
</tr>
<tr>
<td></td>
<td>equations</td>
<td>▪ use for further processor</td>
</tr>
<tr>
<td>▪ selecting interpolation function</td>
<td></td>
<td>▪ results evaluation</td>
</tr>
<tr>
<td>▪ determining the element properties</td>
<td></td>
<td></td>
</tr>
<tr>
<td>▪ assembling the element equations</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
## Contents

1. Introduction
2. Calculation methods for the process design
3. Procedure of Finite Element Analysis
   - 3.1 Domain discretization
   - 3.2 Solution methods
   - 3.3 Element choice
   - 3.4 Determination of element equations
4. Non-linearities in FEM
5. Summary
Procedure of Finite Element Analysis
Domain discretization (1/2)

- Structured rectangular mesh for primitive geometry
- Free rectangular mesh for geometry with singularity (structured mesh is not applicable)
- Structured rectangular mesh for the region with high precision requirements
- Local mesh refinement improves solution precision with lower computation costs
Domain discretization with FEM (on the left) and BEM (on the right)

Source: Institut für Angewandte und Experimentelle Mechanik Universität Stuttgart: Boundary Element Methods-Introduction
In general case in non-linear FEM the load cannot be applied at one step, but should be applied step by step.

Reasons for the continuous load application:
- Material behavior (material properties depend on the loading history)
- Contact conditions are previously unknown

Loading path is divided into discrete steps → Loading path at an arbitrary point within the structure is unknown.

Implicit and explicit solution algorithms can be used to calculate state of the model at the intendent time points t.
Procedure of Finite Element Analysis

**Implicit solution method**

Equation of motion will be evaluated at initially unknown state \((t + \Delta t)\).

\[
K^{t+\Delta t}x = F^{t+\Delta t}_{\text{ext}}
\]

\(K\) is stiffness matrix

\(x\) is displacement vector

\(F_{\text{ext}}\) are external forces

**Principle:**

The simulated process is divided in single steps and the fundamental mathematical equations are solved in the way, that the work piece stays in static equilibrium at the end of each step.

By linear relations it is possible to perform a single solution of equations system during time step \(\rightarrow\) perfect for linear-dynamics or static problem definition.
**Implicit solution method**

Motion equation is evaluated at the known time step $t$.

\[ M^t \ddot{x} + C^t \dot{x} = F^t_{\text{ext}} \]

- $K$ is stiffness matrix
- $M$ is mass matrix
- $C$ is damping matrix
- $x$ is displacement vector
- $\ddot{x}$ acceleration vector
- $\dot{x}$ velocity vector
- $F_{\text{ext}}$ are external forces

Forming process is considered as dynamic process.

Calculation of target variables is done explicitly by means of differentiation scheme, under consideration of former displacement values.
Procedure of Finite Element Analysis

Comparison of explicit and implicit solution methods (1/2)

Source: CADFEM Wiki
## Procedure of Finite Element Analysis

### Comparison of explicit and implicit solution methods (2/2)

<table>
<thead>
<tr>
<th>Method</th>
<th>Procedure</th>
<th>Advantage</th>
<th>Problems</th>
<th>Application</th>
</tr>
</thead>
</table>
| Implicit| Solving of non-linear equilibrium conditions by means of an iterative method (e.g. Newton-Raphson) | • Time steps could be arbitrary big, they are limited only by the convergence behavior and precision requirements  
  • No limitations regarding meshing  
  • Extremely stable numerically | • At any time step a linear system of equations should be solved → computation time  
  • Convergence problems due to contact and instabilities (buckling, penetration) | In the most cases of bulk forming |
| Explicit| Integration of dynamic basic equations by means of central differences method, desired displacements can be calculated directly without iteration | • Solving of system of equations is not necessary  
  • No convergence problems | • Time step size is limited, which leads to a high number of required time steps → increase of computation time  
  • Spring backs (sheet forming) can be only hardly calculated | Sheet forming, highly dynamic applications (e.g. crash simulations) |
## Procedure of Finite Element Analysis

### Element Types

<table>
<thead>
<tr>
<th>1D</th>
<th>2D</th>
<th>3D</th>
</tr>
</thead>
<tbody>
<tr>
<td>Line</td>
<td>Triangle</td>
<td>Tetrahedron</td>
</tr>
<tr>
<td><img src="image1" alt="Line Diagram" /></td>
<td><img src="image2" alt="Triangle Diagram" /></td>
<td><img src="image3" alt="Tetrahedron Diagram" /></td>
</tr>
<tr>
<td>Line with additional node</td>
<td>Triangle with additional nodes</td>
<td>Tetrahedron with add. nodes</td>
</tr>
<tr>
<td><img src="image4" alt="Line with Additional Node Diagram" /></td>
<td><img src="image5" alt="Triangle with Additional Nodes Diagram" /></td>
<td><img src="image6" alt="Tetrahedron with Additional Nodes Diagram" /></td>
</tr>
<tr>
<td>Rectangle</td>
<td>Hexahedron</td>
<td>Hexahedron with add. Nodes</td>
</tr>
<tr>
<td><img src="image7" alt="Rectangle Diagram" /></td>
<td><img src="image8" alt="Hexahedron Diagram" /></td>
<td><img src="image9" alt="Hexahedron with Additional Nodes Diagram" /></td>
</tr>
</tbody>
</table>
# Procedure of Finite Element Analysis

## Element Type

<table>
<thead>
<tr>
<th></th>
<th>Solid (continuum) elements</th>
<th>Shell elements</th>
<th>Membrane elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D elements:</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Plane and axisymmetric problems</td>
<td>Solid (continuum) elements</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Generalized plane strain;</td>
<td></td>
<td>Shell elements</td>
<td></td>
</tr>
<tr>
<td>3D elements:</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Problems with 3D stress state formulation</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3D elements:</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Problems with 3D stress state formulation</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- Applied for plane stress problems
- Only 2 displacement degrees of freedom
- Rotation degrees depends on the element formulation

- 3 displacement degrees of freedom
- No rotation degrees of freedom
- Always defines plane stress state

Source: Abaqus Example Problems Guide
The interpolation functions serve to approximate the progress of the state variables within an element (e.g. displacement, temperature, pressure) from the exact values, calculated at nodes.

Often the polynomials are used for interpolation, according to this, interpolation functions can be classified as:
- linear
- quadratic
- cubic
Procedure of Finite Element Analysis

Linear interpolation Function

Example: one dimensional element of unit length

Two interpolation functions are defined:

\[ h_1(x) = 1 - x \quad \text{and} \quad h_2(x) = x \]

The element function is obtained by weighting the node values \( y_1 \) and \( y_2 \):

\[ y(x) = y_1 h_1(x) + y_2 h_2(x) \]

\[ y(x) = y_1 (1 - x) + y_2 x \]
**Procedure of Finite Element Analysis**

**Quadratic interpolation Function**

- Requires in the middle of the rod an additional node.
- There are represented 3 interpolation functions according to nodes.

\[
y(x) = h_1(x) + h_m(x) + h_2(x)
\]
Procedure of Finite Element Analysis

Cubic interpolation Function

- Require two additional nodes on the rod
### Summary on the Element Choice (1/3)

#### 2D elements can be used if:
- One dimension in the considered structure is significantly smaller than two others
- Stress within the thickness of the structure (perpendicular to the middle surface) is zero

<table>
<thead>
<tr>
<th>Element</th>
<th>Advantage</th>
<th>Disadvantage</th>
<th>Application</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shells (2D)</td>
<td>- Simple formulation</td>
<td>- Limited application area (only plane stress or strain state state)</td>
<td>- Simulation of sheet metal rolling processes</td>
</tr>
<tr>
<td></td>
<td>- Modelling of arbitrary geometrical forms in plane</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Membranes (3D surfaces)</td>
<td>- Simple formulation</td>
<td>- Transfers only axial forces in plane</td>
<td>- Sheet forming, when the bending can be neglected</td>
</tr>
<tr>
<td></td>
<td>- Simple definition of geometry</td>
<td>- No representation of bending moments is possible</td>
<td></td>
</tr>
</tbody>
</table>
## Summary on the Element Choice (2/3)

<table>
<thead>
<tr>
<th>Element</th>
<th>Advantage</th>
<th>Disadvantage</th>
<th>Application</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solid (3D)</td>
<td>- Complete description of the acting forces</td>
<td>- High computation times</td>
<td>- All structures, which should be represented without geometrical simplifications (e. g. tools, massive forming parts)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Should be used only in the case, when any other elements of smaller order could not be applied!</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Edge length ration should not be below 1:4</td>
<td></td>
</tr>
<tr>
<td>Shells (3D surfaces)</td>
<td>- Planar shapes can be discretized by means of smaller number of elements in comparison to solid elements</td>
<td>- In general case normal stress is neglected</td>
<td>- Simulation of sheet forming</td>
</tr>
<tr>
<td></td>
<td>- Better representation of moments distribution</td>
<td>- Thickness change is calculated by means of conservation volume assumption</td>
<td></td>
</tr>
</tbody>
</table>
Procedure of Finite Element Analysis

Summary on the Element Choice (3/3)

- First order elements are recommended when large strains or very high strain gradients are expected.

- If the problem involves bending and large distortions, use a fine mesh of first-order, reduced-integration elements.

- Triangles and tetrahedra are used to mesh a complex shape.

- Triangles and tetrahedra are less sensitive to initial element shape, whereas first-order quadrilaterals and hexahedra perform better if their shape is approximately rectangular.

- Hexahedral mesh provides the results with equivalent accuracy at less costs.
Contents

1 Introduction

2 Calculation methods for the process design

3 Procedure of Finite Element Analysis
   3.1 Domain discretization
   3.2 Solution methods
   3.3 Element choice
   3.4 Determination of element equations

4 Non-linearities in FEM

5 Summary
**Procedure of Finite Element Analysis**

**Continuum discretization**

- **Lagrange:**
  - Nodes and elements move with the material
  - Remeshing is necessary when elements distort
  - Suitable for simulations of instationary processes

- **Eulerian:**
  - Nodes and elements stay fixed in space
  - Material flows through a stationary mesh
  - No remeshing necessary
  - Suitable for simulation of stationary processes and fluid dynamics simulations

- **ALE (Arbitrary Lagrangian Eulerian):**
  - Combination of Lagrange and Eulerian Models
  - Reduction of mesh distortion by permitting material flow through the mesh but only within the object boundary shape
  - Suitable for stationary processes only
Procedure of Finite Element Analysis

**Determination of elements equations**

- **Equilibrium conditions**
  relation between stresses and forces, 3 partial differential equations with 6 unknown stresses

- **Kinematics**
  relation between distortions and displacements, 6 partial differential equations with 6 unknown distortions and 3 unknown displacements

- **Deformation law**
  relation between stresses and distortions, 6 algebraic equations

- Summary of field equations by means of principle of virtual displacements
Procedure of Finite Element Analysis

Solution of element equations

- Subdivision of domain on subdomains, so called “finite elements”, which are connected to each other by means of nodes
- Formulation of interpolation functions for every element
- Approximation of actual displacement fields with:
  \[ u_i = G_i u_M \]
  \( G_i \) is continuous interpolation function
  \( u_i \) components of node displacements
- After elements assembly we obtain the system of equations
  \[ Ku = f \]
  \( K \) is stiffness matrix
  \( u \) is displacement vector
  \( f \) is force vector
- Solution for \( u \) supplies with the kinematic relations \( \varepsilon \)
- By means of deformation law one can obtain stresses \( \sigma \)
Procedure of Finite Element Analysis

Effects consideration

General: the system of equations basically has the form (matrix notation):

\[ K^{t+\Delta t} \mathbf{x} = F_{ext}^{t+\Delta t} \]

**But:** inertia and damping effects are not considered in this formulation!

\[ K \text{ is stiffness matrix} \]
\[ \mathbf{x} \text{ is displacement vector} \]
\[ F_{ext} \text{ are external forces} \]

To consider dynamic problems mass and damping matrices should be additionally represented in system of equations:

\[ M^{t+\Delta t} \ddot{\mathbf{x}} + C^{t+\Delta t} \dot{\mathbf{x}} + K^{t+\Delta t} \mathbf{x} = F_{ext}^{t+\Delta t} \]

\[ M \text{ is mass matrix} \]
\[ \dot{\mathbf{x}} \text{ is acceleration vector} \]
\[ C \text{ is damping matrix} \]
\[ \ddot{\mathbf{x}} \text{ is acceleration vector} \]
Contents

1 Introduction

2 Calculation methods for the process design

3 Procedure of Finite Element Analysis

4 Non-linearities in FEM
   4.1 Geometrical non-linearities
   4.2 Material behavior non-linearities
   4.3 Non-linearities in boundary conditions

5 Summary
Non-linearities in FEM

Overview

- Every real physical process is non-linear, but in many cases it can be simulated as a linear one with the sufficient precision.

- Non-linearities, which could be identified in FEM:
  - geometrical non-linearities
  - material non-linearities
  - non-linearities in the boundary conditions

Source: 3D-Metal Forming BV
## Contents

1. **Introduction**

2. **Calculation methods for the process design**

3. **Procedure of Finite Element Analysis**

4. **Non-linearities in FEM**
   - 4.1 **Geometrical non-linearities**
   - 4.2 **Material behavior non-linearities**
   - 4.3 **Non-linearities in boundary conditions**

5. **Summary**
Non-linearities in FEM

Geometrical non-linearity (1/2)

Problem of large deformations

- Due to large deformations can occur inadmissible mesh distortion, which influence unfavorable on the precision of the computation, also element failure is possible

- Example: Ring crush test
  In the initial state node 2 is situated on the lateral surface, after forming on the front surface. This element is inadmissible!
In this case remeshing is required

- stop the calculation
- remesh deformed geometry with regular elements
- replace field variables (displacements, strains, stresses, etc.) from old mesh to the new one
- continue the calculation

Criteria for the remeshing

- element geometry
- stress gradient
- amount of distortions

In many FE-Programs remeshing is automatized

Source: Transvalor
Contents

1 Introduction

2 Calculation methods for the process design

3 Procedure of Finite Element Analysis

4 Non-linearities in FEM
   4.1 Geometrical non-linearities
   4.2 Material behavior non-linearities
   4.3 Non-linearities in boundary conditions

5 Summary
Non-linearities in FEM

Non-linear material behavior

Material models for large strains

Material models without elastic material behavior

- Ideal plastic
- Viscoplastic

Material models with elastic material behavior

- Elastic-plastic
- Elastic-viscoplastic

Material models with elastic material behavior

Material models without elastic material behavior

Material models for large strains
At small strains, a body deforms purely elastic (reversible).

Increase of the elastic line is described by Young’s Modulus $E$.

Yield Stress $R_{eS}$:
- Elastic strain limit of the material
- Without macroscopic plastic deformation of the tensile specimen
Non-linearities in FEM

Analytical representation of flow curves

**Hollomon: Power law**

\[ k_f = C \cdot \varphi_g^n \]

**Swift: generalized power law**

\[ k_f = C \cdot (D + \varphi)^n \]

\( n \) is strain-hardening exponent

\( C, D \) are material dependent parameters

**Spittel** Additionally considers so-called thermodynamic parameters

\[ k_f = \sigma_F = \sigma_{F_0} \cdot K_T \cdot K_\varphi \cdot K_\varphi \]

\( \sigma_{F_0} \) is fundamental parameter of flow stress defining forming conditions

\( K_T \cdot K_\varphi \cdot K_\varphi \) are correction functions for temperature-, forming rate- and forming velocity flow
Non-linearities in FEM

Yield criteria

- Shear stress hypothesis by Tresca

\[ \sigma_2 = \sigma_3 = 0 \]

\[ \frac{\sigma_1}{A} = k_f = 2k \Rightarrow k = \frac{k_f}{2} \]

with \( \tau_{max} \Rightarrow \sigma_1 - \sigma_3 = k_f \)

\[ k_f = \frac{1}{2} \max(|\sigma_1 - \sigma_2|, |\sigma_2 - \sigma_3|, |\sigma_3 - \sigma_1|) \]

- Form change – Energy hypothesis by von Mises

\[ k_f = \sqrt{\frac{1}{2} \left[ (\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \right]} \]
Non-linearities in FEM

Yield locus curve

For $\sigma_2 = 0$ (plane stress state) the representation of the yield hypothesis in the plane $\sigma_1\sigma_2$ is called yield locus curve.

- Stress points laying on the curves fulfill the corresponding yield condition
- Stress points laying inside the curve does not fulfill the yield condition and correspond to elastic behavior
- Stress points laying outside the curves are not possible
Non-linearities in FEM

Strain-hardening behavior

Example:

von Mises yield criteria

\[ k_f = \sqrt{\frac{1}{2} [(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]} \]
Non-linearities in FEM

Macromechanical damage criteria

<table>
<thead>
<tr>
<th>Damage criterion</th>
<th>Effective strain</th>
<th>Forming process</th>
<th>Cylinder specimen</th>
<th>Collar specimen</th>
</tr>
</thead>
<tbody>
<tr>
<td>Norm. Cockcroft &amp; Latham</td>
<td>Non-linearities in FEM</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- **Cylinder specimen**
  - \( h_0 = 45 \text{ mm} \)
  - \( h_{\text{crack}} = 8.2 \text{ mm} \)
  - \( D_{\text{max}} = 1.17 \)
  - \( D_{\text{max}} = 0.483 \)

- **Collar specimen**
  - \( h_0 = 43 \text{ mm} \)
  - \( h_{\text{crack}} = 22.5 \text{ mm} \)
  - \( D_{\text{max}} = 1.37 \)
  - \( D_{\text{max}} = 0.334 \)
Non-linearities in FEM

Micromechanical damage criteria

Forming process

<table>
<thead>
<tr>
<th>Damage criterion</th>
<th>Cylinder specimen</th>
<th>Collar specimen</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mod. Rice &amp; Tracey</td>
<td>Crack $D_{\text{max}} = 0.522$</td>
<td>Crack $D_{\text{max}} = 0.754$</td>
</tr>
<tr>
<td>Mc Clintock</td>
<td>Crack $D_{\text{max}} = 0.73$</td>
<td>Crack $D_{\text{max}} = 0.424$</td>
</tr>
<tr>
<td>Future damage criterion</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

© WZL/Fraunhofer IPT
Non-linearities in FEM

Cause of material separation

- Void nucleation starts at inclusions

- Mechanisms of void nucleation:
  - Fracture or decohesion of inclusions
  - Nucleation due to decohesion at the grain border
  - Initiation of cracks at existing voids

### Void nucleation
- \( \sigma_Z \)

### Void growth
- \( \sigma_Z \)

### Void coalescence

#### Tensile test
- Decohesion

#### Crush test
- Inclusions of MnS

© WZL/Fraunhofer IPT
Non-linearities in FEM

Cause of material separation

- Numeric prediction of formability through damage criteria
- Time-independent criteria do not consider the stress history of the material
- Time-dependent criteria are based on the calculation of energy
- Formability is reached for:

\[ D = D_{\text{crit}} \]

<table>
<thead>
<tr>
<th>Damage criteria</th>
<th>Calculation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normalized max. principal normal stress</td>
<td>[ D = \frac{\sigma_1}{k_f} ]</td>
</tr>
<tr>
<td>Effective stress</td>
<td>[ D = \sigma_v ]</td>
</tr>
<tr>
<td>Ayada</td>
<td>[ D = \int_0^{\varphi_{\text{V,br}}} \left( \frac{\sigma_m}{\sigma_v} \right) d\varphi_v ]</td>
</tr>
<tr>
<td>Norm. Cockroft and Latham, mod. Oh</td>
<td>[ D = \int_0^{\varphi_{\text{V,br}}} \max\left( \frac{\sigma_1}{\sigma_v}, 0 \right) d\varphi_v ]</td>
</tr>
<tr>
<td>Rice und Tracey</td>
<td>[ D = \int_0^{\varphi_{\text{V,br}}} \exp\left( \frac{3}{2} \cdot \frac{\sigma_m}{\sigma_v} \right) d\varphi_v ]</td>
</tr>
<tr>
<td>Oyane</td>
<td>[ D = \int_0^{\varphi_{\text{V,br}}} \left( 1 + A \cdot \frac{\sigma_m}{\sigma_v} \right) d\varphi_v ]</td>
</tr>
</tbody>
</table>

Source: Dissertation A. Timmer
<table>
<thead>
<tr>
<th></th>
<th>Contents</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Introduction</td>
</tr>
<tr>
<td>2</td>
<td>Calculation methods for the process design</td>
</tr>
<tr>
<td>3</td>
<td>Procedure of Finite Element Analysis</td>
</tr>
<tr>
<td>4</td>
<td>Non-linearities in FEM</td>
</tr>
<tr>
<td>4.1</td>
<td>Geometrical non-linearities</td>
</tr>
<tr>
<td>4.2</td>
<td>Material behavior non-linearities</td>
</tr>
<tr>
<td>4.3</td>
<td>Non-linearities in boundary conditions</td>
</tr>
<tr>
<td>5</td>
<td>Summary</td>
</tr>
</tbody>
</table>
Non-linearities in FEM

Non-linear boundary conditions

- time dependent forces
- time dependent displacements
- contact problems (are very important for forming technique)
  Example: Herz‘ compression of two spheres
Non-linearities in FEM

Contact problem (structural non-linearity)

- **Contact mechanics** is the study of the deformation of solids that touch each other at one or more points.

- Contact depends on the displacement of the parts relatively each other, therefore on structural degrees of freedom, leading to necessity of iterative solution.

- By means of contact other physical fields can be transmitted, such as temperature of electrical field.

- **Takes place in forming:**
  - contact between a part and a tool
  - contact between two deforming bodies

Contact is open
Contact is closed (sticking)
Contact is closed (sliding)
Non-linearities in FEM

Contact definition in FEM simulation

Contact between two bodies can be defined:

- one surface is defined as master (target) surface of elements, representing the limit of admissible movement domain
- other surface is defined as slave (contact) one on the nodes (Gauss points)

At every iteration it is checked if slave surface is situated in admissible movement region (distanced from the master surface → open contact) or in inadmissible movement region (attached or penetrated to the target surface → closed contact)

Source: FEM-Formelsammlung Statik und Dynamik, L. Nasdala 2010
Non-linearities in FEM
Remeshing during contact simulation

- Strong mesh refinement is required especially at unknown edges of contact zones

Source: L. Sun, H. Proudhon, G. Cailletaud, 2011
Contents

1 Introduction
2 Calculation methods for the process design
3 Procedure of Finite Element Analysis
4 Non-linearities in FEM
5 Summary
Summary

- For process design different plasticity-theoretical approaches are used.
- State of the art in terms of the design of real industrial processes is the application of the Finite Element Method (FEM).
- Solution of metal forming problems using FEM requires the description of the material behavior through material models.
- A coupling of FEM and BEM can reduce the computation time for calculations including tool behavior.
- During the process design FEM is inter alia used to predict the formability.
- Ways to describe the formability are inter alia damage criteria and formability graphs.
Backup: New Agenda